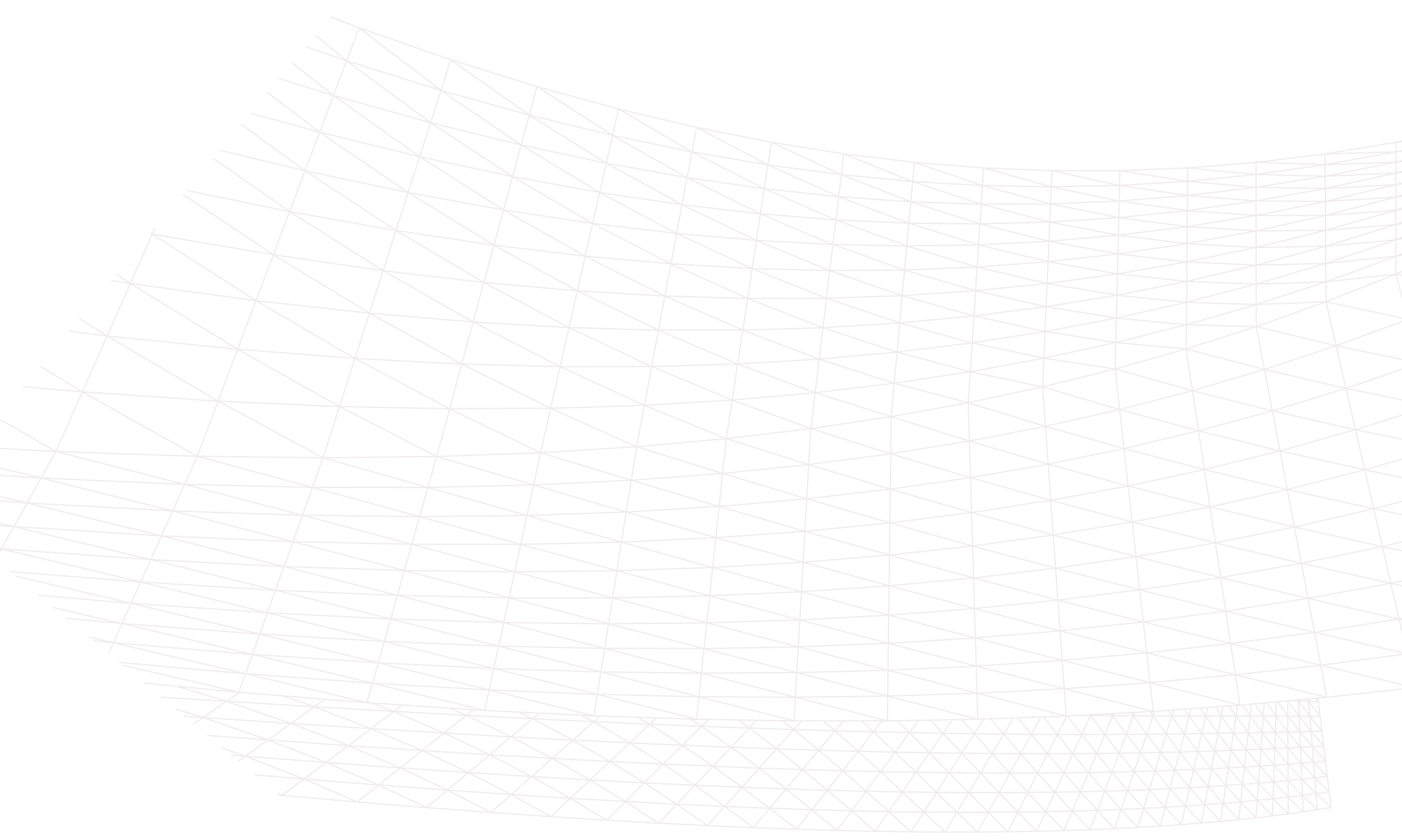


Archicad 27 Reference Guide



Graphisoft®

Visit the Graphisoft website at www.graphisoft.com for local distributor and product availability information.

Archicad 27 Reference Guide

Copyright © 2024 by Graphisoft, all rights reserved. Reproduction, paraphrasing or translation without express prior written permission is strictly prohibited.

Trademarks

Archicad® is a registered trademark of Graphisoft.
All other trademarks are the property of their respective holders.

Contents

Getting Help on Archicad 27	48
License agreement & Trademarks	50
Install Guide for Archicad 27	51
Install Archicad	52
MacOS: Enable Full Disk Access	58
Your Archicad License Key	61
Add/Repair Archicad	62
Updates for Archicad	63
Archicad Tools and Utilities	64
Uninstall Archicad	65
Using a CodeMeter Server	66
Network Installation for BIM Managers	68
Command Line Parameters (Custom Install)	69
Option File (Custom Install)	72
Installer Log Files and Error Codes	73
Unattended Uninstall	74
Third-Party Prerequisites	75
Customized Company Defaults for Unattended Installation (Archicad Only)	76
New Features in Archicad 27	79
Integrated Design Options	80
Enhanced attribute management	83
Tool and workflow improvements	85
Distance Guides	86
Extended search capabilities	87
Custom groups for Graphic Override Rules	88
Improved format options for Interactive Schedule	89
Modernized GDL Editor	91
Additional workflow improvements	93
Visualization	95
Multi-disciplinary design	97
Streamlined Load input and edit	98
Manual adjustment of Structural Analytical Model	100
Detailed Structural Change list in Model Compare	101
MEP Modeler	102
Open BIM	103
File format - Open and save versions Archicad 17 and up	104
BIMcloud - Secure Teamwork login	105

Migrating to Archicad 27	106
Migrating MEP elements	107
Migrating Libraries	108
Migrating a Teamwork Project	112
Update DXF-DWG Translators	115
Update IFC Translators	116
Drawings with Resized Frames are Cropped	117
Replace Curtain Wall Frames/Panels	118
No “Custom” Model View Options Combinations	119
Stair Favorites May Be Reset	120
Stair Tool Replaces StairMaker	121
ID-based Labels with Autotext	122
New “3D Styles” View Option	123
Model View Fill Options Replaced By Graphic Overrides	124
Guide Lines and Snap Points	125
Migration of Lightworks Settings to Cineware	126
Migration of Lights and Lamp Settings	129
Images from QuickTime Formats	130
Element Display in Interior Elevations	131
Building Materials: Collision Detection Setting	132
Configuration	133
Start Archicad	134
Create New Project	135
Open Recent Project(s)	137
Browse for a Project	140
Open & Repair Selected File	141
Opening Legacy or Archive Projects: Manage Library Parts	142
Close Project	144
Save Archicad Project	145
File Compression	147
Bug Reporter	148
Work Environment	149
Default Work Environment Profiles	150
Arranging Palettes, Toolbars and Windows	152
Set Window Background and Grid Line Color	155
Tab Bar	156
Transfer Settings Among Tabs	162
Quick Options Bar	164
Toolbox	166
Info Box	168
Menus	169

Toolbars	170
Shortcuts	171
Palettes	172
Pet Palettes	173
Template Files	175
Backup Files	176
Archive Projects	177
Archicad User Feedback - Net Promoter Score Survey	180
Create Support Package (Troubleshooting)	182
Opening Projects through a Network	184
Add-ons and Goodies	185
Working Units	187
Location Settings	189
Survey Point	193
Survey Coordinate Object	196
Project Preferences	198
Scale	200
Libraries	202
Library Manager	203
Embedded Library	212
Add Objects to Embedded Library	214
Manage Embedded Objects	219
Linked Libraries	221
BIMcloud Libraries	223
Add BIMcloud Library	224
Manage BIMcloud Libraries	226
Remove Library from Project	231
Recommendations for Using Libraries	233
Troubleshooting Library Issues	234
Library Container File	242
Favorites	243
Apply a Favorite	244
Search Favorites	248
Create New Favorite	249
Redefine Favorite	251
Edit Settings of One or More Favorites	253
Missing Content in Favorite	255
Favorites Palette	256
Import/Export Favorites	260
Licensing and sign-in	262
Archicad license types	263

Sign in with Graphisoft ID	266
Log in to BIMcloud	268
License troubleshooting	271
No license found - when starting Archicad	272
No license found - while using Archicad	273
Problem with serial number	274
License Information dialog	276
Using Archicad on multiple devices	279
Attributes	280
Layers	281
Change element's layer	282
Layers dialog	283
Layer status	285
Layer Combinations	287
Quick Layers	290
Use layers to prevent intersections	291
Use separate layer settings for the Layout Book	293
Place all elements on a single "active layer" (Simulate AutoCAD work methods)	294
Lines	295
Edit Line Type (Dashed)	297
Edit Line Type (Symbol)	298
Fills	299
Fill Categories	300
Edit Vector Fill	302
Create/Edit Solid Fill	303
Create/Edit Symbol Fill	304
Create Image Fill	308
Fill Display Mode: True Fill Appearance vs. Bitmap	311
Building Materials	312
Edit and manage Building Materials	313
Intersection Priority (Building Material)	316
Building Material Classification and Properties	318
Environmental Properties	320
Composite Structures	323
Assign composite to wall, slab, roof or shell	324
Edit and manage composites	325
Pen Sets	331
Apply Pen Set	332
Redefine Pen Set	335
Edit and manage Pen Sets	336
Save Pen Set in STB Format	339

Surfaces	340
Edit and manage surfaces	341
Surface Exposure to Light settings (Basic Engine)	344
Surface Cover Fill (Basic Engine)	346
Texture settings (Basic Engine)	347
Get more surfaces	351
Add new surface from catalog	352
Replace surface settings from catalog	353
Search for surfaces in catalog	354
Additional Surface Catalog for SSA/Forward Users	356
Load Cinema 4D material	357
Match surface settings: Cineware and Basic	359
Texture files - naming conventions	361
Align 3D Texture	362
Default Texture Alignments/Reset Texture	365
Override surface in element settings	366
Override Surfaces With Surface Painter	367
Surface Painter Palette	373
Search for Surface (Surface Painter)	377
Profile Manager	379
Edit Complex Profile	382
Save Named Profile	385
Design Layers Panel	386
Modifiers Panel	389
Components Panel	390
Uniform Contours Panel	391
Capture Selected Element as Custom Profile	392
Apply Custom Surface or Line to Profile Edge	393
Profile Origin in Profile Editor Window	394
Stretch Modifiers	396
Offset Modifiers	398
Create New Offset Modifier - Example	400
Edit Offset Modifier	407
Offset Modifiers: Tips and Tricks	414
Using the attribute dialogs	416
Attributes palette	428
Search and sort attributes (Attributes palette)	430
Manage attributes and folders (Attributes palette)	432
Edit attribute settings (Attributes palette)	436
Import attributes	438
Export attributes	442
Purge attributes	444

Delete attributes	445
Missing attributes	449
Find and Select elements with missing attributes	450
Custom attributes of GDL objects	452
Attribute Manager	453
Attribute Management Commands	457
Revert Changes in Attribute Manager	463
Interaction	464
Navigation in Archicad	465
Navigator	468
Navigator Project Map	472
Navigator View Map	474
Organizer	478
Clone a Folder in the View Map	482
Pop-up Navigator	484
Zoom	486
Fit in Window	489
Set Orientation	490
Navigator Preview (2D)	494
Navigation in the 3D Window	497
Accessing 3D Navigation Commands	498
Explore Model (3D Navigation)	499
Orbit (3D Navigation)	501
Navigator Preview (3D)	502
3D Connexion Enabler	505
Origins	506
Elevation Values in the Tracker	509
Selection	510
Selecting Elements	511
Special Selection Commands	513
Deselect Elements	518
Selection Highlights	519
Draw Selection Area	520
Selection Dots	522
Find and Select Elements	524
Define Element Criteria	527
Predefined Criteria Sets	531
Edit Selection Set	534
Store Selection Sets in Selections Palette	535
Marquee Area	536
Marquee Area Definition	538
Save Marquee Area as PDF or DWG	539

Copy/Paste Marquee Area from Project Window	540
Other Editing Operations within the Marquee	542
On-Screen Input Aids	543
The Intelligent Cursor	544
Tracker	546
Ruler	551
Measure Tool	552
Set up Construction Grid	553
Set up Snap Grid	556
Snap to Grid	557
Snap Guides	559
Guide Lines	563
Snap Points	568
Snap Points on Temporary Vector	573
Editing Plane in 3D Window	574
Editing Plane Display	575
Reposition Editing Plane	576
Magic Wand	580
Magic Wand Settings	585
Gravity	586
Mouse Constraints	588
Coordinate Constraints	590
Relative Construction Methods	591
Parallel and Perpendicular Constraints	592
Angle Bisector Constraint	593
Offset and Multiple Offset Constraints (Relative Construction Methods)	594
Aligning Elements to a Surface in 3D	595
Snapping to Existing Elements	596
Cursor Snap Range	598
Distance Guides - Experimental feature	599
Editing Elements	603
Basic Editing Operations	604
Cancel Operations (ESC Key, Backspace)	607
Move Elements	608
Nudging Elements	609
Dragging Elements	610
Rotating Elements	611
Mirroring Elements	613
Elevating Elements	615
Offset All Edges	617
Intersect Two Elements	618
Align Elements	619

Distribute Elements	623
Special Distribute	626
Modify Element Sizes	627
Reshaping Polygons and Chained Elements	628
Stretching with the Marquee Tool	631
Stretch Height	632
Resize (Enlarge or Reduce) Elements	633
Splitting Elements	635
Adjusting Elements	637
Adjust Elements to Slabs	639
Create a Fillet or Chamfer	645
Trim Elements Using Scissors	646
Adding Element Nodes	647
Move Nodes	648
Curve/Straighten Element Edge	649
Explode into Current View	650
Duplicate Elements	653
Drag, Rotate, Mirror Element Copies	654
Multiply Elements	656
Drag & Drop	666
Parameter Transfer	668
Element Transfer Settings	672
Group Elements	676
Lock/Unlock Elements	679
Display Order	680
Virtual Trace: Using References to Edit and Compare Model Views and Drawings	683
Choose a Trace Reference	686
Show/Hide Trace Reference	689
Move/Rotate/Reset Reference	691
Switch Reference with Active: Access Elements within the Reference for Editing or Copying	692
Activate Reference	693
Rebuild Trace Reference	694
Compare Reference with Active	695
Consolidate Lines and Fills in Drawing Windows	699
Check for Linear Overlaps - Experimental Feature	701
Elements of the Virtual Building	706
Construction Elements	707
Place a Construction Element in Archicad	708
Home Story	709
Define Toplink for Wall, Column, Zone or Stair	714
Walls	716
Wall Reference Line	718

Create a Straight Wall	723
Create a Curved Wall	724
Create a Chain of Walls	725
Create a Rectangle of Walls	726
Create a Trapezoid Wall	727
Create a Polygon Wall (PolyWall)	729
Stretching Walls	731
Create a Slanted or Double-Slanted Wall	733
Modify Slanted Wall	735
Create a Log Wall	736
Walls and Other Elements	737
Modify Wall Reference Line	738
Invert Wall Direction	740
Columns	741
Place a Column	742
Create a Slanted Column	744
Create a Multi-Segment Column	745
Delete Column Segment or Beam Segment	747
Define Column/Segment Structure	748
Rotate Join Angle	752
Tapered Column or Segment	754
Editing Columns	756
Edit Mode for Multi-Segment Columns	758
Core and Veneer	760
Columns and Other Elements	762
Column Tool Settings: Preview and Positioning	766
Column End Cuts and Reference Axis	769
Column Model Display	771
Column Display on Floor Plan and Section	772
Beams	775
Place a Beam	776
Create a Slanted Beam	779
Curve a Beam Vertically	781
Create a Multi-Segment Beam	783
Define Beam/Segment Structure	785
Define Join Geometry	789
Tapered Beam or Segment	790
Editing Beams	791
Create a Hole in a Beam	793
Selecting and Modifying Beam Holes	794
Beams and Other Elements	797
Beam Tool Settings: Preview and Positioning	799

Beam End Cuts	802
Beam Reference Axis	803
Beam Model Display	805
Beam Display on Floor Plan and Section	806
Rotating Beam and Columns	809
Complex Profiles	810
Apply Profile to Element	813
Create Complex Profile from Parallel Elements	815
Use Standard Steel Column or Beam Profile	817
Roofs	820
About Roofs	821
Create a Single-plane Roof on the Floor Plan	824
Create a Single-plane Roof in the 3D Window	826
Create a Multi-plane Roof	828
Examples of Multi-plane Roofs	830
Hip Roof	832
Complex Roof	833
Gable Roof	834
Pyramidal Roof	836
Mansard Roof	837
Gablet Roof	839
Gambrel Roof	841
Cross-Gable Roof	843
Editing Roofs	846
Graphical Editing of Roof Geometry	847
Modify the Roof Pitch	849
Add Roof Level	851
Edit Roof Levels	852
Edit Eaves or Gable Overhang	853
Edit Roof Contour	855
Edit Roof Ridge	856
Customize Roof Plane	858
Customize Edge of Roof or Roof Hole	860
Create a Hole in a Roof	863
Create an Atrium	864
Add a Tower to the Roof	865
Create Roof Level Lines	867
Intersect Single-plane Roofs	868
Crop Elements to Single-plane Roof	870
Create Special Roof Objects with RoofMaker	876
Trim Elements to Roof or Shell	882
Define Trimming Bodies	883

Trim Multiple Selected Elements: Automatic Trim	885
Automatic Trim: Example 1	887
Automatic Trim: Example 2	889
Trim Elements to Roof: Manual Trim	890
Merge Trimming Elements	894
Merge Elements: Roofs, Shells, Morphs	896
Managing Element Connections	898
Shells	901
About Shells	902
Extruded Shells	904
Simple Extruded Vault: Parallel to the Floor Plan	905
Simple Extruded Vault: Perpendicular to the Floor Plan	906
Extruded Shell with Freely Defined Profile	908
Revolved Shells	911
Revolved Shell: Simple Input	912
Revolved Shell with Freely Defined Profile	913
Ruled Shells	917
Ruled Vaulted Shell	919
Sample Workflows for Shell Creation	922
Extruded Shell: Wavy Canopy Roof	923
Revolved Shell: Arced Triangular Roof Complex	925
Revolved Shell: Define Contours in Section	929
Ruled Shell: Tennis Court Shell	932
Ruled Shell: Twisted Tower	936
Graphical Editing of Shells	937
Options for Editing Extruded Shells	938
Options for Editing Revolved Shells	943
Options for Editing Ruled Shells	946
Create Hole in Shell	949
Edit Shell Hole	951
Define Shell Contour	955
Edit Shell Contour Polygon	956
Customize Edge of Shell or Edge of Shell Hole	959
Slabs	962
Slab Reference Plane	963
Creating Slabs	968
Placing Holes in Slabs	969
Set Custom Slab Edge Angle and Edge Surface	971
Meshes	974
Create a Mesh	975
Edit Elevation of a Mesh Point	977
Add New Points to the Mesh	978

Create a Hole in the Mesh	979
Display of Meshes	980
Zones	982
Creating Zones	983
Edit Zone Polygon	988
Zone Categories	991
Display of Zones in Floor Plan	992
Display of Zones in Section	994
Zone Stamps	1000
3D Display of Zone Spaces	1002
Crop Zone to Another Element	1003
Calculating Zone Area and Zone Volume	1004
Relation to Zones	1005
Updating Zones	1009
Curtain Wall	1011
Working with Curtain Walls: Overview	1012
Curtain Wall Components	1014
Curtain Wall Frames	1018
Curtain Wall Frame Settings	1028
Curtain Wall Panels	1033
Place Door- or Window-Type Panel in Curtain Wall	1034
Curtain Wall Panel Settings	1038
Curtain Wall Scheme Settings	1044
Set up Grid Pattern in Scheme Settings	1051
Create Grid Pattern Graphically	1062
Curtain Wall Input Methods	1068
Create Curtain Wall on the Floor Plan	1069
Draw Curtain Wall Boundary in the Section Window	1070
Create a Curved Curtain Wall: Geometry Methods	1071
Create Curtain Wall in the 3D Window	1072
Curtain Wall Input Plane in 3D Window	1074
Extrude Curtain Wall from a Slanted Input Plane	1076
Positioning Curtain Wall in Section	1081
Model View Options for Curtain Walls	1083
Curtain Wall Edit Mode	1086
Edit Grid and Grid Lines in Edit Mode	1091
Graphical Editing of Curtain Wall	1095
Add Additional Curtain Wall Boundary	1097
Curtain Wall Junctions	1098
Curtain Wall Accessories	1101
Curtain Wall System Settings	1103
Curtain Walls and Other Elements	1110

Stairs	1115
About the Stair Tool	1116
Stair Input	1117
Graphical Editing of Stair Scheme	1122
Stair Turning Type	1128
Winder Types for Equal-Going Winders	1133
Stair Segment Type	1135
Stair Tool Edit Mode	1139
Automatic “Solver” Options to Resolve Geometric Conflicts	1146
Stair Settings (Main Element Settings)	1147
Stair Settings: Geometry and Positioning	1148
Stair Settings: Rules and Standards	1154
Toplinked Stairs and Changed Story Height	1158
Tread and Riser Finishes	1160
Flight and Landing Structures	1170
Monolithic Flight and Landing Structures	1172
Beam Flight Structure	1177
Beam Landing Structure	1185
Cantilevered Flight and Landing Structure	1195
Stringers Flight and Landing Structure	1198
Split Stair Structure	1204
Custom Sub-Elements for Stairs and Railings	1207
Stair Endings and Connections	1212
Model View Options for Stairs	1217
Detail Level of Stair and Railing Symbols	1221
Floor Plan/Reflected Ceiling Plan Display (Stair Settings)	1222
Edit Stair Symbol Components Graphically	1233
Railings	1235
About the Railing Tool	1236
Place Railing: Basic Polyline Input	1238
Place Associative Railing	1239
Working with Railing Segments and Nodes	1244
Railing Reference Line	1250
Railing Tool Edit Mode	1252
Railing Settings: Overview	1253
Railing Settings: Geometry and Symbol Display	1255
Segment Settings (Railing Tool)	1263
The Pattern Editor (Railing Tool Settings)	1264
Segment Settings (Page of Railing Tool Settings)	1269
Segment Pattern Settings (Panel of Railing Tool)	1272
Toprail Settings (Railing Tool)	1275
Handrail Settings (Railing Tool)	1277

Rail Settings (Railing Tool)	1280
Rail Component Settings (Toprail/Handrail/Rail)	1282
Inner Post Settings (Railing Tool)	1285
Baluster Settings (Railing Tool)	1288
Post Component Settings (Posts/Inner Posts/Balusters)	1293
Panel Settings (Railing Tool)	1296
Node Settings (Railing Tool)	1299
Post Settings (Railing Tool)	1307
Connection Settings (Toprail, Handrail, Rail)	1308
Railing Ends (Toprail, Handrail, Rail)	1312
Model View Options for Railings	1316
Morphs	1318
Morph Selection	1320
Create a 2D Morph	1323
Create a Box Morph	1324
Create a Revolved Morph	1325
Create Morph from Existing Elements	1329
Basic Morph Editing	1330
Basic Morph Editing: Push-Pull	1331
Basic Morph Editing: Bulge	1333
Basic Morph Editing: Add	1334
Basic Morph Editing: Tube	1335
Basic Morph Editing: Offset Morph Edge(s)	1337
Basic Morph Editing: Curve Edge	1339
Basic Morph Editing: Move Morph Sub-Element	1341
Basic Morph Editing: Add Node to Morph	1342
Basic Morph Editing: Fillet/Chamfer a Morph	1343
Useful Commands for Transforming Morphs	1346
Transforming Morphs: Smooth & Merge Faces	1347
Intersect Morph Faces (Generate Edges within a Morph)	1349
Transforming Morphs: Curve & Merge Edges	1350
Combine Morph Faces for Smooth Editing Effect: Use “Soft” Edges	1351
Transforming Morphs: Box Stretch a Morph	1353
Transforming Morphs: Cover with Faces	1355
Transforming Morphs: Custom Face Settings	1356
Morph Edge Type: Hard, Hidden, Soft	1358
Boolean Operations with Morphs	1361
Morph Texture Mapping and Alignment	1364
Fix Modeling Errors: Check and Solidify Morphs	1368
Openings	1370
Create Opening	1372
Opening Display on Floor Plan	1378

Opening Display in 3D	1385
Use Limited Opening to Create Niche or Recess	1387
On-Screen View Options for Openings	1390
Edit Opening	1392
Manage Opening Connections	1394
Edit Openings in Teamwork	1398
Import/Save Openings with IFC	1399
Create Openings from Selection	1400
Model Element Connections	1403
Element Intersections	1404
Basic Intersection Principles	1405
Specific Element Intersections and Junctions	1407
How to Make Sure Elements Collide for Best Cleanup Results	1415
Intersecting Elements That Don't Clean Up	1417
Best Practices for Intersections: Conceptual Design Phase	1418
Fine-Tune Intersections	1420
Legacy Intersection Mode for Older-Version Projects	1421
Parametric Objects	1422
About Parametric Objects	1423
Where to Find Objects for Use in Archicad	1424
BIM Components	1425
Search for Library Part	1429
Placing an Object	1435
2D and 3D Representation of Objects	1437
Stretching Objects	1440
Parameter Transfer Between Objects	1442
Graphical Editing Using Editable Hotspots	1444
Create Patch Object	1446
About GDL Object Subtypes	1447
Create Custom Library Parts and Components	1449
Save 2D Symbols as Objects	1450
Save 2D Elements as Label	1451
Save 3D Elements as Objects	1454
Save Rectangular Doors and Windows from 3D Model Elements	1458
Save Custom-Shape Doors or Windows	1460
Create Custom Components for Doors, Windows, Curtain Wall, Stair, Railing	1463
Create Custom Components: Example Workflows	1465
Example: Create Custom Door Leaf	1466
Example: Create Custom Frame or Cap Profile for Curtain Walls	1469
Editing Custom Library Parts and Components	1471
Open and Edit Library Part	1472
Scripting Custom Objects	1474

Further Resources on GDL Object Creation	1475
Dedicated Object Tools: Doors, Windows, Skylights, Wall Ends	1476
Doors/Windows	1477
Placing Doors or Windows	1479
Previews in Door/Window Settings	1482
Floor Plan Display of Doors/Windows	1483
Setting the Window/Door Plane in Slanted or Complex Walls	1484
Sill or Header Heights	1486
Sill Height Using Subfloor Thickness	1487
Anchor Reveal	1489
Creating an Empty Opening	1491
Moving Wall Openings	1493
Doors/Windows in Curved Wall	1495
Corner Windows	1496
Skylights	1498
Placing and Moving a Skylight	1500
Skylight Constraint Relative to Roof	1503
Wall Ends	1504
Ramp Object	1505
TrussMaker	1509
Creating Trusses in the Floor Plan	1510
Editing Trusses	1511
Creating Trusses in Section/Elevation/IE Windows	1512
Solid Element Operations	1514
About Solid Element Operations	1515
Solid Operation Types	1517
Solid Operation Example	1519
Managing Solid Element Connections	1523
Properties and Classification Systems	1524
Assign Classification	1526
Assign Custom Property Value	1529
Default Property Value	1531
Property Manager	1534
Property Availability for Classifications	1541
Assign Classifications/Properties in Element Schedule	1545
Show Property Data in Labels and Markers	1546
Expression-Defined Element Properties	1547
Create a New Expression-Defined Property: Step By Step	1548
Using Expression Editor: Context-Sensitive Help	1550
Evaluate Expression-Defined Property	1552
Expression-Defined Properties: Examples	1554
Example 1: Simple Mapping	1555

Example 2: Modify Element Parameter, Using an Operator	1557
Example 3: Expand Property with Descriptive Text String	1559
Example 4: Calculate a Quantity and Round Up	1563
Example 5: Calculate a Length and Add Overhang	1564
Example 6: Round Up to Standard Beam Length	1565
Example 7: Display Regulation-Compliance Data for an Element	1566
Example 8: Generate Expanded ID for Openings	1571
Example 9: Calculate Number of Tiles to Purchase	1572
Example 10: List the Total Cost of a Structure in Dollars	1575
Errors in Expressions and How to Fix Them	1577
Result Conflicts with Data Type	1578
All Arguments Must Use the Same Data Type and Compatible Units	1581
Argument's Data Type is Not Correct	1582
Data Type Not Available in Archicad	1585
Mathematical Error	1586
Incorrect Sign and/or Number Type	1587
Data Type Cannot be Changed	1588
Does Not Contain Numeric Characters	1589
Properties in Hotlinks	1590
Export/Import Properties Between Projects	1592
Classification Manager	1595
Library Part Maker	1599
Library Part Maker Palette	1600
Library Part Maker: Detailed workflow	1603
Define Name & Design Area	1605
Add Library Part Components	1608
Component Assignments	1610
Select Property Scheme	1612
Add Library Part Details	1613
Select Library Part Type	1614
Check Library Part	1615
Save Library Part	1617
Library Part Assignments	1618
Creating Objects	1619
Creating Lamps	1620
Creating MEP Objects	1621
Creating Doors and Windows	1622
Creating Skylights	1628
Overview of LOD Assignments	1629
LOD Recommendations	1631
Customizing Data Properties	1634
Create Custom Property Scheme	1635

Import Property Data from Excel	1636
Views of the Virtual Building	1637
Archicad Model Views	1638
Floor Plan Window	1639
Stories	1640
About Stories	1641
Navigate Among Stories	1642
Defining Story Display in 3D	1643
Defining Story Display in Section/Elevation/IE	1644
Create or Delete or Rename Stories	1645
Story Level Lines	1647
Floor Plan Cut Plane (Global Setting)	1651
Element Display in Floor Plan and Section	1653
Show On Stories	1654
Floor Plan Display (Element Settings)	1655
Show Projection	1657
Cut Surfaces and Outlines (Element Settings)	1659
3D Window	1660
Show Selection/Marquee in 3D	1662
Redefine Image Settings (3D View)	1664
3D Engines	1665
3D Projections	1666
3D Navigation Extras	1669
3D Cutting Planes	1670
Show or Hide 3D Cutaway	1671
Create a 3D Cutaway on the Floor Plan or Section	1672
Create a 3D Cutaway in the 3D Window	1675
Edit Cutaway	1679
3D Cutaway Attributes	1681
Hide 3D Cutting Planes	1682
Delete Cutting Planes	1683
3D Window Size	1684
Physically based renderer in 3D Window - Experimental feature	1685
Sections	1687
Create a Section Viewpoint	1689
Open a Section Viewpoint	1691
Horizontal/Vertical Range of Section Viewpoint	1692
Define Marker Reference for Source Marker	1695
Floor Plan Display of Section Markers	1697
Model Appearance (Section-Type Views)	1698
Editing Section/Elevation/IE Lines on Floor Plan	1704
Place a Linked Section Marker	1707

Place Unlinked Marker	1709
Create an Independent Section Viewpoint	1710
Assign Section Rebuild Status	1711
Updating Sections	1712
Summary of Rebuild Commands	1714
Elevations	1715
Interior Elevations (IE)	1717
About Interior Elevations	1718
Create Interior Elevation Group	1719
Create Single Interior Elevation Viewpoint	1722
Interior Elevation Horizontal Range	1723
Interior Elevation Vertical Range	1726
Display and Edit Interior Elevation Marker Lines and Marker Range	1728
Interior Elevation IDs and Names	1729
Delete/Restore IE Viewpoint	1732
Display of Elements in Interior Elevation	1734
3D Document	1735
About the 3D Document	1736
Create a 3D Document	1738
Redefine the 3D Document	1741
Details	1744
About Details	1745
Create a Detail Drawing Viewpoint with Model Source	1746
Contents of the Detail Viewpoint	1747
Place a Linked Detail Marker	1748
Create an Independent Detail Viewpoint	1749
Display of Detail/Worksheet Boundary	1751
Update Detail/Worksheet Marker Boundary	1752
Updating the Detail Window	1753
Worksheets	1754
About Worksheets	1755
Worksheet vs. Detail	1756
Create a Worksheet Drawing with Model Source	1757
Contents of the Worksheet Window	1758
Editing in the Worksheet Window	1759
Create an Independent Worksheet Viewpoint	1760
Place a Linked Worksheet Marker	1761
Managing Markers in Archicad	1762
About Markers	1763
Display of Marker Range Lines	1764
Source Marker Highlight	1765
Changing Marker Type	1766

Transfer Marker Parameters	1767
Copying a Marker	1768
Find Linked Markers	1769
Check Markers Palette	1770
Deleting a Viewpoint/View/Drawing with a Marker	1772
Deleting a Source Marker	1773
Interactive Schedule	1774
Types of Interactive Schedules	1775
Units in Interactive Schedules	1776
Open and View a Schedule	1777
Show Schedule Data for Selected Floor Plan Items	1779
Editing and Updating Schedule Items	1780
Select Schedule Items on Floor Plan/3D	1781
Define Schedule Content Using Scheme Settings	1782
Surface Schedule	1784
Surface Schedule: Predefined schemes	1785
Exposed Area parameter in Surface Schedules	1788
Opening surface in Surface Schedule	1790
Format Options (Interactive Schedule)	1793
Scheduled Elements on Locked or Hidden Layer	1803
Merge Schedule Items	1804
Schedule Headers	1809
Previews and Pictures in the Interactive Schedule	1812
Restructure Schedule to Fit Layout	1816
Split Schedule into Multiple Layouts	1818
Project Indexes	1820
Model View Options	1823
Define Model View Options	1824
Model View Options Combinations	1825
On-Screen View Options	1828
Partial Structure Display	1831
Applying Partial Structure Settings to Model Windows	1834
Save Partial Structure Settings by View	1835
How to Define Components as “Core” or “Finish”	1836
Visualizing the Entire Model	1838
Partial Structure Display Settings: Effect on Elements	1840
Partial Structure Display Settings in Windows Based on a Source Marker	1842
Renovation	1844
Assign Renovation Status to Elements	1845
Renovation Filters	1847
Apply Renovation Filter	1848
Show an Element in a Single Renovation Filter Only	1849

Renovation Filter Options	1850
Examples of Renovation Filters	1855
Graphic Overrides By Renovation Status	1858
Openings By Renovation Status: Override Examples	1860
Best Practices for Creating Renovation Views	1861
Workflows for Tricky Renovation Situations	1862
Adding Insulation to Walls: Renovation Workflow	1863
Adding New Door Panel to Existing Door Frame: Renovation Workflow	1864
Zone Stamps for Renovated Rooms: Renovation Workflow	1865
Listing Material Quantities for Renovation Projects	1866
Reset Renovation Status	1867
Graphic Overrides	1868
Graphic Override Rules	1870
Graphic Override Combinations	1875
Exceptions: When Overrides Don't Apply	1880
Surface Criteria in Graphic Override Rules and Collision Detection	1881
Element Listing Parameters	1886
General Listing Parameters	1887
Beam Listing Parameters	1896
Beam Segment Listing Parameters	1899
Column Listing Parameters	1900
Column Segment Listing Parameters	1906
Curtain Wall Listing Parameters	1909
Curtain Wall Accessory Listing Parameters	1912
Curtain Wall Frame Listing Parameters	1913
Curtain Wall Junction Listing Parameters	1915
Curtain Wall Panel Listing Parameters	1916
Window/Door Listing Parameters	1919
Fill Listing Parameters	1927
Lamp Listing Parameters	1928
MEP Listing Parameters	1929
Mesh Listing Parameters	1930
Morph Listing Parameters	1931
Object/Lamp Listing Parameters	1932
Opening Listing Parameters	1933
Railing Listing Parameters	1935
Railing Segment Listing Parameters	1936
Railing Members Listing Parameters	1937
Roof Listing Parameters	1938
Shell Listing Parameters	1942

Skylight Listing Parameters	1948
Slab Listing Parameters	1951
Stair Listing Parameters	1953
Stair Finish Listing Parameters	1957
Stair Structure Listing Parameters	1959
Wall Listing Parameters	1962
Zone Listing Parameters	1970
Component Data in Archicad	1977
Component Listing Parameters in the Interactive Schedule	1979
Calculating Component Areas	1981
Component Areas of Basic and Composite Elements	1982
Component Areas of Complex Profiles	1986
Functions in Expressions	1990
CONSTANTS	1991
Pi [PI]	1992
True[TRUE] False[FALSE]	1993
OPERATORS	1994
Add [+]	1995
Divide [/]	1996
Equals [=]	1997
Greater Than [>]	1998
Greater Than or Equal to [>=]	1999
Less Than [<]	2000
Less Than or Equal to [<=]	2001
Multiply [*]	2002
Not Equal [<>]	2003
Power [^]	2004
Subtract [-]	2005
LOGICAL FUNCTIONS	2006
And [AND]	2006
Or [OR]	2007
Not [NOT]	2008
If [IF]	2009
If Multiple [IFS]	2010
Exclusive Or [XOR]	2012
Undefined [ISUNDEFINED]	2013
MATHEMATICAL FUNCTIONS	2014
Absolute Value [ABS]	2014
e Exponent [EXP]	2015
Natural Logarithm [LN]	2016
Base Logarithm [LOG]	2017

Base 10 Logarithm [LOG10]	2018
Remainder [MOD]	2019
Nth Root [NTHROOT]	2020
Power [POWER]	2021
Quotient [QUOTIENT]	2022
Sign [SIGN]	2023
Square Root [SQRT]	2024
Sum [SUM]	2025
Round to Multiple [MROUND]	2026
Round Down to Multiple [MROUNDDOWN]	2027
Round Up to Multiple [MROUNDUP]	2028
Round [ROUND]	2029
Round Down [ROUNDDOWN]	2030
Round Up [ROUNDUP]	2031
TRIGONOMETRICAL FUNCTIONS	2032
Arccosine [ACOS]	2032
Arccotangent [ACOT]	2033
Arcsine [ASIN]	2034
Arctangent [ATAN]	2035
Cosine [COS]	2036
Cotangent [COT]	2037
Sine [SIN]	2038
Tangent [TAN]	2039
STATISTICAL FUNCTIONS	2040
Average [AVG]	2040
Maximum [MAX]	2041
Minimum [MIN]	2042
TEXT EDITING FUNCTIONS	2043
Compare [COMPARE]	2043
Merge Strings [CONCAT]	2044
Contains [CONTAINS]	2045
Find String [FIND]	2046
String from Left [LEFT]	2047
Length of String [LEN]	2048
Lower Case [LOWER]	2049
String from Position [MID]	2050
Replace in String [REPLACE]	2051
Repeat String [REPT]	2052
String from Right [RIGHT]	2053
Split String [SPLIT]	2054
Split String to Left [SPLITLEFT]	2055
Split String to Right [SPLITRIGHT]	2056

Substitute Strings [SUBSTITUTE]	2057
Join Strings [TEXTJOIN]	2058
Trim [TRIM]	2059
Upper Case [UPPER]	2060
DATA CONVERSION	2061
Convert to String [STR]	2061
Convert Unit to String [STRCALCUNIT]	2062
Number from String [STRTONUM]	2063
PARAM-O Reference Guide	2064
Installation	2065
Getting Started	2066
PARAM-O Manager	2067
PARAM-O Editor	2068
Creating an Object	2070
Making an Object Parametric	2072
PARAM-O Node Documentation	2074
Parameters	2075
Dimension 1	2076
Dimension 2	2077
Height	2078
Shapes	2079
Block	2080
Cylinder	2081
Cone	2082
Sphere	2083
Ellipsoid	2084
Elbow	2085
Wedge	2086
Regular Prism	2087
Profile Extrude	2088
Inputs	2089
Boolean	2090
Integer	2091
Number	2092
Length	2093
Angle	2094
Random Number	2095
Global Value	2096
List Inputs	2097
Integer Series	2098
Number Series	2099

Random Series	2100
Number Distribution	2101
Attribute Inputs	2102
Pen	2103
Surface	2104
Profile	2105
Point Inputs	2106
Point	2107
Vector	2108
Deconstruct Point	2109
Deconstruct Vector	2110
Line Points	2111
Grid Points	2112
Arc Points	2113
Transformations	2114
Move	2115
Move By Vector	2116
Rotation X	2117
Rotation Y	2118
Rotation Z	2119
Scale	2120
Combination	2121
Shape Transformations	2122
Transform Shape	2123
Move Shape	2124
Move Shape By Vector	2125
Rotate Shape X	2126
Rotate Shape Y	2127
Rotate Shape Z	2128
Scale Shape	2129
Maths	2130
Addition	2131
Subtraction	2132
Multiplication	2133
Division	2134
Lower	2135
Lower Or Equal	2136
Greater	2137
Greater Or Equal	2138
Equal	2139
Not Equal	2140
Expression	2141

Abs	2142
Floor	2143
Ceil	2144
Sqrt	2145
Negative	2146
Min	2147
Max	2148
Trigonometry	2149
Sine	2150
Cosine	2151
Tangent	2152
Arc Sine	2153
Arc Cosine	2154
Arc Tangent	2155
Solid Operations	2156
Union	2157
Difference	2158
Intersection	2159
Extras	2160
GDL Object	2161
GDL Shape	2162
Viewer	2163
Documentation	2164
Fills	2165
Draw a 2D Fill	2166
Assign a Cut Fill (GDL Elements)	2169
Assign a Cover Fill	2170
Add Area Text to a Fill	2172
Display Vector Fills in project views	2173
Set Orientation of Vector or Symbol Fill Patterns	2175
Using Gradient Fills	2183
Drafting	2185
Lines, Circles, Polylines, Splines	2186
Line Categories	2187
Drawing a Single Straight Line Segment	2188
Stretching or Shrinking Lines	2189
Drawing Arcs and Full Circles	2190
Stretching Curved Elements	2191
Editing an Arc Using its Tangent	2193
Drawing Elliptical Arcs and Full Ellipses	2194
Convert Ellipse to Circle	2195
Drawing Splines	2196

Editing Splines	2198
Drawing Freehand Curves	2199
Drawing Polylines and Chained Lines	2200
Editing Polylines	2202
Hotspots	2204
Figures	2206
Drawings in Model Views	2208
Place Drawing in Model View	2209
Delete Drawing	2211
Annotation	2212
Dimensioning	2213
Dimensioning Standards	2215
Glossary of Dimensioning Terms	2216
Static Dimensions	2217
Linear Dimensions	2218
Dimension Wall or Slab Thickness	2219
Linear Dimension Geometry Methods	2222
Linear Dimensions in the 3D Document Window	2226
Elevation Dimensions	2228
Radial Dimensions	2232
Level Dimensions	2234
Place Level Dimension on element	2235
Level Dimensions on Tread, Landing, or Landing Structure	2239
Angle Dimensions	2240
Edit Dimensions	2242
Modify the Witness Line	2248
Edit Text Box	2249
Pointer Line	2251
Automatic Exterior Dimensioning	2253
Automatic Interior Dimensioning	2256
Secondary Dimensions (Add-on)	2258
The Grid Tool	2259
Components of a Grid Element	2261
Visibility of a Grid Element	2262
Creating a Straight Grid Element	2264
Creating a Curved Grid Element	2265
Editing a Grid Element	2267
Editing Grids in Teamwork	2268
Place a Grid System	2269
Text Blocks	2272
Placing Text Blocks	2273
Formatting Text Blocks As a Whole	2275

Resizing Text Graphically	2276
Applying Favorite Text Settings	2277
Formatting Individual Components of Text Blocks	2278
Autotext	2279
How to Insert Autotext	2280
Autotext Reference Drawing	2282
Autotext Keywords	2284
Labels	2289
Place an Associative Label	2291
Text/Autotext Label Content	2293
Symbol Labels with Element Information	2295
Custom Settings for Symbol Labels	2299
Label Multiple Selected Elements	2305
Labeling Curtain Walls	2306
Labeling Stairs	2309
Place an Independent Label	2311
Find and Select Text Blocks/Labels	2312
Text Editing in Archicad	2313
Search and Replace Text	2314
Spell Checker	2315
Editing Commands in Text-Type Windows	2316
Navigator - Layout Book	2317
Working with Layouts	2321
Master Layouts	2323
Drawings in the Layout Book	2327
Place Drawing on Layout	2328
Arranging Multiple Drawings on the Layout	2333
Drawing Frame	2334
Drawing Titles	2340
Managing and Updating Placed Drawings	2344
Layout and Drawing IDs	2347
Layout Workflow Schemes	2352
Layout Workflow: Smaller Projects	2353
Layout Workflow: Larger Projects	2354
Revision Management	2357
Revision Management Workflow	2358
Revision Management: Default Schemes in the Template	2360
Working with Changes	2362
Change Manager Palette	2363
Create Change	2365
Link/Unlink Change Elements	2371
Add/Remove Changes on Layout Revisions	2373

Change Details	2375
Archive Changes in Change Manager	2378
Feedback on Layout Revisions	2379
Revision History Data on Layout and in Layout Settings	2381
Revision IDs	2384
Managing Transmittal Sets	2386
Change List and Transmittal History	2398
Revisions in the Publisher	2402
Changes from External Files	2404
Publishing	2405
Print	2406
Plot	2408
Publisher	2409
Define Publishing Method: Print, Save, Plot, BIMx	2416
Publisher File Formats	2419
Publishing Process	2422
Element Information	2426
ID Management	2429
Element ID Manager	2430
Calculation (Legacy)	2433
Design Options	2437
Suggested workflow for Design Options	2438
Design Options interface	2440
Create and manage Design Options and Sets	2442
Doors, Windows, and Skylights in Design Options	2444
Add 2D elements to Design Options	2445
Set a default Option	2446
Activate an Option	2447
Focused editing	2448
Relink elements to a different Option	2450
Working with Design Option Combinations	2452
Switch among Design Option Combinations	2454
Save Design Option Combination in a view	2455
Design Options in Teamwork	2456
Hotlinks and Design Options	2457
Choose which Options to place (from source file)	2458
Choose an Option for the placed Hotlink (in host file)	2459
Place different Options from the same module	2461
Which Design Options are part of the Hotlink source file?	2462
Comparing Design Options	2463
Element properties for Design Options	2465
Merge Design Option to Main Model	2467

Collaboration	2468
Teamwork	2469
Teamwork Functions at a Glance	2470
Graphisoft BIMcloud: Features and Benefits	2471
Teamwork Setup/Preparation	2472
Share a Teamwork Project in Archicad	2473
Libraries Setup for Teamwork Project Sharing	2476
Network Diagnostic Information (BIMcloud)	2479
Open or Join Teamwork Project	2483
Close Teamwork Project	2485
Save Local Copy of Teamwork Project	2487
Force User to Leave a Teamwork Project	2488
Join or Leave a Teamwork Project	2490
Send/Receive Changes in Teamwork	2491
Manage Local Teamwork Data	2493
Move Local Teamwork Data to a Different Computer	2496
Delete Local Teamwork Data	2497
Schedule Teamwork Data Cleanup Reminder	2498
Working Online/Offline in Teamwork	2499
Remote Access to Teamwork Projects	2500
Working from Home and the Office: Suggested Workflow for a Teamwork Project	2501
Travel Pack: Move Your Teamwork Project and Libraries to a Different Computer	2502
Avoid Conflicting Work Sessions in Teamwork Projects	2509
Teamwork Interface	2510
Reserving Elements in Teamwork	2512
Overview of Teamwork Reservation	2513
Reserve Elements by Selection (Teamwork)	2514
Reserve Elements by Criteria (Teamwork)	2516
Reserve Library Objects (Teamwork)	2520
Feedback on Element Ownership (Teamwork)	2521
Reservation Results (Teamwork)	2527
Reserving Teamwork Project Data (Non-Elements)	2529
Feedback on Ownership of Teamwork Project Data	2531
Reservation in Attributes palette; Reserve All Attributes (Teamwork)	2533
Reserve Individual Views and View Map Folders	2535
Reserve Settings of Layout Book Items (Teamwork)	2538
Moving Views and Layouts in the Navigator: Reservation Requirements (Teamwork)	2540
Reserve Publisher Sets (Teamwork)	2541
Reserve and Delete Layers (Teamwork)	2542
Reserve Find and Select Criteria (Teamwork)	2543
Reservation in Interactive Schedules (Teamwork)	2544
Reserve List Scheme Dialog Boxes (Teamwork)	2548

Reserve All Elements and Data (Teamwork)	2549
Review Your Workspace (Teamwork)	2550
Creating and Sending Project Data (Teamwork)	2551
Assign Elements (Teamwork)	2552
Request Elements or Project Data (Teamwork)	2554
Release Elements or Project Data (Teamwork)	2558
Messaging (Teamwork)	2560
Create a New Message (Teamwork)	2563
Receive and Open Message (Teamwork)	2566
Review Elements (Teamwork)	2567
Add Comment to Activities (Teamwork)	2568
Libraries in Teamwork	2570
Hotlink Modules and XREFs in Teamwork	2572
Hotlink Modules	2575
About Hotlink Modules	2576
Place Hotlink	2579
Edit Hotlink Module Content	2584
Hotlink Settings	2586
Hotlink Module Manager	2592
Create Module (.mod) File	2599
Save Selection as Module	2602
Save Selection in PLN Format	2603
Collision Detection	2607
Archicad Design Checker	2614
Model Compare	2617
Compare Teamwork Model Versions (Before and After Receiving Changes)	2619
Compare Two Saved Versions of Current Model	2621
Compare Two Views of a Single Project	2623
Compare Archicad Models Before and After Hotlink Update	2625
Compare Structural Analytical Models	2627
Select View for Compare	2633
Filter Changes Shown in Model Compare Palette	2636
Model Compare: Review the Changes	2640
Workspace Colors for Model Compare	2644
Issue Handling	2646
Add Selected Elements to New Issue	2648
Show Containing Issue	2650
Add Views to Issue	2651
Issue Manager	2653
Suggestion Elements in Issue Manager (Highlight, New, Modify, Delete)	2655
Assign Issue to User	2659
Accept Issue Element(s)	2661

Issue Management Workflow: Example	2663
Issue Organizer	2667
Issue Details	2672
Create Issue from Model Check Report	2674
Create Issue from Model Compare Change List	2676
Import Issues (BCF format)	2677
Export Issues (BCF format)	2679
Export/Import Issues Using PDF Markup	2680
Change Tracking in Teamwork	2685
Create New Subscription	2686
Subscribe to Changed Elements - by Criteria	2687
Subscribe to Changed Elements - by Selection	2688
Subscribe to Changes in Project Data	2690
Filter Element Changes by Parameter Type	2691
Filter Change Notifications by Teams and Users	2692
View and Manage Tracked Changes	2695
Manage Subscriptions	2698
MEP Modeler	2700
Apply MEP Work Environment	2701
Place Route element	2702
Edit Route graphically	2706
Edit individual segments and nodes	2711
Route Settings	2713
Route Preferences	2720
Place Terminal, Accessory, or Equipment	2723
Select Connected MEP Elements (Command)	2724
MEP Documentation	2725
MEP-compatible Library Parts	2726
MEP Systems	2729
Create custom MEP object	2731
MEP elements in IFC exchange	2733
Structural Analytical Model	2734
Structural Engineering Work Environment and Commands	2738
Elements of the Structural Analytical Model	2739
1D Members (Column and Beam)	2740
Connection Range (Extend Core to Enable Connection)	2744
Beam/Column Profiles in SAF export	2747
2D Members (Slab, Wall, Roof)	2751
Structural Supports	2754
Place Structural Support	2755
Support Elements: Symbol Options	2759

Structural Supports at Openings and Holes	2760
Structural Links	2761
Structural Loads	2765
Load Case Palette	2767
Load Manager (for Load Cases and Load Groups)	2768
Load Case Settings	2771
Load Group Settings	2773
Load Combination Dialog Box	2775
Place Loads	2778
Place Point Load	2779
Place Line Load	2781
Place Surface Load	2783
Create/Update Live Loads	2785
Set up Load Categories	2789
Load Tool Settings	2790
Representation of Loads	2794
Loads in SAF Import/Export	2796
Representation of Structural Analytical Model	2797
Structural Analytical Model Generation Rules	2804
Adjustment Rules	2806
Set Up Adjustment Rules	2808
Adjuster Types and Options	2811
Stretch 1D Member	2812
Stretch 2D Member	2813
Cutback 1D Member	2814
Cutback 2D Member	2815
Maximum Cutback/Maximum Overhang	2817
Stretch & Cutback	2818
Adjust Connected Ends	2819
Offset 1D to 1D	2821
Offset 1D to 2D	2825
Offset 2D Member	2828
Offset Options In Case of Multiple Targets	2830
Snap (1D to 1D)	2832
Examples of Adjustment Rules	2833
Example 1: Offset Beams to Slabs	2834
Example 2: Stretch Beams to Columns	2838
Example 3: Snap	2840
Example 4: Cutback	2842
Example 5: Stretch & Cutback	2845
Filter Openings for Structural Analytical Model	2846
End/Edge Release (Generation Rules)	2848

Eccentricity	2852
Manual editing of analytical members	2854
End/Edge Releases	2859
Model Check - Overview	2863
Physical Model Quality Check	2864
Physical Model Check Parameters	2865
Structural Analytical Model Quality Check	2869
Model Check Report	2876
Collaboration with Structural Analysis Software	2878
IFC Structural Analysis Model Import/Export - Experimental Feature	2879
Structural Analysis Format	2881
Export SAF File from Archicad	2882
Import SAF File to Archicad	2884
SAF Translators - Overview	2886
Use external database to translate unmapped profiles	2887
Download SAF Data Files optimized for structural applications	2889
Prepare and export a SAF Data File to ensure standard mapping	2890
Detailed SAF Translator Settings	2891
Material Mapping - Import	2892
Additional Material Mapping: By Element Property Value	2894
Add More SAF Materials (Material Mapping - Import SAF)	2895
Material Mapping - Export	2897
Map Archicad Building Material (By Name)	2898
Use Element Property for More Precise Material Mapping	2900
Add More SAF Materials to Mapping List (Export to SAF)	2903
Profile Mapping - Import	2905
Map SAF Cross Sections By Name	2906
Add More SAF Cross Sections (Import SAF)	2908
Mapping Imported "General" or "Parametric" SAF Cross Sections	2910
Parametric SAF Cross Sections Recognized by Archicad	2912
Default Mapping for "Other" Cross Sections (Import SAF)	2914
Profile Mapping - Export	2915
Map Complex Profiles (By Name)	2916
Add More SAF Cross Sections to Mapping List (Export to SAF)	2918
Default Mapping for Exported Cross Sections	2920
Miscellaneous Settings - Import	2921
Miscellaneous Settings - Export	2922
Create new SAF Translator	2926
Import/Export SAF Translators	2927
Visualization	2928
Cameras	2929

PhotoRendering	2930
PhotoRendering Settings Palette	2931
Cineware Basic Settings	2937
Render Settings (Basic) Panel	2938
Cineware Basic Settings: Environment Panel	2942
Cineware Basic Settings: Background Panel	2944
Create Rendered Image	2946
Visualization Extras	2952
Fly-Through	2953
Sun Study	2956
Align View	2959
Graphisoft BIMx	2963
Publish BIMx Hyper-Model from Archicad	2964
Publish BIMx Hyper-Model: Publisher method (detailed method)	2965
Publish BIMx Hyper-Model Using a Wizard	2975
Republish Hyper-model	2979
BIMx: Notes on Model Data, 3D Model Settings, and Textures	2981
Energy Evaluation	2983
Energy Evaluation Workflow: Overview	2984
General Settings for Energy Evaluation	2986
Energy Evaluation: Direct BIM to BEM	2988
Internal Space Zones for Energy Evaluation	2989
Building Energy Model View	2992
Automatic Model Geometry and Material Property Analysis in Energy Evaluation	2993
Energy Model Review Palette	2994
List and Tree Tab Page View Options	2995
Thermal Blocks Tab Page	2996
Structures Tab Page	2998
Openings Tab Page	3001
Building Energy Model Visualization	3006
Additional Data Assignment and Input	3012
Thermal Block Property Settings (Energy Evaluation)	3013
Structure Property Settings (Energy Evaluation)	3015
Opening Property Settings (Energy Evaluation)	3019
Climate Data (Energy Evaluation)	3021
Environment Settings (Energy Evaluation)	3024
Operation Profiles (Energy Evaluation)	3030
Building Systems (Energy Evaluation)	3035
Energy Source Factors (Energy Evaluation)	3043
Energy Costs (Energy Evaluation)	3044
Energy Performance Evaluation	3045

Start Energy Simulation (Energy Evaluation)	3046
Interoperability with Green Software (Energy Evaluation)	3047
Advisory Messages (Energy Evaluation)	3051
VIP-Core Dynamic Simulation Engine (Energy Evaluation)	3054
Energy Evaluation Report Dialog	3055
Reading the Energy Performance Evaluation - PDF Report	3059
Reading the Energy Performance Evaluation - XLS Report	3065
Interoperability	3067
File Types Opened by Archicad	3068
Archicad File Types (Open)	3069
CAD Formats Opened by Archicad	3070
Image Formats Opened by Archicad	3072
Model/3D Formats Opened by Archicad	3073
Other Formats Opened by Archicad	3076
File Types Saved by Archicad	3077
Archicad File Types Saved by Archicad	3078
CAD Formats Saved by Archicad	3080
Image Formats Saved by Archicad	3082
Model/3D Formats Saved by Archicad	3083
Document and Other Formats Saved by Archicad	3089
Merging Files to Archicad	3090
Import/Export Property Data via Spreadsheet	3095
Update Properties in Hotlink Modules - Experimental Feature	3101
Import BIM Content	3103
Working with PDF Documents	3105
Create PDF Output Using the Publisher	3106
PDF Options	3108
Save Document in PDF Format	3111
3D Content in PDF	3113
Working with DWG/DXF Files	3117
Opening DWG/DXF Files	3118
Saving DWG/DXF Files	3121
Merge DXF-DWG	3125
Round-Trip Conversion (Smart Merge)	3127
Working with XREFs	3129
Attributes of XREF Files	3132
Using XREFs in Round-Trip Communication	3134
Working with Point Clouds	3135
Import Point Clouds	3136
Trace Point Cloud to Create Model Elements	3141
Optimize Display of Point Cloud Object	3143

Redefine Origin of Point Cloud Object	3145
Place Point Cloud Object in Teamwork Project	3146
Working with Rhino 3D Models	3147
Open or Merge Rhino Model	3148
Import Rhino 3DM File as Object	3155
Hotlink Rhino File	3158
Export Model to Rhino	3160
Working with RFA & RVT Files	3163
Import RFA as GDL Object	3164
Hotlink 3D RVT Model	3167
Export 3D Model as Revit 3D Geometry	3171
Working with FBX files	3173
Export to FRILO	3175
Export to Artlantis	3176
Interoperability with MEP Applications	3178
Export Archicad Files to MEP Software	3179
Import MEP Data to Archicad	3181
Interoperability with Energy Analysis	3182
Integrated Energy Evaluation	3183
Export Files from Archicad to Energy Analysis Software	3184
Import Energy Analysis Results Into Archicad	3185
Working with IFC	3186
IFC File Types Supported by Archicad	3187
Archicad Classifications and Categories in IFC	3188
Export IFC Model from Archicad	3191
Import IFC Model to Archicad	3194
Open IFC Model	3196
Merge IFC Project	3198
Hotlink IFC File	3202
Update with IFC Model	3206
IFC Project Manager	3209
Functions of the IFC Project Manager	3210
Navigating in the IFC Project Manager: Tree Structure	3211
Navigating in IFC Project Manager: Attributes and Properties	3216
Assign or Edit IFC Property/Attribute Values	3218
Create New, Custom IFC Property	3221
Create New Classification Reference	3224
Apply Predefined Rules	3226
Using Assignments in IFC Project Manager	3229
Using Type Products in IFC Project Manager	3238
Manage IFC Data at Element Level	3244

IFC Local Preferences	3246
Define Element Criteria using IFC Data	3247
IFC Translators: Overview	3252
Translators for Import (Detailed Settings)	3259
Model Filter for IFC Import	3262
Type Mapping for IFC Import	3264
Property Mapping for IFC Import	3266
Using the Mapping Table	3269
Assigning Property Values in Archicad (Based on IFC Property Mapping)	3272
Geometry Conversion for IFC Import	3276
Layer Conversion for IFC Import	3279
Material and Surface Conversion	3282
Translators for Export (Detailed Settings)	3285
Model Filter for IFC Export	3290
Type Mapping for IFC Export	3293
How to Fix Error Warnings in the Type Mapping Dialog	3300
Geometry Conversion for IFC Export	3305
Property Mapping for IFC Export	3312
How to create a Mapping Rule	3317
Map Component-Level Properties	3320
Examples of Property Mapping Rules	3322
Data Conversion for IFC Export	3328
Unit Conversion for IFC Export	3333
Filter Model at Import (Secondary Filter)	3334
Filter Model at Export	3337
Appendix: IFC Terms and Concepts	3339
Model View Definitions	3340
IFC Data Types	3343
Appendix: IFC Data Mapping in Archicad	3351
Predefined Property Mapping (Export Archicad to IFC)	3352
Built-in IFC Type Mapping for Archicad	3357
Data Types for Properties in IFC and Archicad	3362
User Interface Reference	3363
Controls	3364
Action Center	3365
Project Chooser	3369
The 2D Navigator Preview Palette Controls	3370
The 3D Navigator Preview Palette Controls	3371
Control Box	3374
Coordinates Palette	3376
Search and Replace Text Dialog Box	3378

Formatting Palette and Text Editor Controls	3380
Renovation Palette	3383
Teamwork Palette	3385
Project Info	3391
Project Notes	3393
Project Preview	3394
Architect Credits in BIMx	3396
Session Report Window	3398
Solid Element Operations Palette	3399
Classic 3D Navigation Toolbar	3401
Dialog Boxes	3402
No license found - when starting Archicad	3404
Work Environment Dialog Box	3405
Work Environment Schemes	3407
Work Environment Profiles	3412
Dialog Boxes and Palettes	3417
Selection and Element Information	3420
Tracker and Coordinate Input	3422
Input Constraints and Guides Dialog Box	3424
Model Rebuild Options	3426
More Options	3429
Advanced Redraw Options	3432
On-Screen Options	3434
Data Safety & Integrity	3438
Network And Update	3441
Special Folders	3443
Shortcut Customization Dialog Box	3444
Toolbox Customization Dialog Box	3447
Tool Settings Dialog Box Customization	3449
Info Box Customization Dialog Box	3450
Toolbar Customization Dialog Box	3451
Menu Customization Dialog Box	3454
Project Preferences Dialog Boxes	3459
Dimensions Preferences	3460
Calculation Units Preferences	3467
Calculation Rules Preferences	3469
Zones Preferences	3471
Reference Levels Preferences	3473
Legacy Preferences	3476
Structural Analytical Model Representation (Preferences)	3480
Structural Analytical Model Units	3481
Add-on Manager	3482

Story Settings Dialog Box	3484
Edit Elements by Stories Dialog Box	3486
Floor Plan Cut Plane Settings Dialog Box	3487
Copy Dialog Box	3488
Linework Consolidation Settings	3490
Fill Consolidation Settings	3494
Model View Options for Construction Elements	3496
Detail Level of Door, Window and Skylight Symbols	3499
Miscellaneous Settings for Library Parts	3501
3D Document Settings	3504
Filter and Cut Elements in 3D Dialog Box	3509
3D Styles	3511
Hardware Acceleration Options	3518
3D Projection Settings	3520
Parallel Projection Settings	3521
Perspective Projection Settings	3526
Sunlight Settings	3528
RoofMaker Settings	3529
TrussMaker Settings	3538
Grid System Settings	3540
GDL (Geometric Description Language)	3544
GDL Object Editor	3545
Object Preview Picture	3547
Details	3548
Parameters	3550
Migration	3553
Components and Descriptors	3554
Script Windows	3555
2D Symbol	3560
Preview Picture	3561
2D and 3D Views	3562
GDL Editor Shortcuts	3564
Subtype Hierarchy Dialog Box (GDL Master Window)	3565
Scheme Settings Dialog Box (Interactive Schedule)	3566
Index Settings Dialog Box	3579
View Settings Dialog Box	3580
Book Settings Dialog Box	3583
Subset Settings Dialog Box	3587
Layout Settings	3589
Master Layout Settings	3592
Drawing Manager	3596
Paste Options Dialog Box	3602

Print 2D Document	3603
Print 3D Document	3605
Print Picture	3606
Print Layout	3607
Print Interactive Schedule	3609
Header/Footer Settings	3610
Plot Setup	3612
Plot 2D Document	3614
Plot Layout	3615
PhotoRendering Settings	3616
Basic Rendering Engine	3617
Sketch Rendering Engine	3620
XREF Management Dialog Box	3623
Attach XREF	3626
DXF/DWG Translation Setup	3629
Drawing Unit (DXF/DWG Translation Setup)	3632
Open Options (DXF/DWG Translation Setup)	3633
Save Options (DXF/DWG Translation Setup)	3636
Attributes (DXF/DWG Translation Setup)	3641
Layers (DXF/DWG Translation Setup)	3642
Pens & Colors (DXF/DWG Translation Setup)	3646
Line Types (DXF/DWG Translation Setup)	3649
Fills (DXF/DWG Translation Setup)	3650
Font Conversion (DXF/DWG Translation Setup)	3652
Miscellaneous (DXF/DWG Translation Setup)	3653
Custom Functions (DXF/DWG Translation Setup)	3654
Merge DXF-DWG Model Space Dialog Box	3655
Smart Merge Options	3657
Custom Functions (DXF/DWG Translation Setup)	3661
Merge DXF-DWG Model Space Dialog Box	3662
Smart Merge Options	3664
Surfaces (Cineware)	3668
Cineware Surface Channels	3670
Color (Cineware Surface Channel)	3673
Diffusion (Cineware Surface Channel)	3676
Luminance (Cineware Surface Channel)	3678
Transparency (Cineware Surface Channel)	3679
Reflectance (Cineware Surface Channel)	3685
Environment (Cineware Surface Channel)	3717
Fog (Cineware Surface Channel)	3720
Bump (Cineware Surface Channel)	3721
Normal (Cineware Surface Channel)	3724

Alpha (Cineware Surface Channel)	3727
Glow (Cineware Surface Channel)	3729
Displacement (Cineware Surface Channel)	3731
Grass (Cineware Surface Channel)	3739
Illumination (Cineware Surface Channel)	3742
Textures (Cineware Surfaces)	3746
Special Settings for Image Textures (Cineware Surfaces)	3750
Mix Mode and Mix Strength (Cineware Surfaces)	3754
Shaders List (Cineware Surfaces)	3756
Procedural Shaders (Cineware Surfaces)	3767
Shader Effects (Cineware Surfaces)	3770
Sketch Shaders (Cineware Surfaces)	3798
Additional Effects for Shader Layers/Folders (Cineware Surfaces)	3800
Using Layers to Combine Shaders (Cineware Surfaces)	3802
Detailed Cineware Settings	3807
General Considerations When Using Detailed Settings	3808
Navigating in the PhotoRendering Settings Palette	3812
List of Detailed Cineware Settings	3814
Light Adjustments (Cineware)	3815
Shadow Casting (Cineware)	3816
Environment: Physical Sky (Cineware)	3818
Environment: HDRI Sky (Cineware)	3831
Cineware Effects	3833
Ambient Occlusion (Cineware Effect)	3834
Caustics (Cineware Effect)	3840
Cel Renderer (Cineware Effect)	3842
Color Mapping (Cineware Effect)	3844
Color Correction (Cineware Effect)	3846
Depth of Field (Cineware Effect)	3847
Lens and Filters (Cineware Effect)	3850
Distance Fog (Cineware Effect)	3852
Monochrome Model (Cineware Effect)	3853
Tone Mapping (Cineware Effect)	3854
Vignetting (Cineware Effect)	3859
Global Illumination (Cineware)	3860
Preset (Cineware Global Illumination)	3861
Primary and Secondary Method (Cineware Global Illumination)	3863
Sampling (Cineware Global Illumination)	3868
Radiosity Maps (Cineware Global Illumination)	3874
How Does QMC Work?	3885
Irradiance Cache (Global Illumination)	3886
Irradiance Cache (Legacy) (Cineware Global Illumination)	3893

Details (Cineware Global Illumination)	3898
Physical Renderer (Cineware)	3900
Physical Camera	3901
Physical Renderer Options	3904
Physical Renderer Advanced	3909
General Options (Cineware)	3911
Antialiasing (Cineware)	3916
Grass (PhotoRendering Option)	3921
Spherical Camera (Cineware)	3926
Stereo Rendering (Cineware)	3930
Using Redshift Renderer	3943
Redshift Renderer: Current limitations	3944
Tool Settings Dialog Boxes	3949
Working in Tool Settings Dialog Boxes	3951
Wall Tool Settings	3954
Roof Tool Settings	3962
Shell Tool Settings	3967
Mesh Tool Settings	3974
Zone Tool Settings	3976
Morph Tool Settings	3985
Settings of Library Part Elements	3989
Object Tool Settings	3999
Panels of the Object Settings Dialog Box	4000
Special Lamp Parameters	4007
Basic Light Parameters	4008
General Light Sources and Effects	4012
Door/Window Tool Settings	4022
Door/Window Preview and Positioning Panel	4024
Where to Set Door/Window Parameters	4027
Floor Plan Display of Door/Window: Symbolic vs. Projected	4028
Other Panels in Door/Window Settings	4029
Skylight Tool Settings	4031
Dimension Tool Settings	4035
Linear Dimension Tool Settings	4036
Dimension Text Settings	4044
Text Content	4045
Convert Dimensions Dialog Box (Add-on)	4048
Text Tool Settings	4049
Text Style	4051
Label Tool Settings	4055
Drawing Settings	4058
Fill Tool Settings	4066

Line-type Tool Settings	4069
Section and Elevation Settings	4071
Interior Elevation (IE) Tool Settings	4077
Worksheet and Detail Tool Settings	4081
Change Tool Settings	4083
Grid Tool Settings	4085
Wall End Tool Settings	4088
Spline Tool Settings	4089
Hotspot Tool Settings	4090
Figure Tool Settings	4091
Camera Tool Settings	4093
Custom Settings Panel for a Door/Window	4096
Nominal Sizes and Tolerance	4099
Shape	4102
Door/Window Settings and Opening	4103
Frame Style and Dimensions (for Door)	4106
Natural Ventilation	4107
Door Leaf Type	4108
Handle and Kick Board (for Door)	4109
Leaf and Sash Dimension (for Door)	4110
Sash Type/Sash Options (Door or Window)	4111
Frame and Sash (for Windows)	4113
Frame Widths (for Windows)	4114
Opening Type and Angle (Door/Window)	4115
Opening Lines (Door/Window)	4117
Opening Lines - Sidelight and Transom	4118
Model Attributes of Door/Window Opening	4119
Wall Opening	4121
Reveal Settings	4122
Wall Closure	4123
Wall Inset (for Windows)	4126
Masonry Arch	4128
Tolerances	4130
Model Attributes of Wall Opening	4131
Fixtures and Fittings	4132
Threshold Settings	4133
Sill Settings	4134
Board Settings	4135
Casing Outside/Inside Settings	4136
Sunshade Settings	4137
Model Attributes of Fixtures and Fittings	4138
Floor Plan and Section Tab Page (Door/Window Settings)	4139

Minimal Space	4142
Custom Corner (for Windows)	4144
Extra Topics	4145
Solibri Connection	4146
Install Solibri Connection Add-on	4147
Connect to Solibri	4148
Export Model to Solibri	4150
Select Archicad Elements in Solibri	4152
Get Selection Basket	4153
Import Issues from Solibri	4155
Break Solibri File Link	4156
Bluebeam Connection	4157
Merkmalserver-Verbindung Add-on	4164
Area List Add-on	4177
Check for Area Conflicts	4189
Environmental Properties	4196

Getting Help on Archicad 27

Online Reference Guide

In Archicad, open from Help > Online Resources > Archicad Reference Guide.

PDF Reference Guide

Click [here](#) to access in **PDF** format.

Context-Sensitive Help

Do one of the following:

- Press F1 (Windows) or Cmd+? (Mac)
Note: Shortcut may vary depending on your language version.
- Right-click on any user interface element and choose the 'What's this' command

The Help will open to a topic relevant to the Archicad interface item you have clicked. (In some cases, the Help will open to its starting page.)

Graphisoft Community Site

Knowledgebase articles, tutorials and videos are available at [Graphisoft Community](#).

Download Offline Help

When you do have an internet connection, click the links below to download the files (either the PDF, or the ZIP file, or both). That way, you will be able to access the Help even while working offline.

Archicad Help (PDF document)

[Link to PDF](#)

Download the PDF to your computer.

Archicad Help in ZIP format (for context-sensitive Help)

[Link to Help.zip](#)

Follow these steps **carefully** to enable context-sensitive Archicad Help while working offline:

1. Download **Help.zip** to your computer and unzip it.
2. Now go to the original Help folder where Archicad is installed (for example: Program Files/Graphisoft/ Archicad 27/Help).
3. If the original Help folder contains a **Files** folder, delete it. (If not, continue.)
4. Go back to the new Help folder you just unzipped, and open it.
5. From there, move the **Files** folder into the Archicad Help folder.

Note for Safari Users

The Safari browser is not supported.

However, if you are still using a Safari browser (v. 11 and later), follow these steps in order to view the offline Archicad Help.

1. Go to Safari Browser Settings
2. Choose Preferences
3. Choose Advanced
4. Choose "Show develop menu in menu bar"

5. From the Develop menu: select “Disable local file restrictions.”

Additional Resources

See additional PDF resources in the Archicad Help menu:

- **GDL Reference Guide**

This manual is a complete reference to the Graphisoft’s proprietary scripting language, GDL (Geometric Description Language).
- **Calculation Guide**

Archicad’s Calculate function interacts with the project database to calculate the number of elements in a project, their spatial disposition and the quantity of element components. This is a legacy function. Most users will use the Interactive Schedule.

License agreement & Trademarks

Archicad Licenses

By installing and using Archicad, you are legally bound by the provisions of the End User License Agreement accepted by you during the Installation process.

You can access a copy of the EULA for your version of Archicad

- from the Help menu of Archicad, or
- from the Documentation folder inside the Archicad folder where you installed the program on your computer's hard drive

Third Party Software Licenses

Archicad leverages technologies and components licensed from third party companies. Major functions powered by third party software are:

Dwg I/O and DGN I/O powered by Open Design Alliance (ODA[®]).

<http://www.opendesign.com>

PDF technology powered by PDFNet SDK copyright © PDFTron™ Systems Inc., 2001-2013, and distributed under license. All rights reserved.

<http://www.pdftron.com>

IFC interoperability powered by Jotne EPM Technology's EDM Toolkit.

<http://www.epmtech.jotne.com>

Cineware Rendering Engine - Copyright MAXON Computer GmbH. All rights reserved.

<http://www.maxon.net>

Embedded web browsing - created with Chromium Embedded Framework (CEF). Copyright © 2008-2014 Marshall A. Greenblatt. Portions Copyright © 2006-2009 Google Inc. All rights reserved.

Integrated Energy Evaluation powered by StruSoft's VIPCore engine

<http://www.strusoft.com>

Software license Protection by Wibu

<http://www.wibu.com>

You can find additional third Party license information on the Graphisoft webpage:

<http://www.graphisoft.com/licenses/>

Install Guide for Archicad 27

[Install Archicad](#)

[MacOS: Enable Full Disk Access](#)

[Your Archicad License Key](#)

[Add/Repair Archicad](#)

[Updates for Archicad](#)

[Archicad Tools and Utilities](#)

[Uninstall Archicad](#)

[Using a CodeMeter Server](#)

Related:

[Network Installation for BIM Managers](#)

Install Archicad

Download Archicad

Archicad is available for download at:

<http://www.graphisoft.com/downloads/>

Archicad Hardware/Software Requirements

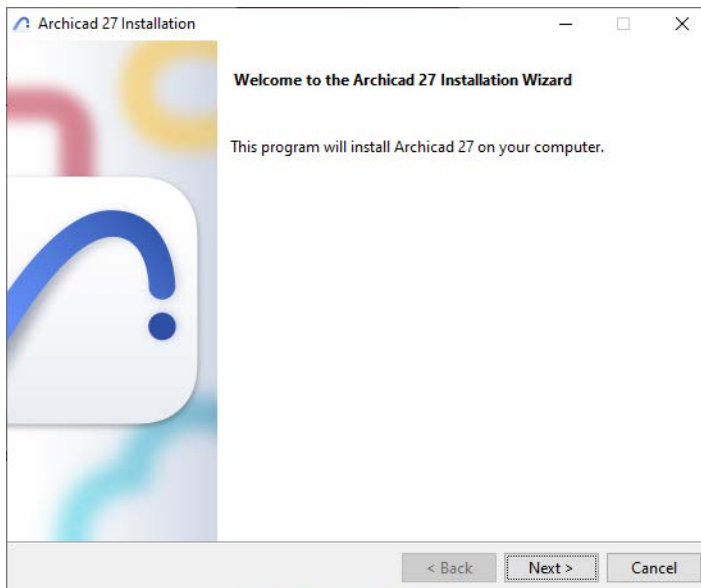
Hardware/Software requirements are modified from time to time. For up to date information on system requirements, please consult the following link:

http://www.graphisoft.com/support/system_requirements

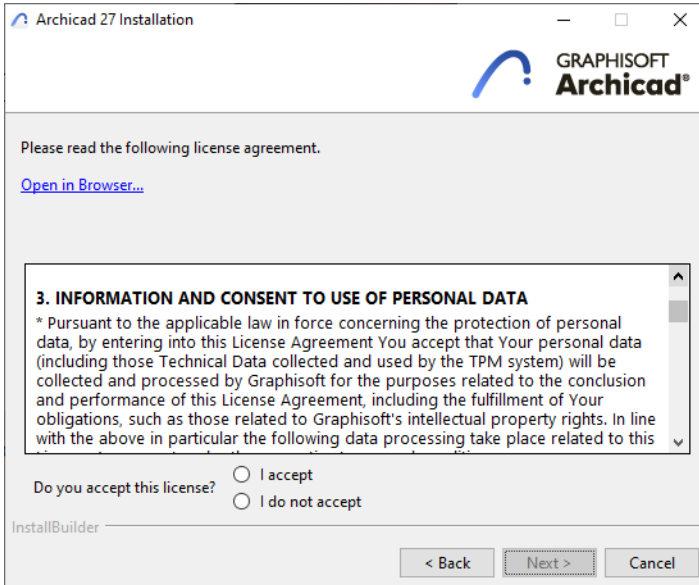
Installation Process

1. Start the installer to open the Welcome Screen.

Click **Next** to continue.



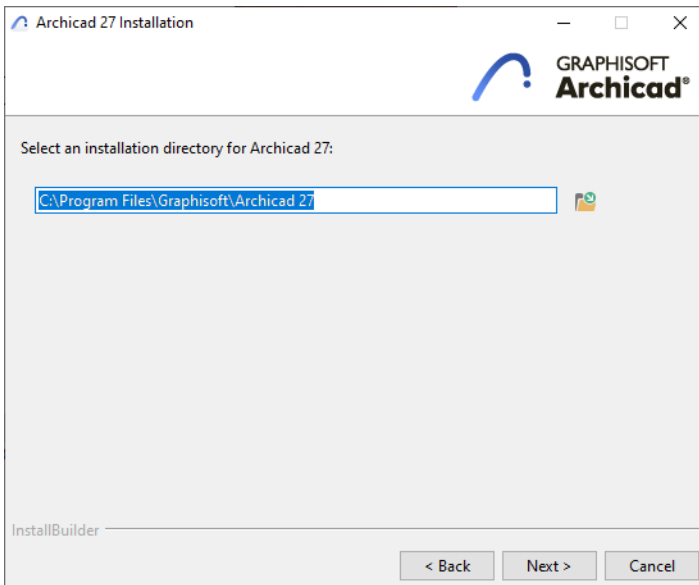
2. License Agreement



Read the license agreement carefully and accept it. (Click **Open in Browser** to see the agreement in a new window.)

Click **Next**.

3. Select Installation Directory

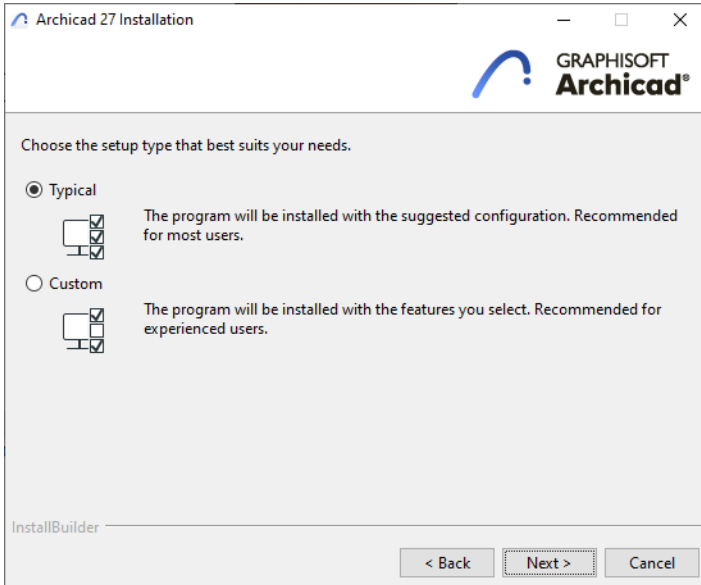


On this page, define the file path of the installation. As a rule, you should choose the path suggested by default, to simplify the installation of any future updates.

– On Mac: Installation must be in the Applications folder or any of its sub-folders.

To confirm the file path, click **Next**.

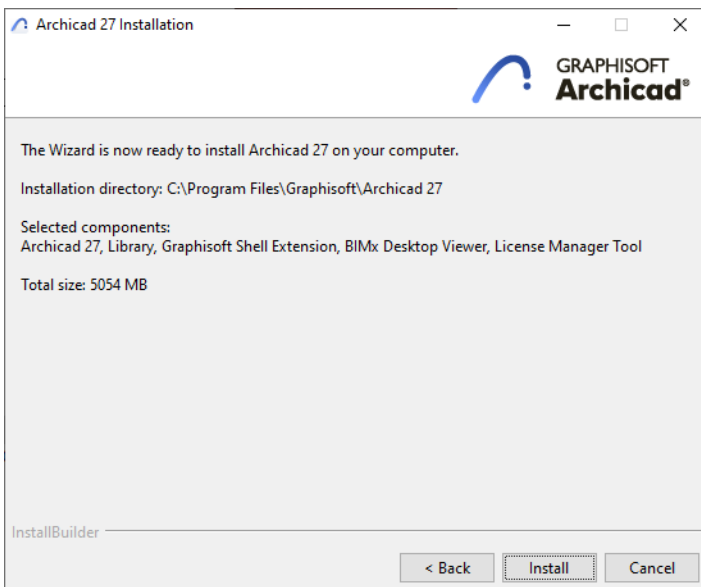
4. Choose Setup Type



- Typical (recommended): Installs Archicad 27 together with all of its components (e.g. Library).
- Custom: Installs only specific program components.

Click **Next**.

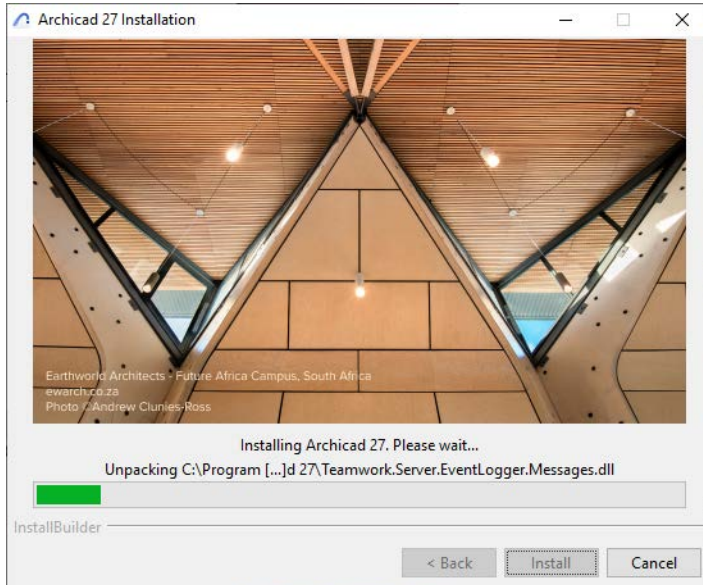
5. Installation Summary



This screen summarizes your chosen configuration.

Click **Install** to begin installation.

6. Installation Underway

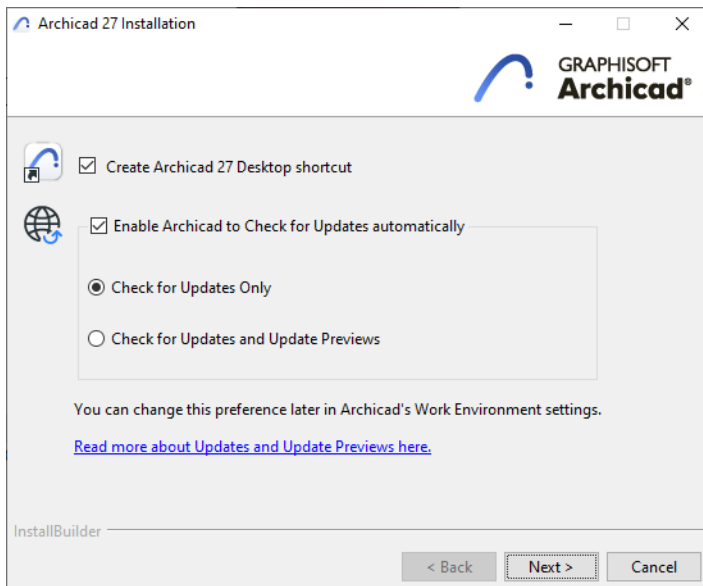


Depending on your hardware configuration, this process may take several minutes. In addition to other components, CodeMeter, the BIMx Desktop Viewer, and the License Manager Tool are all installed in the background.

7. For Mac users: if you are installing Archicad on a macOS system, you will see a warning: for Archicad to run properly, you must enable Full Disk Access in macOS settings. Follow the steps described in the warning dialog.

See [MacOS: Enable Full Disk Access](#)

8. Create Shortcut; Set Notification Preferences



- Optionally: **Create Archicad desktop shortcut** (Win only).

- Optionally: **Enable Archicad to Check for Updates automatically.**

If you check this, then Archicad Action Center will notify you whenever a new Archicad and/or Library update is available.

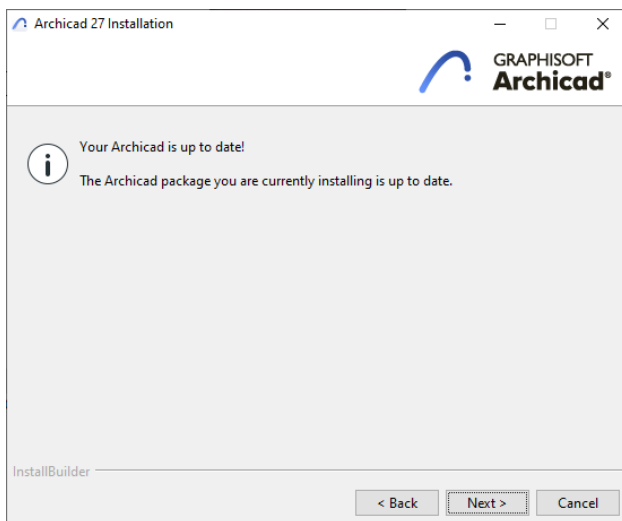
You can limit the type of notification:

- **Updates Only** (default): Get notified only if a final (fully tested) update is available
- **Updates and Update Previews:** Also get notified whenever a preview version of an update is available. Update previews are internally tested but not yet ready for wide release.

(You can change your notification preferences at any time in Archicad's Work Environment dialog. See [Check for Updates and/or Update Previews](#))

Click **Next**.

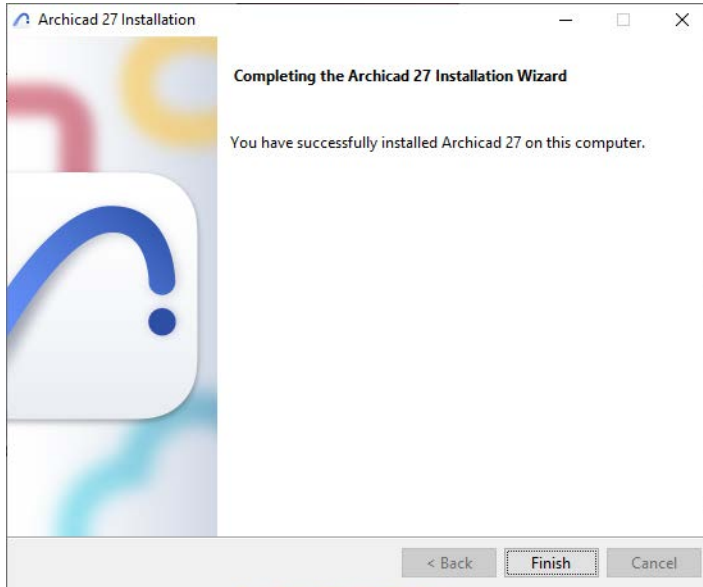
9. If you enabled automatic Check for Updates: the installer now runs an initial check and informs you if any updates are available for the version you are now installing.



If updates are available, finish installing this package, then install the update.

Click **Next**.

10. Complete Installation



Click **Finish**. You may be prompted to restart your computer.

Silent Install for Archicad and Add-ons

It is possible to use a silent install process to install Archicad using a customized, company-preferred configuration.

For details, see [Network Installation for BIM Managers](#).

MacOS: Enable Full Disk Access

When installing or updating Archicad on a macOS, you will see a warning: for Archicad to run properly, you must enable Full Disk Access in macOS settings.

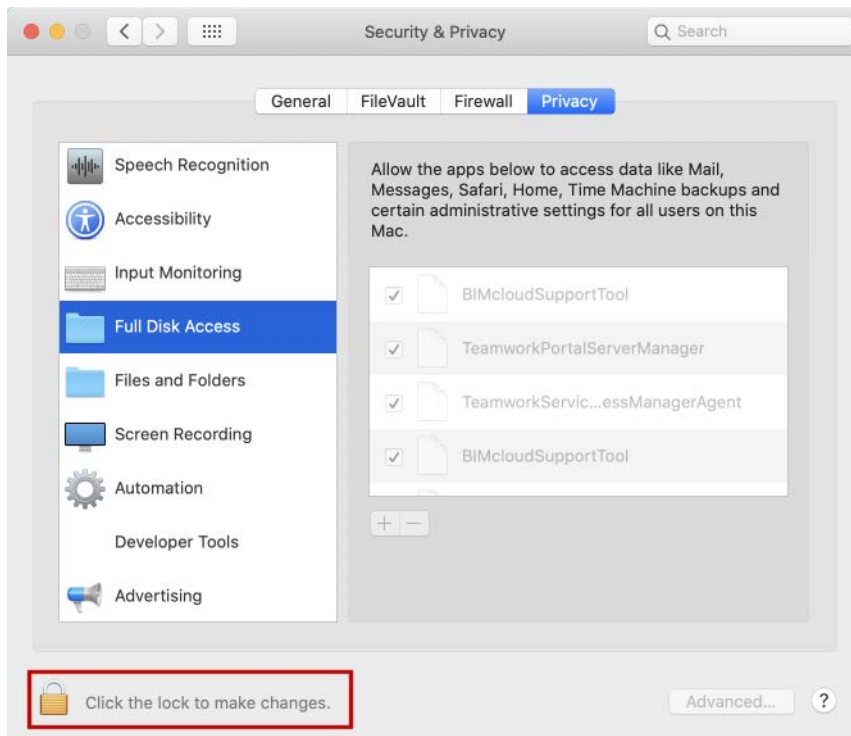
Follow these steps:

1. In the Archicad Warning dialog, click the link: **Go to Security & Privacy Settings**.



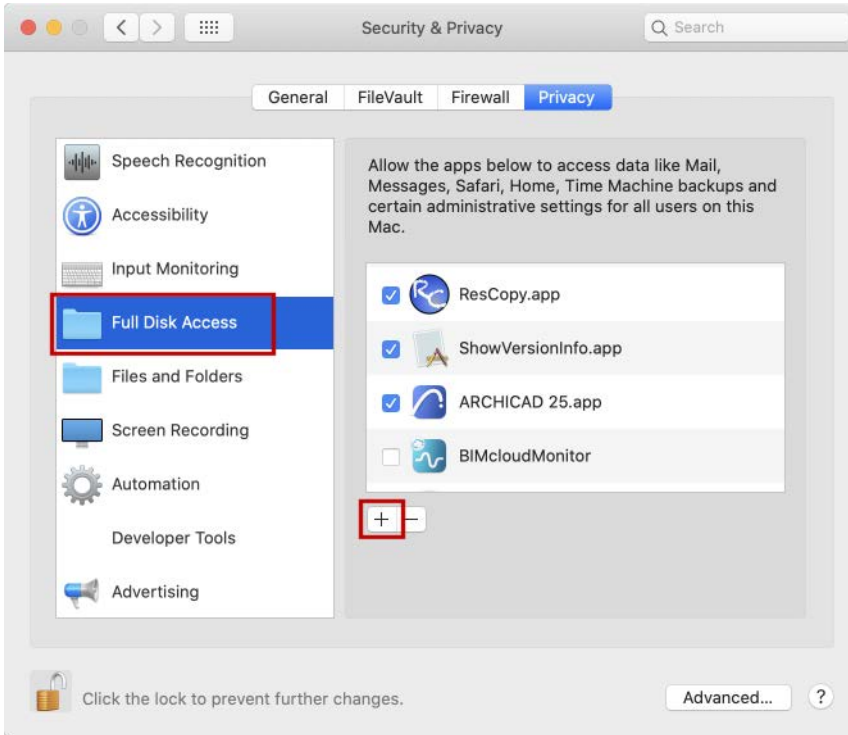
The Privacy panel of your computer's Security & Privacy Settings opens.

2. At bottom left, click the Lock icon so you can make changes. You may need to enter a password.

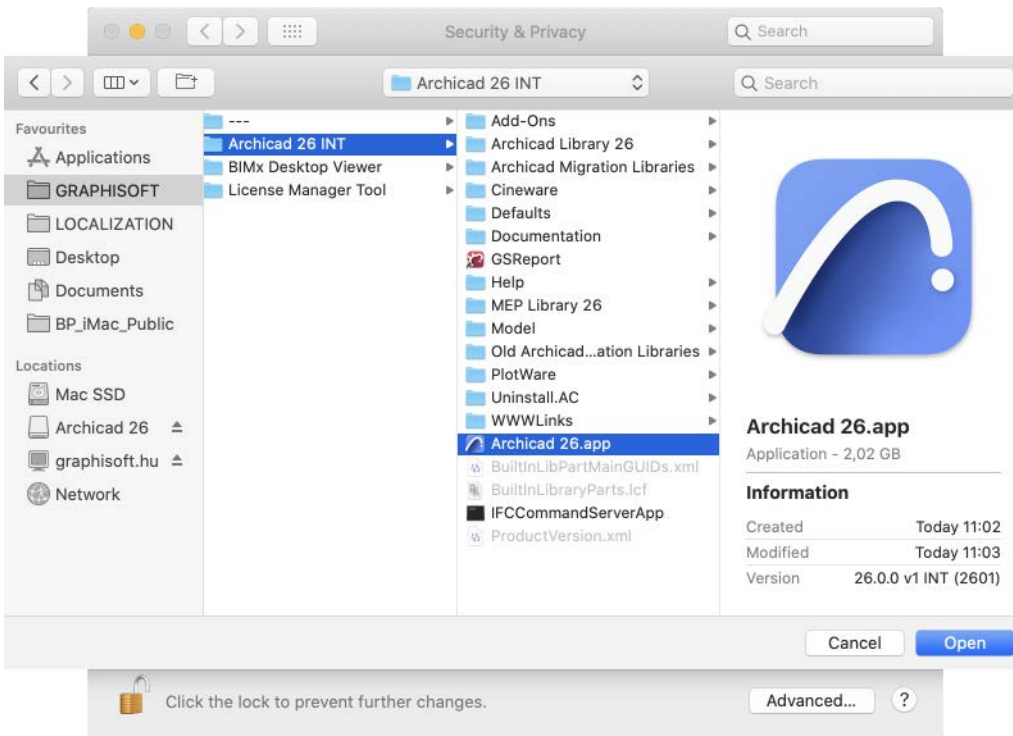


3. Select **Full Disk Access** from the list of applications in the left column of the panel.

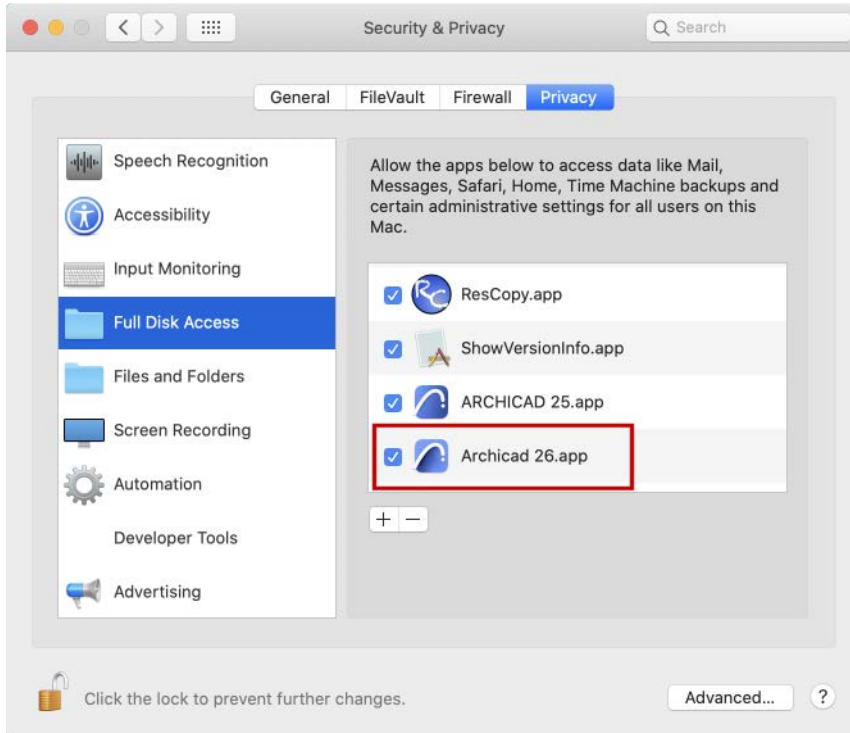
4. At the bottom of the list, click the **Plus** sign.



5. Navigate to **Applications > Graphisoft > Archicad Folder > Archicad Application**.



6. Click **Open** or drag and drop to add it to the list.



7. Return to the Archicad Installer and click **Next** to continue.

Your Archicad License Key

When you start Archicad, the program automatically checks to see if you have a license available. If so, you are ready to work.

To update, download and otherwise manage your license keys, use the License Manager Tool.

The License Manager Tool is automatically installed to your computer as part of Archicad installation. It is also available for download at

<http://www.graphisoft.com/downloads/licensing/>.

Hardware Key

The Hardware Key (Dongle) contains your license information on a USB stick. Before you start using the latest version of Archicad, this Hardware Key must be updated.

For the update process, use the License Manager Tool:

<http://www.graphisoft.com/downloads/licensing/>.

Start the License Manager Tool (LMT) and follow the directions to [update your Hardware Key](#).

Now you are ready to use Archicad.

Note: After updating your license key to Archicad 27, you can still use older-version Archicads (versions 17-26).

Software Key

The Software Key is a software-based license. Using the License Manager Tool, a software key can be downloaded from the License Pool to a local client computer. To update your software key to the latest Archicad version, start the License Manager Tool and follow the directions to [update your license](#) to the latest version.

If you are using a single license and plan to work on a different computer, you should upload the software key to the License Pool, so that you can access it from a different location. (Internet connection is required.)

Note: If you are using a network license, there is no need to upload the software key to the license pool.

Lost or Defective License Key

In case of a lost or defective License Key, contact your local distributor.

Related Topics

[Archicad license types](#)

Add/Repair Archicad

If you already have Archicad installed, and start the **Installation Wizard**, you are given two options:

1. **Add** will add new features to your current installation. You will see all available features in a list and you can check features to be added to your current installation.
2. **Repair** will reinstall Archicad with the features of your current installation.
 - The Repair process will revert your Archicad version to its default settings. After Repair, you must reinstall the latest Archicad and Library updates.
 - Project files you saved into the Archicad folder will not be deleted or overwritten by the repair process.

Updates for Archicad

Check for updates manually

To check for available updates of your localized version of Archicad or Archicad libraries, start Archicad and do any of the following:

- Use **Help > Check for Updates**
- Open the Action Center (**File > Info > Action Center**) and select the Updates item to see notifications on any updates available for download.

Check for updates automatically

Set the Check for Updates options in **Options > Work Environment > Network and Update**. You will be notified if a new library or program version matching your Archicad language version is available, which you can then download.

See [Check for Updates and/or Update Previews](#).

Install the update

The **Check for Updates** command takes you to the Graphisoft update website. If a new update patch is available, the web page automatically offers you the updater package that fits your current Archicad installation.

Note: If you have an updater patch installed and use the **Add/Repair** function of the Archicad Installer, you will be prompted to re-apply the updates after the modification or repair is carried out. The Archicad update option is available to holders of the Commercial or Education Archicad license.

Other downloads

You can find other downloadable materials (example files, goodies etc.) at

www.graphisoft.com/support/archicad/downloads.

Archicad Tools and Utilities

The following utilities are automatically installed on your computer together with Archicad.

CodeMeter system components	A Hardware Key driver (CodeMeter) will be installed as part of the installation process for Archicad.
Plotter Drivers	Archicad includes a number of built-in drivers for a wide variety of HPGL compatible plotters. If you are using a large-format printer, make sure you use a printer driver specified by the manufacturer, because its plotter function is only an emulation.

The following utilities are also installed with Archicad. If you prefer not to install them, use Custom Installation and uncheck these options.

Graphisoft Shell Extension	Optional, for Windows only. Utility for extended Graphisoft file property information on Windows.
BIMx Desktop Viewer	Explore the interactive, walk-through BIMx Hyper-model in the context of the hyperlinked 2D project documentation and embedded BIM data.
License Manager Tool	The License Manager Tool can be used to manage the Graphisoft protection keys you have purchased, whether hard or soft.

For more information about additional utilities, open the folders of the applications in the Archicad main folder to find detailed descriptions.

Uninstall Archicad

On Windows

It is best to use the “Programs” or “Add or Remove Programs” utility, accessible from the Control Panel.

On Mac

Navigate to the “Uninstall.AC” folder in your Archicad folder, and select the “Uninstall.app” application.

Uninstall process

The Uninstall Wizard removes files which were installed by the Archicad Installation Wizard.

CodeMeter and associated components are not removed.

The uninstaller will not remove any files you have created in the Archicad main folder - e.g. Archicad project files that you saved into the Archicad main folder.

Silent Uninstall for Archicad and Add-ons

[See *Unattended Uninstall*.](#)

Using a CodeMeter Server

If you would like to use network licenses (e.g. instead of single licenses), you will need a CodeMeter hardware key or software key. Contact your local Graphisoft partner to order one or to exchange your existing single key(s).

Single or network keys

A so-called **single key** can hold one license for each Graphisoft product (Archicad, EcoDesigner STAR) and can be used locally. You connect the hardware key to your computer's USB port or download the software key with License Manager Tool (LMT); if you have the CodeMeter driver installed (it is installed with Archicad), your Graphisoft product will find a license and will run in commercial mode. Each workstation in an office needs to have an individual key.

A **network key** can hold multiple licenses related to each product (Archicad, EcoDesigner STAR). To share them across multiple computers you need a CodeMeter server. This arrangement provides more flexibility than having individual keys, as the licenses are allocated only when they are being used on one of the clients.

System requirements for CodeMeter Server

The CodeMeter server is part of the CodeMeter driver, so you only need to select a computer on your network that will act as a CodeMeter server. To set up a CodeMeter server as a network license server, you will need the following:

- A computer with network connection (should be running during work hours).
- The computer must have at least one free USB port, for hardware key only.

Note: There is no need for a server-level operating system for the CodeMeter server. You can use one of the workstations as a CodeMeter server, and you must keep it running during work hours. Any computer that has recent Windows or OS X versions can be used.

Set up the server

After you have your server running and the CodeMeter driver is installed, start the server as described below. All CodeMeter settings can be managed in CodeMeter WebAdmin. To start it:

- Open CodeMeter Center (from Applications or using the Start menu) and click on WebAdmin;
- or simply open localhost:22350 on your browser.

To enable the CodeMeter server:

1. Go to Configuration/Server Access tab.
2. In the Network Server section, check **Enable**.
3. In the CmWAN Server section, check **Enable**, if you want to use the computer in a wide area network (WAN) and allow license accesses
4. Click **Apply**.
5. Restart CodeMeter service (on Mac: System Preferences/CodeMeter, on Win: CodeMeter Control Center)

From that point, the server runs and shares licenses on the local network.

Client-side settings

All clients on the local network should find the server automatically. In case this does not work (e.g. your server is in a different subnet, or you are connecting remotely through the Internet), then you may need to specify the server's hostname IP address. On the client:

1. Open WebAdmin.
2. Go to Configuration/Basic/Server Search List.
3. Add the server to the Server Search List.

This field can have multiple entries; the client searches for available licenses in the specified order and only on the specified servers. Only when the list is empty does the client search for other available servers.

Access control

With the default settings, all computers on the same local network (same subnet) can use a license from the server.

To restrict access:

1. Open WebAdmin.
2. Go to Configuration/Server/License Access Permissions.

If the Clients' list is empty, anyone on the local network can use a license. If at least one entry is listed here, only the listed entries can request a license.

Note: If the server is public on the Internet, or no one is allowed to use a license, make sure that this list is filled out correctly to avoid unwanted license usage.

You may want to check who is using a license:

1. Open WebAdmin on the server computer.
2. Go to License Monitoring/Sessions.

Here you can see a list of active clients and clients who have borrowed licenses.

License Borrowing

If your network key has a license borrowing product, the clients can borrow a license.

For more information, see article on [License Borrowing](#).

Network Installation for BIM Managers

The standard installer for Archicad, Archicad Updates, and Graphisoft Add-ons can be deployed in unattended mode.

You can use unattended installation with either the default installation options or custom options where applicable - such as a different installation path, or a customized AC Work Environment.

This article describes how to deploy unattended installation with either default or custom options.

Note: Administrator permission is required to run the executable file.

Unattended Installation - Default Options

In the command line, add the executable file followed by the parameter `--mode unattended`

For example:

Win

```
Archicad-27-INT-3000-1.0.exe --mode unattended
```

Mac (ARM processor)

```
Archicad\ 27\ Apple\ Silicon\ Archicad\ Apple\ Silicon\ Installer.app\ Contents\ MacOS\ installbuilder.sh --mode unattended
```

Mac (Intel processor)

```
Archicad\ 27\ Archicad\ Installer.app\ Contents\ MacOS\ installbuilder.sh --mode unattended
```

This will launch the installation without user interface and without any required user input.

If no further parameters are set, the installation will run with default settings.

To run the installation using custom options, please keep reading.

Unattended Installation - Custom Options

There are two ways to set up custom settings for the unattended installation:

- Add command line parameters, in any order
- Add an option file (containing a list of command line parameters). Use an option file if you are using multiple parameters whose character count might exceed the command line limit.

Topics in this section:

[Command Line Parameters \(Custom Install\)](#)

[Option File \(Custom Install\)](#)

[Installer Log Files and Error Codes](#)

[Unattended Uninstall](#)

[Third-Party Prerequisites](#)

[Customized Company Defaults for Unattended Installation \(Archicad Only\)](#)

Command Line Parameters (Custom Install)

The following table lists the parameters that can be used to customize unattended installation of Archicad, Archicad Updates, and/or Graphisoft Add-ons, as applicable.

In the command line, type the executable file name followed by one or more parameters as detailed below.

Note: There are two dashes in front of each parameter. Be careful when adding parameters using copy/paste, because sometimes the two dashes are converted into a single dash, which will cause errors.

To do this	Type this in the command line
Run installation in Unattended Mode	<code>--mode unattended</code>
List available standard parameters See a list of general command-line parameters for customizing unattended installation.	<code>--help</code>
Specify installation folder (if you don't specify, the program will be installed to the default location.)	<code>--installdir</code> followed by the path of the preferred installation folder. If the path contains spaces, enclose the entire path in quotation marks. Note: The uninstaller will also be located at this path. See also Unattended Uninstall
Install additional (non-default) components Use this option to select additional ("allowed") components for installation, which would not be installed by default (e.g. certain Goodies in the Goodies Suite installer).	<code>--enable-components</code> followed by a comma-separated list of the allowed (non-default) components to install. To see list of allowed components: type <code>--help</code> .
Disable installation of (default) components Use this option to prevent installation of certain components, which otherwise would be installed by default.	<code>--disable-components</code> followed by a comma-separated list of default components to omit from the installation. To see list of default components: type <code>--help</code> .
Identify Current User (Mac only) Installation on Mac is run via a "root" user. In case of deployment installation (e.g. Apple Remote Desktop), use this parameter to specify the current user - for example, to create desktop and dock shortcut for a specific user (rather than the "root" user).	<code>--currentUserName</code> followed by the username.

To do this	Type this in the command line
Use an Option file containing command line parameters	<code>--optionfile</code>
	followed by name of the option file (as well as its directory path, if it is not in the same folder as the executable). For info on how to set up the Option File, see Option File (Custom Install) .

Additional Command Line Parameters (for Archicad Only)

These parameters are not listed in the --help file, but can be used to customize unattended Archicad installation.

Default installation option is shown in red.

To do this	Type this in the command line
Add AC Shortcut to Desktop Added by default. If you don't want to add it, replace 1 with 0	<code>--desktopshortcut 1</code>
	<code>--desktopshortcut 0</code>
Add AC Dock Icon (Mac only) Added by default. If you don't want to add it, replace 1 with 0	<code>--dockshortcut 1</code>
	<code>--dockshortcut 0</code>
Enable Automatic Update Enabled by default. To disable it, replace 1 with 0	<code>--enableautomaticupdate 1</code>
	<code>--enableautomaticupdate 0</code>
Automatic Update - Notification Preference If <code>enableautomaticupdate = 1</code> : this additional parameter defines which update type triggers a notification in Archicad. Default value is <code>regular</code> . Notifies for Updates only. To notify for both Updates and Update Previews: use the <code>insider</code> value.	<code>--updatechannel regular</code>
	<code>--updatechannel insider</code>
Install predefined company-standard defaults for <ul style="list-style-type: none"> • Work Environment, • Archicad Template, and/or • DXF-DWG Translators 	<code>--customEnvironment</code>
	followed by the path of the Defaults folder containing your preferred files. For info on how to set up the Defaults folder, see Customized Company Defaults for Unattended Installation (Archicad Only) .

Additional Command Line Parameters: EDU-University and Regional Languages Only

To do this	Type this in the command line
<p>Archicad EDU - University Licenses only</p> <p>Provide User ID and Serial Number for unattended install. Not available for individual students with EDU licenses.</p>	<pre data-bbox="545 275 831 300">--eduSerialNumber</pre> <p data-bbox="545 310 1334 336">followed by 20-character serial number (no spaces, no separators)</p> <pre data-bbox="545 369 735 394">--eduUserID</pre> <p data-bbox="545 405 1254 430">followed by the edu User ID (usually your registration email)</p> <ul data-bbox="545 447 1433 682" style="list-style-type: none"> • If the User ID contains spaces, enclose the User ID in quotation marks. • If these parameters are incorrect, installation will still take place, but Archicad will run in Demo mode only. Reinstall as needed using the correct data. • Test the installation with your user data before deploying unattended install. <p data-bbox="545 701 668 726">Examples:</p> <pre data-bbox="545 760 1169 785">--eduSerialNumber SY6NQKC0K19P06V89S1B</pre> <pre data-bbox="545 808 1139 833">--eduUserID JohnSmith@graphisoft.com</pre>

Option File (Custom Install)

As an alternative to using command line parameters to customize the unattended install, you can create an option file.

Thus, instead of typing parameters into the command line, you will list the needed parameters in a text file, then add this text file to the command line.

This can be useful if you are using multiple parameters whose character count might exceed the command line limit.

1. Create an option file.

- The file must be a utf8 text file
- In the first line, type: `# encoding: utf-8`
- The file must contain a list of command line parameters, each followed by an equals sign, followed by the custom value(s).
- Start a new line for each parameter
- Do not use `--` in front of the parameters
- Do **not** use quotation marks in parameters - not even if the specified value contains spaces

Example

Contents of an option file to install the Archicad26.exe file, using unattended install, to a customized folder location on your D drive, and including all components except Archicad Help:

```
# encoding: utf-8
```

```
mode=unattended
```

```
installdir=D:\DATA folder\
```

```
disable-components=ArchicadHelp
```

2. Save the file as a utf8 text file, e.g. "OptionFile.txt"

3. Add the resulting option file to the command line preceded by `--optionfile`

```
Archicad-27-INT-3000-1.0.exe --optionfile OptionFile.txt
```

If either the executable or the Option file is not located in the current directory, you must specify its full path:

```
C:\Users\CurrentUser\Downloads\Archicad-27-INT-3000-1.0.exe --optionfile  
D:\DATA\OptionFile.txt
```

Note: If the file path (specified in the command line) contains spaces, enclose the entire path in quotation marks.

Installer Log Files and Error Codes

After running unattended installation, the associated log files are found here:

Win

```
C:\ProgramData\Install.GS\yyyy.mm.dd.hh.mm.ss.log
```

Mac

```
/Users/Shared/Install.GS/ yyyy.mm.dd.hh.mm.ss.log
```

Error Codes

- 1: Not Supported Operating System, OptionFile syntax error, Invalid parameter, misc.
- 2: InstallDir is not empty
- 3: Add / Repair not allowed in unattended mode, already installed
- 4: InstallDir is in an Archicad or BIMcloud folder
- 5: VCRedist 2015 cannot be installed
- 6: There is not enough disk space in the selected installDir
- 7: In Add-on Installer: Archicad search detection error (Archicad not found...)
- 8: InstallDir's path contains invalid characters: <>"|?*

Unattended Uninstall

Use the `--mode unattended` parameter to uninstall Archicad or Add-ons in unattended mode.

Notes:

- Be careful when adding parameters using copy/paste, because sometimes the two dashes are converted into a single dash, which will cause errors.
- The following examples assume that the program(s) were installed to their default location. If the `--installdir` parameter was changed for installation, then the path shown below must be modified accordingly.

Win

```
C:\Program Files\Graphisoft\Archicad 27\Uninstall.AC\Uninstall.exe --mode unattended
```

Mac

```
/Applications/Graphisoft/Archicad 27/Uninstall.AC/Uninstall.app/Contents/MacOS/installbuilder.sh --mode unattended
```

Third-Party Prerequisites

The installation process for Archicad includes a number of additional third-party software products which are necessary to run Archicad.

When you run the Archicad installer, it will automatically install those third-party products which are not yet installed, or which are out of date.

If you don't want to install these automatically during unattended installation, make sure they are installed separately beforehand.

For a list of these products, see:

[Third Party Installers for Archicad](#)

Customized Company Defaults for Unattended Installation (Archicad Only)

This description applies only to Archicad installation.

It details how to set up a Defaults folder for use with the `--customEnvironment` parameter, used to customize the unattended installation.

Introduction

When installing Archicad in unattended mode, the default installation includes predefined settings for:

- Work Environment
- Template file
- DXF-DWG Translators

However, you may prefer to use custom options (e.g. your own company preferences) for any or all of these factory defaults.

To do this, you must

- Define your preferred custom settings in Archicad
- Set up a “Defaults” folder location using the hierarchy described below
- Add your customized settings to this folder/sub-folders as described below
- Add the path of this “Defaults” folder to the command line or the Option file.

[See more at *Command Line Parameters \(Custom Install\)* and *Option File \(Custom Install\)*.](#)

Hint: If you want the installed Archicads to contain *only* the Templates and Work Environment profiles that you have customized, then - once you have added your customized files to the “Defaults” folder - you can delete the factory defaults from there.

Step by Step

1. Complete a Standard Single Installation

The first step is to do a Standard installation of Archicad on a single client machine.

Next, set up your preferred Work Environment, Template, and/or DXF-DWG translators:

2. Customize Work Environment Settings

Use the dialog boxes at **Options > Work Environment** to set up your company standard Work Environment settings. Most such options are found in three dialog boxes of Company Standard Schemes: **Data Safety & Integrity**; **Network and Update**; and **Special Folders**.

Once you have set these options, save these settings as schemes, then save the schemes in a company-default profile. To save a profile, go to **Options > Work Environment > Work Environment Profiles**.

3. Customize Templates

Create and save any number of template files. Template files include links to libraries loaded through Library Manager (e.g. “Office Standard Library”). Templates can also include Project Preferences (e.g. Working Units, Zone calculation preferences), and such items as Master Layouts, Project Info, placed elements (e.g. Elevation markers, entire buildings).

Save each template using **File > Save as...** (Archicad Template file *.tpl).

4. Create “Defaults” Folder

Create a temporary “Defaults” folder. Make sure it is in a location accessible by all users.

See also [Note on Ensuring Network Access to “Defaults” Folder \(WIN Only\)](#):

Within this “Defaults” folder, create sub-folders and add your saved, customized defaults as described in the next steps.

5. Create “Defaults/Archicad” sub-folder.

- To the Archicad sub-folder, add your saved “WorkEnvironment” folder and its contents.

Where to find saved WorkEnvironment folder:

Users\<>Username>\AppData\Roaming\Graphisoft\Archicad xx.0.0 yyy **(Windows)**

~/Library/Preferences/Graphisoft/AC xx.0.0 yyy **(Mac)**

where xx stands for the version number of Archicad and yyy is the language version of Archicad.

- Also to the Archicad sub-folder, add your saved Template file(s) (extension .tpl).



6. Create “Defaults/DXF-DWG Translators” folder.

- To the DXF-DWG Translators sub-folder, add the contents of your saved Translator folders.

Where to find saved Translator folders:

Users\<>Username>\Graphisoft\DXF-DWG Translators xx.0.0. yyy **(Windows)**

~/Library/<Username>/Graphisoft/DXF-DWG Translators xx.0.0. yyy **(Mac)**

7. Add the DXF/DWG Translator List.xml file to the “Defaults” folder.

Where to find the Translator List XML file:

Users\<>Username>\AppData\Roaming\Graphisoft\Archicad xx.0.0 yyy R1\Dxf-Dwg Preferences\
(Windows)

~/Library/Preferences/Graphisoft/AC xx.0.0 yyy v1. **(Mac)**

8. Provide the path of the “Defaults” folder as a command line parameter: `--customEnvironment`

- If the file path contains spaces, enclose the entire path in quotation marks.

Example

Note: Make sure you type the name of the “Defaults” folder to correspond to your language version.

Win

```
Archicad-27-INT-3000-1.0.exe --mode unattended --customEnvironment
"D:\...\Defaults"
```

Note on Ensuring Network Access to “Defaults” Folder (WIN Only):

- If your “Defaults” folder is in a network location, you should include *the full network path* in this parameter.
- If the folder location is on a mapped network drive, make sure the folder path is accessible regardless of user permissions (e.g. Administrator).

Mac

```
Archicad\ 27\Installer.app/Contents/MacOS/installbuilder.sh --mode unattended  
--customEnvironment "userhome/.../Defaults"
```

New Features in Archicad 27

Integrated Design Options

Enhanced attribute management

Tool and workflow improvements

Additional workflow improvements

Visualization

Multi-disciplinary design

Open BIM

File format - Open and save versions Archicad 17 and up

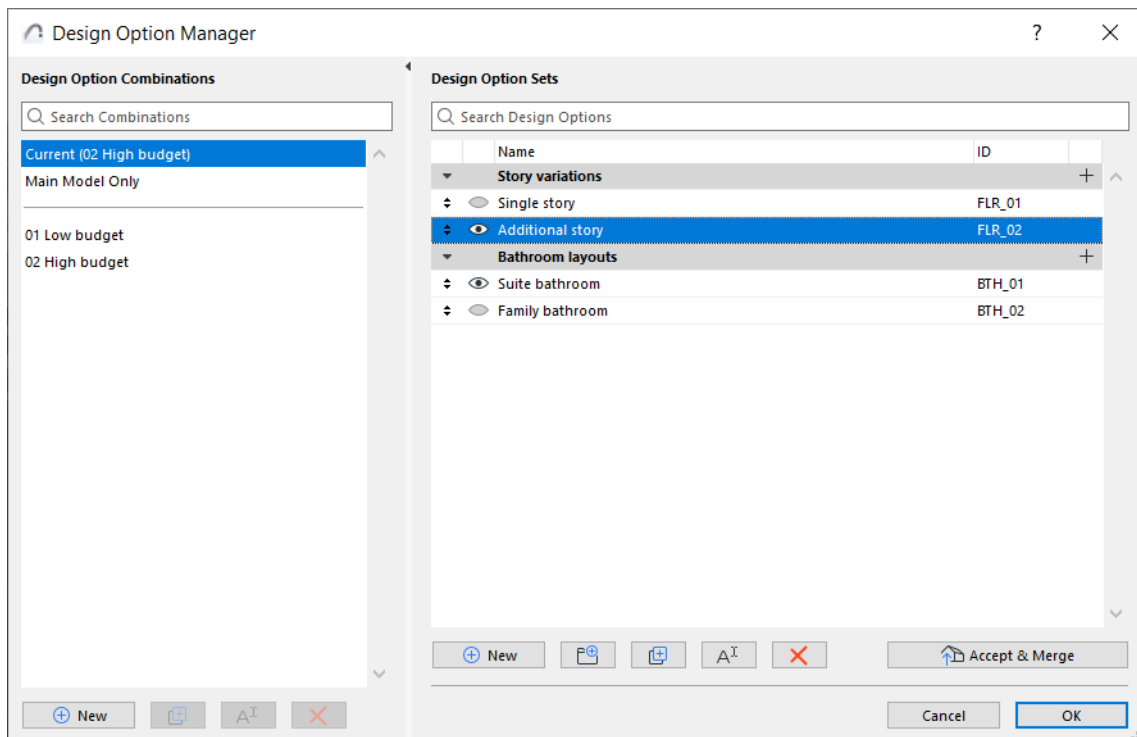
BIMcloud - Secure Teamwork login

Integrated Design Options

A new, dedicated solution in Archicad 27 supports managing multiple design options in an integrated and efficient way.

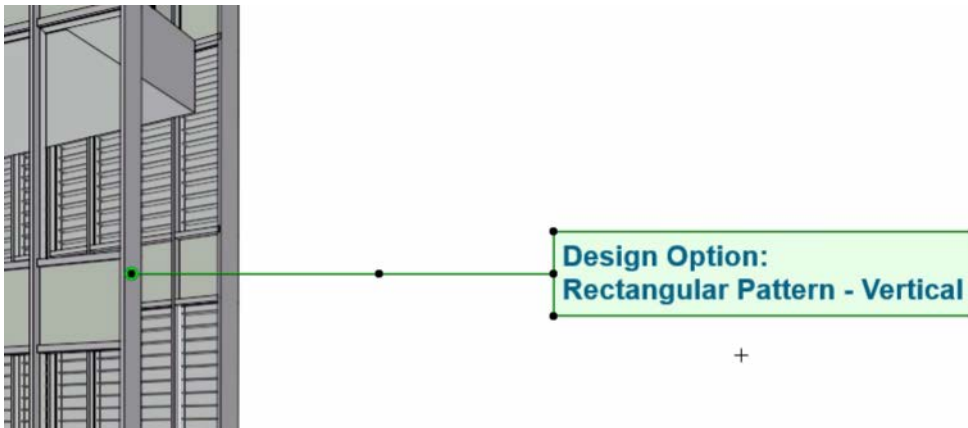


- Manage constantly changing designs and switch among alternative Design Options
- Make better design decisions in significantly less time, by evaluating and communicating design options efficiently -- for any project type and scale



Workflow highlights

- Create design variations and easily switch among them – whether they include the whole building, just parts of it, or multiple parts at the same time.
- Choose the best and most efficient design alternatives, in 3D or any 2D view – without having the different options intersecting each other through through solid element options, merge, crop or element junctions.
- Study designs in the context of their surroundings and evaluate them for aesthetics, performance, efficiency.
- Design Options are a view-based setting: document your options via any 2D or 3D output.
- Use the Design Options palette to easily turn on and off Options on-the-fly as you work.
- Focus on editing the current Option by locking, fading, or colorizing other elements outside of the current Option being worked on.
- Finalize Options by integrating a selected Option into the main model, or adapt Options along the lifecycle of a Project by merging, duplicating or rearranging Options between Option sets.
- Design Options are integrated into the Hotlinks workflow. Work with embedded variations, place a module with selected options, or simply change it later, all at the instance level. Save and publish Hotlink modules with all Options included, or just the visible ones for that view.
- Filter and extract BIM data according to different Options: create Option-based Graphic Overrides, and extract data via Schedules and Labels.
- Options are widely integrated into all 2D and 3D elements, including Annotation and Markers such as Sections, Elevations, Details and Worksheets.



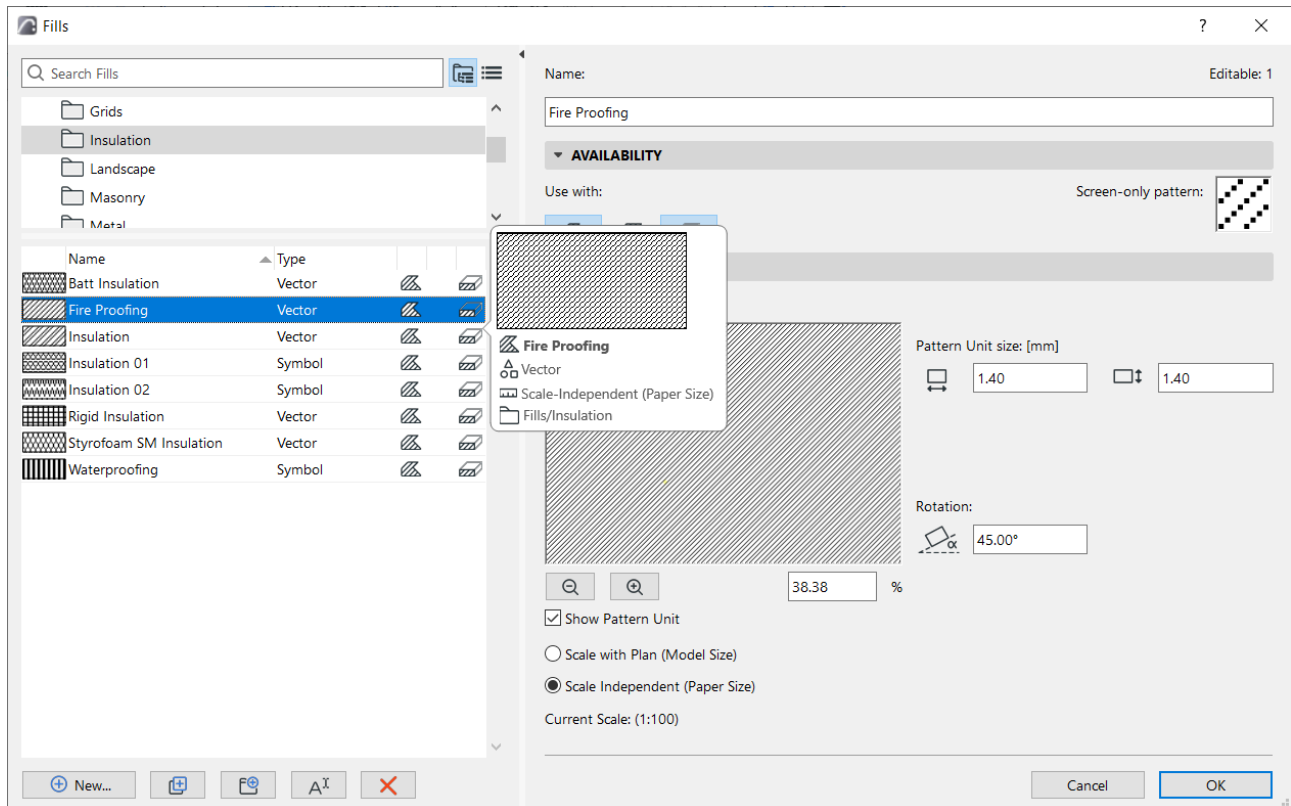
Curtain Wall Surface Areas by Design Option		
Design Option Name	Home Story	Surface Area
Rectangular Pattern - Vertical		
	L2	4,596.6
	L8	2,334.0
	L12	1,228.1
	L15	1,088.0
	L19	106.7
		9,353.4 m²

[Design Options](#)

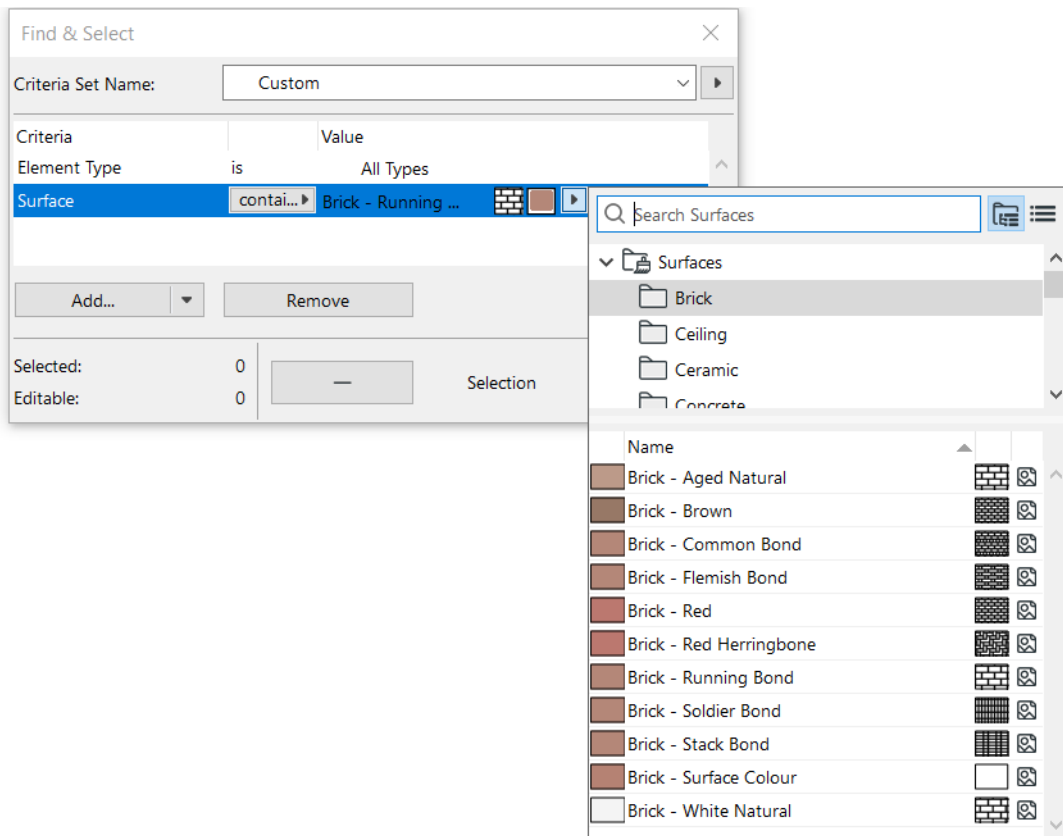
Enhanced attribute management

Improving the management of complex BIM project has been a continuous focus for Graphisoft in recent years.

Archicad 26 simplified the attribute management workflows by introducing folders for key attributes. Archicad 27 continues this development, introducing folders and search bar for additional attributes such as Fills and Lines.



- Attributes are organized in folders in the criteria dialogs, too - making it easier to set up criteria (for example, in Interactive Schedules or Find & Select).



- Besides extending workflows, we also optimized the usability and consistency of attribute-related interfaces to speed up attribute management.
- At attribute import: By default, now import the associated attributes too (previously you had to check them manually)
- All attributes lists: easier to choose the attribute you need, thanks to better previews, more pop-up info, smart search
- All attributes dialogs: use left-right arrow keys to navigate
- All attributes dialogs: fewer steps to create and name a new folder or new attribute (fewer dialogs in the process). To rename: just double-click inline and type.
- Easier access to associated attributes (keyboard shortcut instead of clicking)
- Layers dialog: rearrange the columns as you like (for example, move the Show/Hide icon to be next to Lock/Unlock)
- Attributes Palette: Expanded bubble provides more on-the-fly info about attributes, helping users identify and choose them more quickly (no need to navigate to separate dialog)

[Attributes](#)

Tool and workflow improvements

Distance Guides

Extended search capabilities

Custom groups for Graphic Override Rules

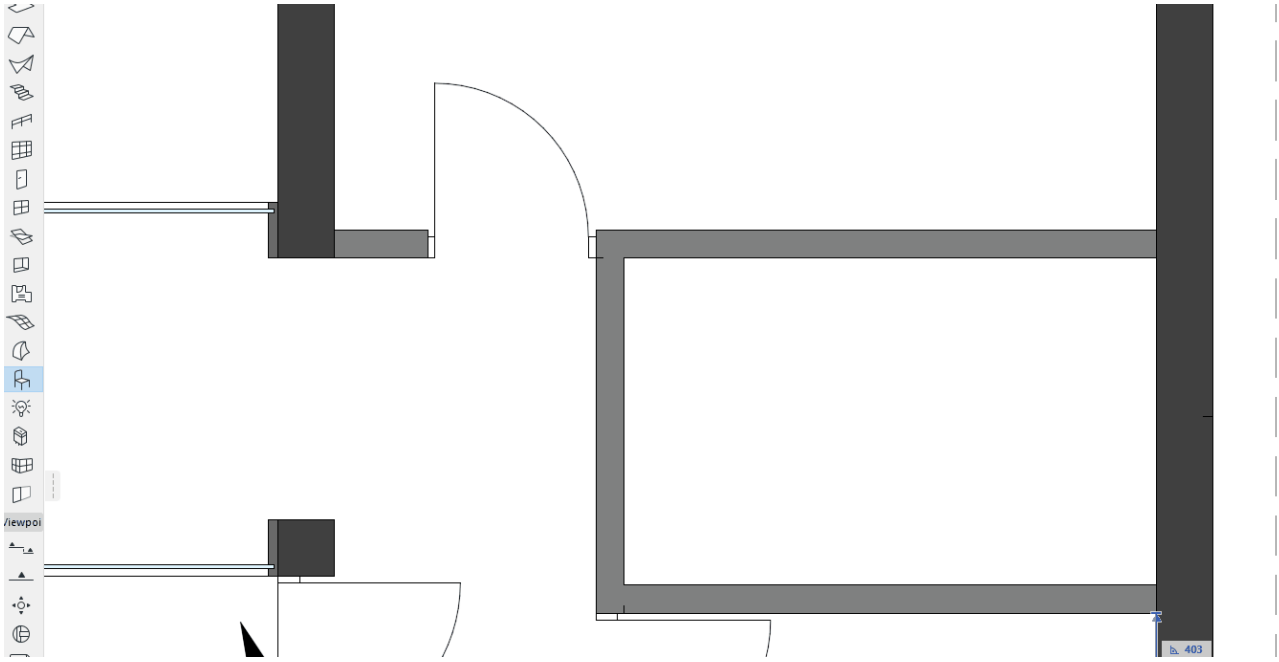
Improved format options for Interactive Schedule

Modernized GDL Editor

Additional workflow improvements

Distance Guides

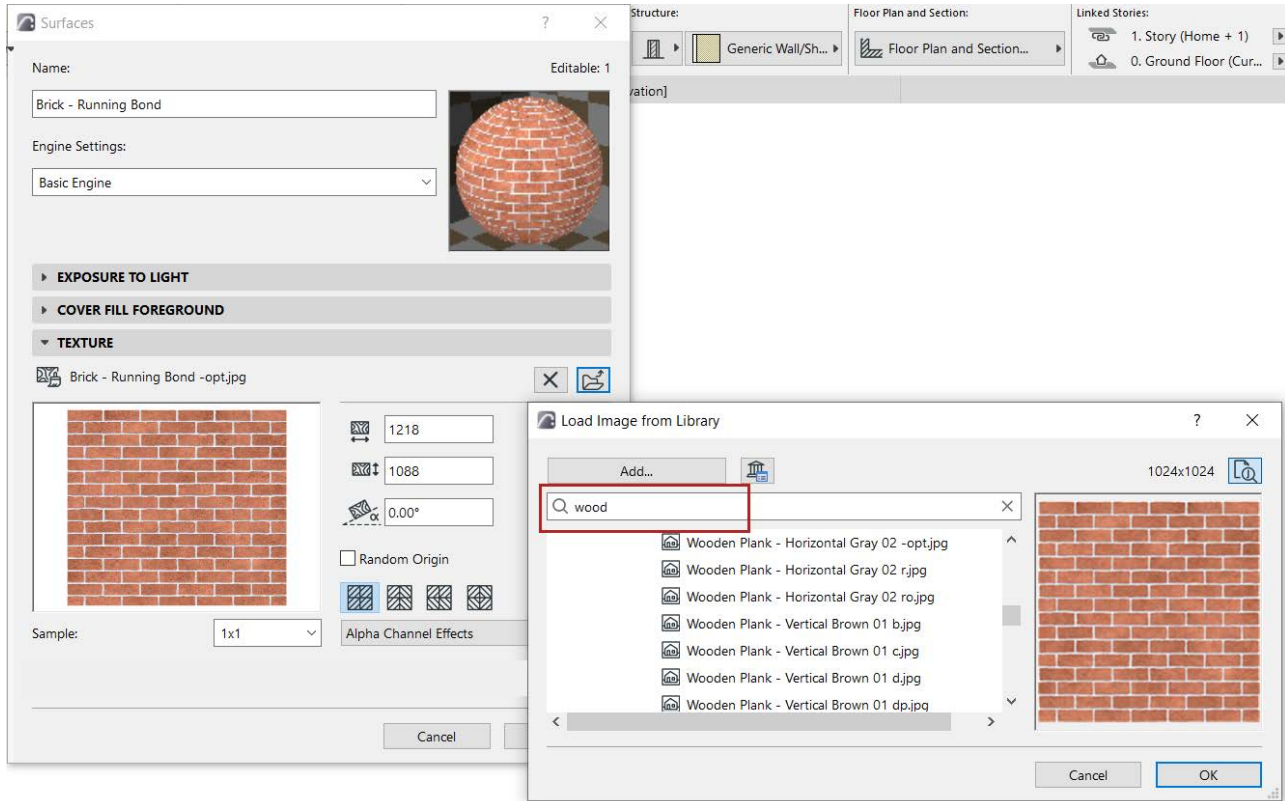
Place new elements or move selected elements more accurately, thanks to pop-up guides showing dynamic distances from surrounding elements. You can show Distance Guides during or after element placement. Works when inputting or editing Wall, Column, Beam, Window, Door, Opening.



[Distance Guides - Experimental feature](#)

Extended search capabilities

Enjoy new and improved Search in many essential Archicad dialogs for a faster, more consistent workflow. No more scrolling through long lists to find what you need. For example:



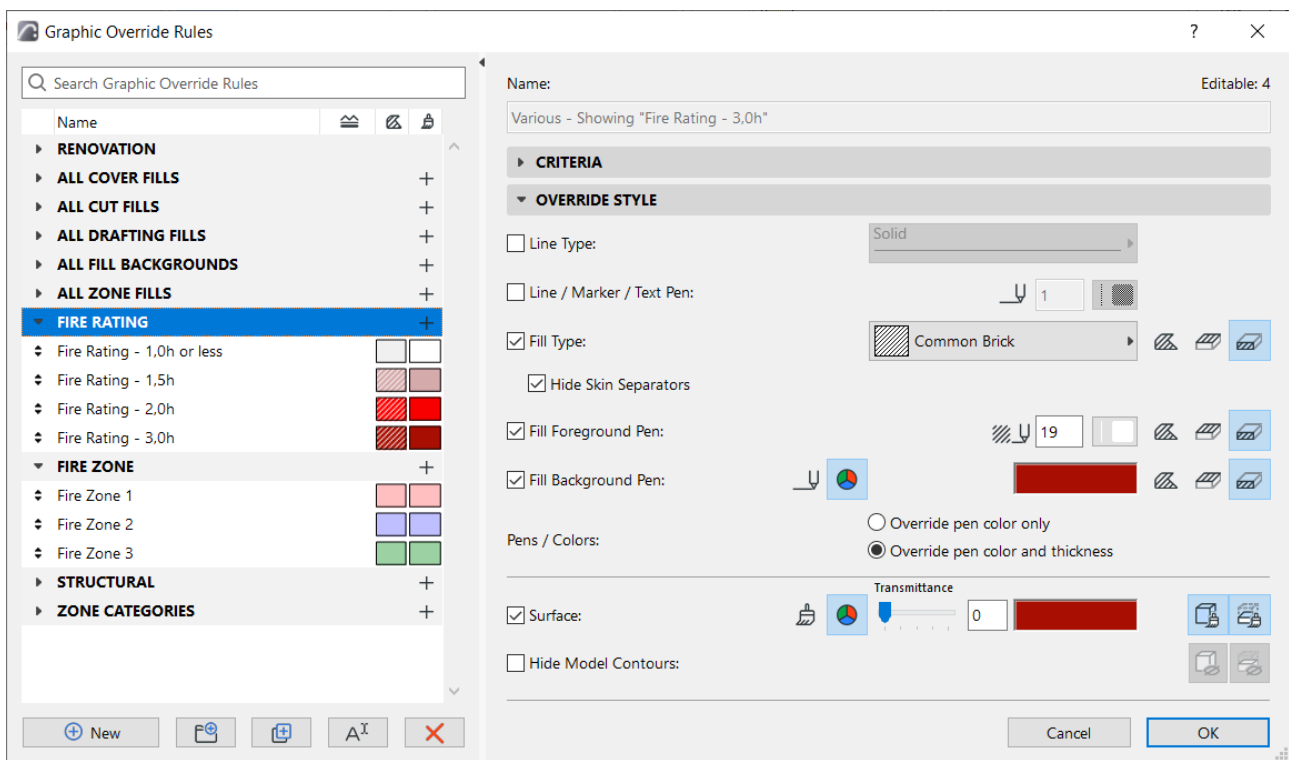
- Load Image from Library: Easily find any texture image to attach to a Surface
- Drawing Manager: Search for any drawing or external document
- Search in all attribute dialogs and lists (which use folders)
- Link Drawing: Search project views
- Graphic Override Combinations and Rules
- 3D Styles
- Rendering Scenes (Select and Manage Rendering Scenes)
- Favorites: Existing search has been improved - it now waits for additional keystrokes instead of generating reviews after every keystroke (this could cause slowdowns in long lists of Favorites).

After search, hit Escape in all search locations, to clear the search term and start again.

Custom groups for Graphic Override Rules

Archicad's Graphic Override feature is widely used for project audit and visual representation, based on BIM. Archicad 27 adds new ways to organize these settings:

- Instead of long lists of rules, create any custom grouping for Graphic Overrides Rules - organize, sort, and easily find what you need
- Drag and drop rules among groups
- Open groups to see the rules you need, or close them to avoid clutter
- New: context menus with all relevant commands
- Create new or duplicate combinations and rules directly in place, without additional dialogs
- Add entire groups of rules to a combination



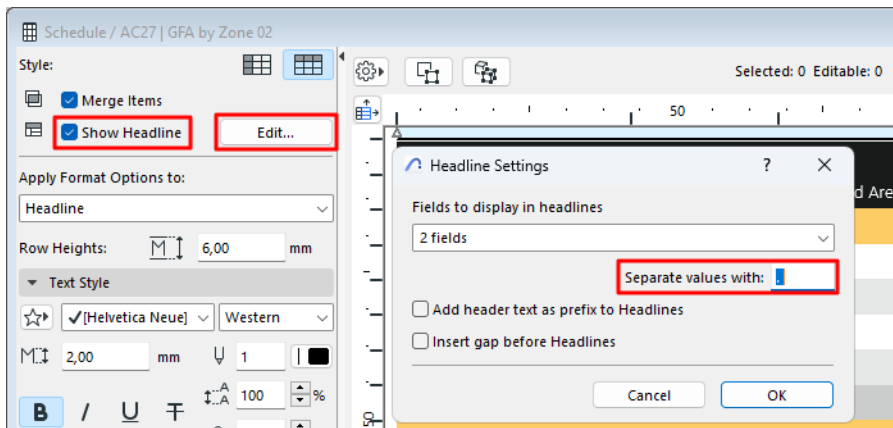
Improved format options for Interactive Schedule

New and improved formatting options for Schedules improve the visualization of data.

The screenshot displays the Archicad 27 interface with a schedule table for 'IES-01 Walls - 03 Yellow'. The table has columns for Element ID, 2D Plan Preview, Height [m], Thickness [m], Area [m2], Net Volume [m3], Perimeter [m], and Construction Type. The table is divided into sections for different wall types: '100 Block Insulated Cavity Plastered', '140 Block Insulated Cavity Plastered', 'Basement Wall', and 'Generic Wall/Shell'. The style panel on the left shows various formatting options such as 'Merge Items', 'Show Headline', 'Text Style' (Arial), 'Background' (Cells, Stripes), 'Border', and 'Print Footer & Format Change'.

Element ID	2D Plan Preview	Height [m]	Thickness [m]	Area [m2]	Net Volume [m3]	Perimeter [m]	Construction Type
Wall: 100 Block Insulated Cavity Plastered							
SW - 001		3,000	0,29	1,21	3,64	9,03	<Undefined>
SW - 002		3,000	0,29	1,21	3,64	9,03	<Undefined>
SW - 003		3,000	0,29	1,21	3,64	9,03	<Undefined>
				3,63 m²	10,92 m³		
Wall: 140 Block Insulated Cavity Plastered							
SW - 004		3000	0,29	1,23	3,68	9,04	<Undefined>
SW - 005		3,000	0,29	1,23	3,68	9,04	<Undefined>
				2,46 m²	7,36 m³		
Wall: Basement Wall							
SW - 006		3,000	0,29	1,23	3,70	9,04	<Undefined>
SW - 007		3,000	0,29	1,23	3,70	9,04	<Undefined>
				2,46 m²	7,40 m³		
Wall: Generic Wall/Shell							
SW - 008		3,000	0,30	1,27	3,81	9,06	<Undefined>
SW - 009		3,000	0,30	1,27	3,81	9,06	<Undefined>
SW - 010		3,000	0,30	1,27	3,81	9,06	<Undefined>
SW - 011		3,000	0,30	1,27	3,81	9,06	<Undefined>
				5,08 m²	15,24 m³		
				13,63 m²	40,92 m³		

- Cell background colors
 - Background colors visually differentiate value types, such as headers and totals
 - Alternating striped row color to clearly differentiate rows
 - Cell colors can be set to display only or also for output
- Hide columns by setting their width to zero - this lets you remove unnecessary clutter when merging headline cells. Double-click the ruler mark to reset column width, or modify width in the Schedule Cell Size dialog.
- Choose any character to separate the headline fields (such as comma, period, vertical line). Add a space if needed

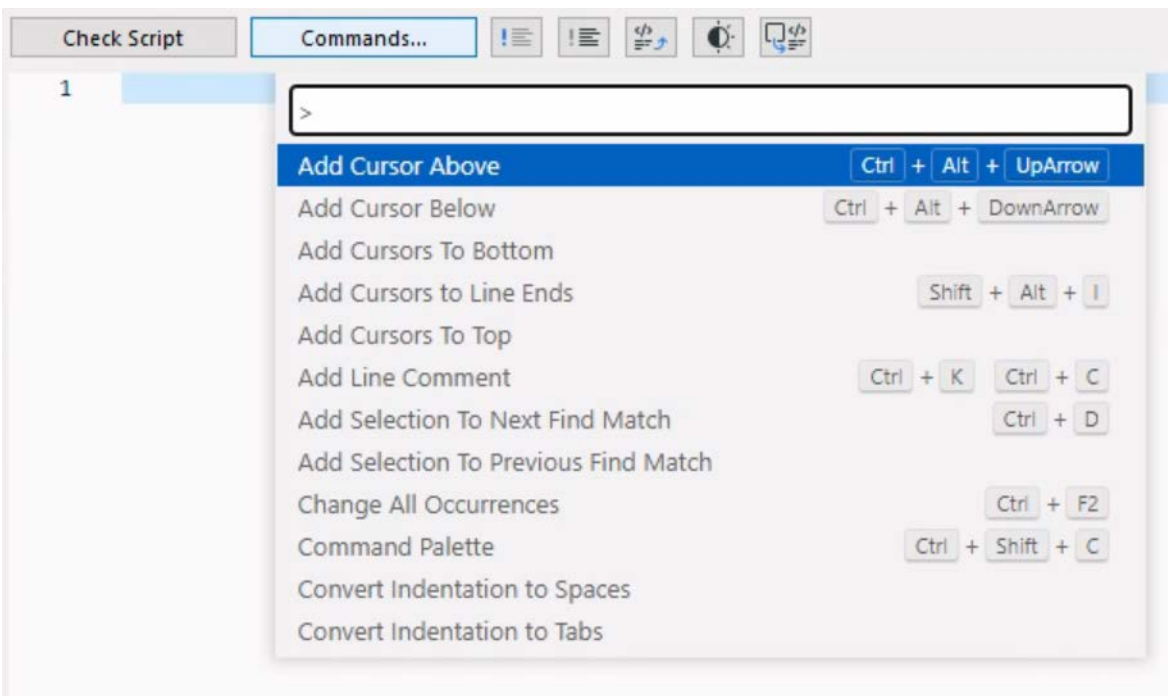
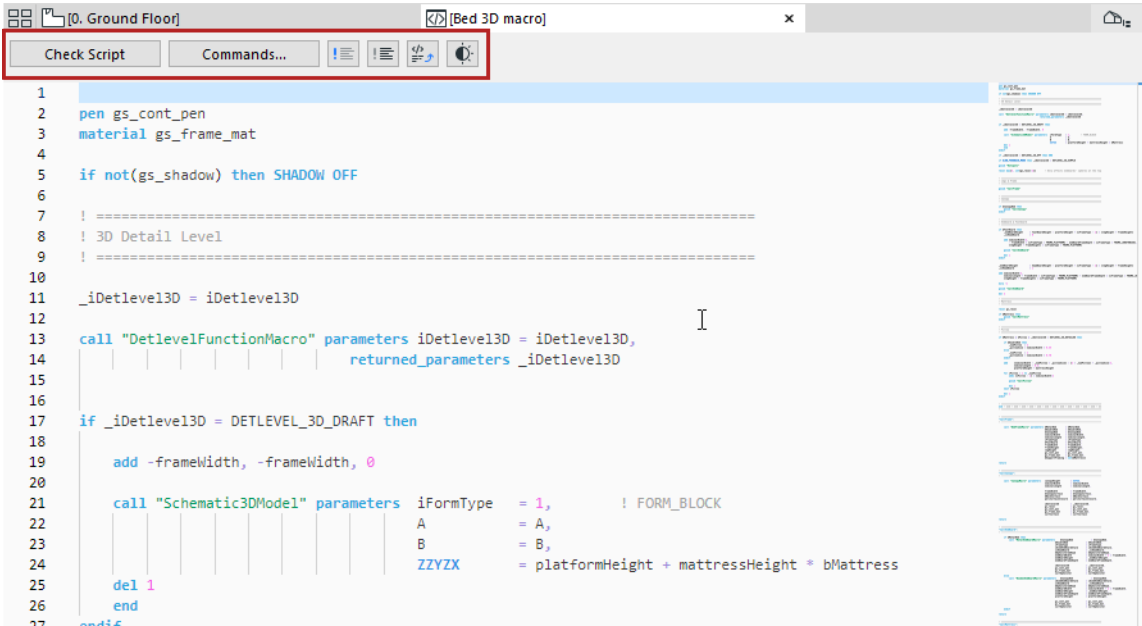


- Improved selection appearance of cells, by creating a bold outline of the currently selected cell and row/column edge highlights helps to know where you are without cluttering the interface with whole row/column selection as in previous versions.

Modernized GDL Editor

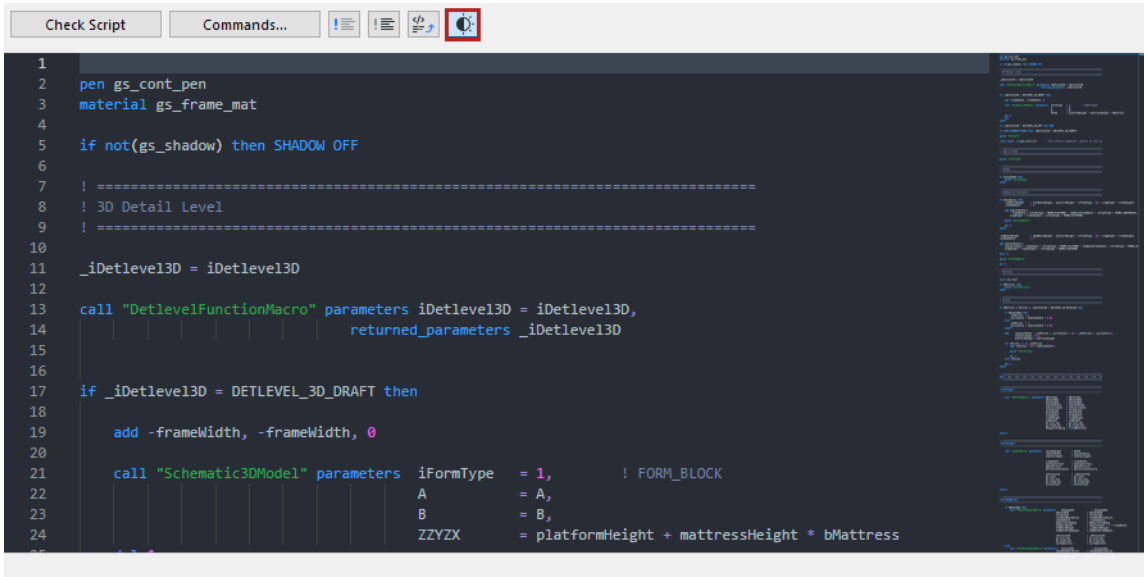
The scripting interface in Archicad’s GDL Editor window has gotten an overhaul, making life much more pleasant for content creators. Standard editing enhancements are now available, including

- Syntax highlight: Highlights code elements with different colors to make it easier to read and understand
- Auto-completion: Suggests code completions based on what you are typing
- Code folding: Collapse and expand sections of code for easier navigation
- Multiple cursors: Place multiple cursors, so you can edit multiple lines at the same time



- Auto-generate Script - Select element from Floorplan or 3D, click to generate its GDL script

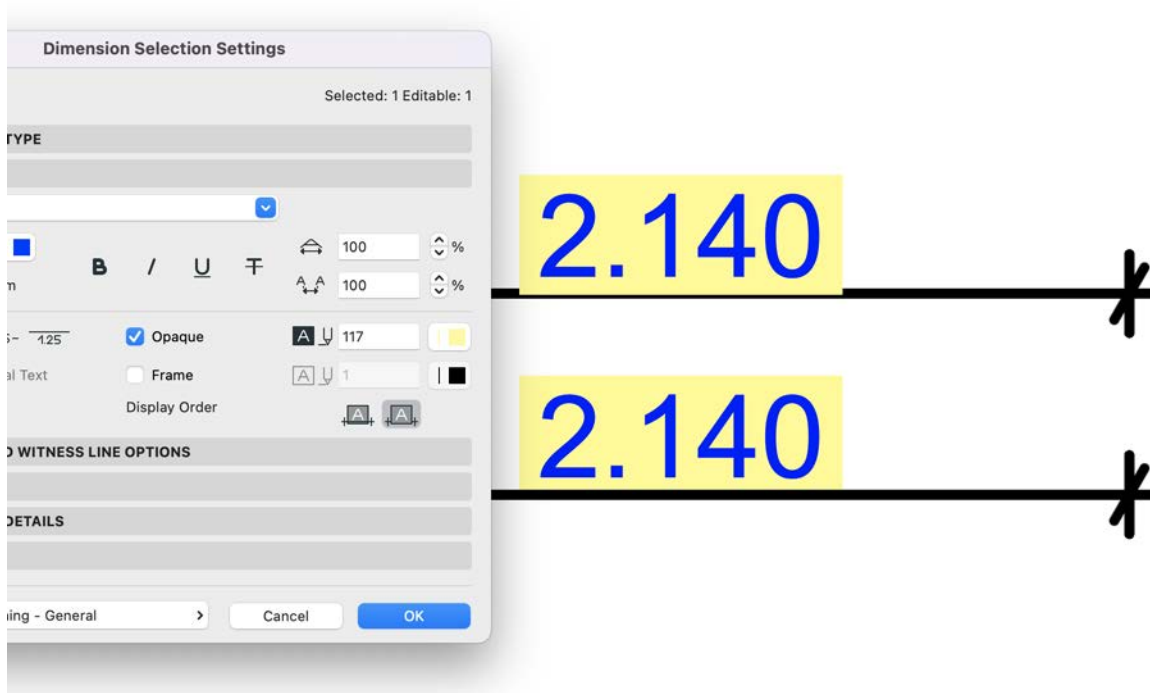
- Find and replace
- Dark Mode



[Script Windows](#)

Additional workflow improvements

Set display order for the background of dimension text field, to avoid texts that overlap dimension lines.



Element Transfer Settings

You can now disable the transfer of 'Home Offset' for Slab and Roof. This was already available for Walls, Columns, Beams and Objects. (Previously, a Favorite or inject applied to Slabs and Roofs would always transfer the Home Offset, causing their elevations to change.)

Editing lines and polylines together

Now works reliably and consistently

Grid Tool

The ID and Properties panel is now available in the Info Box, consistent with other tools.

Criteria and Field popups

With the same content and behavior as in previous versions, but now with consistent ordering and grouping of items throughout the interface.

Tracker

New option to multiply or divide tracker values (in addition to existing add and subtract function). For example: with distance value at 2000, type 2/ and enter, the result will be 1000. Or type 2* for a result of 4000. The input field has also been made wider, to more easily see the full value.

Archicad updates

Centrally distribute Archicad updates for all users in a company

Labels

Performance improvement with the selection and redraw of Labels, and the evaluation of user-defined **Properties** in Labels.

Quick Options Bar

Resizing behavior has been updated, with the addition of the new Design Options. When resizing the Archicad window, Quick Option panels will no longer disappear (as in previous versions) if the window area becomes too narrow. Panels now shrink further to always fit, with an instant tooltip when the panels become narrow.

Dimension text

The “multi-line” field has been renamed back to “Custom” based on user feedback.

Duplicating Profiles

Now opens the Profile Editor like it does when creating a New Profile. In previous versions, the Profile Editor wouldn't open, and required an extra click to open and edit.

Visualization

Our answer for the complex needs of visualization is to provide a versatile and flexible toolset, which includes integrated rendering engines, such as Cinerender and Redshift by Maxon and connection to external tools such as Enscape, Twinmotion, NVIDIA Omniverse, and many more.

Import-export FBX

Archicad 27 adds native support for importing and exporting files in FBX format - the de facto standard for 3D file transfer to AR/VR applications.

FBX integration with Archicad lets users connect to all kind of visualization applications, ensuring a fluent and open visualization workflow and the ability to offer an immersive VR/AR experience for your clients.

[Working with FBX files](#)

Physically Based Rendering in Archicad 3D views (Experimental feature)

Physically based rendering in the Archicad 3D window delivers a much more vivid and realistic appearance while you explore the 3D model. This initial, experimental release allows you to test and explore 3D views. The feature simulates the soft shadows that naturally occur when indirect or ambient lighting is cast onto a scene.



Physically based rendering ON



Physically based rendering OFF

[Physically based renderer in 3D Window - Experimental feature](#)

Archicad-Enscape workflow

An Archicad-Enscape connection plug-in will be available for MacOS in 2023, allowing our Mac-based users to enjoy the benefits of a modern, real-time rendering application tightly integrated with Archicad. (This plug-in was already available for Windows.)

Multi-disciplinary design

Siloed work across disciplines is increasing coordination challenges. Graphisoft is committed to introduce efficient integrated design workflows between the three major disciplines: Architecture, Structural and MEP Engineering.

[Streamlined Load input and edit](#)

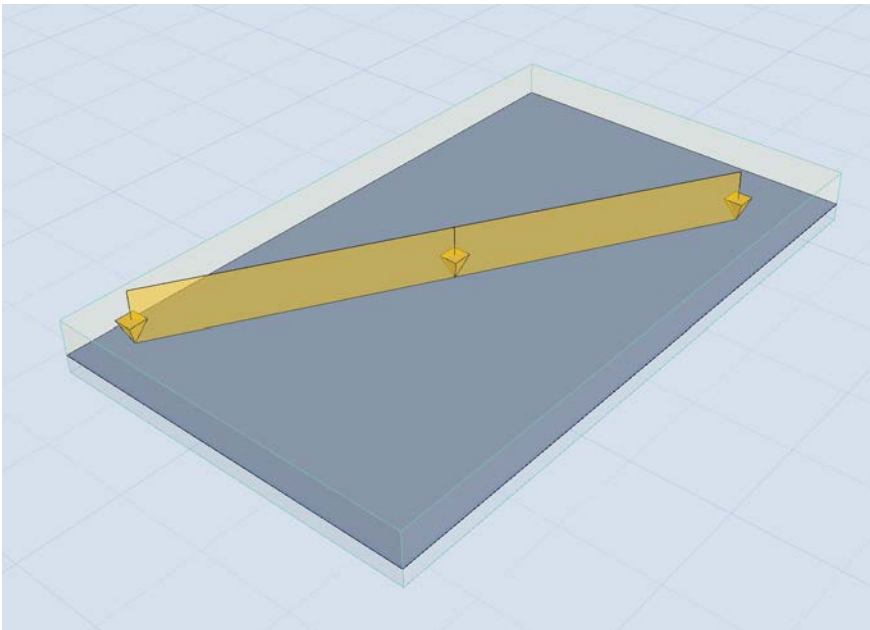
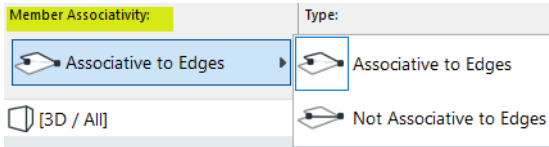
[Manual adjustment of Structural Analytical Model](#)

[Detailed Structural Change list in Model Compare](#)

Streamlined Load input and edit

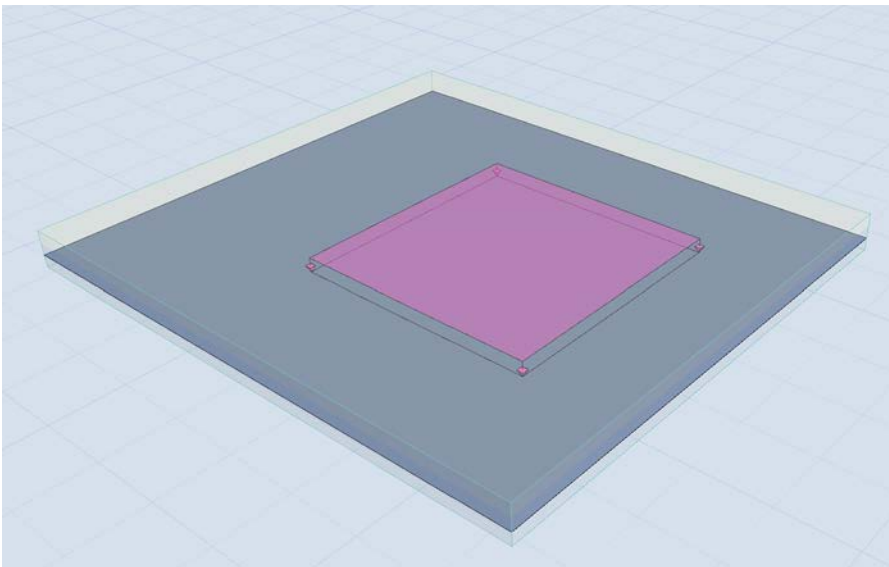
Archicad 27 enables you to generate and place Loads much faster.

- Place Line Loads easier along Member edges
- Place Line Loads that are not associative to Member edges - including anywhere on the element surface

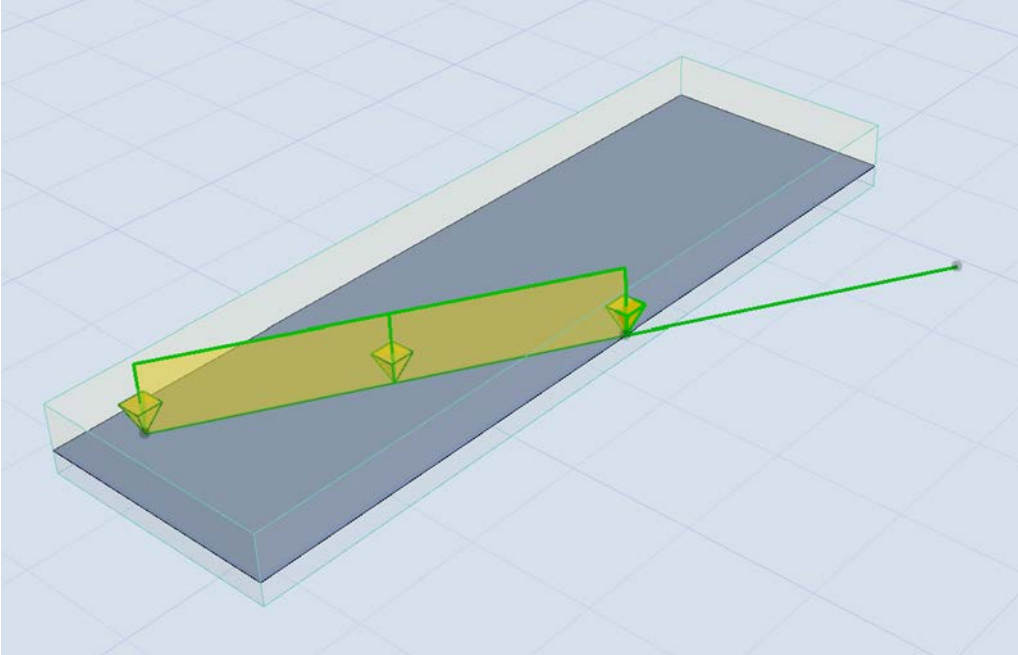


Non-associative Line Load

- Place a Surface Load on a sub-region of a 2D member



- Display overhanging parts of Loads



[Place Loads](#)

Manual adjustment of Structural Analytical Model

Archicad 27 now lets structural engineers fine-tune connections through manual adjustments, as a complement to the already existing adjustment rules. Now it's easier to achieve a continuous analytical model for complex building geometries.

Use the pet palette to intuitively adjust members by hand:



Stretch Member length



Angular Stretch Member length (curved Beam): Adjust end of 1D Member while keeping it on the same curve



Move node of 2D Member



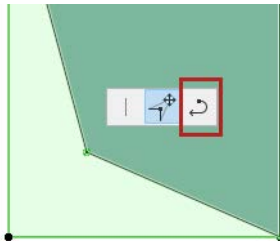
Offset 2D Member edge



Insert new node in 2D Member

Revert manual editing

Revert any manual editing to the Member's original geometry.

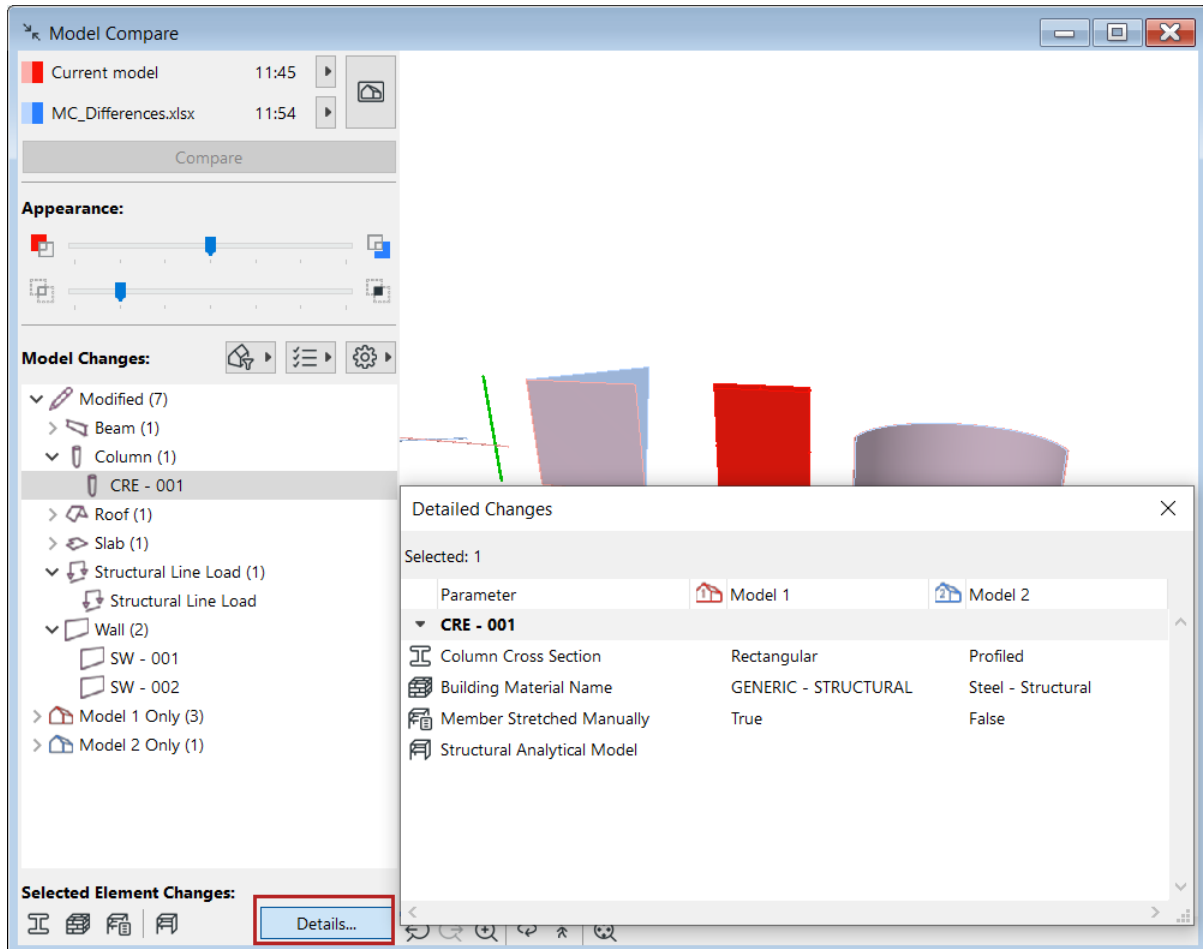


[Manual editing of analytical members](#)

Detailed Structural Change list in Model Compare

Model Compare is an essential project management and quality checking tool for multidisciplinary design teams.

Now you get detailed information on changes when you run Model Compare with structural views: the list shows which parameters have changed between the two models, including their values, where applicable.



[Compare Structural Analytical Models](#)

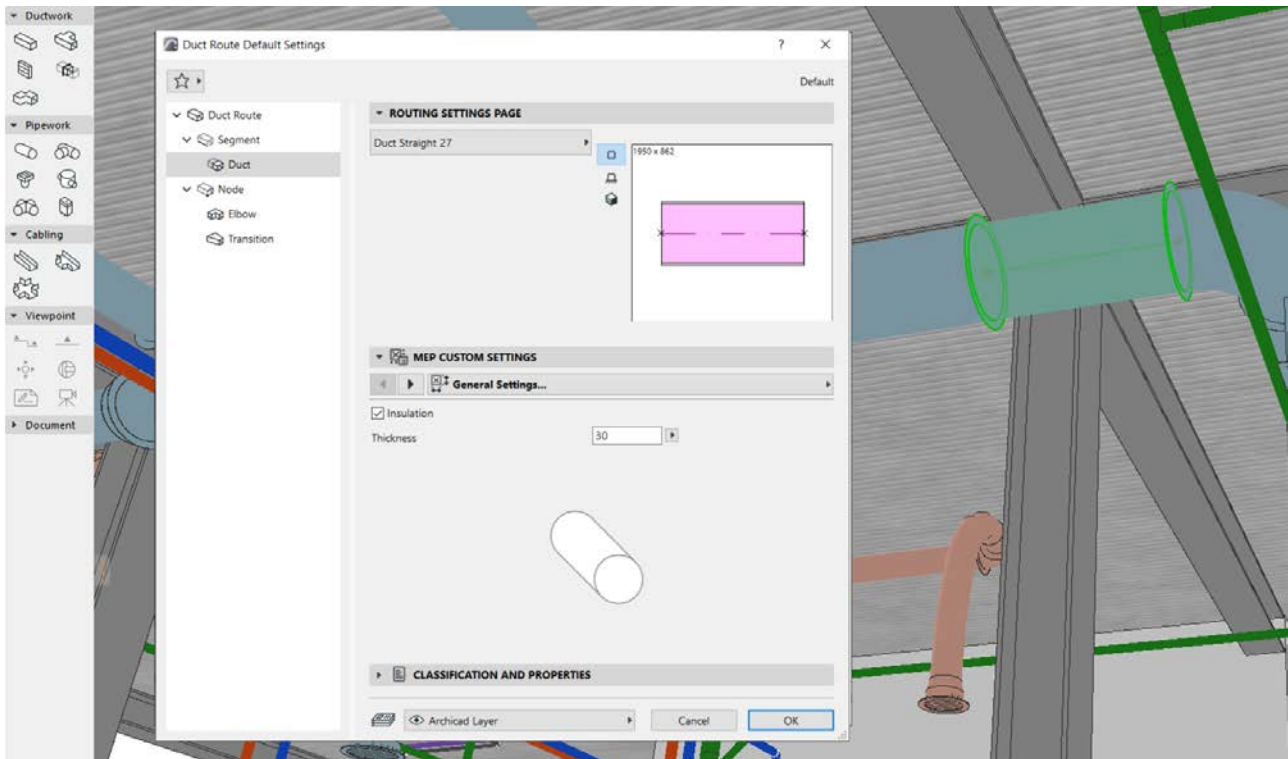
MEP Modeler

MEP Modeler has been renewed in terms of interface, modeling and technology.

- The routing process is intuitive - just click in series (like Spline input) to place the Duct route, Pipe route, or Cable Carrier route.
- Transitions and junctions are generated automatically.
- The route is handled as a whole, but edit any segment or node individually, in updated dialogs.

For Teamwork users, predefined “MEP Engineer” and “Lead MEP Engineer” roles are available. New specific permissions have been created for “MEP tool elements” and “MEP Preferences.”

A modern API is available for MEP Modeler, enabling custom Add-ons.



[MEP Modeler](#)

Open BIM

IFC4 certification for Architectural Reference Exchange for Import

In 2023, Archicad is on track to be certified by buildingSmart for IFC4 Architectural Reference Exchange for Import.

The IFC4 standard promotes consistency and efficiency in data exchange and communication, leading to better overall results. With the buildingSMART Certification for IFC 4 Architectural Reference Exchange for Import, we support all users working in IFC-based workflows to work with a qualified software.

RFA and RVT Geometry Exchange with Revit 2023

This tool supports collaboration with Revit engineers, if your planning partner cannot import/export IFC models. It imports RFA based families directly into Archicad so that you can use them as native Archicad objects.

The latest update of this tool brings Revit 2023 version support for Archicad 27 users.

[Working with RFA & RVT Files](#)

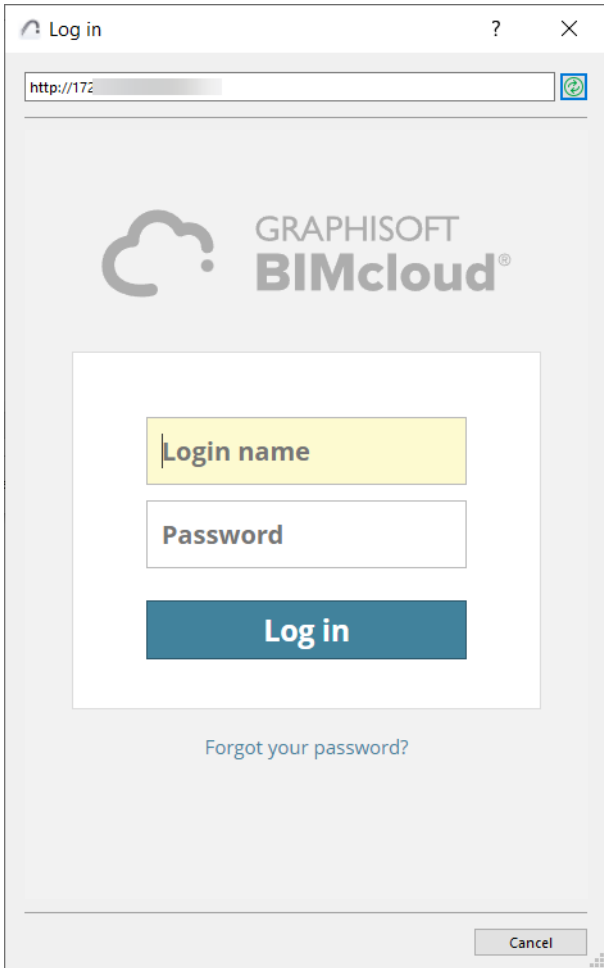
File format - Open and save versions Archicad 17 and up

To improve the sustainability and code of Archicad, Archicad 27 has dropped support and compatibility for files older than version 17 (Archicad 27 supports files with version 17-27). Therefore, older files can no longer be opened.

To migrate older files (prior to Archicad 17), open and save the project in Archicad version 17-26. Then open the project in Archicad 27.

BIMcloud - Secure Teamwork login

For Teamwork users logging in to BIMcloud from Archicad: A revamped login method ensures more secure authentication.



[Log in to BIMcloud](#)

Migrating to Archicad 27

Archicad can open projects from version 17 and up.

You cannot open project files from earlier versions directly in Archicad 27; you have to convert them first. To convert, open and save the project in Archicad version 17-26. Then you can open the project in Archicad 27.

See also [Archicad file and protection compatibility](#).

When you open the project in Archicad 27, you might notice the following changes in your project because of new Archicad features. Click on an issue to read more.

Migration Topics (click topic for info)	26	25	24	23	22	21	20	19	18	17
Migrating MEP elements	+									
Migrating Libraries	+	+	+	+	+	+	+	+	+	+
Migrating a Teamwork Project	+	+	+	+	+	+	+	+	+	+
Update DXF-DWG Translators	+	+	+	+	+	+	+	+	+	+
Update IFC Translators			+	+	+	+	+	+	+	+
Drawings with Resized Frames are Cropped					+	+	+	+	+	+
Replace Curtain Wall Frames/Panels						+	+	+	+	+
No "Custom" Model View Options Combinations						+	+	+	+	+
Stair Favorites May Be Reset						+				
Stair Tool Replaces StairMaker							+	+	+	+
ID-based Labels with Autotext							+	+	+	+
New "3D Styles" View Option							+	+	+	+
Model View Fill Options Replaced By Graphic Overrides								+	+	+
Guide Lines and Snap Points									+	+
Migration of Lightworks Settings to Cineware										+
Migration of Lights and Lamp Settings										+
Images from QuickTime Formats										+
Element Display in Interior Elevations										+
Building Materials: Collision Detection Setting										+

Migrating MEP elements

The MEP Modeler for Archicad 27 has been entirely renewed in terms of interface, modeling and technology. It is not compatible with any older-version Archicad projects containing MEP routes and elements.

When you open an older-version project, you will see the MEP elements as ordinary GDL objects.

You can attach new MEP routes to these GDL elements. If you back-save to Archicad 26, your original MEP model is unchanged, but any new MEP elements added in Archicad 27 will disappear.

Migrating Libraries

Migrate from Archicad 26 and earlier

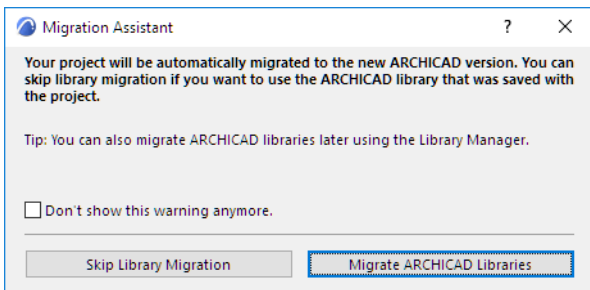
The new version of Archicad will work perfectly well with any older-version Archicad libraries. However, if you stay with your older libraries, you will not get the full benefit of library improvements and fixes.

We recommend that when you migrate a project to Archicad 27, you also migrate your older-version Library to the Archicad 27 Library, using the easy and automatic **Migrate Archicad Libraries** function.

At any time, to view information about possible Library-related issues in your current project, check the Libraries item in the Action Center (**File > Info > Action Center**).

Migrate Archicad Libraries when Opening Older-Version Project

When you open an older-version project in Archicad 27, and if your project includes an Archicad Library, you are prompted to use the **Migrate Archicad Libraries** function:



Click **Migrate Archicad Libraries**. This process will do the following:

- Load the new Archicad library
- Load the Archicad Migration Libraries folder ([What is a Migration Library? See below.](#))
- Automatically replace placed objects from the older libraries (versions 17-26) with their latest compatible version
- Remove Archicad libraries with lower version numbers

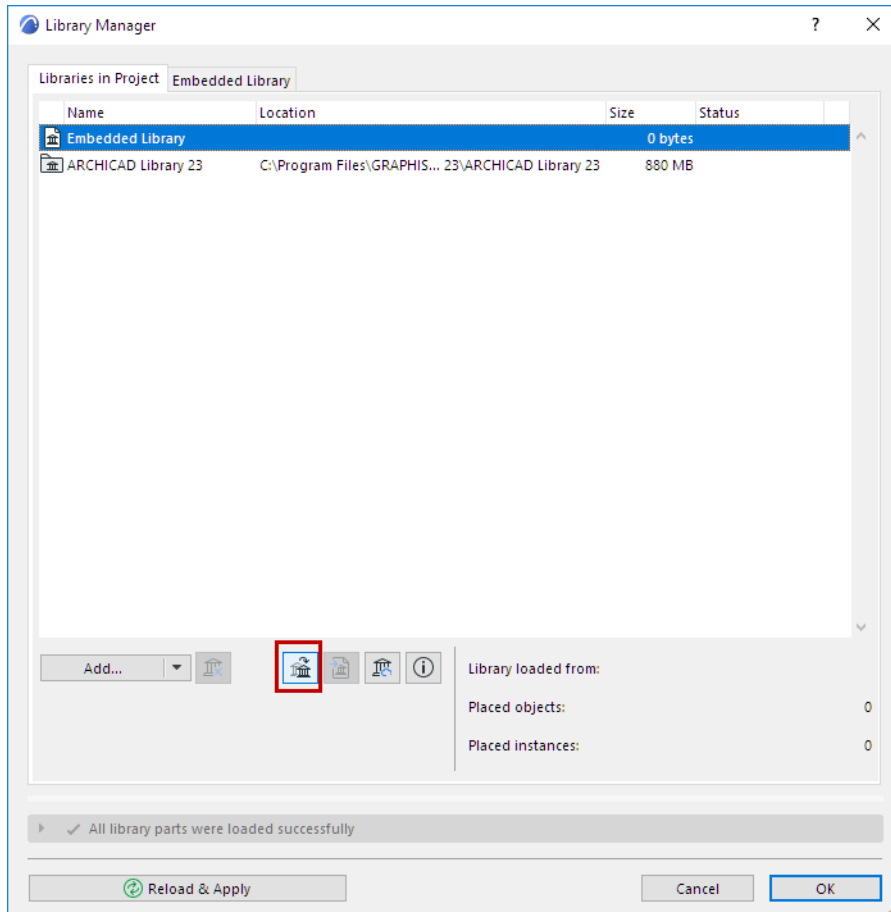
The Migrate Archicad Libraries process may take several minutes. Once it is complete, if your project includes objects from pre-Archicad 17 libraries, you may still have missing library parts. In this case, use Library Manager to load Old Archicad Migration Libraries. ([What is a Migration Library? See below.](#))

If you choose **Skip Migration**, you will load the old Library that was saved with the project. All placed objects will be available from that library, but you will not be able to use any library improvements and fixes from the latest version.

Migrate Archicad Libraries for a Running Project

You can use the Migrate Archicad Libraries process at any time, not only when migrating an older-version project.

Use the **Migrate Archicad Libraries** button in Library Manager.



Note: If the button is not shown in Library Manager, this is because all needed libraries (latest Archicad library plus Migration libraries) have already been loaded; and the project has no missing library parts.

Replace Older-Version Surfaces and Textures

To replace the older-version surfaces in your project with new ones from the AC27 Library's Surface Catalog, do the following:

1. Open the **Surfaces** dialog (Options > Element Attributes > Surfaces)
2. Use **New > Replace Settings from Catalog**.

What is a Migration Library?

Each new version of the Archicad Library contains many objects that have been updated since the previous version. For the most part, these updated objects are compatible with placed older versions of the same object, and will automatically replace the older versions.

Some objects from older Archicad libraries, however, have no counterpart in the new Archicad 27 library (typically, these are “discontinued” objects deemed obsolete). But if you have such objects placed in your project, you want them available even if after migrating.

Archicad Migration Libraries serve this purpose: they contain only those objects that do *not* have counterparts in the latest-version Archicad Library. A Migration Library is much smaller than full-version Archicad Library.

When Are Migration Libraries Used?

Migration Libraries are loaded automatically with two workflows:

- **Migrate Archicad Libraries**

[See *Migrate Archicad Libraries when Opening Older-Version Project*](#).

- **Consolidate:** If you have placed objects from multiple Archicad libraries and have loaded all the libraries, you will be prompted to **Consolidate your libraries**. The Migration Libraries are loaded automatically.

[See *Consolidate Multiple Archicad Libraries*](#).

If you do *not* use these workflows, then some of your placed objects from older-version projects may not have a compatible corresponding object in Archicad 27. In this case:

- Those incompatible older objects will be “Missing Objects” if you have removed your old libraries. In this case, you must replace these missing objects with new ones from the Archicad 27 library, and reset their parameters as needed.
- Keeping your old libraries alongside your new libraries is not recommended, because many of the loaded objects will be duplicates and/or superfluous, and increase library loading time and potential errors.
- If you still have missing library parts, you may be using library parts from Archicad libraries earlier than version 17. If so, you must manually add the **Old Archicad Migration Libraries** folder to the project, using Library Manager.

Where to Find Migration Libraries

Migration Libraries are installed on your computer as part of Archicad installation. They are divided into two folders:

- Archicad Migration Libraries (for Archicad library versions 17 and up)
- Old Archicad Migration Libraries (generally available for Archicad library versions 10-16)

You can also download Migration Libraries from the Graphisoft website.

Note: Available migration libraries may vary, depending on your localized version.

Obsolete Objects

Objects are “obsolete” in two cases:

- it has been discontinued entirely (e.g. a computer or TV object that is no longer current)
- the object has a counterpart in the new Archicad version, but due to changed parameters or functions (e.g. a window whose sash has been altered), the old object will not be replaced automatically

If you use the **Migrate Libraries** or **Consolidate** functions as described above, the Migration Libraries will be loaded automatically, together with the necessary obsolete objects, and you won’t have any missing objects.

Replace Obsolete Object with Latest Library Part

If you are using an obsolete object (e.g. window from an earlier version) and wish to replace it manually with the newest corresponding library part, be aware that the new window will not be exactly the same as the older one (otherwise, it would have been replaced automatically).

Therefore, when you do replace the old window with the new one, you should reset the new window’s parameters as needed.

To expedite this, use **Parameter Transfer**: this way, you will pick up the old window's parameters and inject them into the new window before placing it. This still won't produce an exact replica, but everything that can be replicated will be, and this will save you manual work.

See [Parameter Transfer Between Objects](#).

Migrating a Teamwork Project

Migrate from Archicad 26 and earlier

To migrate an older-version Teamwork Project (version 17-26) to Archicad 27, you will need:

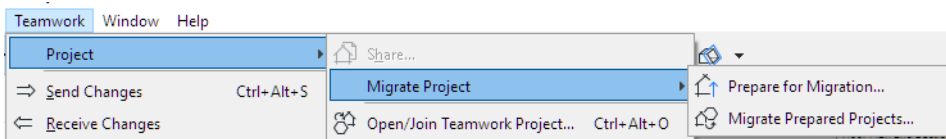
1. Both the older Archicad and Archicad 27 installed on your client machine.
2. Access to BIMcloud for version 27, either on the same or on different machines.

The migration process entails two phases:

- Prepare Project for Migration (you do this in the older Archicad); and
- Migrate Prepared Project (you do this in Archicad 27).

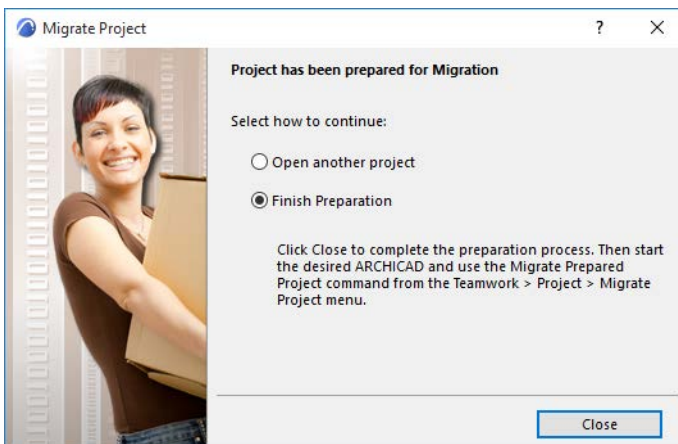
Prepare Project for Migration

1. Start the older Archicad.
2. Open your Teamwork project in the older Archicad.
3. Make sure that all other users send in their changes and leave the project. (If they do not, they will lose any unsent changes.)
4. Go to **Teamwork > Project > Migrate Project > Prepare for Migration**.



5. Click **Prepare**.
6. The project is prepared for migration.
7. Do one of the following:
 - Choose **Finish Preparation**, then Close
 - To migrate another project, choose **Open and Prepare another Project**.

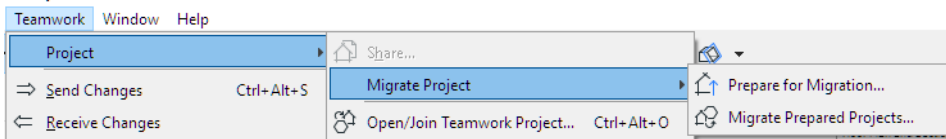
Do this if you have several Teamwork projects to migrate.



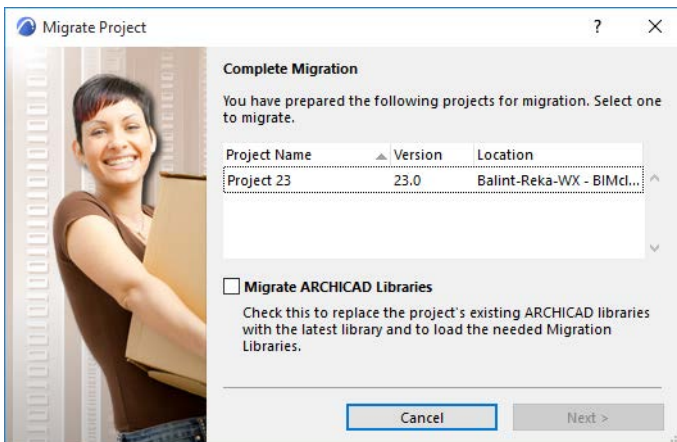
8. Once you have prepared all the projects you need, move on to the next phase:

Migrate Prepared Project

1. Start Archicad 27.
2. Choose **Teamwork > Project > Migrate Project > Migrate Prepared Projects**.



3. The dialog box lists all the prepared projects: select one to migrate it.
 - If the listed project is shown in gray, it means you cannot complete that project's migration now - for example, because the server is offline.

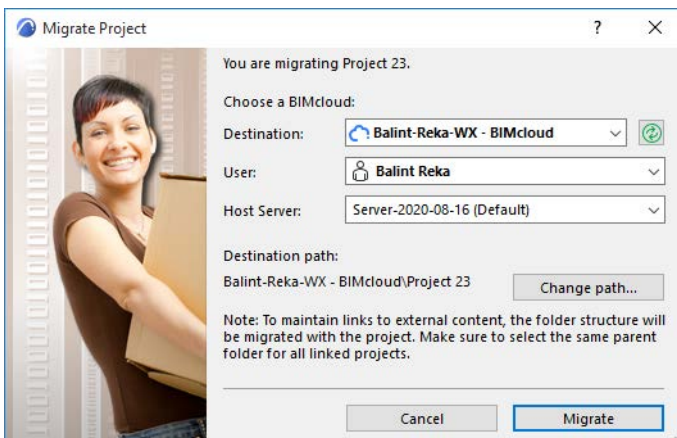


Optionally, check the **Migrate Archicad Libraries** box: this will replace the project's currently used Archicad libraries with Archicad Library 27, plus the necessary migration libraries. (These libraries will be uploaded to the BIMcloud if they are not there already.) Otherwise, the migrated project will continue to use its current Archicad libraries.

4. Click **Next**.
5. Choose a destination server for the project.

Note: If you are uploading to a BIMcloud: you have an additional option to define the Host Server for the migrated project.

You must be logged on to this server, and you must have the correct permissions to add a project to the BIMcloud.



6. The project content will be uploaded to the chosen BIMcloud, including existing users and their roles.
Note: The Project Log, Comments, messages and any reservations will be *deleted* during project migration.
7. Now that the original project is migrated, you can manually delete it from the old BIM Server.

Hotlinked Modules and Drawings

If the migrated project contains *hotlinked modules or drawings linked from older Archicad Teamwork projects*, you must also migrate each of those Teamwork projects to Archicad 27, following the steps above.

If those hotlink/drawing source projects were on the *same* Server as the migrated host project, then the host project will automatically locate and link the source projects, after they are all migrated.

If those hotlink/drawing source projects were on a *different* Server than the migrated host project, you must re-link the hotlinked modules and drawings manually, once the source projects are all migrated.

Temporarily Missing Library Objects

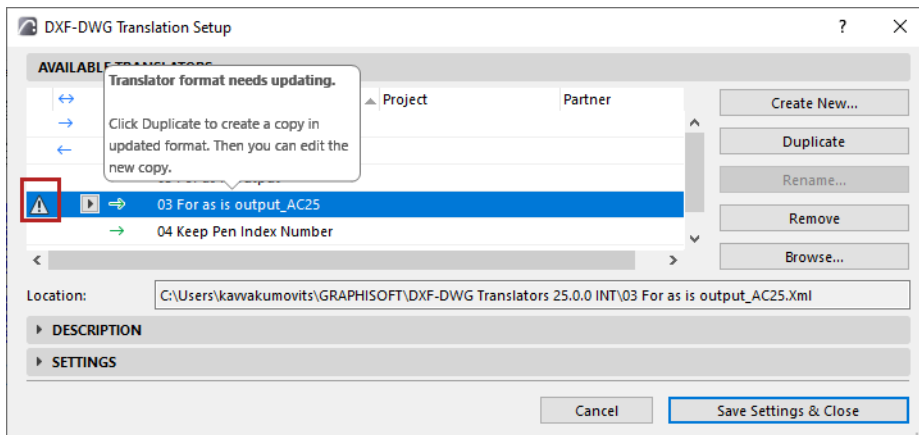
Some objects in your migrated Teamwork project may be listed as “**Temporarily Missing**” in the Library Manager. As soon as other users do a Send and Receive, this problem will be solved automatically.

Update DXF-DWG Translators

Migrate from Archicad 25 and earlier

The DXF-DWG translator file (xml format) now uses an updated file format. Dialog-level feedback helps you convert your translators to ensure a smooth migration whenever needed.

- Translators imported from earlier-version Archicad projects are listed in the DXF/DWG Translator Setup, but they are outdated and therefore uneditable. They are marked with an icon and tooltip. Just create a duplicate - the new copy is updated to the new format.



- If the outdated (and therefore uneditable) translator happens to use a template file that is inaccessible: it is also marked with an icon and tooltip with instructions.
 - First open the translator in an earlier (pre-27) version of Archicad - now the translator settings are editable
 - Browse for the template file - now the template file and its settings are accessible
 - Reopen the translator in Archicad 27: now you can update the translator (create a duplicate), based on the template file settings

Update IFC Translators

Migrate from Archicad 24 and earlier:

You can no longer import Translator files in XML format.

To use XML-format translators from older-version projects:

1. Open the project in AC version 21-24
2. Go to **File > Interoperability > IFC > IFC Translators**
3. In the IFC Translators dialog box, click **Import**
4. Select the .XML file format, then browse for the Translator
5. Save the project, then open it in AC27. Translator data will be recognized.

Migrate from Archicad 20 and earlier:

- IFC Scheme Settings have been replaced by Translator-specific Property Mapping.
 - The “Property Mapping for IFC Export” preset contains the older-version project’s Scheme Settings
- Type Mapping is customizable by the user
 - The “Type Mapping for IFC Export” preset contains the default Classification Mapping used in older-version projects.

Drawings with Resized Frames are Cropped

Migrate from Archicad 22 and earlier

When opening an older-version project, Drawings whose Frames were “Manually Resized” (as seen in Drawing Settings) will be cropped.

Cropped Drawing content was a new feature in Archicad 23 to enhance performance. Cropped content can be restored at any time, as long as its source remains available.

See [Drawing Frame](#) for more information.

Replace Curtain Wall Frames/Panels

Migrate from Archicad 21 and earlier

Your existing GDL Frames and Panels are migrated as-is to Archicad 27.

It is recommended that you replace them with corresponding Panels and Frames provided in the Archicad 27 Library.

Advantages:

- In Archicad 27, these components can be assigned Building Materials separately
- The new Frames will intersect intelligently - including Frames that meet at angles (e.g. hexagonal patterns)
- The new Frames and Panels can use the new Model View Options available for Curtain Walls (Schematic and Simplified as well as Full)

Note: Once you replace a GDL Frame-Panel with one in Archicad 27, you cannot change it back to a GDL-based element.

No “Custom” Model View Options Combinations

Migrate from Archicad 21 and earlier

Edited MVO Combinations are no longer “Custom” - they are always named.

“Custom” MVO Combinations migrated from earlier versions automatically become named MVO Combinations.

Stair Favorites May Be Reset

Migrate from Archicad 21

When you open an Archicad 21 project, you may find that the 2D attributes which you customized in your Stair Favorites have been reset to their built-in defaults.

These attributes (e.g. Break Mark, Walking Line, Numbering) are found in Stair Settings, under Floor Plan Display/Reflected Ceiling Plan.

As needed, open your Stair Favorite Settings and readjust the 2D attributes.

Stair Tool Replaces StairMaker

Migrate from Archicad 20 and earlier

The StairMaker Add-on is no longer supported.

StairMaker Stairs from older-version projects are converted into GDL Stair-type objects, with limited GDL parameters available from Object Settings.

ID-based Labels with Autotext

Migrate from Archicad 20 and earlier

ID-type Labels (ID Label, Unique ID Label) from earlier versions are transformed into the renewed Text/Autotext-type label, which uses Autotext to display the element's ID.

New “3D Styles” View Option

Migrate from Archicad 20 and earlier

The appearance of 3D views depends on **3D Styles**, a View Option. (View > 3D View Options > 3D Styles).

These settings replace the 3D Window Settings of earlier versions.

If your project contains a large number of 3D views with varying settings, be aware that a large number of 3D Styles may be generated, each with a unique name.

Model View Fill Options Replaced By Graphic Overrides

Migrate from Archicad 19 and earlier

The Override Fill Display panel of Model View Options has been eliminated. Instead, overrides are handled by Graphic Override Rules (GOR). These Rules are applied to individual views/viewpoints as Graphic Override Combinations.

The Renovation Override Styles (from earlier versions) are now predefined as Graphic Override Rules.

Guide Lines and Snap Points

Migrate from Archicad 18 and earlier

The Snap Point setting formerly called “Along Entire Element” is now called “Between Nodes”.

Migration of Lightworks Settings to Cineware

Migrate from Archicad 17.

The Lightworks Rendering engine is no longer available. Instead, the Cineware engine, which is based on Maxon Cinema 4D R21 and incorporates most of its parameters, is integrated in Archicad.

Photorendering Settings

Lightworks Photorendering Settings saved in 3D views are not transferred to the Cineware engine (except for the size parameter). For such views, a default rendering scene is provided.

Photorendering settings for Basic and Sketch engines, however, are retained. If you open such a 3D view in Archicad:

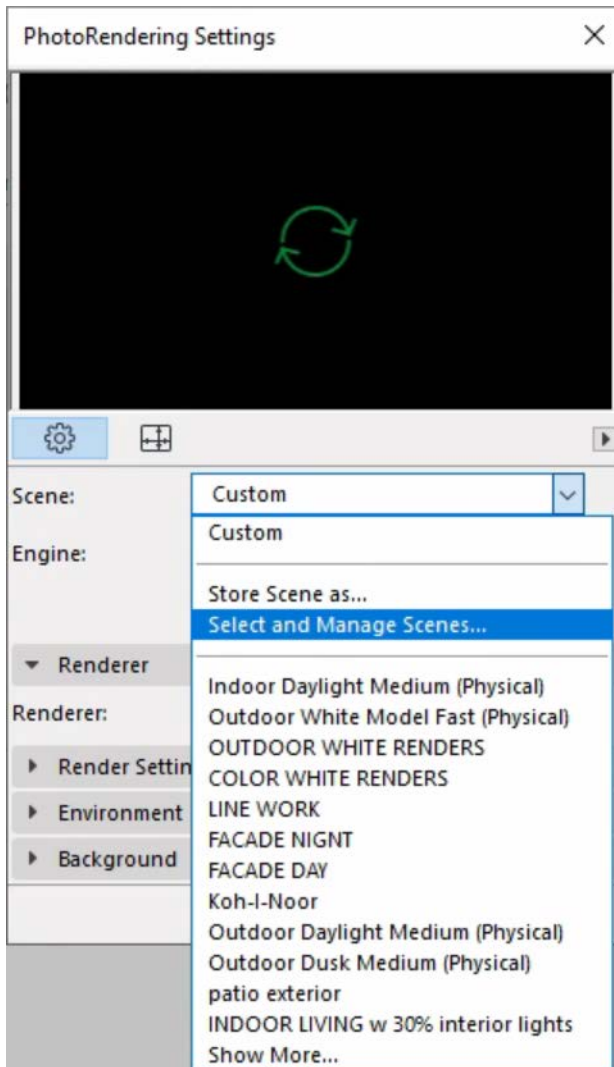
- In the 3D only panel of View Settings, the Rendering Scene is set to “Custom”
- In Photorendering Settings, the scene is also set to “Custom”, whose settings correspond to those saved with the current view. Only one Custom scene at a time is possible in the “Select and Manage Scenes” dialog box in Archicad, so you should rename these custom scenes if you want to re-use them.

Predefined Rendering Scenes in Archicad

The Archicad template for each language version includes predefined Rendering scenes, with parameters configured for specific rendering scenarios.

If you are using your own company template instead of the Archicad template, you will not access these scenes by default. To use these scenes with your own template, do the following:

1. Open the template file
2. In PhotoRendering Settings, go to the Select and Manage Scenes dialog box



3. Select any or all of the predefined rendering scenes or folders
4. Click Export to save them to a location on your computer. (File extension: renderingscene)
5. Open your own project/template.
6. In the Select and Manage Scenes dialog box, click Import to bring in the scenes.

Surface Settings

- Surface Settings configured for the Basic rendering engine *are compatible*: they will be applied to their Surface counterpart in Cineware
- When you open the older project, Archicad will automatically generate your Basic Surface settings in Cineware: your older renderings will, by default, look similar to the model in the 3D window
- Surface Settings configured for Lightworks *will not be converted* to Surface Settings for Cineware

Recommended Workflow for Migrating Surfaces to Cineware

Use this workflow if you have an existing template file, and you want to upgrade your surfaces to take advantage of Cineware's more sophisticated surface parameters.

1. In Archicad, go to the Surfaces dialog box.
2. Select an existing Surface that you want to upgrade.
3. In Surfaces, click **New**.
4. In the **New Surface** dialog box, choose **Replace Settings from Catalog**.
5. Search the built-in Surface catalog or the BIM Components portal for a surface that is most similar to your existing surface, and click OK.
6. The newly selected surface settings replace the previous surface settings (for the currently selected surface only).

Notes:

- The original surface's Name and Index number (as listed in the Attributes palette) will remain *unchanged* even after the settings have been overridden.
- Although surfaces can include a cover fill (used to display surfaces in certain Archicad views), this **Replace Settings** function will *not* overwrite the cover fill (if any) of your original surface; that cover fill (formerly called vectorial hatch) remains unchanged in the project (and continues to be listed as before in Attributes palette).

Migration of Lights and Lamp Settings

Migrate from Archicad 17.

Archicad includes many new and updated Lamp objects (including light sources) for better illumination of your project and more realistic renderings.

The Lightworks Rendering engine is no longer available in Archicad. Instead, the Cineware engine, which incorporates most of the parameters of the Cinema4D renderer, is integrated in Archicad.

When you migrate the older project, you will see that your general lights, interior and street lamps are *compatible*: their geometry and position, and their color and intensity parameters, remain unchanged.

- Lights that were configured for the Basic Rendering Engine have *retained their parameters*.
- Lights that were configured for the Lightworks Engine are, to the extent possible, *compatible*: their parameters are converted into their equivalents for the Cineware Engine.
- The Lightworks Sun Object is replaced by the Sun Object.
- The Lightworks Window Light and Sky objects are not migrated. If your older-version project contains these objects, they will be missing after you migrate the project. The solution is to load the older-version Lightworks Library which contained these objects, or to replace them in Archicad using the Physical Sky (Environment setting, including a sun) and the new Window Light object.

Recommendations

- Due to the compatible migration noted above, a project migrated from Archicad 17 will have lamp effects that closely approximate their appearance in the earlier version. However, since the Cineware engine is fundamentally different than Basic and Lightworks engines, you should do a quick test render of your scene before generating the full rendering output.
- Due to the Global Illumination capability of the Cineware engine, you may find that the project is better-lit than in the old version, and you can remove or change the parameters of some of your light/lamp objects.

Images from QuickTime Formats

Migrate from Archicad 17.

Image files saved in QuickTime formats are no longer supported in Archicad, to ensure compatibility for image files on both Windows and Mac.

Supported image formats are: JPEG, PNG, GIF, TIFF and BMP. You can save and place image files only to/from these formats.

If you migrate an older project that contains unsupported figures or drawings:

1. Figures and embedded drawings from Archicad 17: these images are converted automatically into a supported image file format (e.g. png); no action is needed.
2. Linked drawings: The drawing content is not visible; their place in the project is an empty white box. The icons of these drawings are shown in red in Drawing Manager.

You should manually convert these files to a supported file format (e.g. PNG or JPG), then re-link them.

3. Image Files in the Embedded Library:

Any image files that are in unsupported formats are shown with the “unsupported file format” icon in the Library Manager. You should open and resave these files to a supported file format, and then Add these files to the Embedded Library.

If these image files are used in the project (e.g. in a Surface or Image Fill), then they are also listed in the Unsupported Objects folder of the Library Manager.

Converting image files

It is recommended to convert your QuickTime-format image files to either PNG or JPG format.

- If you convert a drawing to PNG format, you will not lose any data.
- If you convert a drawing to JPG, you will lose some data during the file compression process, but the file size will be smaller.

Element Display in Interior Elevations

Migrate from Archicad 17.

Interior Elevations from older-version projects will be migrated to ensure that they contain the same elements as before.

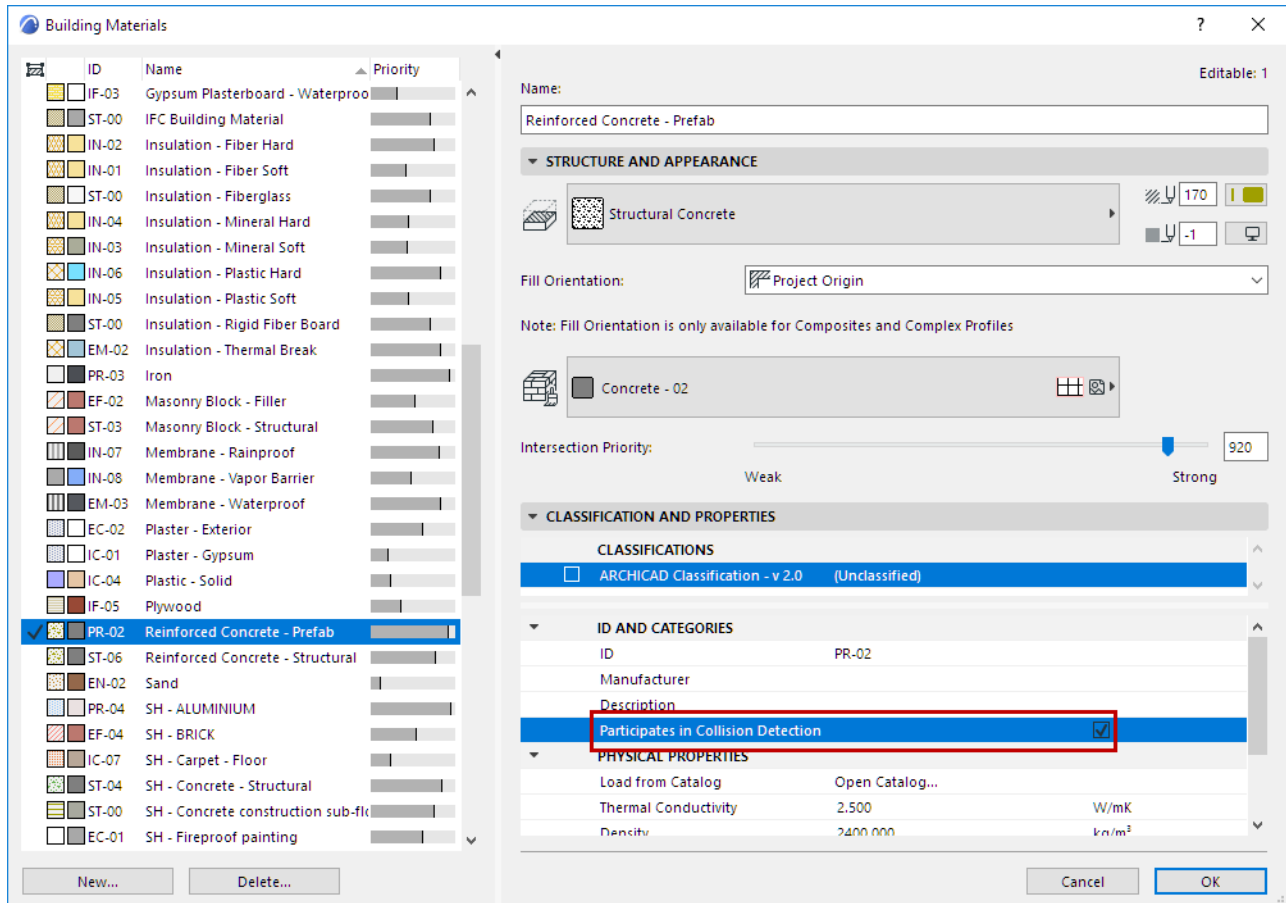
Any newly placed Interior Elevations will display elements according to the new logic: if you use the Limited horizontal range, elements are shown which fall within the limit. If you use Infinite horizontal range, the entire model space within the Marker's range is displayed (as in the case of Section or Elevation views).

Building Materials: Collision Detection Setting

Migrate from Archicad 17.

The Building Materials dialog box has a “Participates in Collision Detection” checkbox in the Properties panel. This is useful for defining whether particular building components - parts of composite or complex elements - should be exported in IFC, and/or whether they should participate in collision detection.

[See Collision Detection.](#)



All Building Materials are migrated with the checkbox ON. (All migrated Building Materials are set to participate in Collision Detection by default.)

To exempt any Building Material, uncheck the Collision Detection box.

Configuration

This section of the Archicad Reference Guide includes information on starting the program, setting up your workspace, choosing preferred project settings, and defining the basic project components that you will use while working in your projects (templates, attributes, Favorites). It also describes how to manage libraries in Archicad.

[Start Archicad](#)

[Create New Project](#)

[Open Recent Project\(s\)](#)

[Browse for a Project](#)

[Open & Repair Selected File](#)

[Opening Legacy or Archive Projects: Manage Library Parts](#)

[Close Project](#)

[Save Archicad Project](#)

[File Compression](#)

[Licensing and sign-in](#)

[Bug Reporter](#)

[Work Environment](#)

[Template Files](#)

[Backup Files](#)

[Archive Projects](#)

[Archicad User Feedback - Net Promoter Score Survey](#)

[Create Support Package \(Troubleshooting\)](#)

[Add-ons and Goodies](#)

[Working Units](#)

[Location Settings](#)

[Survey Point](#)

[Project Preferences](#)

[Scale](#)

[Libraries](#)

[Favorites](#)

Start Archicad

1. Do one of the following:
 - Double-click the Archicad program icon
 - Click the Archicad Starter application
2. At first start-up, sign in with your Graphisoft ID.
3. The **Start Archicad** dialog appears. Do any of the following:

[Create New Project](#)

[Open Recent Project\(s\)](#)

[Open or Join Teamwork Project](#)

If you don't have a license

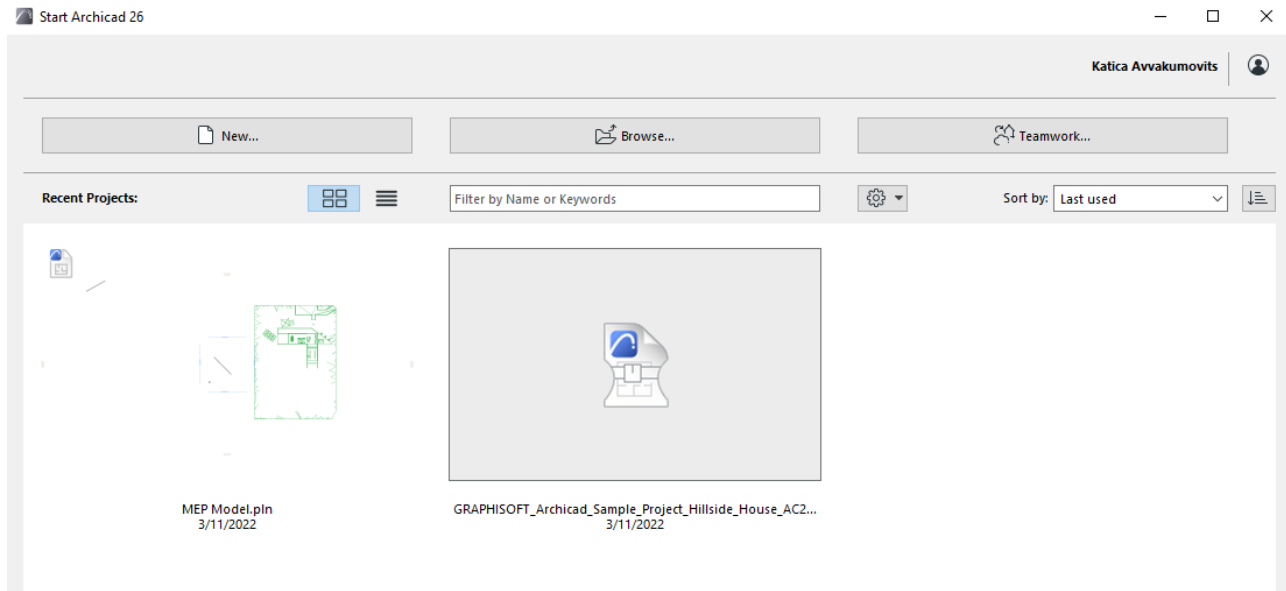
If you don't have a license available, a dialog box prompts you with options for continuing.

See [License troubleshooting](#).

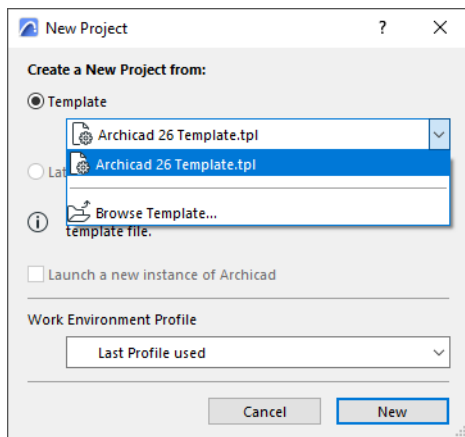
[Sign in with Graphisoft ID](#)

Create New Project

1. From the Start Archicad dialog, click New.

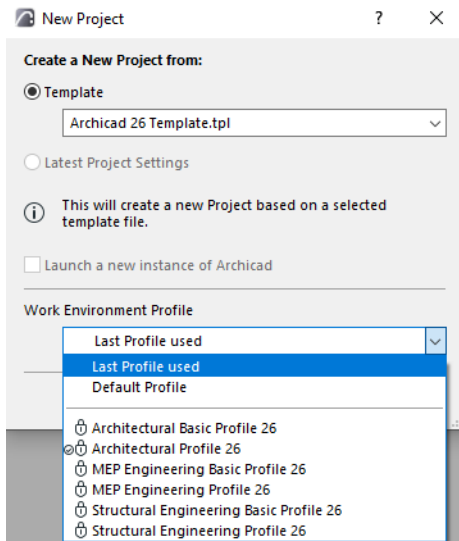


2. From the appearing dialog, choose a template from the list, or browse for a different template file.



See also [Template Files](#).

3. Choose a **Work Environment Profile** for the new project.



- **Current Profile:** Applies the profile that was in effect when you last closed Archicad, including any unnamed Custom schemes and *undefined* schemes.
- **Default Profile:** The default one is listed with a checkbox. You can change this default at Options > Work Environment > Profile Options.

[See Profile Options.](#)

- Or choose any of the predefined Work Environment Profiles shown in the list.

[See Default Work Environment Profiles](#)

4. Click **New** to create the new project.

Create New Project From a Running Archicad

1. Use **File > New > New**
2. If you already have Archicad running: you can choose **Latest Project Settings**, rather than a template, for the new project.

Note: To automatically use the default project settings: press Alt (Opt) key with the **New** command. The command changes to **New and Reset All**.

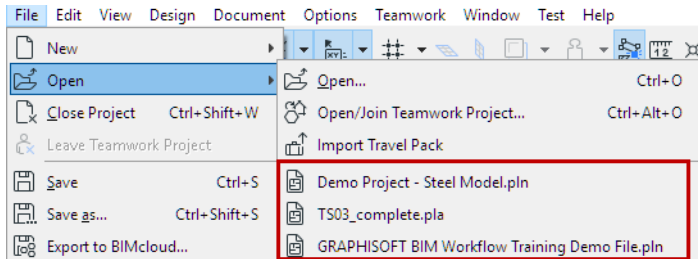
Launch a new instance of Archicad

1. From a running Archicad, do one of the following:
 - **File > New > New**
 - **File > Open > Open**
2. From the New Project or Open Project dialog, check **Launch a new instance of Archicad**

Open Recent Project(s)

Do one of the following:

- From a running Archicad:
 - Use **File > Open**
 - Select a recent project from the pop-up

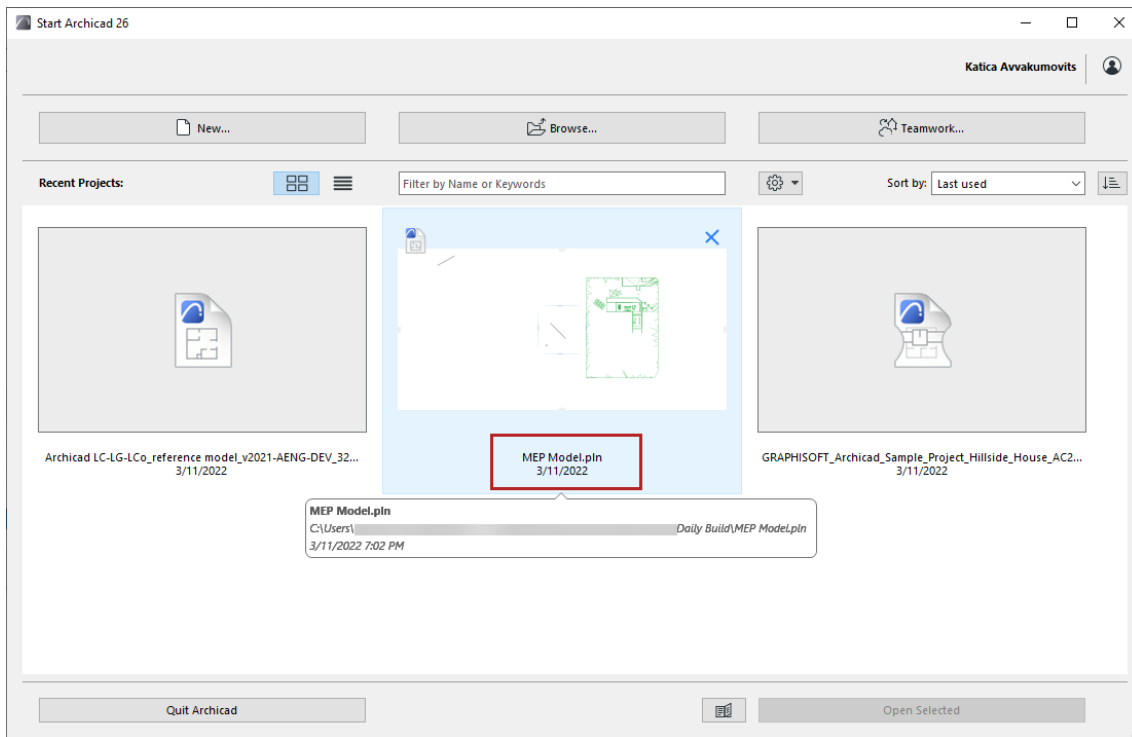


- From the **Start Archicad** dialog: Select one or more recent projects (see below)
 - Note:** To open the Start Archicad dialog, do one of the following:
 - Double-click the Archicad program icon
 - Click the Archicad Starter application

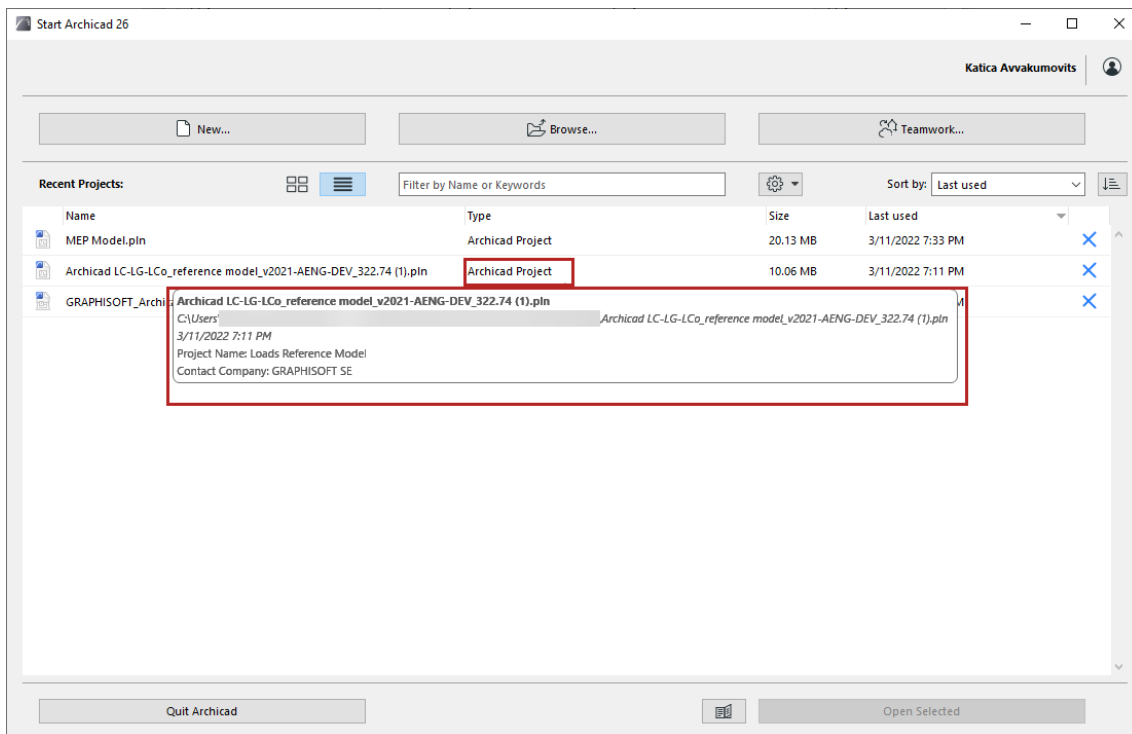
Start Archicad Dialog: List of Recent Projects

The **Start Archicad** dialog lists all the Archicad projects, including Teamwork and module (.MOD) files, that have been opened by the program.

- Click the list or tile icon to see projects in **tile** view or **list** form
- Hover over a project to see its **Project Info**
 - Note:** These data are user-defined tags entered at **File > Project Info**.



Recent Projects in Tile View



Recent Projects in List View

- **Recovery** files are in red type
Note: If you remove a Recovery file from the Recent Projects list (with the red X), it is deleted entirely, and you cannot retrieve it again.
- **Not Available:** The project is located on an external drive or network that is not currently available

- **Remove** a project from the Recent List: click Delete, Backspace, or the [blue X](#)

In List view:

- **Sort** by name, last used, size or file type, in ascending or descending order
- **Filter** projects by name, keyword, or any other Project Info data

In Tile view:

- The Preview cycles among the project's last-opened tabs and its Project Preview picture (if any).

Related Topics

[Project Preview](#)

[Project Info](#)

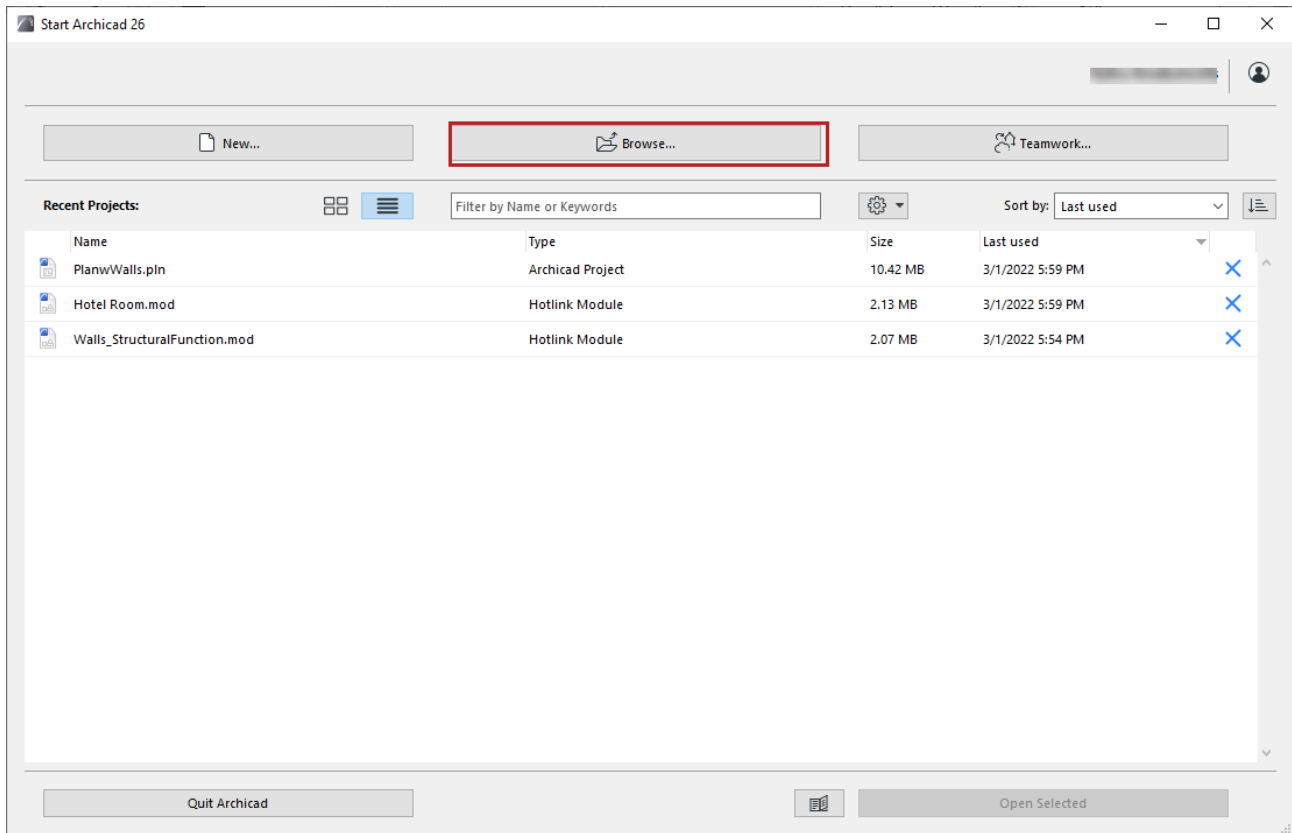
[Opening Legacy or Archive Projects: Manage Library Parts](#)

[Open & Repair Selected File](#)

Browse for a Project

Do one of the following:

- From a running Archicad: Use **File > Open > Open**
- From the **Start Archicad** dialog, click Browse.



Note: To open the Start Archicad dialog, do one of the following:

- Double-click the Archicad program icon
- Click the Archicad Starter application

Browse for a project, then click **Open**.

Related Topics

[Launch a new instance of Archicad](#)

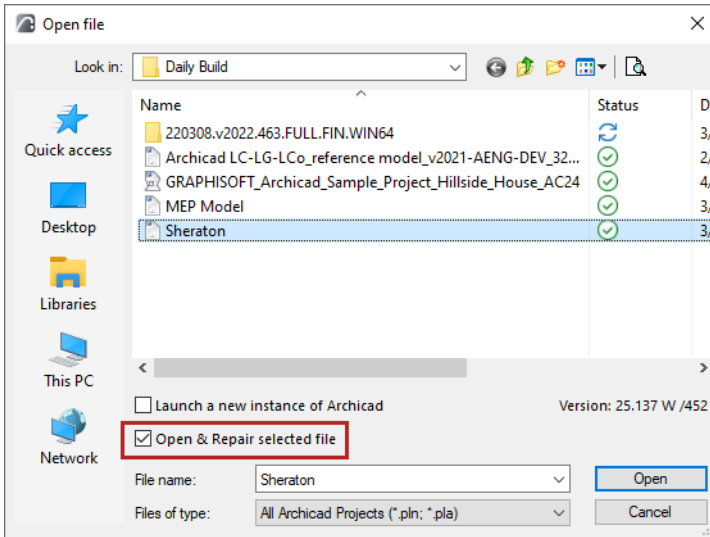
[Opening Legacy or Archive Projects: Manage Library Parts](#)

[File Types Opened by Archicad](#)

[Open & Repair Selected File](#)

Open & Repair Selected File

If the selected project cannot be opened the first time, check this box and try again.



Opening Legacy or Archive Projects: Manage Library Parts

In the **Start Archicad** dialog, the **Settings** pop-up contains your preferences for handling Libraries and Library parts when opening Legacy (version 17 and up) or Archive projects.

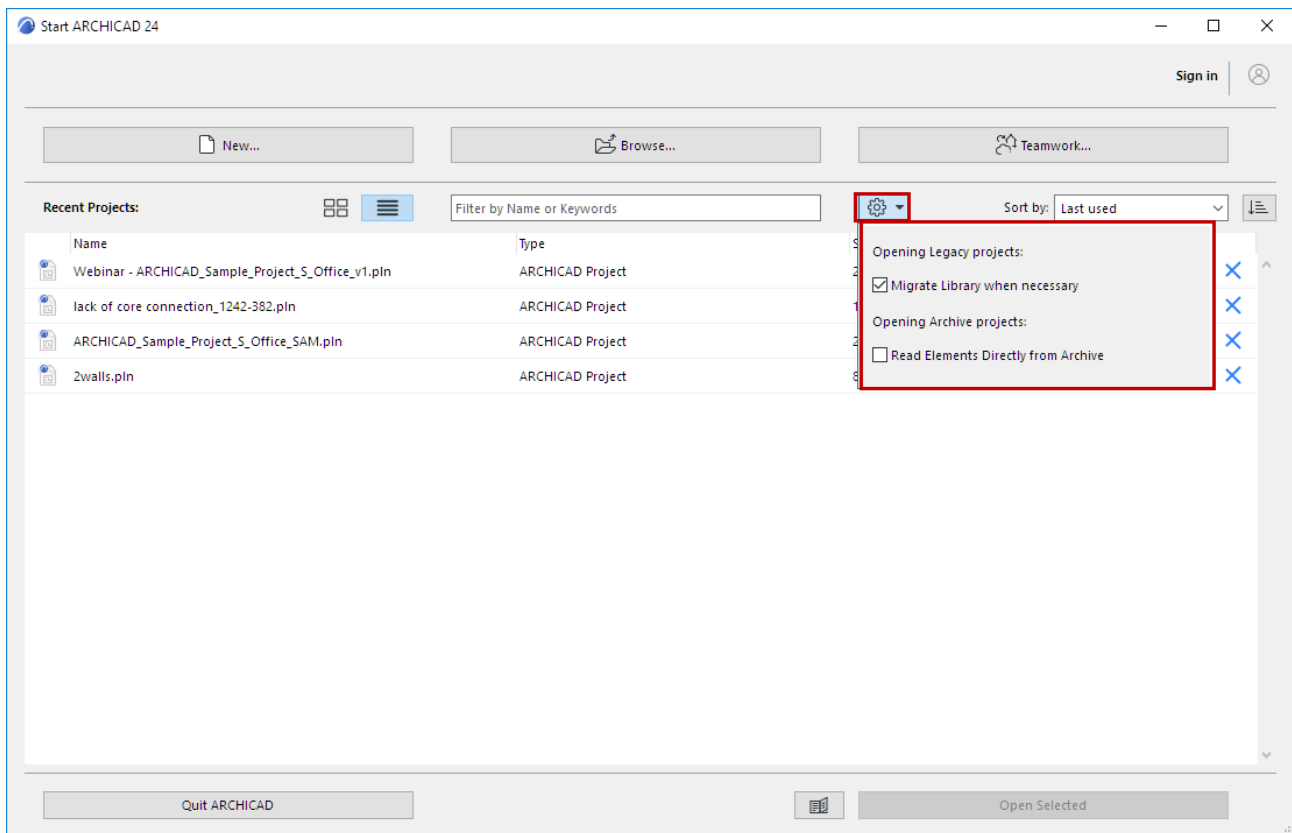
Note: To open the Start Archicad dialog, do one of the following:

- Double-click the Archicad program icon
- Click the Archicad Starter application

Legacy Projects

By default, Archicad will **Migrate Library when necessary**.

With this setting, the migration process takes place in the background every time you open a Legacy project (version 17 and up), with no user input required.



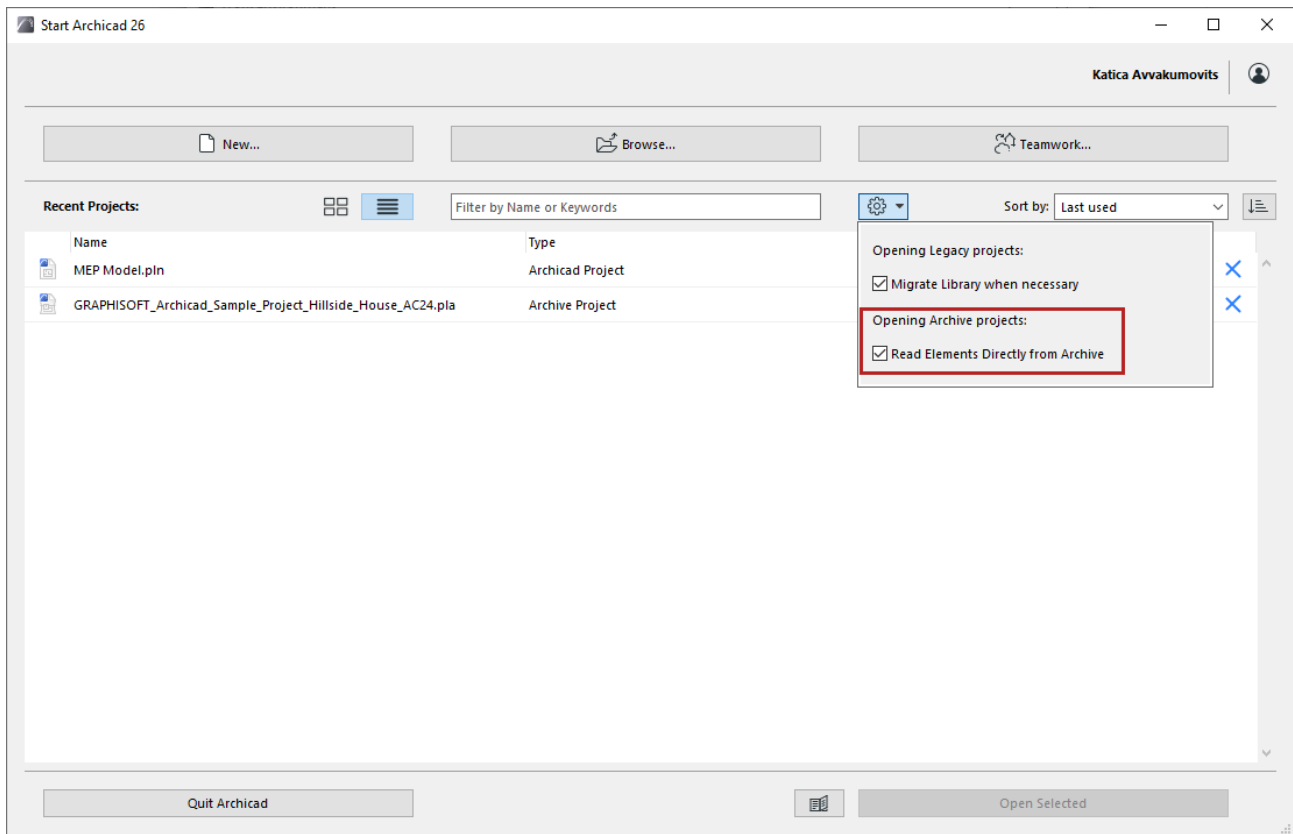
Archive Projects

Another option is available for the startup of Archive (pla) projects: Read Elements Directly from Archive.

If you check this box, library parts are read automatically, with no user input required.

If this box is not checked: each time you open an Archive file, the **Open Archive** dialog appears, where you must define how to handle the library parts.

See [Open Archive Project to learn about these options](#).



To view information about possible Library-related issues, check the Libraries item in the Action Center (**File > Info > Action Center**).

Related Topics

[Action Center](#)

Close Project

From Floor Plan: **File > Close** closes the entire Project.

External files (GDL Object windows, pictures) remain open until you close them separately.

Reopen Project Tabs/Windows of Closed Project

When you close a project with many tabs/windows open simultaneously, Archicad will reopen these the next time you open the project.

Re-opening all these tabs/windows may take time, especially if they must be rebuilt.

If the number of tabs/windows to be re-opened exceeds 20 (not including PhotoRendering windows), a warning dialog lets you re-open just the frontmost tab/window (plus the Floor Plan).

Related Topics

[Save Archicad Project](#)

[Close Teamwork Project](#)

Save Archicad Project

Use **File > Save** to save your project.

Choosing **File > Save as** allows you to save the Project under a new name, archive it or save the current window's contents in a variety of file formats. Some of the available options may depend on the set of currently loaded Add-ons. Some file formats are 2D-only or 3D-only formats.

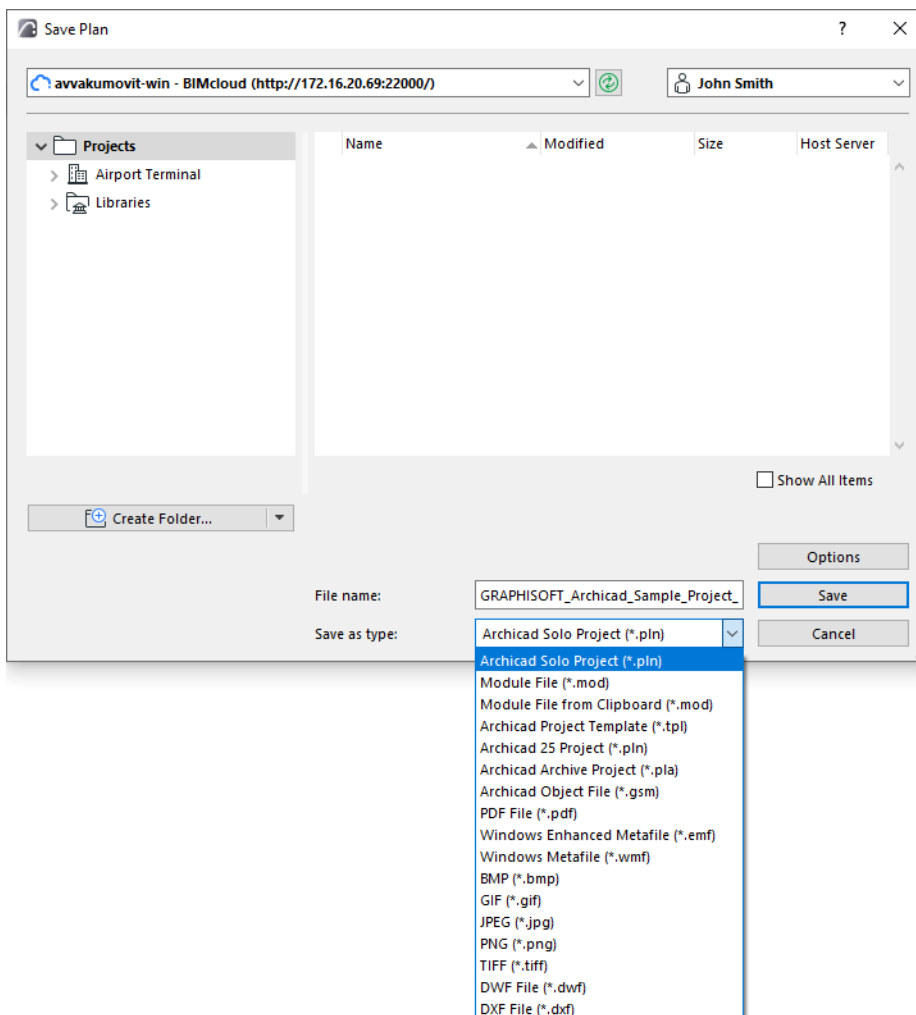
See [File Types Saved by Archicad](#).

Export to BIMcloud

Use **File > Export to BIMcloud** to upload the Archicad Project to the BIMcloud.

Choose any of the supported file types.

To use this function, you must be logged on as a BIMcloud user.



Save from List Window

- Basic **Element**, **Component** and **Zone Lists** can be saved as Plain Text, Tabulated Text, HTML tables or PDF. Click either radio button to include the file's **Used Column only** or **All Columns** in the file.
- Formatted lists including graphic information can be saved in.rtf (Rich Text Format), Archicad Project format, or PDF.

Save from Schedule Window (Interactive Schedule)

Schedules can be saved in the following formats: Tabbed text, DWF, .xls and PDF.

From a PC, with MS Office, you can also save in .doc format.

Related Topics:

[File Types Saved by Archicad](#)

[File Compression](#)

[Template Files](#)

[Backup Files](#)

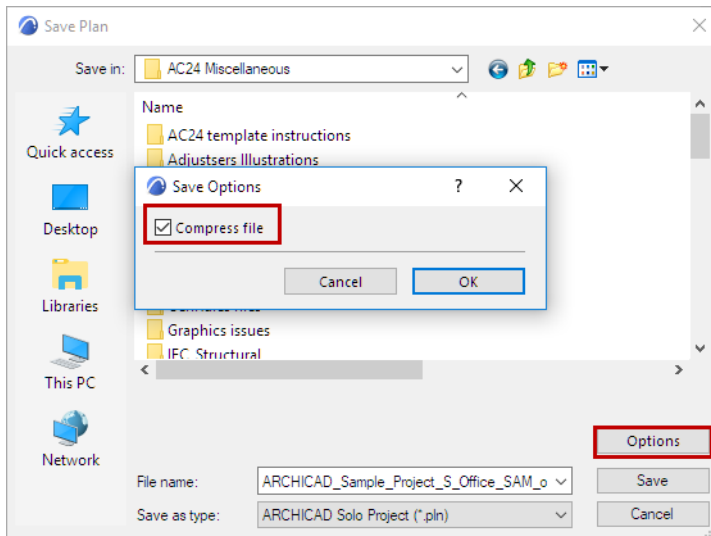
[Archive Projects](#)

File Compression

When saving certain file types from Archicad (.pln, .tpl, .pla, .mod), the **Options** button in the Save dialog box presents you with the option to “Compress file.”

This option is enabled by default, and is recommended in most cases, since file compression reduces file size considerably (by 60-70%). Using this option also increases download speed if you are working from a slow network, because the files are smaller.

However, note that using the Compress file option means increased file saving time, as a consequence of increased calculation needs. If the saving procedure takes too long, try turning this option off.



Bug Reporter

Graphisoft Bug Reporter is a program that creates an electronic report of a bug, error, crash or other incorrect functionality of a Graphisoft product. By sending in the Bug Report, you provide valuable information which helps Graphisoft fix problems and improve your experience. Thank you for your cooperation!

What Does Bug Reporter Do?

Bug Reporter starts automatically whenever your program crashes.

From the appearing dialog, choose to send either a Detailed or an Anonymous bug report.

In both types of reports, Bug Reporter gathers information about your hardware configuration, system software and Graphisoft applications you have installed on your computer. Unlike the Anonymous Report, the Detailed Report also includes your description of the problem, plus computer name, user name and license data.

[See the *Privacy Notice* for additional information.](#)

Send Detailed Report

1. In the text field, describe what you were doing when the problem occurred. This detail is very useful for Graphisoft.

For example:

“I tried to Click-to-Trim a circular wall with a closed spline intersecting it at 4 points, when bogus lines appeared on the screen.”

2. Optionally, add your email address.
3. Check the box: “I agree to send a detailed report...”.
 - This authorizes Bug Reporter to gather additional system data as detailed in the Graphisoft Privacy Notice.
 - Sending a detailed report also provides you with a Bug ID number, which you can use to follow up on the status of your crash report.
4. Click **Send Report**. If you are online, the report is sent to Graphisoft.
5. The feedback dialog displays the Bug ID.

If you wish to follow up on this bug with Technical Support, use this Bug ID to reference the issue.

Send Anonymous Report

You can leave the box unchecked, then click **Send Report**. This creates a Bug Report including limited system information, as detailed in the Privacy Notice. It does not include any additional data or description of the problem.

If You Are Offline

A Bug report is generated even if you are offline. In this case, you are prompted to save the report file to your computer, then send it manually to Technical Support.

Change Your Bug Report Preference (from Anonymous to Detailed)

By default, the Bug Report preference is “Anonymous.”

You can change this preference to “Detailed”.

To do this, double-click GSReport in the folder where Archicad is installed.

Work Environment

The following sections contain information on the main features of the Archicad user interface, and how to customize them.

[Default Work Environment Profiles](#)

[Arranging Palettes, Toolbars and Windows](#)

[Set Window Background and Grid Line Color](#)

[Tab Bar](#)

[Pop-up Navigator](#)

[Quick Options Bar](#)

[Toolbox](#)

[Info Box](#)

[Menus](#)

[Toolbars](#)

[Shortcuts](#)

[Palettes](#)

[Pet Palettes](#)

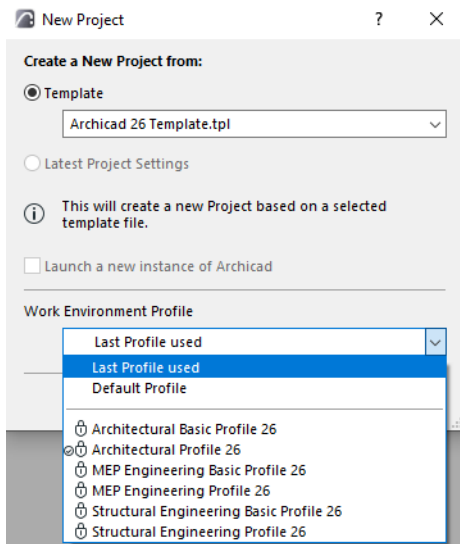
[Work Environment Dialog Box](#)

Default Work Environment Profiles

Archicad is shipped with default Profiles:

- **Architectural**
- **MEP Engineering**
- **Structural Engineering**

When you first start Archicad, the New Project dialog lets you choose from any of these profiles.

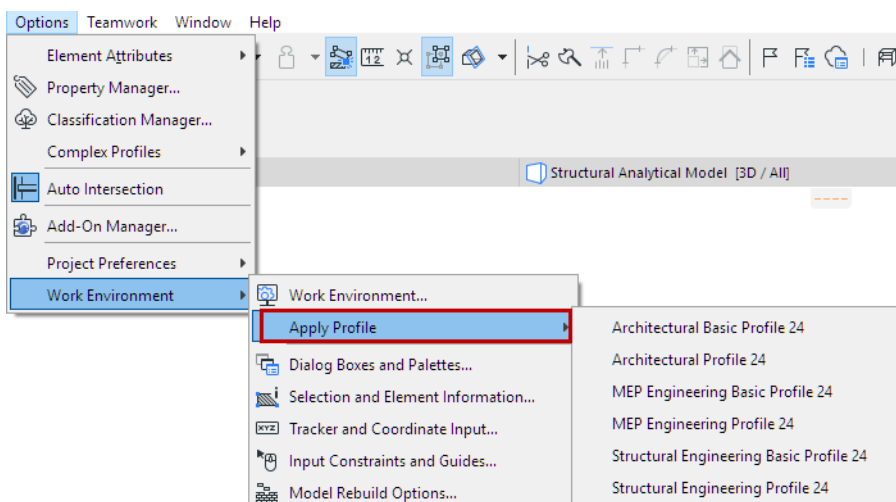


Each Profile, optimized for a particular discipline, is set up to make it easy to find the tools and commands you will probably need most often.

Naturally, you or your office CAD manager can reconfigure these profiles as needed.

Apply a Different Work Environment Profile

Use **Options > Work Environment > Apply Profile** and choose a profile.



Customize Work Environment Settings

As you get used to working with Archicad, you will develop personal preferences for using the program's features and arranging the various palettes, toolbars and menus on screen. You will set most of these options

in the many sub-dialog boxes opened from the **Work Environment** dialog box at **Options > Work Environment > Work Environment**.

You can customize any of these settings while you work: just open the needed dialog box, change the setting, and press OK. The setting will take effect.

See [Work Environment Dialog Box](#).

Arranging Palettes, Toolbars and Windows

Use the following Archicad commands of the **Window** menu to optimize your workspace.

Enter/Exit Full Screen

Use Full Screen to maximize the workspace.

Show/Hide Tab Bar

The Tab Bar is visible by default. Use this command to turn it on or off.

See also [Tab Bar](#).

Show/Hide Palettes and Toolbars

Use this to hide all the open palettes and toolbars.

Close All Other Tabs and Windows

Closes all Tabs and Windows except for the currently active one. (Inactive Floor Plan, Rendering, GDL Editor windows will not be closed.)

Palettes > Show Main Palettes Only

Shows only the main palettes: Toolbox, Info Box, Status Bar, Navigator, Quick Options.

Toolbars are not affected.

The arrangement and show/hide and docked status of your Toolbars, Palettes and the Tab Bar can be stored as part of the “Workspace Scheme” of your Work Environment.

See [Work Environment Schemes](#).

Unified Window Mode on Mac

By default, Archicad works in unified window mode, as the preferred working method for Mac users. (Archicad has always worked this way on PC.) All windows, palettes and toolbars are contained within a single application window, so you can open, close and move all of them together.

Exception: When migrating projects from pre-19 versions, the Legacy Window mode is on by default.

In unified window mode, the header displays the project’s file name. Cmd+click this file name to display its file path.

If you close the project, and no project windows remain open, the application frame will not be visible, but the Archicad application will not close (the menu remains on screen).

To use Archicad with floating windows (instead of a unified window) on a Mac, as in Archicad 18 and earlier, you can turn on “Legacy Mode” at **Options > Work Environment > Legacy Window Mode** (Mac only).

Dock or Undock Palettes or Toolbars

To dock/undock a palette or toolbar, do one of the following:

- Double-click the header. If the palette/toolbar is undocked, it will jump to a docked position, and vice versa.
- Click on its header (on PC, the drag symbol will appear) and drag it to one side or the top or bottom of the screen. Release the cursor to dock it.
- To undock: click the header and drag it out.

To **suspend** the docking function while dragging a palette or toolbar, press Ctrl (PC only).

To **cancel** the docking/undocking, press ESC.

Note: To dock all undocked windows (Mac only), use **Window > Dock All Windows**. This command is not visible by default; you can add it to your Window menu by customizing your Work Environment (Options > Work Environment > Menus).

Toolbars can only be docked at the very edge of the screen (left/right/top/bottom), but they can be next to each other in multiple rows; docked toolbars won't overlap each other. They do not snap together or move together the way palettes do, and they cannot be stretched.

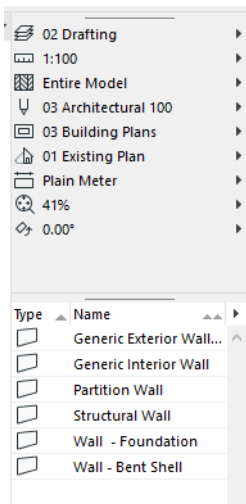
Limitations on Docking Palettes

The following palettes *cannot* be docked: Mark-Up Tools; Find and Select; RoofMaker; Solid Element Operations; Profile Manager, PhotoRendering Settings, MEP Routing, IFC Project Manager, Energy Model Review.

Docking Groups of Palettes

You can snap several docked palettes together, forming a group.

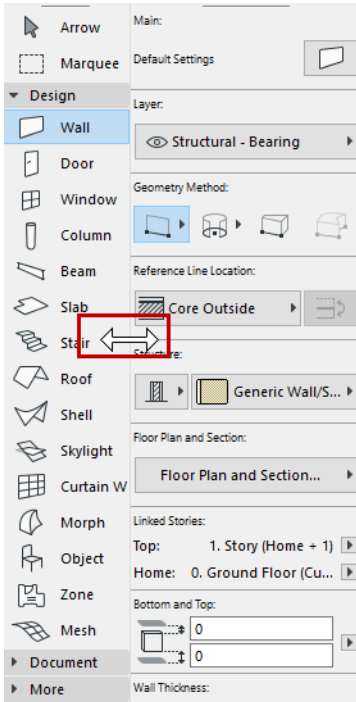
- To insert a second palette *above* the first, click and drag the second palette onto the header of the first. To insert a second palette *below* the first, click and drag the second palette to the line at the bottom of the first one.



Quick Options Palette docked with the Favorites Palette

- Within a group, you can arrange palettes next to each other vertically and horizontally in the desired arrangement.
- If you resize any palette that is part of a palette group, the rest of the palettes in the group will be resized automatically.

- To set how much of each palette in a group should be visible on screen, drag the splitter bars.



Snap Undocked Palettes Together (Mac Only)

On Mac, you can snap undocked palettes to each other.

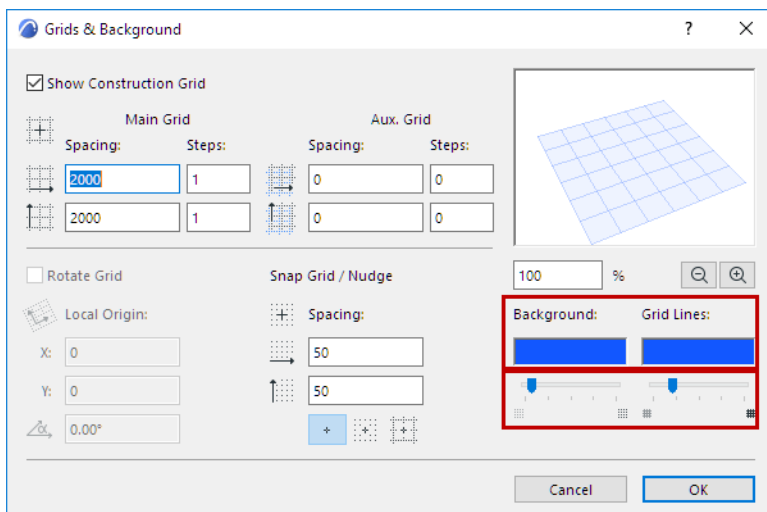
Palettes will snap to each other if you pull them within a given range of each other. The palette edge (not the cursor) is the docking anchor. Once snapped together, you can drag all of them at once, provided you grab the palette at top left.

To **suspend** the snapping function while dragging a palette, press Cmd.

Set Window Background and Grid Line Color

1. Open **View > Grid & Editing Plane Options > Grids & Background**.
2. Double-click the **Background** field to open the **Set Color** (on Mac: **Color**) dialog box to select a different color.
3. Double-click the **Grid Lines** field to open the **Set Color** (on Mac: **Color**) dialog box to select a different color for the Grid Lines.

Note: The color you select here will also be used for the dimmed Project Origin when it is temporarily displaced.

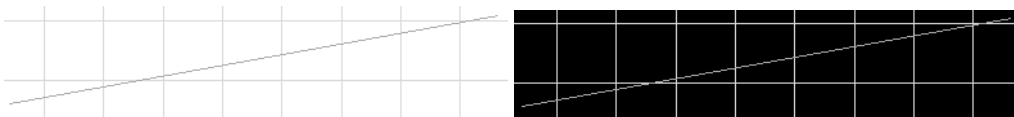


The next two controls are available only if the 3D window is active:

Grid Background/Grid Opacity: Use the sliders to set the intensity of the grid background (editing plane) and the grid lines in the 3D window only.

Automatic Pen Color Visibility Adjustment for Model Views

When the luminance value of a particular background color falls below a threshold value - that is, if your background is sufficiently dark - black pens will be shown as white on your Archicad screen. (This is useful if you use a dark or black background to imitate AutoCAD methods.)



However, when printed, pen colors will print according to their actual settings in the pen set.

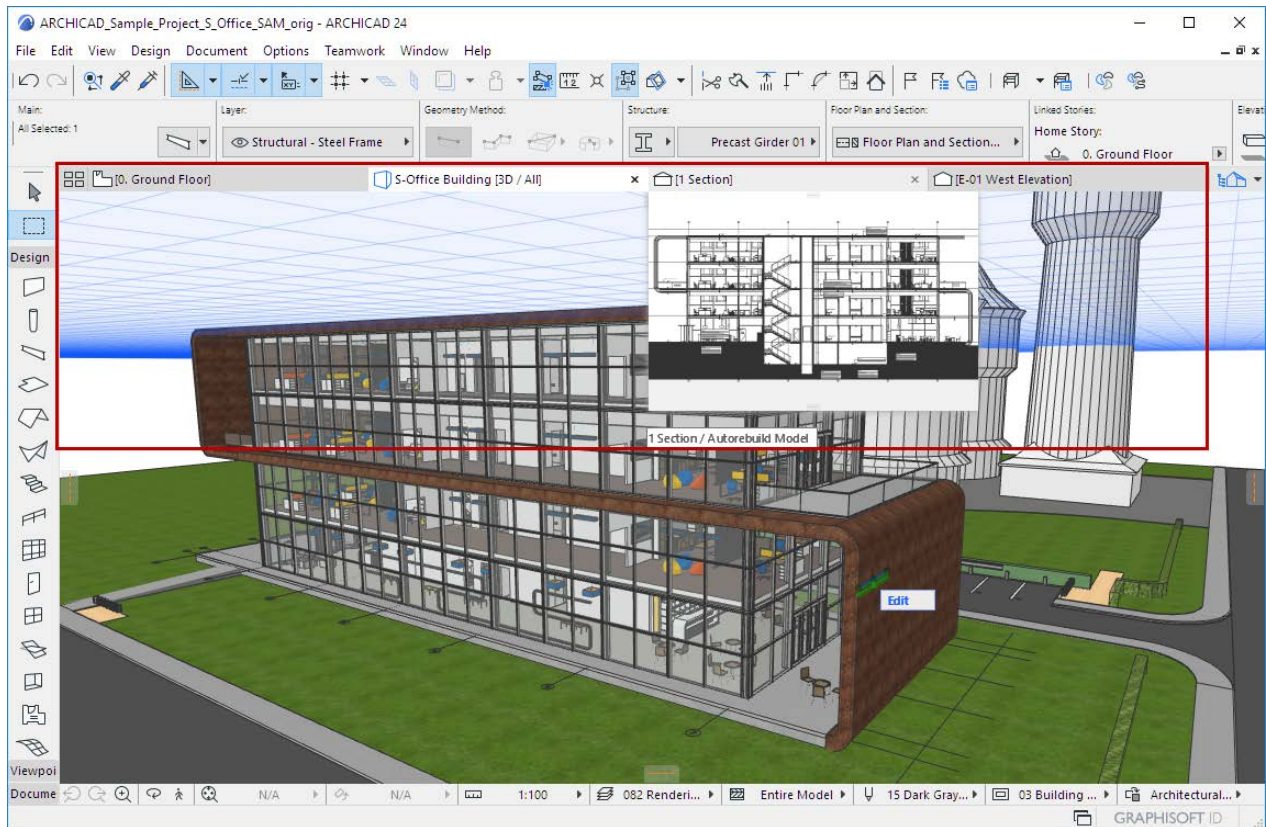
Non-black pens that would be difficult to see against a particular background will automatically shift to a similar, but more easily visible color. On a white background, elements in white pens are adjusted to a light gray to enhance visibility. Again, when printed, pen colors will print according to their actual settings in the pen set.

To disable this automatic color adjustment, uncheck the **Automatic Pen Color Visibility Adjustment** checkbox in **Options > Work Environment > On-Screen Options**.

[See On-Screen Options.](#)

Tab Bar

The Tab Bar at the top of your Archicad workspace shows all of your open views/viewpoints.



- Click any tab to activate that window.
- Hover over any tab to see a preview of its last opened view, with its name.

Turn off Tab Bar

The Tab Bar is visible by default. To turn it on and off, use **Window > Show/Hide Tab Bar**.

Turn off Tab Preview

By default, Tabs display a preview when you hover over them. Change this preference in **Options > Work Environment > More Options**.

Topics in this Section

[Click Tab to Activate](#)

[Tab Overview](#)

[Tab Navigation Shortcuts and Commands](#)

[Tab Bar Context Menus](#)

[Open a Tab from a Marker](#)

[Overflow Items on the Tab Bar](#)

[Close Tabs](#)

[Dock/Undock Tabs \(Mac\)](#)

[Dock/Undock Tabs \(Win\)](#)

Open New View/New Layout Preference

Click Tab to Activate

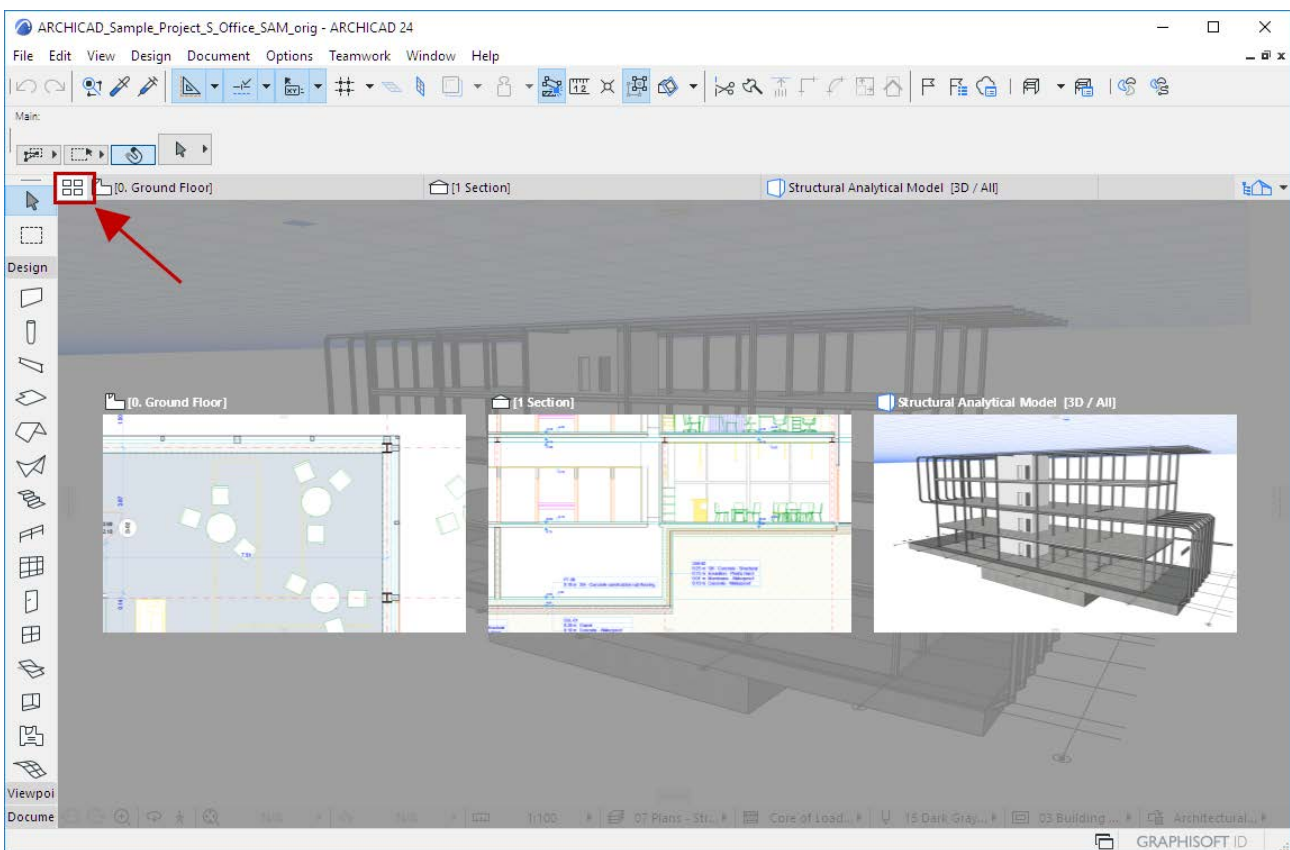
Click any tab, or click on its preview, to activate it.



Note: A tab of a view that has not been opened yet has no generated preview, and displays a view icon instead.

Tab Overview

Click the Tab Overview icon to show/hide all tab previews as an overview, then click the one you need:



Show/Hide Tab Overview

Tab Navigation Shortcuts and Commands

- Cycle among the tabs with the shortcuts:
 - Ctrl/Cmd+Tab (from left to right)

- Shift Ctrl/Cmd + Tab (from right to left)

See also [Overflow Items on the Tab Bar](#).

- Use the **Window > Show Next Tab** or **Window > Show Previous Tab** commands

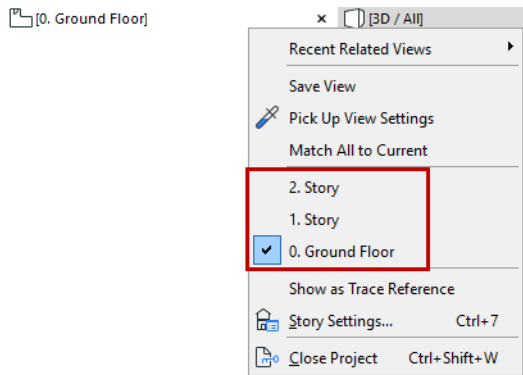
Note: If you do not see these commands in the menu, use the **Options > Work Environment > Menus** dialog box to add them.

Each new Tab is appended to the right end of the Tab Bar.

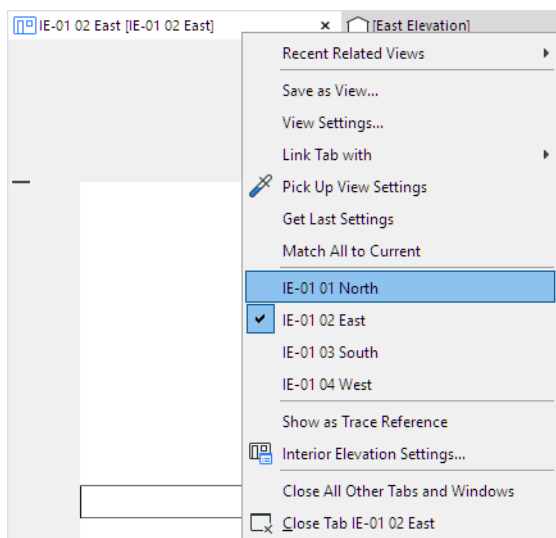
Tab Bar Context Menus

Available items vary depending on the type of view/viewpoint.

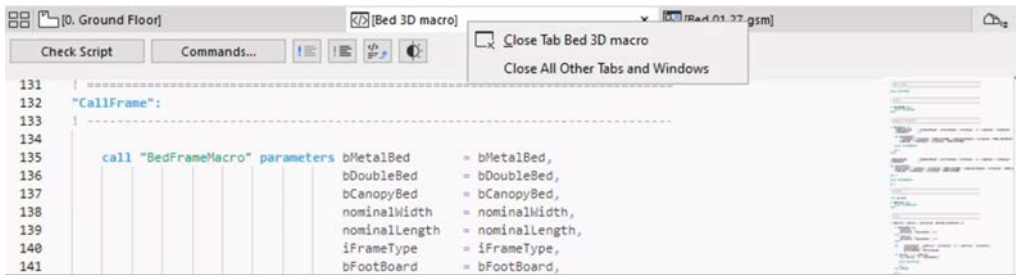
- From the Story tab, navigate to any story



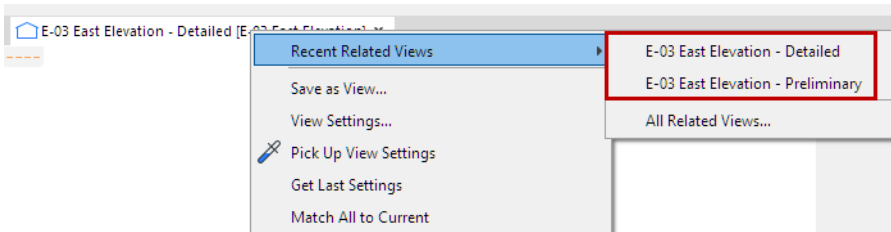
- From an Interior Elevation view tab, navigate to any other IE in the same group



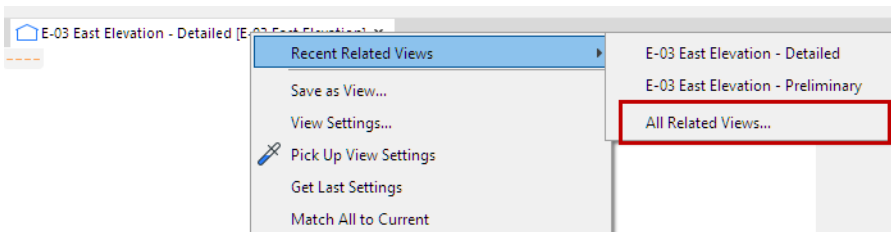
- From a 3D window tab, switch between Axonometric and Perspective views
- From a GDL editing window, switch back to the Master window



- From any viewpoint or view, find recent views created from the same viewpoint.

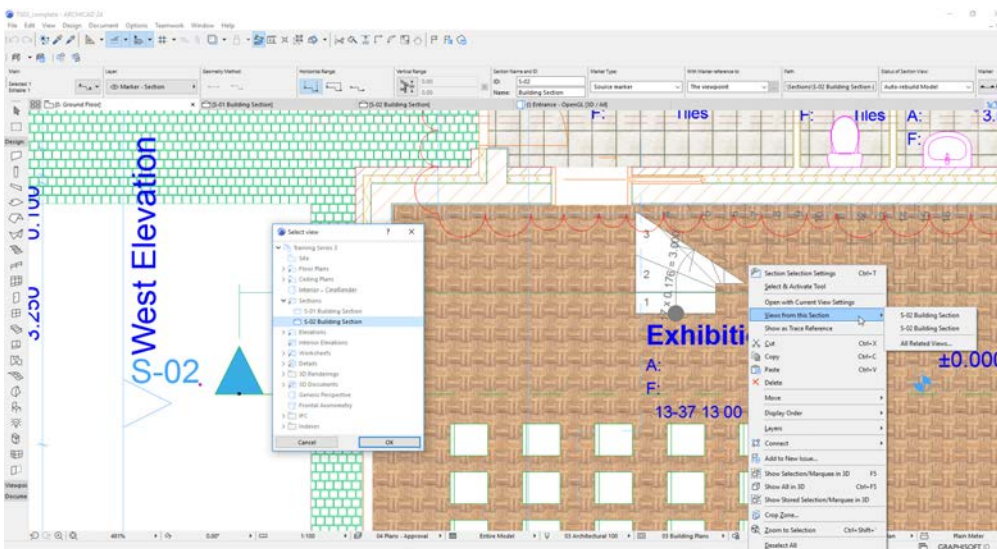


- Click **All Related Views** to access, in a tree structure, all the views created from the same viewpoint as the tab:



Open a Tab from a Marker

Use a placed marker to navigate to any view created from a particular viewpoint.



Select Placed Marker, Then Right-Click to Open Any Related View

Overflow Items on the Tab Bar

You can have any number of tabs open at a time. If they no longer fit into the width of the tab bar, a double-arrow pop-up appears at the right edge of the Tab Bar.

Click to see a list of the additional tabs.

The last visible tab is also listed here, shown with a checkmark.



Click on any of these to activate it and add it to the visible tabs on the Tab Bar.

Close Tabs

Do one of the following:

- Click the X on the tab
- Use the **Close Tab** command from the tab's context menu
- Use the **Close Tab** command from the Window menu
- Click the mouse-wheel over the tab

To close all tabs except the currently active one, use **Window > Close All Other Tabs and Windows**.

Dock/Undock Tabs (Mac)

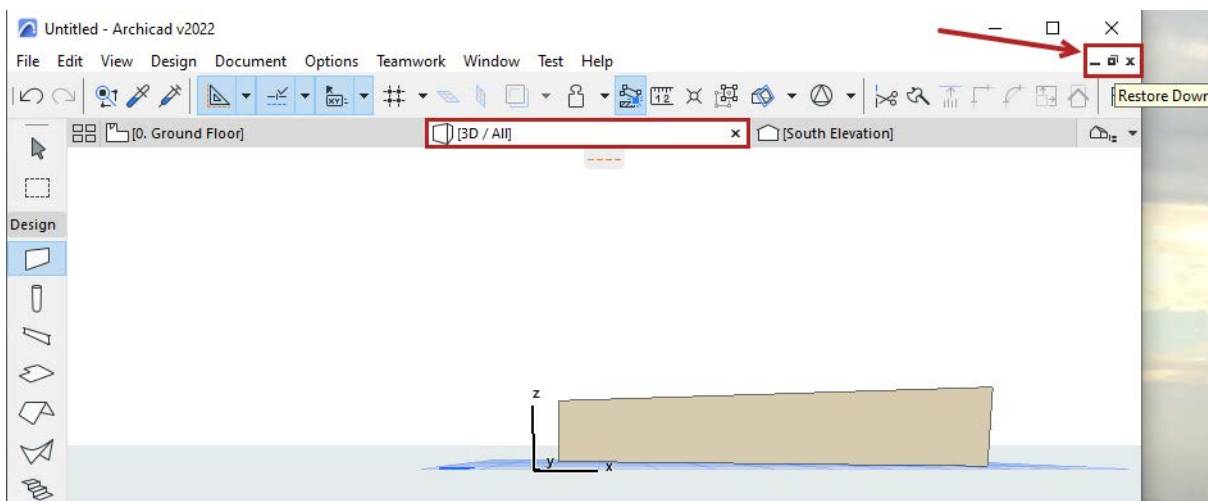
Do one of the following:

- Use the Undock command from the tab's context menu
- Use the **Window > Undock** command.

To dock a tabbed item, use the **Window > Dock** command.

Dock/Undock Tabs (Win)

Undock or minimize the active tab using the icons in the upper right corner.



Open New View/New Layout Preference

When you open a new view or new layout, it can either replace the opened one (of the same type, in the case of views), or be opened in a separate new tab. Change this tab-opening preference in **Options > Work Environment > More Options**.

Exception: You can have only one Floor Plan and one 3D window open at a time.

Related Topics:

[Transfer Settings Among Tabs](#)

[Watch the Video](#)

[Watch the Video](#)

Transfer Settings Among Tabs

Several explicit commands and shortcuts enable you to transfer or otherwise control tab settings.

Shortcut: Use Current Settings When Switching Tabs

In contrast to the general rule that tabs retain their own settings, you can use this shortcut to apply the currently active settings when you click to open a new tab:

- on Mac: Cmd + click
- on Windows: Ctrl + click

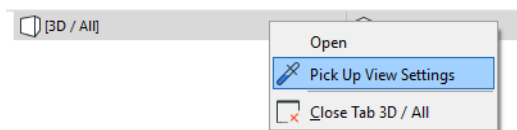
Get Last Tab's Settings

From a tab's context menu, use the **Get Last** command to activate the settings of the last-opened tab for the currently open tab.

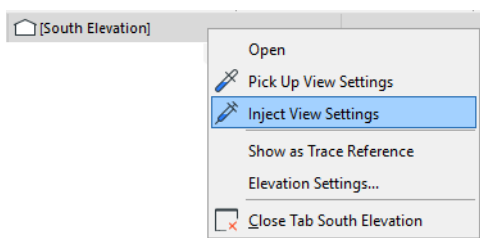
Pick Up/Inject View Settings

Use the Pick Up/Inject commands to transfer the View Settings from one tab to another.

1. Right-click on any tab whose settings you want to transfer. (It does not have to be the currently opened tab.)



2. Click **Pick Up View Settings** (or Alt+Shift on the tab).
3. Right-click the tab of the view/viewpoint where you want to inject the settings.
4. Click **Inject View Settings** (or Ctrl+Alt+Shift on the tab).



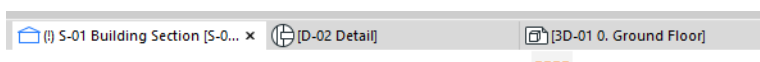
5. The clicked tab is now a viewpoint, which uses the View Settings you injected. If needed, use the **Save View** command from the tab's context menu to save it as a view.

Match All Settings on Tab Bar

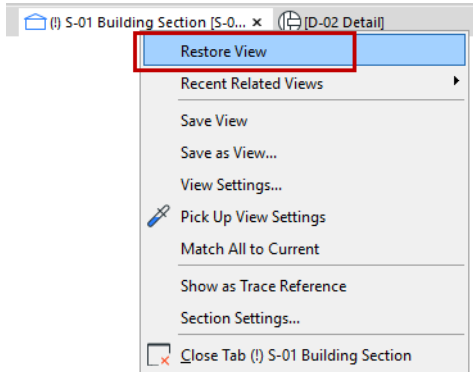
1. Right-click the tab of the currently opened view/viewpoint
2. Click **Match All to Current**. This applies the View Settings, *except the Zoom and Scale settings*, of the current tab to all the tabs (including grouped ones).

Restore View Settings from View Tab

If the current view that you are editing no longer matches its saved View Settings, the tab is marked with a warning.



To restore the view's saved status, **Double-click** the tab (or click **Restore View** from the tab's context menu). If the view is linked to other tabs in a group, view settings **change identically** in all linked tabs.



[Watch the Video](#)

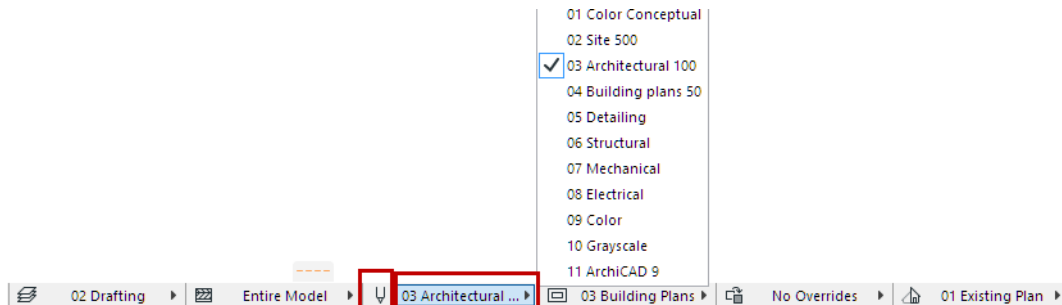
[Watch the Video](#)

[Watch the Video](#)

Quick Options Bar

The Quick Options Bar is shown by default at the bottom of the Archicad window.

Its controls show the *current* settings of the active tab. Use these as a quick way to apply changes among these settings, and to access the dialog box used to set up the relevant options set.



Use **Window > Show/Hide Quick Options Bar** to display or hide it as needed.

Controls available from the Quick Options Bar:

Zoom

See [Zoom](#).

Orientation

See [Set Orientation](#).

Layer Combination

- Applies a new Layer Combination to the view.

See [Layer Combinations](#).

Scale

- Applies a new scale to the currently active window.

See [Scale](#).

Partial Structure Display

- Changing this setting applies a new Partial Structure Setting to the currently active window.

See [Partial Structure Display](#).

Pen Set

- Applies a new pen set to the model. (Not available in the Layout window.)

See [Pen Sets](#).

Model View Options Combination

- Changing this setting applies a new Model View Options Combination to the whole project.

See [Model View Options Combinations](#).

Graphic Overrides Combination

- Applies saved set of Graphic Override Rules to the view.

See [Graphic Overrides](#).

Renovation Filter

- Choose a Renovation Filter to assign to the view.

See [Renovation](#).

Option Combination

- Applies a Design Options Combination to the view.

See [Design Options](#).

Dimensions

- Applies a new Dimensioning standard to the whole project. (Not available in the 3D window)

See [Dimensions Preferences](#).

3D Styles

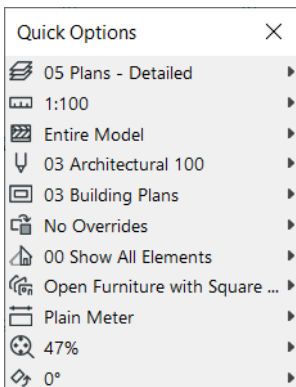
- Applies a 3D Style to the 3D window.

See [3D Styles](#).

Quick Options Palette

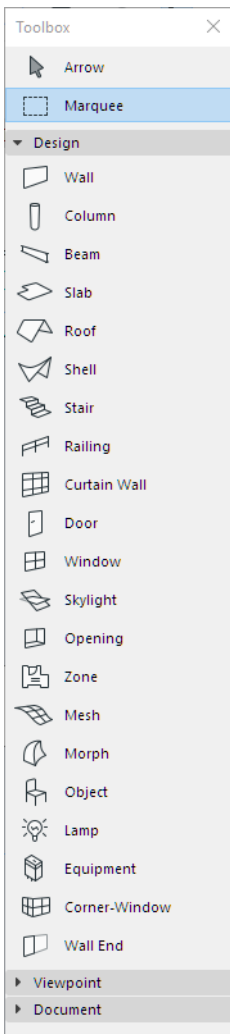
Use this palette as an alternative to the Quick Options Bar.

To hide/show the Quick Options Palette, use **Windows > Palettes > Quick Options Palette**.



Toolbox

The **Toolbox** shows a variety of tools for selection, 3D construction, 2D drawing and visualization.



The Toolbox is divided into Tool Groups to make it easier to locate the tool you need.

Toolbox contents vary, depending on the current Work Environment Profile and any Add-ons.

Display the Toolbox

Use **Windows > Palettes > Toolbox**

Using the Toolbox

Do any of the following:

- Click any tool to activate it.
- Double-click to open its Tool Settings dialog box.
- Hover over the tool icon to access the Favorites chooser pop-up for that tool.

[See also Favorites.](#)

Customizing the Toolbox

Use **Options > Work Environment > Toolbox** to customize the contents and arrangement of your toolbox.

See [Toolbox Customization Dialog Box](#).

Store your customized Toolbox settings as part of a Tool Scheme in your Work Environment.

For details, see [Work Environment Schemes](#).

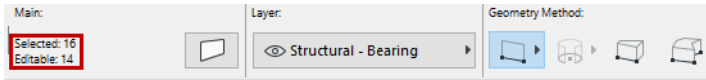
Info Box

The Info Box is a condensed collection of input and parameter controls for the currently active tool (default Settings) or selected element (Selected/Editable Settings).

Unlike Element Settings, the Info Box stays on screen while you work.

If several elements are selected, the Info Box displays the controls for the last selected element.

- **Selected/Editable:** The current Info Box gives you feedback on the number of Selected Elements, as well as how many of those are Editable. Changes made to Info Box settings will affect the Editable elements. (An element might not be Editable if it is on a locked layer.)



Header texts indicate what each Info Box control represents.

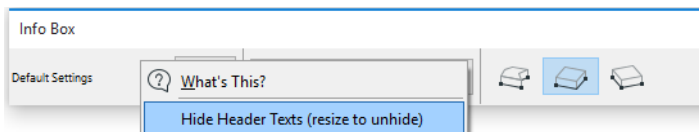


Display Info Box

Windows > Palettes > Info Box

Hide Header Texts

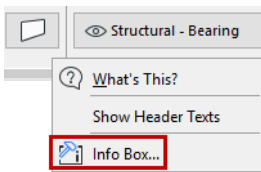
Right-click anywhere in the Info Box and choose Hide Header Texts.



Customizing the Info Box

You can customize the order and visibility of panels in each Tool's Info Box: go to **Options > Work Environment** and open the Info Box Customization Dialog Box.

An easy way to access this dialog box is to open the Info Box context menu by right-clicking anywhere in the Info Box, then click Info Box...



See [Info Box Customization Dialog Box](#).

You can store your customized Info Box settings as part of a Tool Scheme in your Work Environment.

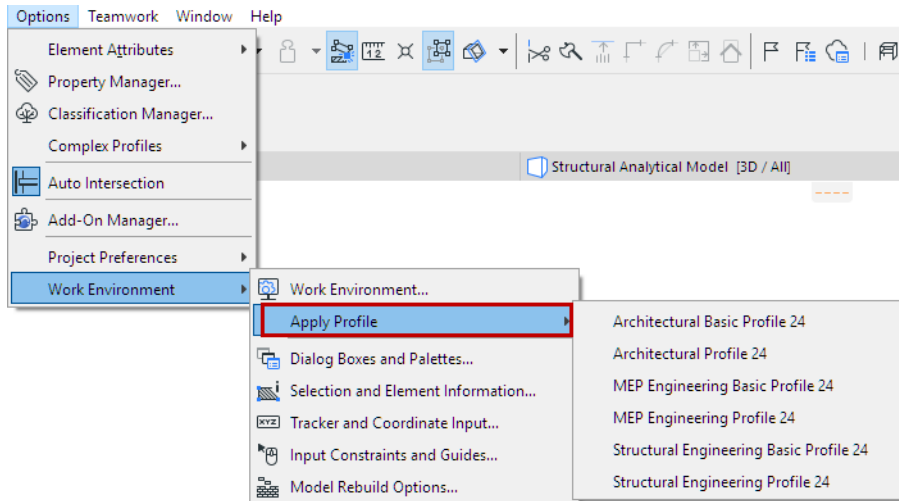
For details, see [Work Environment Schemes](#).

Menus

Menus are displayed across the top of the Archicad window.

If you don't see the menu you expect, try changing your Work Environment Profile:

Use **Options > Work Environment > Apply Profile** and choose a profile.



Customizing Menus

Use the settings in the **Options > Work Environment > Menus** dialog box to customize the content of your menus.

Any command or menu can be placed into or removed from any menu; the order of the commands within any menu is entirely customizable.

For details, see [Menu Customization Dialog Box](#).

Exception: Context menus are not customizable.

You can store your customized menu command settings as part of a Command Layout Scheme in your Work Environment.

For details, see [Work Environment Schemes](#).

Toolbars

A toolbar is a collection of commands and/or menus displayed in icon or text form and grouped by topic. It's a good way to easily access frequently used commands, by theme.

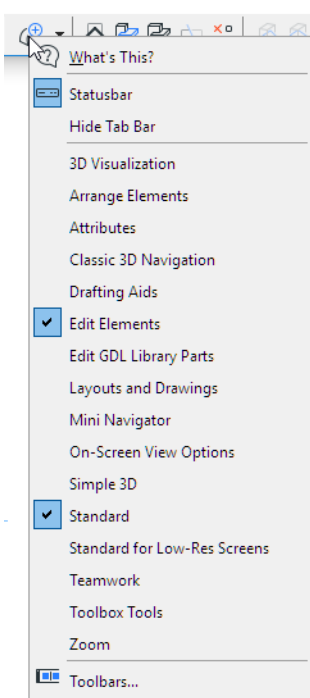


The “Edit Elements” Toolbar

Display Toolbar

Open a Toolbar from **Window > Toolbars**.

Or from the context menu of any displayed toolbar.



See also [Arranging Palettes, Toolbars and Windows](#).

Create or Customize Toolbars

Use **Options > Work Environment > Toolbars**

See [Toolbar Customization Dialog Box](#).

You can store your customized toolbars as part of a Work Environment scheme:

- The *content* of named toolbars are saved as part of a Command Layout Scheme.
- The *on-screen display* of toolbars is saved in a Workspace Scheme.

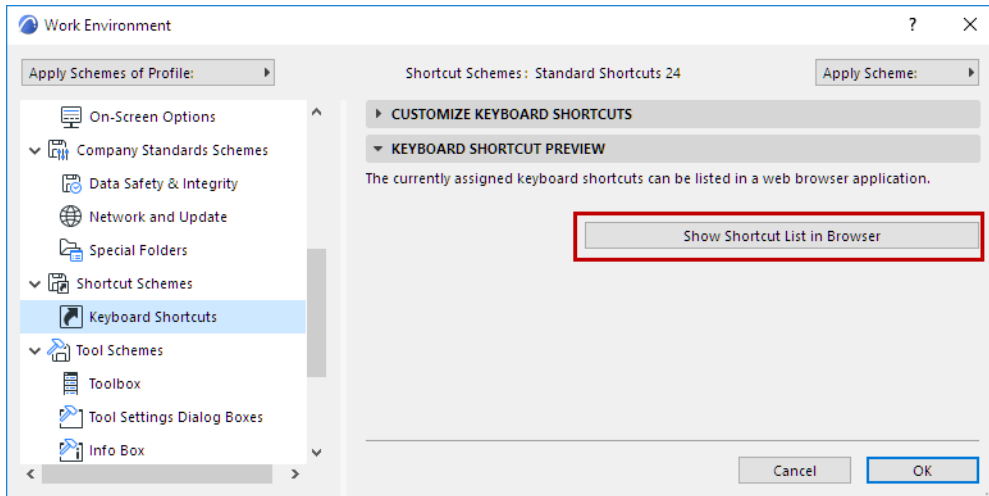
For details, see [Work Environment Schemes](#).

Shortcuts

The shortcut keys in effect are defined by your Work Environment settings.

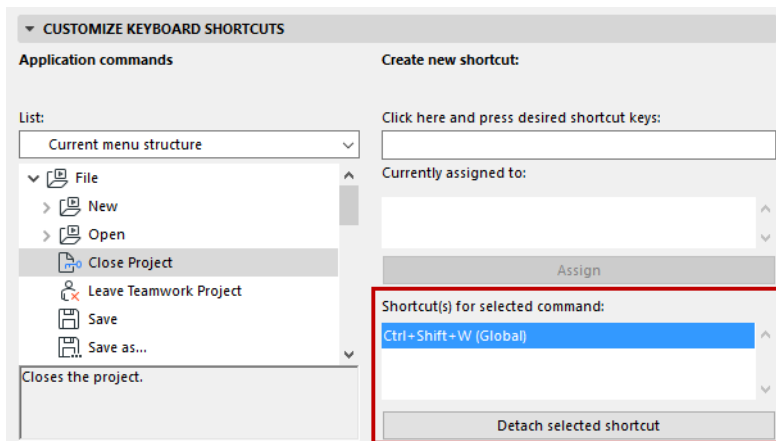
View/Print List of Keyboard Shortcuts

1. Go to **Options > Work Environment > Keyboard Shortcuts**
2. Click the **Show Shortcut List in Browser** button at the bottom of the Keyboard Shortcut Preview Panel.



Customize Keyboard Shortcut

1. Go to **Options > Work Environment > Keyboard Shortcuts**
2. Choose a command from the list on the left.
3. Enter the desired shortcut combination in the field at the right.



4. Click **Assign**.

For more details, see [Shortcut Customization Dialog Box](#).

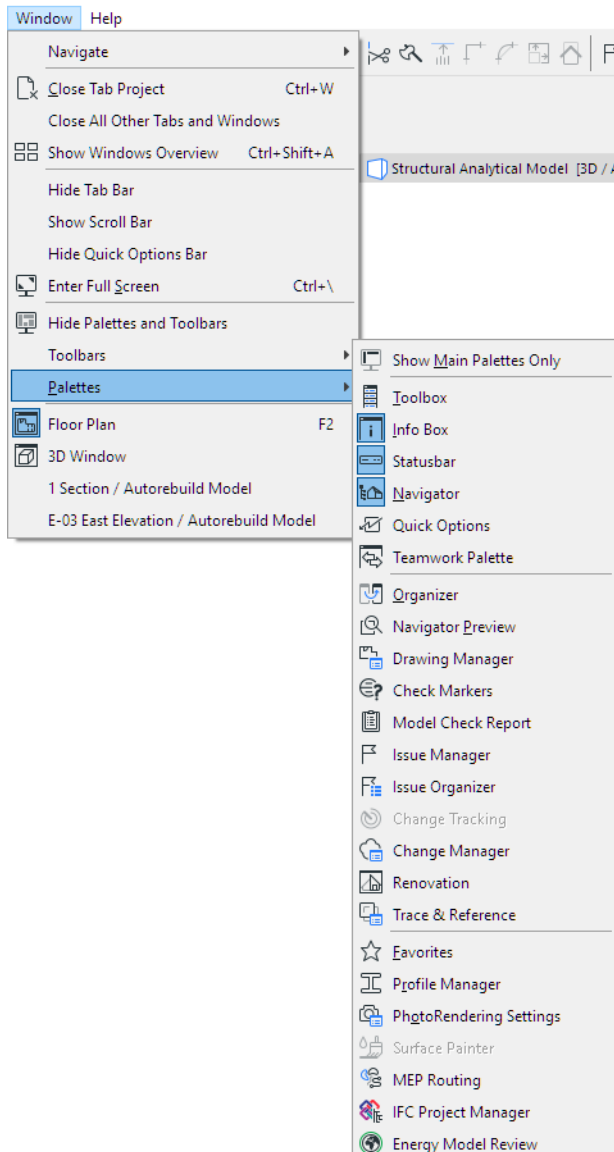
Apart from a few non-customizable shortcuts (listed at the bottom of the Show Shortcut list), all shortcuts in any scheme are customizable. Changes to the selected Shortcut scheme are applied when you press **OK** and close the dialog box.

You can store your customized Shortcut settings as part of a Shortcut Scheme in your Work Environment.

For details, see [Work Environment Schemes](#).

Palettes

Archicad's palettes help you construct, modify and locate elements. Each palette can be shown or hidden separately using the **Window > Palettes** command.



Show Main Palettes Only

The main palettes (Toolbox, Info Box, Status Bar, Quick Options and Navigator) can be enabled all at once, using the **Window > Palettes > Show Main Palettes Only** command.

See also [Arranging Palettes, Toolbars and Windows](#).

Each of these main palettes is described in detail in the Archicad documentation:

Toolbox

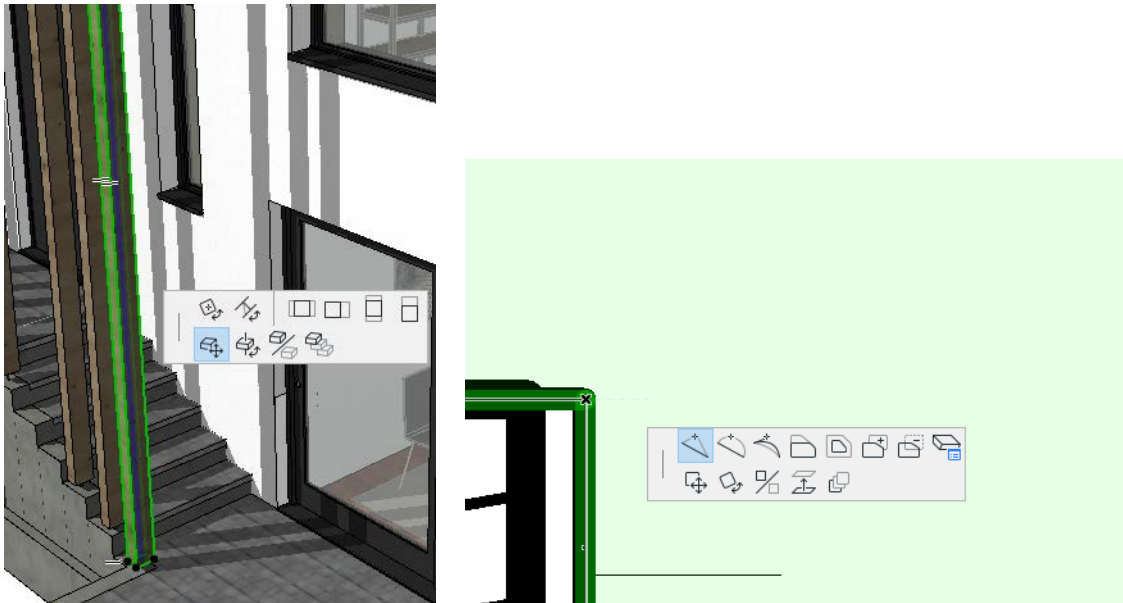
Info Box

Navigator

Quick Options Palette

Pet Palettes

Use the pet palette while editing elements on the fly: the pet palette will always display the relevant editing commands you need for that particular element.



To access the pet palette:

1. Select the element
2. Move your cursor on an editable edge, node, or surface
3. Click to see pet palette
4. Click the command you need.

Sometimes, a pet palette appears during input (as when inputting a polyline)

Pet palette commands vary depending on:

- selected element
- type of selection (edge, node or surface)
- window (e.g. 3D vs. Floor Plan)

Use the shortcuts “**F**” and **Shift+F (Opt+F)** to move to the next/previous icon

You cannot customize the contents of a pet palette.

You can change your mind and choose a different function from the pet palette as long as you have not completed the editing operation.

The pet palette automatically disappears when the operation is finished.

Pet Palette Movement

The pet palette will either follow your cursor around on screen as you work (like a pet on a leash), or be placed in a preferred spot (and told to “stay”), depending on which **Pet Palette movement** option you specify.

Set these preferences at **Options > Work Environment > Dialog Boxes and Palettes**.

See [Dialog Boxes and Palettes](#).

▼ DIALOG BOXES AND PALETTES

Dialog Box Auto-Update Delay: sec

Hide Locked Layers in Pop-Up Palettes

Use scroll effects for dialog box panel opening

Speed: (Slow to Fast)

Move up dialog box if it cannot accommodate more panels

Close panels if the dialog box cannot accommodate more panels

Pet Palette movement: Follow Cursor Jump to preferred position

Distance from cursor:

when appearing on screen: (Near to Far)

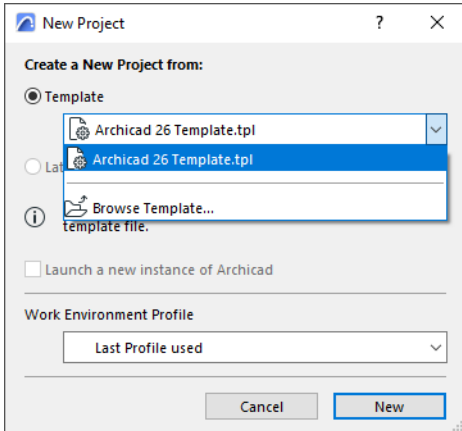
when following the cursor: (Near to Far)

Position of Text Formatting Palette: Above Text Editor Jump to preferred position

Template Files

A template is a read-only project file with extension `.tpl`. It contains all project preferences settings, placed elements and tool default settings of the project.

When you create a new project in Archicad using a template file, this default template is at the top of the pop-up list when you choose a template:



See also [Create New Project](#).

Template File Location

The default template is located in the Archicad Defaults folder.

On PC: \Program Files\Graphisoft\Archicad 27\Defaults\Archicad

On Mac: \Applications\Graphisoft\Archicad 27\Defaults\Archicad

To create a customized template:

1. Open a new empty project file.
2. Edit your project preferences, IFC Translator settings, and set up the project structure and/or place elements.
3. Save this project file as a template: use **File > Save as**, and choose “Archicad Project Template (*.tpl)” as the file type.

To open a template:

Do one of the following:

- Use **File > Open**, and browse for an existing template file
- Use **File > New**
 - In the appearing New Project dialog, use **Create New Project from Template**
 - Select the desired template file.

Note: Template File selection also occurs when you import or hotlink an IFC-type file to Archicad, because IFC Translator files are saved in a template.

For working with IFC Translators, see [Important Note on Translators and Template Files](#)

Note: Settings of the Work Environment Profile used to open your project will override Template file settings.

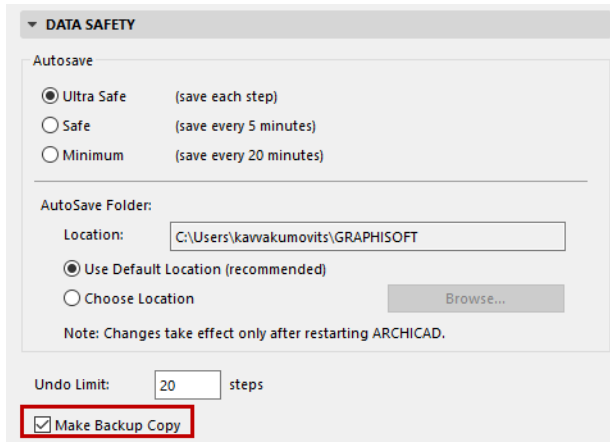
Backup Files

Open a Backup Project file (.bpn) to recover the latest backup copy of a damaged project file.

To open a backup file, do one of the following:

- Browse and double-click its name in the file system
- From the Archicad Open dialog or other system dialogs: use the dialog's "All file types" filter to include BPN files in the available list

Make sure that **Make Backup Copy** is active in **Options > Work Environment > Data Safety & Integrity**.



Related Topics:

[Data Safety & Integrity](#)

Archive Projects

Documents saved as Archives are stored together with their Library Parts (in a Library Container File, with extension .lcf), and properties contained in and defined with the Project. The Archive format is recommended if you want all files referenced by the project's placed elements to be included.

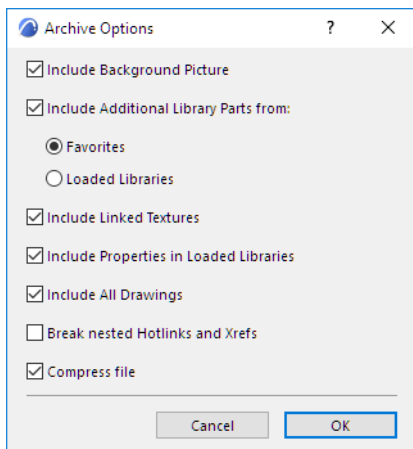
The Archive file format is recommended when:

- Moving a Project to another computer
- Storing a completed Project

The **Archive Options** dialog box allows you to add selected additional items to the Archive file (see below).

Save Project in Archive format

1. Use the **File > Save as** command.
2. From the appearing **Save** dialog box, choose Archicad Archive Project (*.pla) from the Save as type dropdown.
3. Click the **Options** button to access the Archive Options dialog box:



- **Include Background Picture** (if loaded in the Background Panel of PhotoRendering Settings)

Use next three checkboxes to include additional loaded library parts as part of the Archive file (in addition to library parts placed in the project).

- **Include All Parts in**
 - **Favorites:** Save all library parts that are included in the project's Favorites (to avoid missing objects when trying to use Favorites)
 - **Local Libraries:** Save all parts from all loaded libraries
- **Include Linked Textures**
- **Include Properties in Loaded Libraries**
- **Include All Drawings:** This will include *all* drawings placed on layouts, even those which are NOT marked as "Store Drawing in the Project file" in its Drawing Settings dialog box.
- **Break nested Hotlinks and Xrefs**

See [Nested Modules \(Hotlink Module Manager\)](#).

- **Compress file**

[See File Compression.](#)

[Watch the Video](#)

Important Notes about Archive Projects:

- If any Library Parts are **missing** from your Project at the time you save the Project as an Archive, these missing parts will not be included in the Archive document either.
- If you **add a .pla file to your Linked Libraries** folder using Library Manager, make sure you add the explicit .pla file itself, not a folder that contains the .pla. Otherwise the parts located within the .pla will not be read.

[See Library Manager.](#)

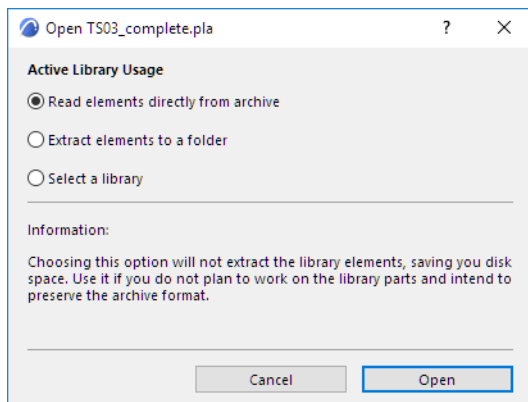
- Because of the larger volume of information it stores, an **Archive file size** is somewhat larger than the corresponding Project document.
- Archives store the **font information** used in the project, but not the fonts themselves. This means that if you wish to display textual information the same way as it appears in the original environment, you have to install the appropriate fonts.
- Archive files do not store **Add-ons**. Therefore, if your project contains elements affected by Add-ons, these elements may lose certain features or behavior if these Add-ons are not present.

Open Archive Project

Do one of the following:

- Use **File > Open > Open** to browse for an archive (PLA) project
- Use the **Start Archicad** dialog to browse for an archive project

By default, when you open an archive project, this dialog appears:



Choose one of the three options:

- **Read elements directly from archive (DEFAULT):** Library parts will not be extracted from the archive file. You can place Objects from the Archive file, but you cannot edit the Library parts themselves (unlike the elements in an .lcf file). This also applies to list schemes used in calculations. This method saves a lot of disk space, but limits the modification possibilities.
- **Extract elements to a new folder:** The external library parts required for the project will be extracted from the archive and placed in a new folder that you can name. This solution is recommended if you need to actually work on the project objects. (You cannot extract objects to the Embedded Library.)

- **Select a library:** Instead of extracting the library parts saved with the project, you can indicate a folder to store the required elements. This solution is recommended if you wish to update the Project's objects with their latest versions.
 - To *skip* the Open Archive dialog, check **Read Elements Directly from Archive** from the Settings pop-up of the Start Archicad dialog. As a result, when you open an archive project from the Start Archicad dialog, the library parts will be automatically read during project start-up, with no user input required.

To view information about possible Library-related issues, check the Libraries item in the Action Center (**File > Info > Action Center**)

Related Topics

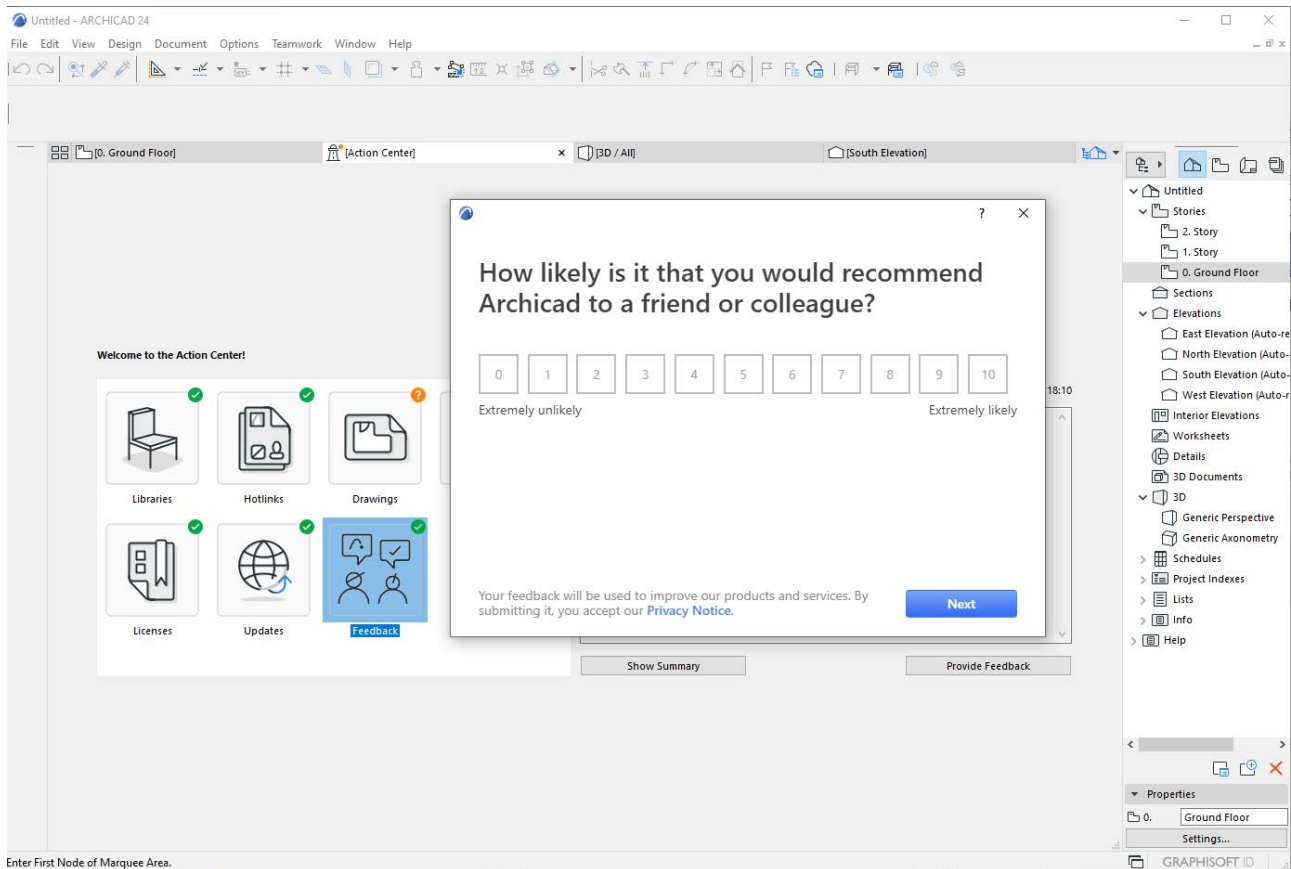
[Opening Legacy or Archive Projects: Manage Library Parts](#)
[Action Center](#)

Archicad User Feedback - Net Promoter Score Survey

We have introduced a new survey feature, so we can make Archicad work even better for you.

We know your time is precious: our surveys are short and take just a minute or two.

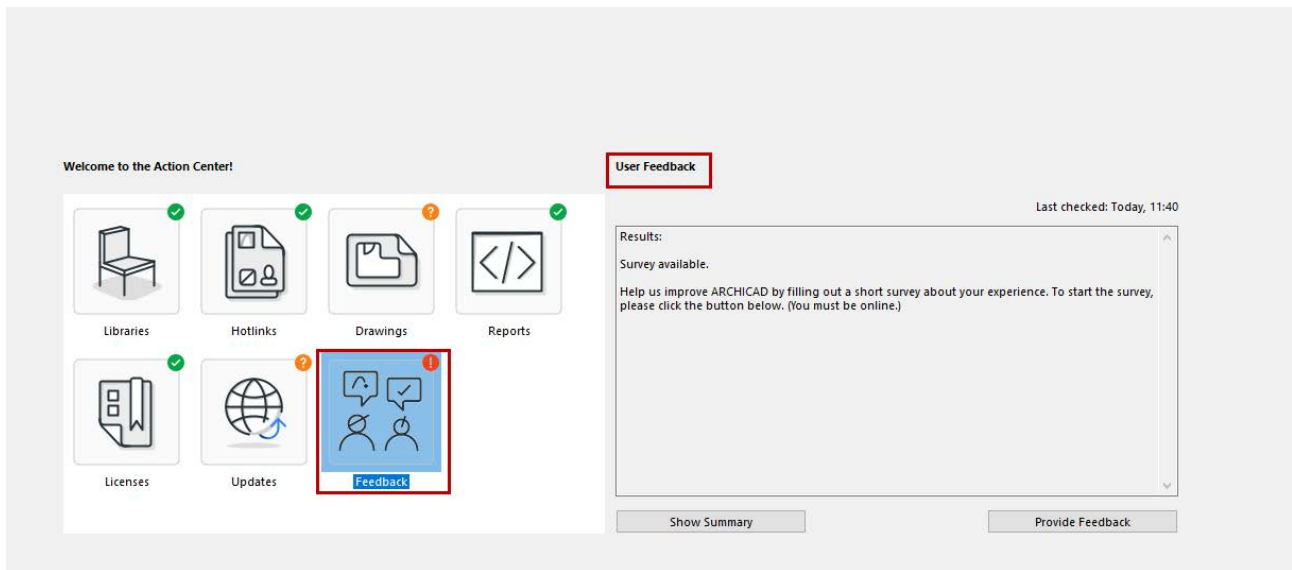
The first survey is available now: the “Net Promoter Score” is a key metric that helps us to gauge your overall experience with Archicad.



Start Survey from Action Center

To start the survey, go to the Action Center. (File > Info > Action Center)

The new “Feedback panel” indicates that a new survey is available. Just click to open and submit the survey directly from the Action Center.



Check for New Survey

At any time: check to see if a survey is available by clicking Check All Items in the Summary tab of the Action Center. (Show the Summary tab, then Check All Items)

Your feedback helps us to improve our products and meet your needs. From time to time, you will receive a friendly reminder about available surveys when you close your Archicad project. You can either respond to the survey right then, or just close up and revisit the survey in the Action Center at a later time.

To read more about how we collect and handle data, see our [Privacy Notice](#).

Related Topics:

[Action Center](#)

Create Support Package (Troubleshooting)

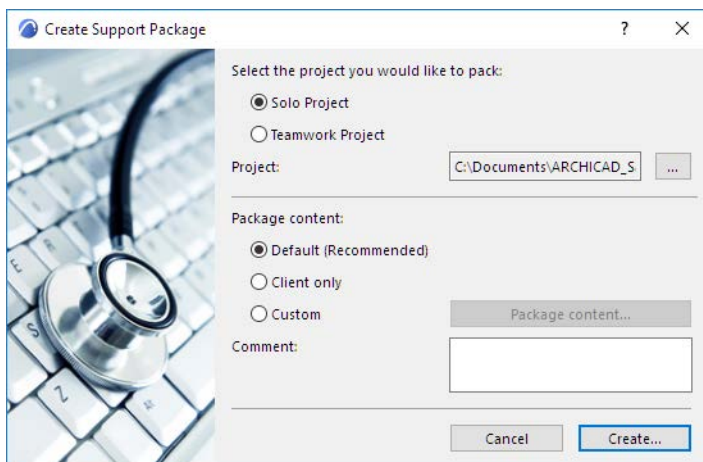
If you encounter an error in a project that requires technical support, the problem can be resolved much more efficiently if the support personnel can view the problematic project files. The relevant files might be located on a BIMcloud as well as on client machines, in various folders.

The **Support Package** function is a utility that automatically gathers the necessary data and stores it in a single file (.support for solo projects or .twsupport for Teamwork projects) that you can easily send to Graphisoft for troubleshooting.

In the case of Teamwork projects, certain problems - errors during Send, Receive, or Saving the local copy - will automatically bring up the Support Package interface. (At this point, you can proceed or cancel as you prefer.)

Create Support Package

1. From the Archicad Help menu, select **Create Support Package**.
2. You are prompted to close any open projects. (You will also be prompted to Save or Send and Receive changes, where applicable.)
3. The Create Support Package dialog box appears:



Choose either Solo Project or Teamwork Project.

Your most recently closed project is shown in the Project field. To choose a different project, click the Browse (...) button.

For a Teamwork project: You must select a BIMcloud, then select the Teamwork Project whose files you want to send.

4. Select the package content to be sent.

Default

Default content is recommended. For a solo project, the Default content includes log files and crash info files. For a Teamwork project, the Default content includes log files, crash info files, and your local copy of the project.

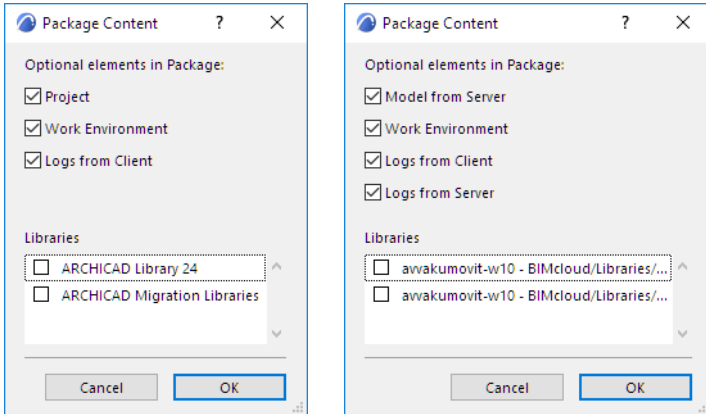
Client Only

The Client Only option is for Teamwork projects: it excludes the logs and Model files from the Server, and sends the Model and Log files only from the Client.

Custom

To include other items as well, choose **Custom**, then Package Content. In the appearing dialog box, check the boxes of the items to include. (The Default content is included automatically; here, choose any additional items to include in the package.)

The optional content differs for solo and for Teamwork projects:



5. Enter any comments that could be helpful to support staff. For example, describe what happened; list the last few actions you took.
6. Click OK to return to the Create Support Package dialog box.
7. Click **Create**.
8. Select a destination folder for the Support Package file.
9. Click OK to start the file creation process.
10. Send the file to your local Graphisoft Technical Support.

Opening Projects through a Network

If the file that you wish to open is already in use by someone else on the local network, Archicad will warn you about this and let you know the name of the user, that is, the name defined in the Sharing Setup Control Panel (Mac) or as the User Name (Windows).

You have the following choices:

- Open the file as read-only
- Open the file with exclusive access
- Cancel opening the file

If you choose to **open the file as read-only**, you can see and modify the whole file, but you can only save it under a different name or into another location. If you try to overwrite the original file, you will again be notified that it is in use and that you cannot replace it with your modified Project.

You can also choose to **open the file with exclusive access**. However, you should be very careful about using this option.

There are a number of reasons why Archicad may state that a file is in use although it actually is not:

- The person who last used the file did not properly open and close the file in Archicad, e.g., because of a system crash.
- You have made a copy of a file that was in use at the moment, and the copy of the file also includes the name of the person who was using it during the copying operation.

Note: If you do open a file with full access while another person is using it, you will both overwrite each other's work without getting any further warning message from Archicad. Therefore, be extremely careful when opening a file with exclusive access in any situation other than the emergencies mentioned above.

Add-ons and Goodies

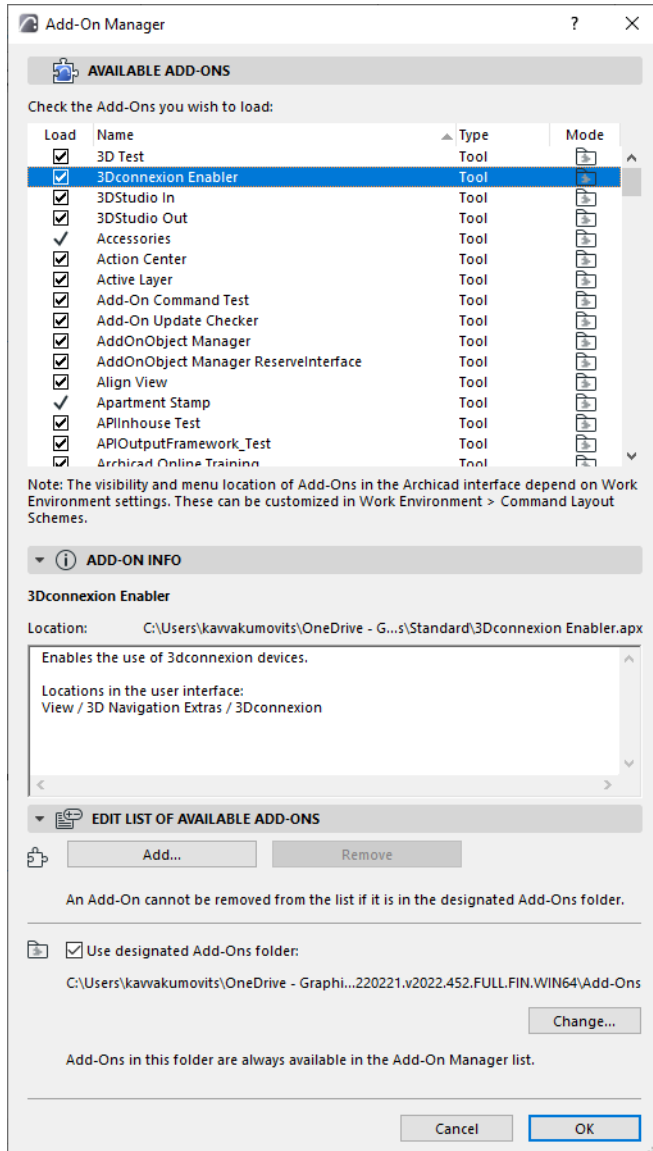
Add-ons are small applications that extend Archicad's core functionality. Many add-ons are integrated as Archicad menu commands.

Goodies are other Add-ons that are not integrated into Archicad's default interface.

[See Goodies, below.](#)

Managing Add-ons

Open the Add-on Manager using **Options > Add-on Manager** command.



Use the Add-on Manager to:

- Load Add-ons from any location. They will be loaded after the next startup of Archicad
- Permanently remove Add-ons while Archicad is running
- Control which Add-ons should load automatically when Archicad is started

- Show information about the loaded Add-ons

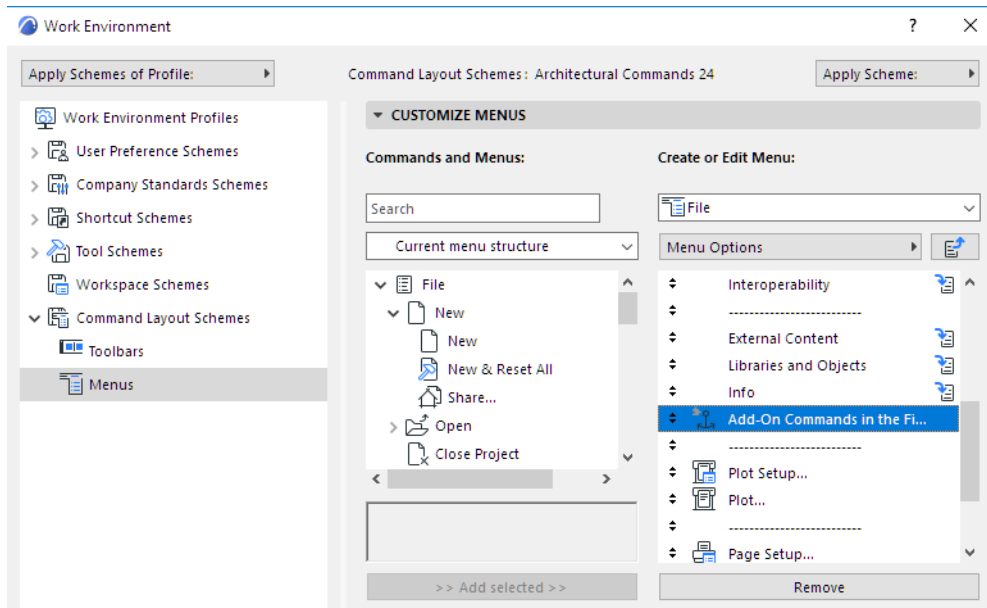
See [Add-on Manager for more details](#).

Menu Location of Add-on Functions

The menu/toolbar location of add-ons is defined in the settings dialogs of **Options > Work Environment > Menus/Toolbars**.

In the Customize Menus (Toolbars) page, add an “Add-on Anchor Point” item to any menu (e.g. File menu) or toolbar. The respective Add-ons will be displayed at this customized menu/toolbar position.

If the Add-on Anchor Point is not part of your menu structure, then the Add-ons will not be displayed in the menu.



Goodies

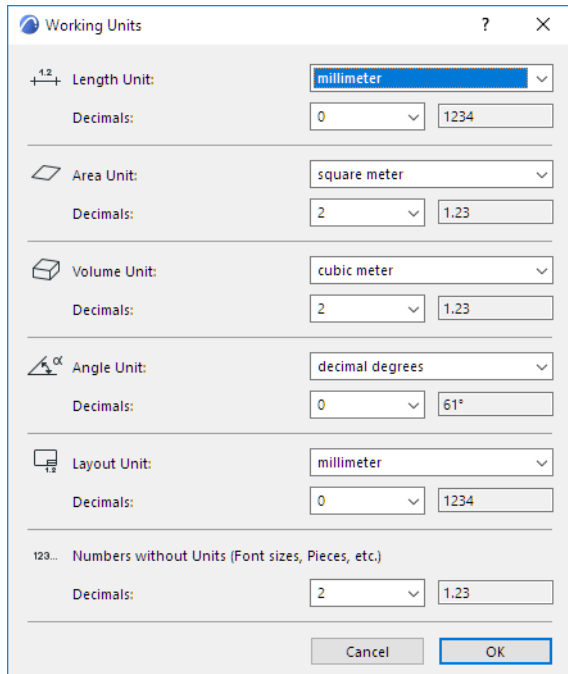
“Goodies” are another source of Archicad Add-ons. They are not integrated into Archicad’s default interface. To access them, do the following:

1. Go to the Archicad **Help** menu.
2. Choose **Online Resources > Archicad Downloads**.
3. From the Downloads page, scroll down to Add-ons, and click “Add-ons and Goodies.”
4. Select your Language version to access available Goodies.
5. Click on the Goodies installer to install all Goodies from a single file.

Working Units

Use the dialog box at **Options > Project Preferences > Working Units**.

Generally, working units are displayed in dialog boxes and during input (e.g. in the Tracker or Control Box).



Length Unit

Select from standard Length Measurement Units.

- If the model is saved in BIMx, this model unit will be used by default when the BIMx model is opened in the application.

Notes:

- If you change the Working Units (for example from metric to imperial), some of the other default values such as grid spacing or default element sizes may need adjusting for meaningful, round values.
- The Length Unit entered here does not apply to Dimensioning functions. To set Dimension units, use **Options > Project Preferences > Dimensions**.

Area Unit

Select from standard Area Measurement Units.

Volume Unit

Select from standard Volume Measurement Units.

Angle Unit

Select from standard Angle Measurement Units.

Note: Regardless of the angle unit set here, the angle measurements of core data in dialog boxes - such as Slant angle - are always displayed in Decimal Degrees. The number of decimal places for these angles is defined by the “Numbers without Units” preference below.

Notes on Surveyor’s Unit:

- The Surveyor's Unit preference set here is reflected during input in the Control Box, Tracker and Measure Tool.
- The initial Surveyor's Unit angle is measured from the North direction (set in the **Options > Project Preferences > Location Settings**).

See [Location Settings](#).

Layout Unit

Select from standard Length Measurements units for all editing functions associated with the Layout Book (e.g. Layout size, 2D drawing elements added to Layouts, Title size, etc.).

Decimals/Fractions: Set the number of decimals or size of fractions for the chosen Length Measurement Unit.

Numbers without Units (Font sizes, Pieces, etc.)

Set the number of decimals displayed for Text Height and Element Property values without units. Also used for the display of core angle measurements (e.g. Slant angle)

Note: Text height is measured in mm if the Length Unit is metric and in points if it is feet and inches.

Working Units in Teamwork

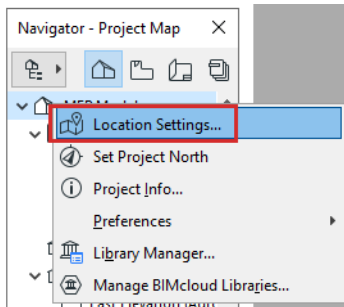
If you open a Teamwork project from the server, its Working Units will have default values based on the template used to create the Teamwork project. However, you are free to change these Working Units while you work on the Teamwork project. Your Working Units preferences specific to each user, and are not saved to the server when you Send or Receive changes.

See also [Structural Analytical Model Units](#).

Location Settings

Locations Settings is available:

- from **Options > Project Preferences > Location Settings**
- from the context menu of the Project Name in the Navigator



Location Settings has three panels:

Project Location

Survey Point Panel

Project North

Project Location

Project Name and **Site Full Address** are taken from the Project Info dialog box.

Click **Edit** to modify these data by opening the Project Info dialog box.

See [Project Info](#).

Enter the **Latitude** and **Longitude** of the Project Location.

- For maximum accuracy (e.g. for Energy Evaluation purposes), you should identify and fill in precise latitude/longitude data for the location of your project.
- Click the pop-ups to switch between Latitude North and South, and Longitude East and West.
- Choose your preferred unit of measure from the fly-out menu.

Time Zone

Choose a time zone at the Project Location, expressed as a positive or negative offset from UTC (Coordinated Universal Time), together with the name of the corresponding time zone.

Altitude

Enter the altitude of the project location in the desired units.

- Altitude is used as the Sea Level Reference Level for your project, at Options > Project Preferences > Reference Levels.
- Altitude data is used when you export the project to SketchUp and Google Earth formats.

[See SketchUp \(.skp\) and Google Earth \(.kmz\).](#)

Use Google Maps to View and Query Location Data

Provided you have an Internet connection: click on **Show in Google Maps** to view the project location as defined by your latitude/longitude data.

To query precise data using Google Maps, zoom in to the desired location. Right-click to bring up the context menu, then choose “What’s Here?” to view (and copy, as needed) latitude/longitude data.



Import/Export Location Settings (XML, SketchUp, Google Earth)

Use the **Import** and **Export** buttons to transfer Location Settings data in XML, Google Earth (.kmz) or SketchUp (.skp) format.

All three formats can be used to transfer information between Archicad projects. The Google Earth and SketchUp formats also allow users to align their Archicad models with SketchUp and Google Earth models.

- The **XML** format stores and reads longitude & latitude, angle display format, time zone, altitude & altitude unit, north direction and display format.
- The **Google Earth** (.kmz) format stores longitude, latitude & altitude as a single place mark. The import function will also read longitude and latitude data from .kmz files containing a single place mark. The altitude will also be imported if it is expressed in absolute mode (not relative or clamped).
- The **SketchUp** (.skp) format exports and imports longitude, latitude, altitude, North direction and time zone. The Site Address:-city and Site Address:-country data are exported but NOT imported).

For further information on SketchUp:

- import: [see SketchUp \(.skp\)](#)
- merge: [see Merge SketchUp \(.skp\) File](#)
- export: [see SketchUp \(.skp\)](#)

For further information on Google Earth:

- import: [see Google Earth \(.kmz\)](#)
- merge: [see Merge Google Earth \(.kmz\) File](#)
- export: [see Google Earth \(.kmz\)](#)

Survey Point Panel

- Choose a Symbol Type for the project Survey Point.

- Set Position and Georeferencing data.

See [Survey Point](#).

Project North

Define Project North (Location Settings dialog)

Enter an Azimuth (compass) angle value here, or drag the compass pointer.

From the pop-up, choose the display unit for Project North.



Note: Unit accuracy depends on the Angle Unit accuracy in **Options > Working Units**.

Set Project North (Graphically)

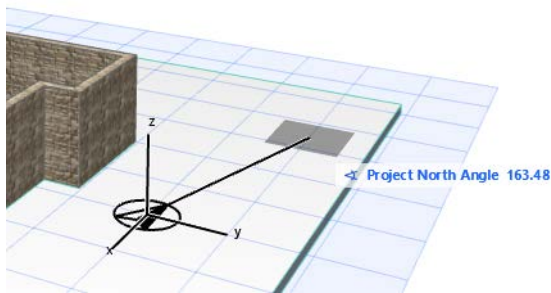
The Set Project North command is available from

- **Options > Project Preferences > Set Project North**
- the Standard toolbar (Survey Point menu)
- from the context menu of the Project Name in the Navigator

In Floor Plan or the 3D window:

1. Click to display the Project North icon.
2. Move the pointer to set Project North.

The Survey Point's North Angle coordinate is adjusted accordingly.



The Project North value affects the following:

- The calculations of the Energy Evaluation function.
- Some Interior Elevation Markers include text which display the Project North value.
- The North Symbol object (in the Archicad Library), when placed in the project, reflects the Project North value. (You must check the "Follow Project North" box in the object's Settings.)
- When using surveyor's units (**Options > Project Preferences > Working Units**), the North of the Coordinates Palette and the Tracker will be the same as the one set here.
- The Sun azimuth will be calculated with respect to Project North (where East is at 0.00 degrees).

Sun position is set in 3D Projection Settings. See [Sun Position](#).

Survey Point

Archicad's Survey Point can be used as common reference point to facilitate coordination of models from different programs.

In some countries, use of a particular reference point is a standard requirement when defining the project.

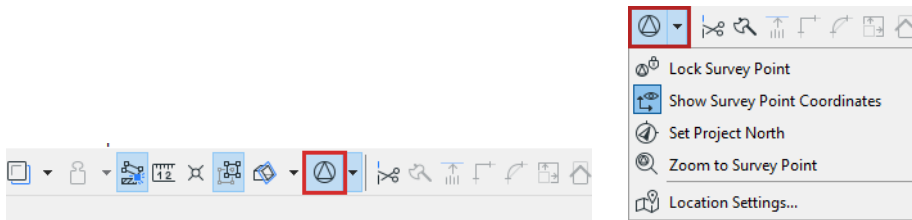
Survey Point can also be used

- to align the XREF origin (in the Attach Xref Options dialog)
- to align a Hotlink Module (in the Place Hotlink dialog)
- to align the IFC Site location of an exported IFC model ([see Define IFC model position by:](#))

The Survey Point is also integrated into Archicad's workflows with BCF, DXF, DWG, and IFC, which also use the project's native Survey Point data.

In Teamwork: To edit the Survey Point, first reserve Locations Settings.

Survey Point Commands in the Standard Toolbar



Show/Hide Survey Point

Click the toggle to display the Survey Point.

By default, the Survey Point is placed at the project origin.

Click and drag to move it.

Lock Survey Point

Use this to keep from accidentally moving or editing it.

Zoom to Survey Point

Zooms to the Survey Point on the Floor Plan.

Show Survey Point Coordinates

Displays the Survey Point's coordinates in the project, to the right of the Survey Point.

The coordinates use the project's Working Units for length dimensions.

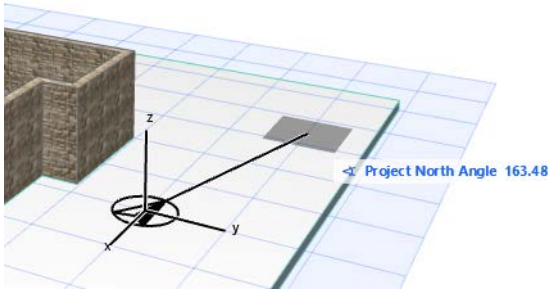
To set position data: see [Survey Point Symbol and Position](#).

Set Project North

In Floor Plan or the 3D window:

1. Click to display the Project North icon.
2. Move the pointer to set Project North.

The Survey Point's North Angle coordinate is adjusted accordingly.



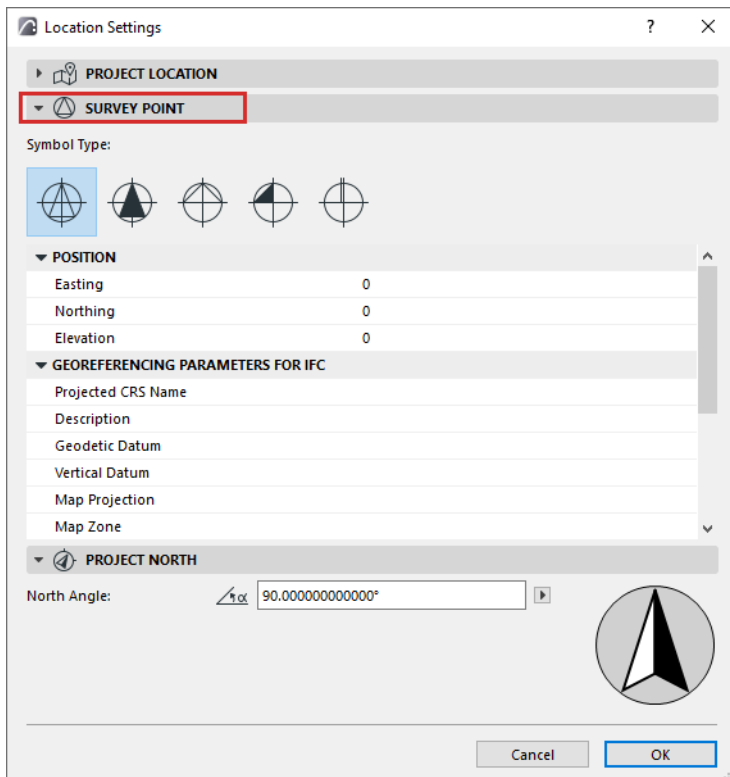
Edit Survey Point Coordinates

Do any of the following:

- Left-click the Survey Point and enter coordinates in the appearing Tracker
- Click and drag the Survey Point
- Rotate the Survey Point (see below)
- Use Location Settings (Options > Project Preferences)

Survey Point Symbol and Position

1. Go to the Survey Point panel of Location Settings
 - from **Options > Project Preferences > Location Settings**
 - from the context menu of the Project Name in the Navigator



2. Choose a Symbol Type for the project Survey Point
3. Set Position data. Easting, Northing and Elevation data are measured relative to the Project Origin.

Georeferencing

These are a set of location data used in IFC data exchange; they are exported as IFC Properties. They do not affect the Survey Point location in Archicad.

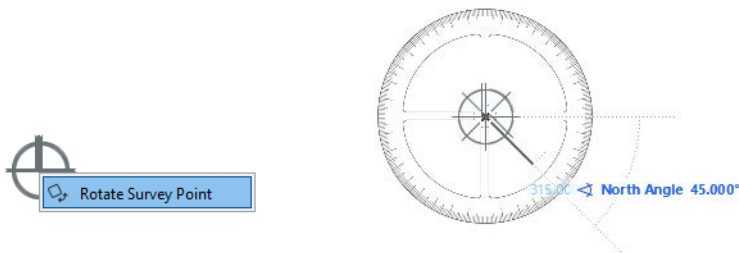
- Export via IFC 2x3: The data are exported as two new IFC Property Sets.
- Export via IFC4: New IFC entities are created: IFC Map Conversion, and IFC Projected CRS.

For more information, see [User Guide for Geo-referencing in IFC](#).

Rotate Survey Point

On the Floor Plan:

1. Hover over the Survey Point to preselect it.
2. From the context menu, click Rotate Survey Point, then rotate on screen



You can also use the regular Edit > Rotate command, or adjust Tracker values.

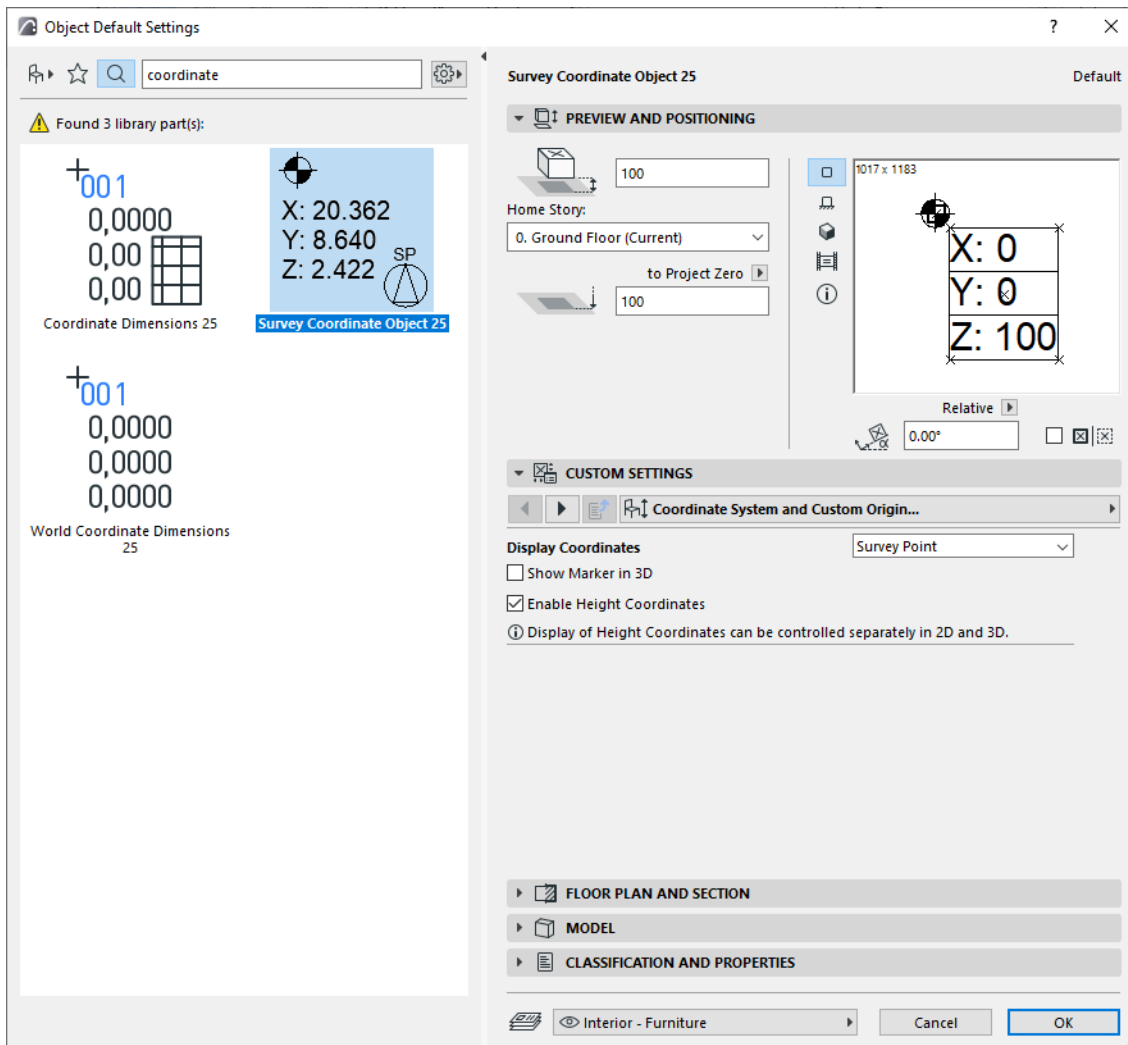
Related Topics:

[Location Settings](#)

[Survey Coordinate Object](#)

Survey Coordinate Object

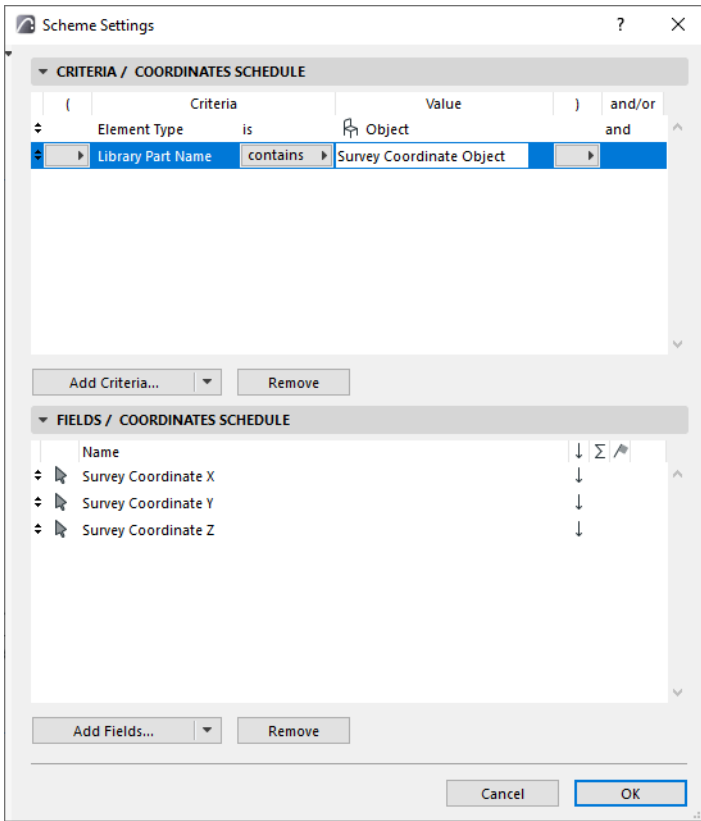
This object is in the standard Archicad Library.



It automatically displays the XYZ values based on the Survey Point position.

If the Survey Point moves, the Survey Coordinate Object values will be updated automatically.

For each placed Survey Coordinate Object, you can display its values in an Element Schedule:



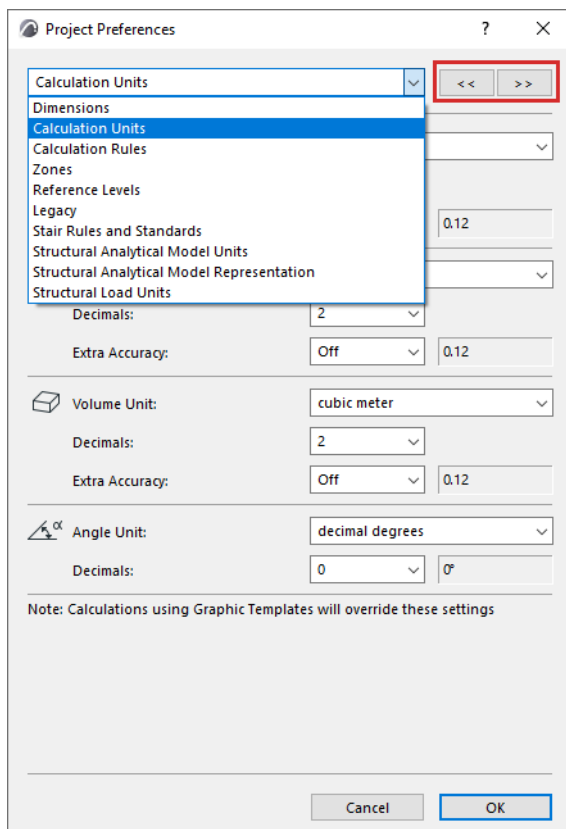
Project Preferences

The **Options > Project Preferences** menu contains the Project Preferences tab pages where you set standards and working methods specific to the project you are working on, and which are saved along with the project.

If another user opens the Project on their own computer, the same settings will be applied.

In Teamwork, you must have access rights and must reserve the Project Preference dialog box in order to change these preferences.

The **Project Preferences** dialog box contains a pop-up menu at top left listing each preference category (which you can also access directly from **Options > Project Preferences**). Click **Next** and **Previous** to navigate among screens.



For details on each of these controls, see:

[Dimensions Preferences](#)

[Calculation Units Preferences](#)

[Calculation Rules Preferences](#)

[Zones Preferences](#)

[Reference Levels Preferences](#)

[Legacy Preferences](#)

[Stair Rules and Standards: Project Preferences](#)

[Structural Analytical Model Representation \(Preferences\)](#)

[Structural Analytical Model Units](#)

[Route Preferences](#)

Related Topics:

[Working Units](#)

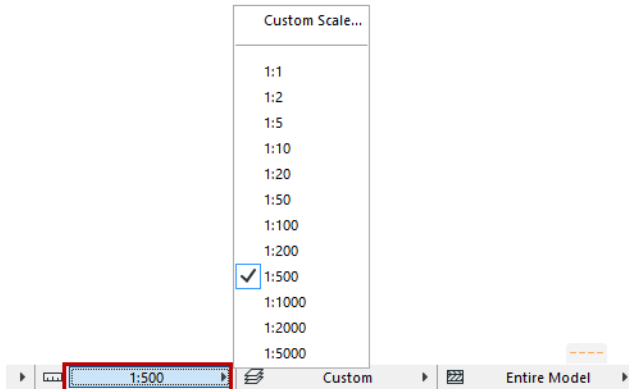
[Location Settings](#)

[Working Units](#)

Scale

To set the scale of the currently active model window, do one of the following:

- Use the Scale pop-up from the Quick Options Bar at the bottom of the window

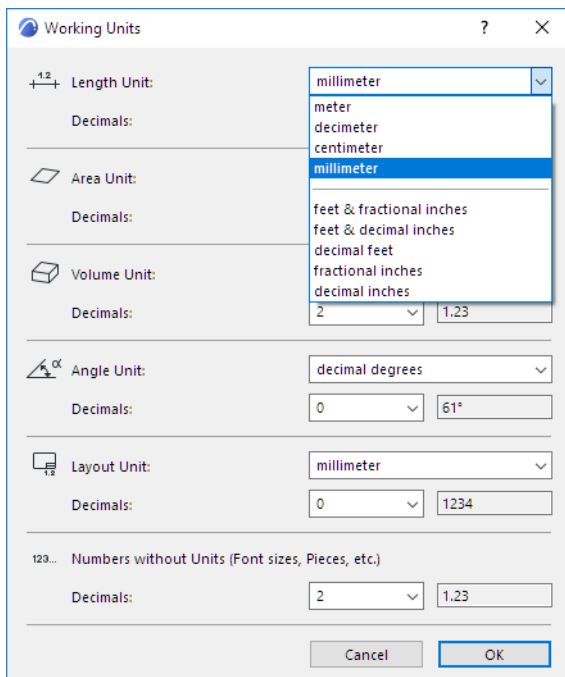


- Use **Document > Floor Plan Scale** (the command varies depending on the window)

Select a standard scale from the pop-up list, or choose **Custom Scale** and type one in.

Scales are saved as part of View Settings.

Scales are displayed using metric or US standards, depending on the Length Unit setting defined at **Options > Project Preferences > Working Units**.



To make a zoomed view match the view at the currently set scale, choose **View > Zoom > Actual Size** after setting the scale.

Actual Size is the equivalent of the 100% scale value.

The 2D Symbol of some GDL Objects can be set to be sensitive to the current scale: the symbol varies depending on the current scale.

Model Size vs. Paper Size Elements

Model Size elements are rescaled along with the model whenever you change the project scale. Model size elements include all construction elements such as walls, objects, slabs etc.

Paper Size elements are printed or displayed on the screen at the size you specify, regardless of the scale selected for the Project. For elements that do not have any real size, such as dimensions and arrowheads, you can specify a fixed size defined in either points or millimeters.

Either Model Size or Paper Size: The following elements may be either model size (scaled with the plan) or paper size (scale independent):

- Text Blocks created with the Text tool: Choose Model or Paper size.

[See *Text Tool Settings*.](#)

- Dashed and symbol lines: Choose Model or Paper size.

[See *Lines*.](#)

- Vectorial, symbol and image fills: Choose Model or Paper size.

[See *Create Image Fill*.](#)

Set a Separate Printing Scale for 2D Document

The Scale options in the Print 2D Document and Plot 2D Document dialog boxes (File menu) allow you to specify a Custom printing scale each time you print or plot your work from a Floor Plan or other 2D window.

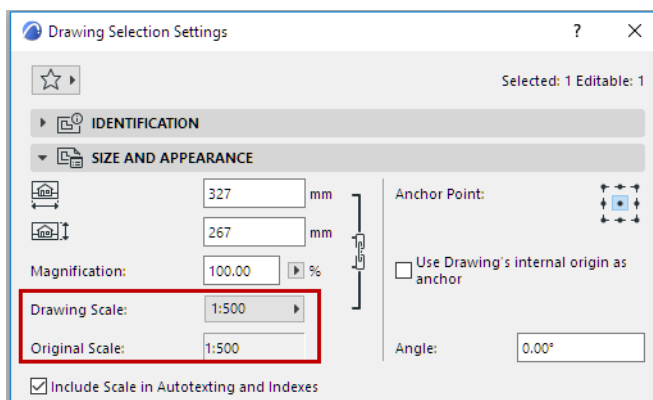
The **Text and Markers** options enable you to reduce/enlarge text and markers as the printing scale is changed, or keep them at a fixed (paper) size.

[See *Print 2D Document and Plot 2D Document*.](#)

Drawing Scale

Drawings based on an Archicad view have a **Drawing Scale**. By default, this Drawing Scale is the same as the **Original Scale** (the scale of the Drawing's source view), but you can customize the Drawing Scale in Drawing Settings.

[See *Drawing Settings*.](#)



Customizing the Drawing scale has no effect on the scale of objects within the drawing; it is equivalent to a graphical resizing of the Drawing, like the effect of magnifying a document with a copy machine.

Libraries

Each version of Archicad is shipped with a standard object library containing hundreds of preconfigured, editable objects (also known as GDL objects or Library parts).

When creating a new project, Archicad will load the library contained in the template used for the new project. Each subsequent version of Archicad comes with an updated Archicad Library. There are different Libraries corresponding to different national standards.

You can also add additional libraries to the project as needed.

View and manage your project's libraries in the Library Manager (**File > Libraries and Objects > Library Manager**).

[Library Manager](#)

[Recommendations for Using Libraries](#)

[Embedded Library](#)

[Linked Libraries](#)

[BIMcloud Libraries](#)

[Remove Library from Project](#)

[Troubleshooting Library Issues](#)

[Missing Library Parts](#)

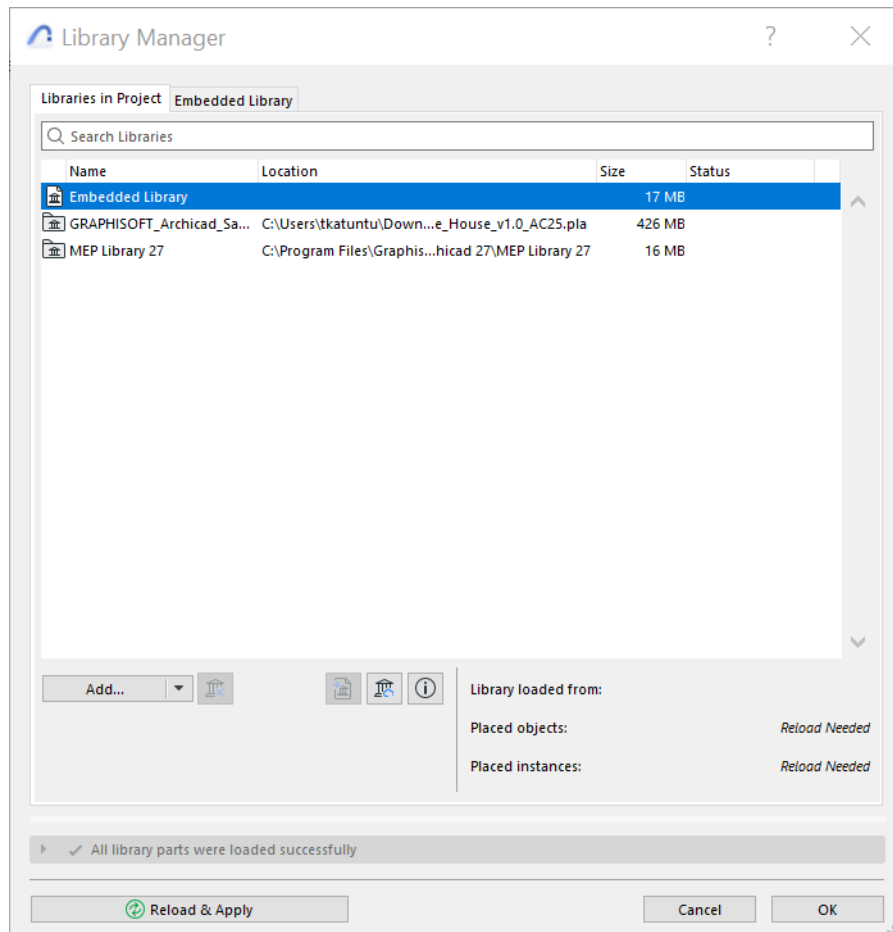
[Library Container File](#)

[Updates for Archicad](#)

Library Manager

Use Library Manager to view and manage project libraries.

Library Manager (**File > Libraries and Objects > Library Manager**) also provides feedback on any missing or outdated Libraries or Library Parts, and provides information about how to resolve these issues.



Topics in this Section

[Open Library Manager](#)

[List of Project Libraries](#)

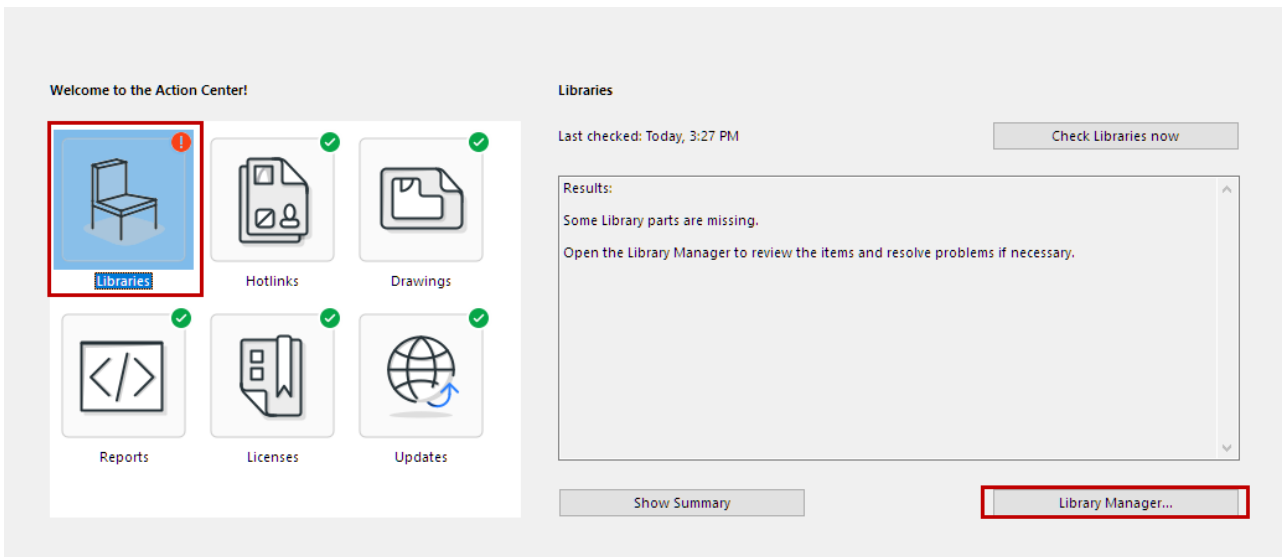
[View Contents of Selected Library](#)

[Access and Manage Library Contents](#)

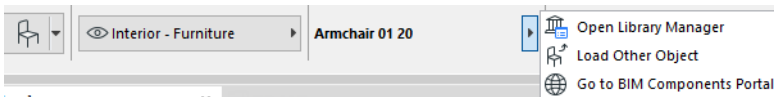
Open Library Manager

Do one of the following:

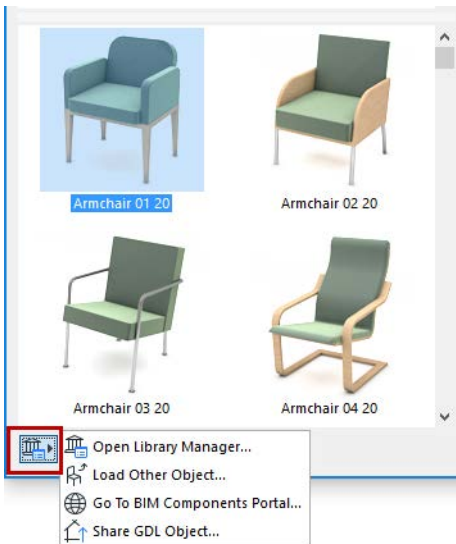
- Use the **File > Libraries and Objects > Library Manager** command
- Right-click any Library Part in its Settings dialog box and use **Show in Library Manager**
- In the Action Center (**File > Info > Action Center**), click on the Libraries item for feedback on Library-related issues, then click **Library Manager**



- Open Library Manager from the Info Box pop-up of any Library Part



- Use the Library Manager icon at the bottom of any Library Part Settings dialog box



- With a library part selected, use **File > Libraries and Objects > Show in Library Manager**

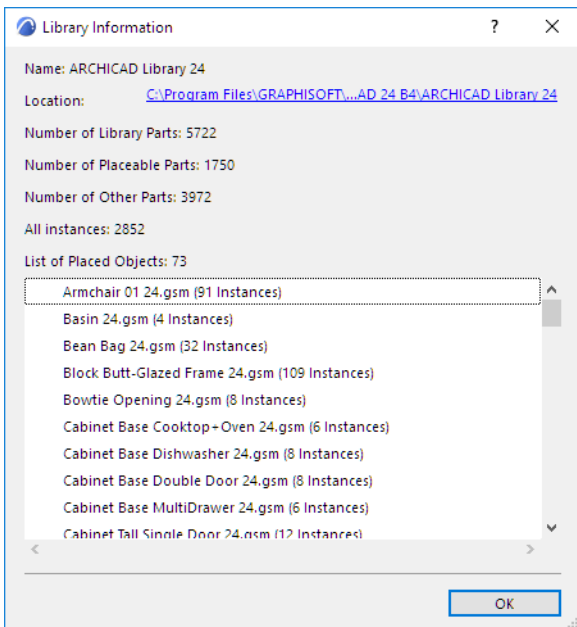
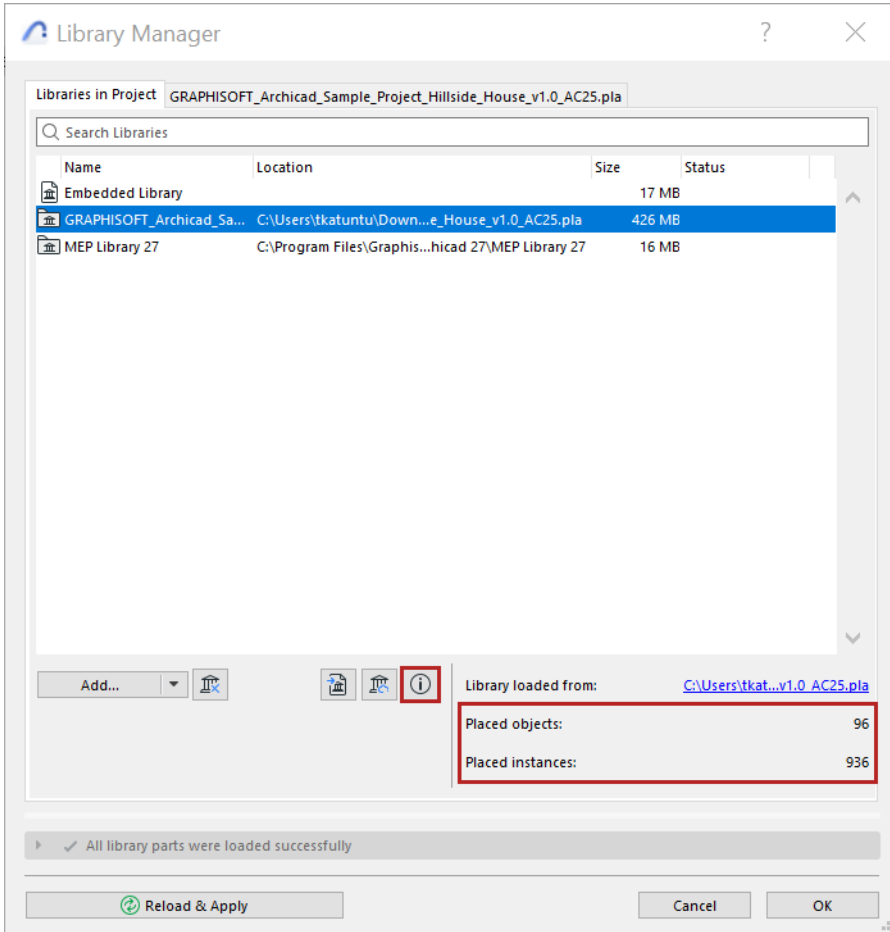
List of Project Libraries

The **Libraries in Project** tab lists each library with its location, size and server status (where relevant).

The icon indicates the type of Library (Linked, Embedded, or BIMcloud Library).

Note for Teamwork users: Reserving the Library List on the **Libraries in Project** tab page enables you to add or delete libraries to/from this list. It does not allow you to modify any library’s contents. (To do so, go to the content tab of a selected library. Select and reserve the specific content you wish to modify.)

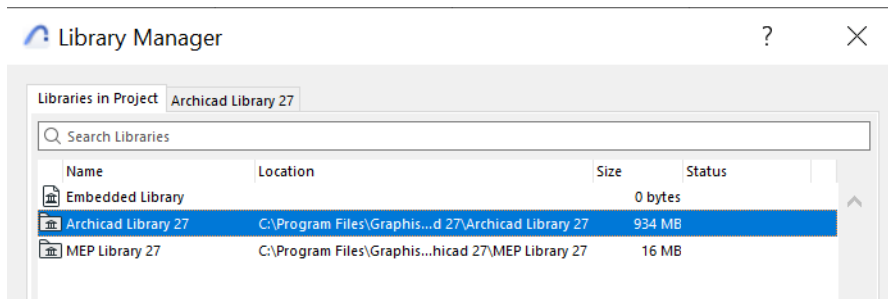
Select any library, and/or the Info icon, to view information about it.



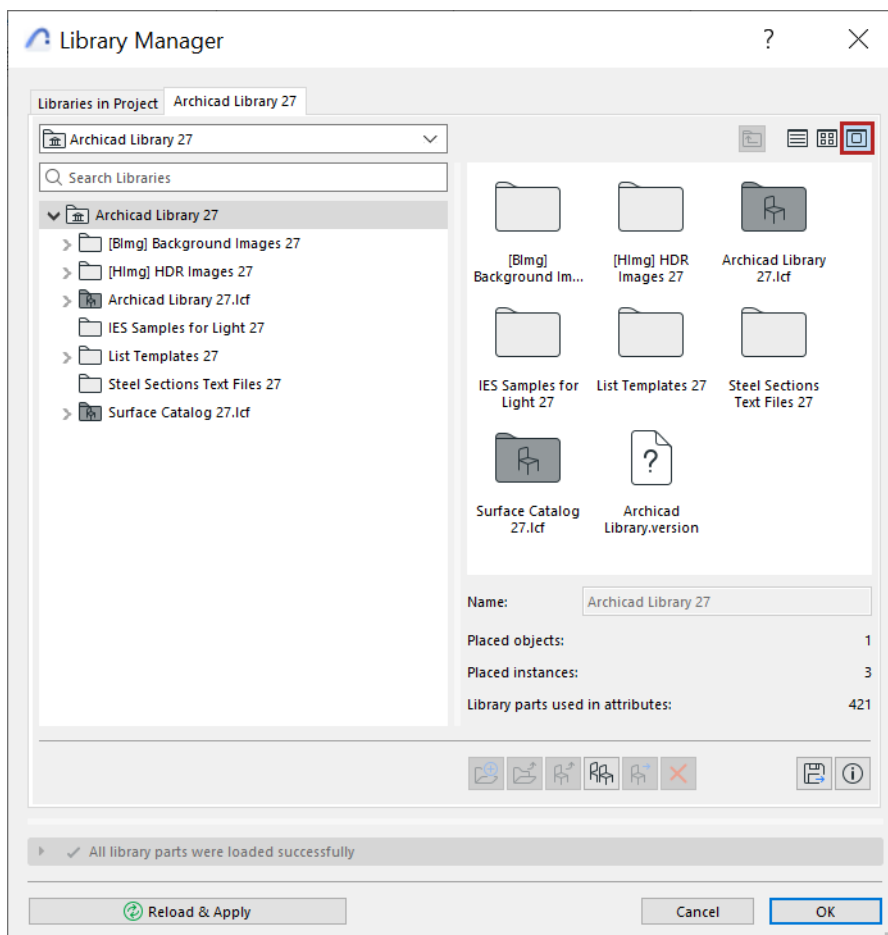
View Contents of Selected Library

To see a library's contents:

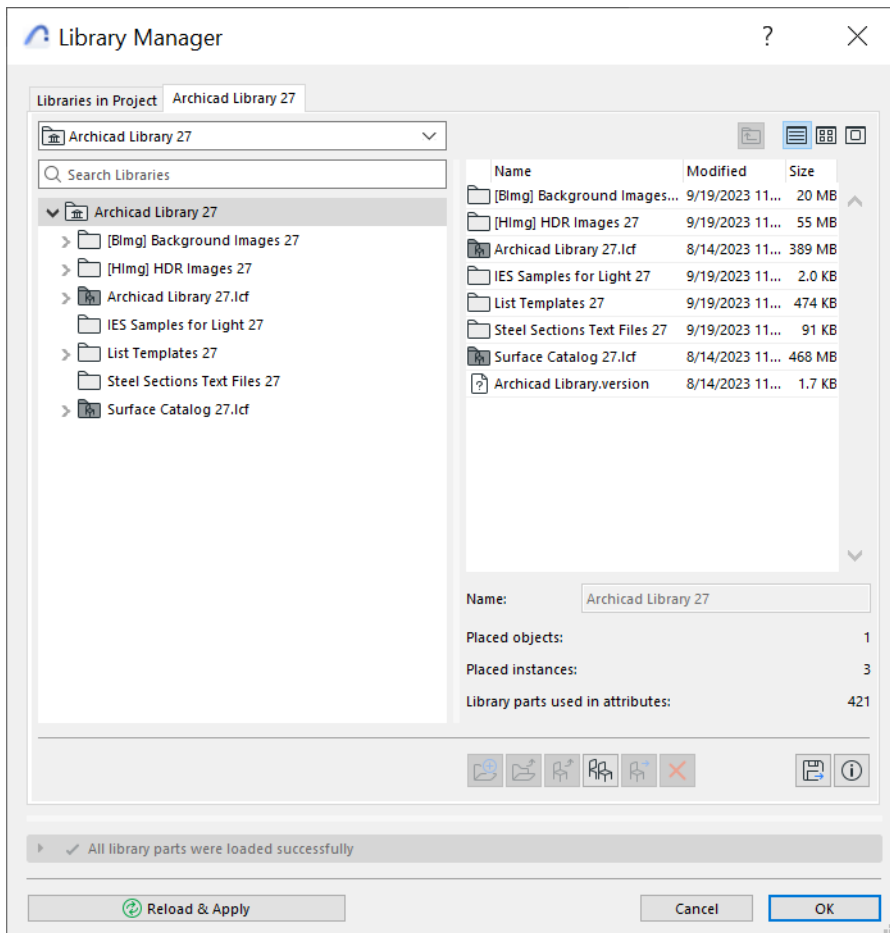
1. Select the name of the library from the list on the **Libraries in Project** tab.



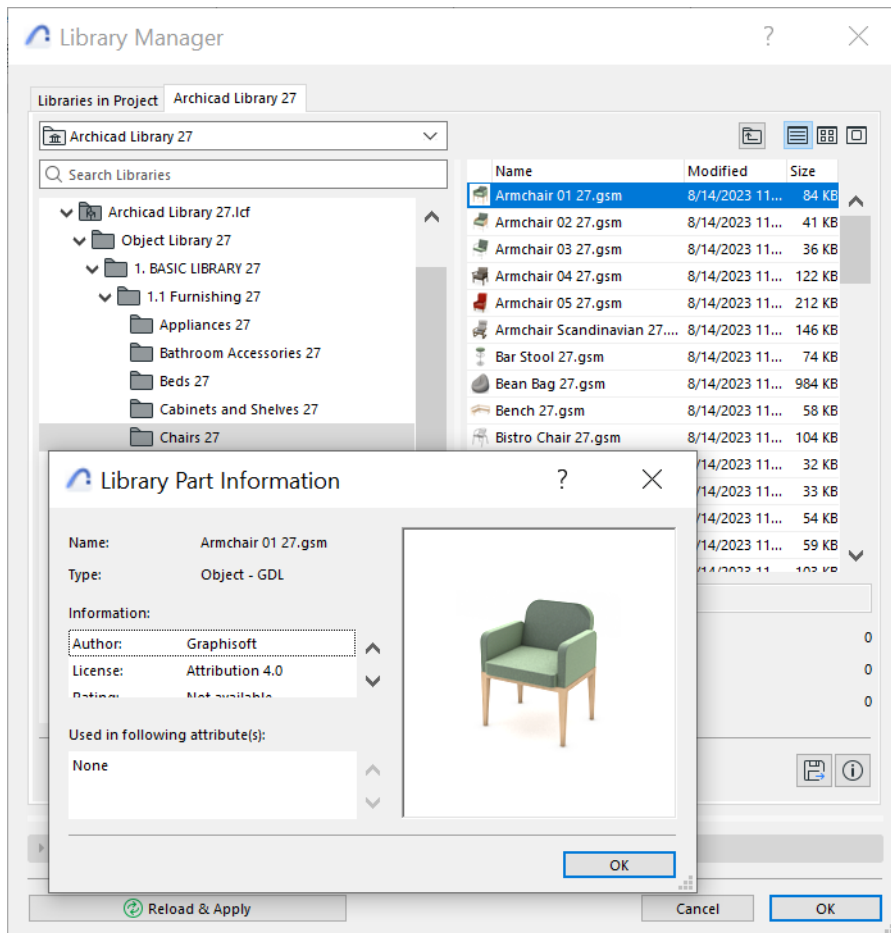
2. Double-click to open the right-side tab (or click on the content tab) to see the selected library's contents, in icon view or list view:



Note: Filenames that consist only of a file extension (such as “.gsm” or “.DS_STORE”) are not visible in Library Manager



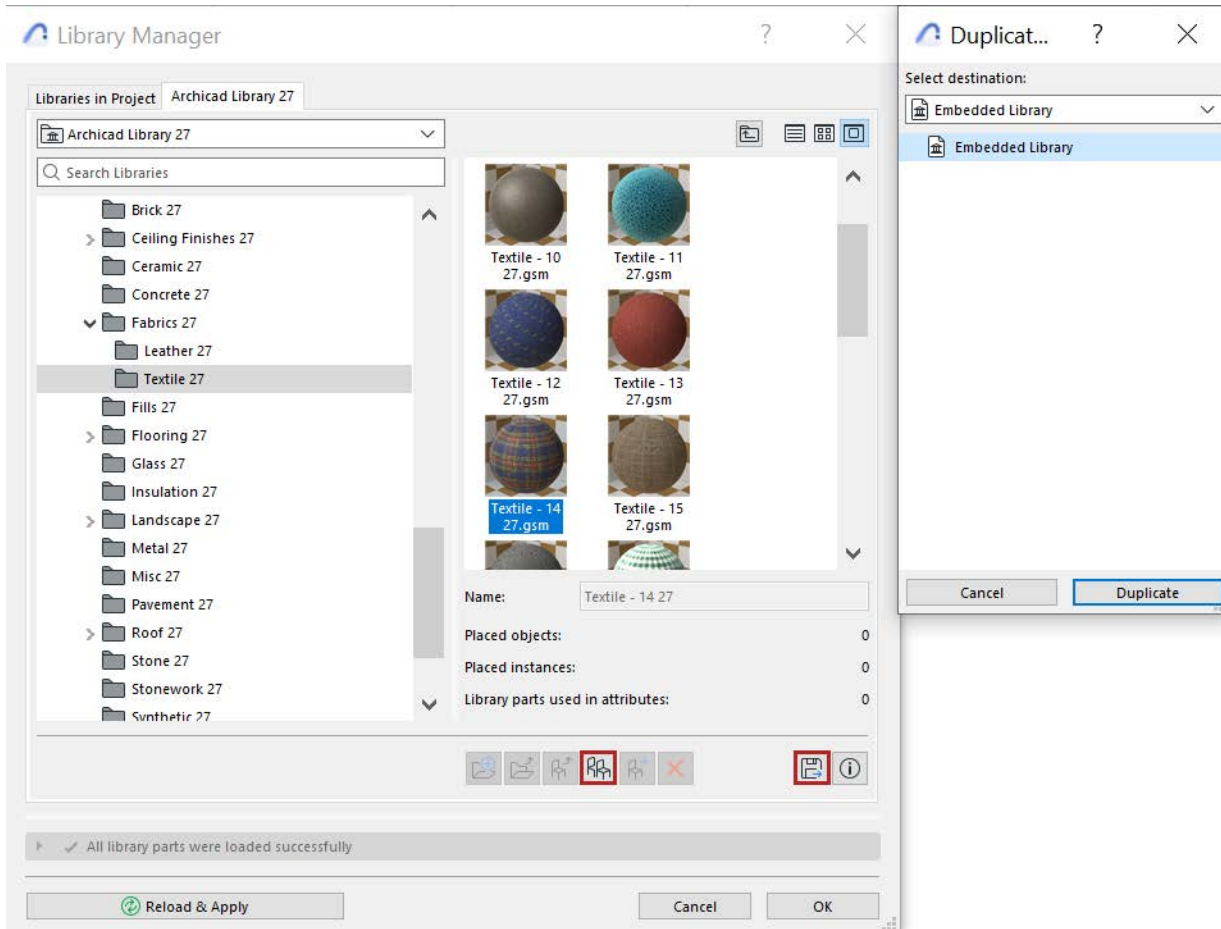
Click **Info** to see a large-sized preview of a library part.



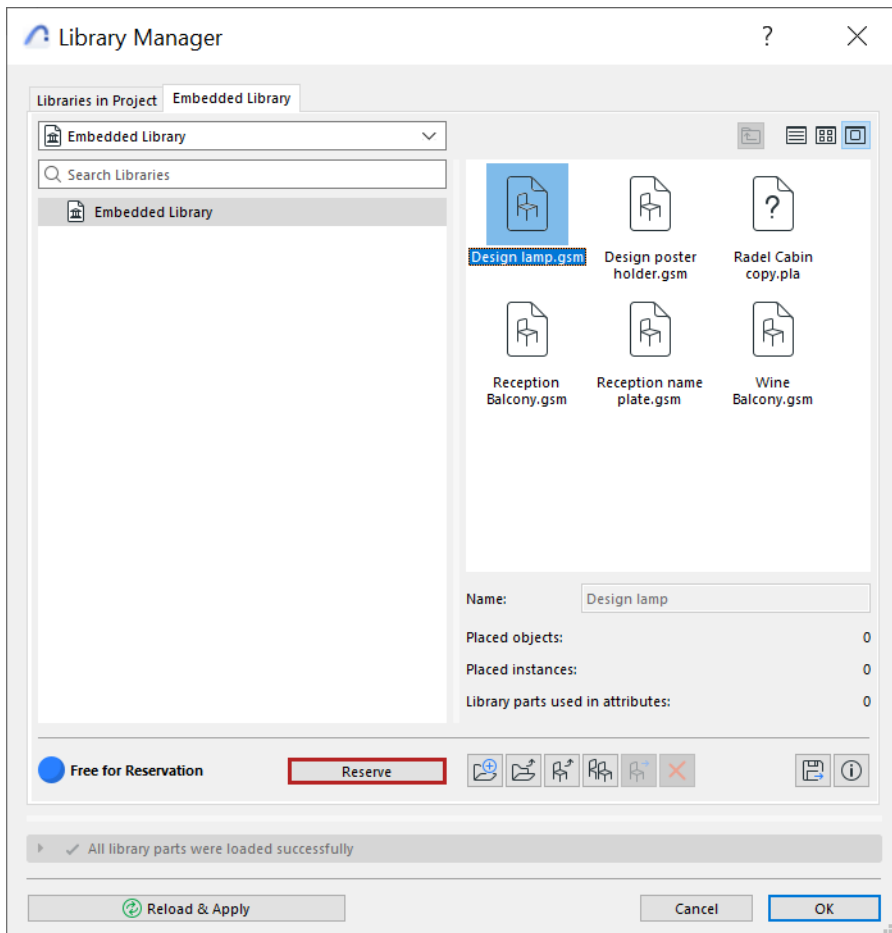
Access and Manage Library Contents

Linked Library folders cannot be edited from the Content tab. All of the management commands, except the Duplicate, Export and Info buttons, are grayed out. Use a file manager interface, outside of Library Manager, to manage these Linked Library folders.

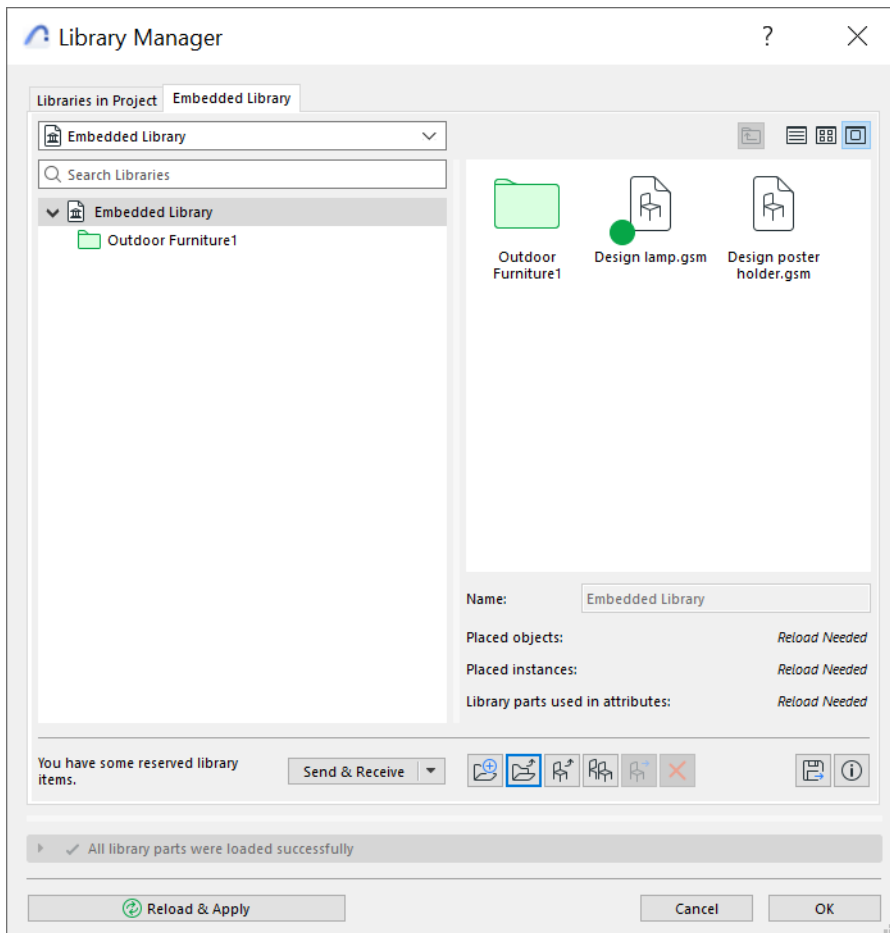
However, you can use **Duplicate** or **Export** to copy a Library Part file (such as a surface) to an accessible folder, then edit it in an external application (e.g. Photoshop). Then use Add to add it back to your library.



Note for Teamwork users: To modify any library item, you must reserve the item in the Content tab of a selected library. Select and reserve the specific content you wish to modify.



- Library Parts reserved by you are shown with a green dot. Newly added library parts or folders that have not yet been sent to the server are displayed in green.



See also [Open and Edit Library Part](#).

For details on each type of item listed in Library Manager, see the links below:

[Embedded Library](#)

[Linked Libraries](#)

[BIMcloud Libraries](#)

[Remove Library from Project](#)

[Recommendations for Using Libraries](#)

[Troubleshooting Library Issues](#)

[Library Container File](#)

Embedded Library

The **Embedded Library** stores custom, project-specific objects in the project itself (rather than in your file system or on the BIMcloud), to ensure that they are always available and editable.

The Embedded Library may contain:

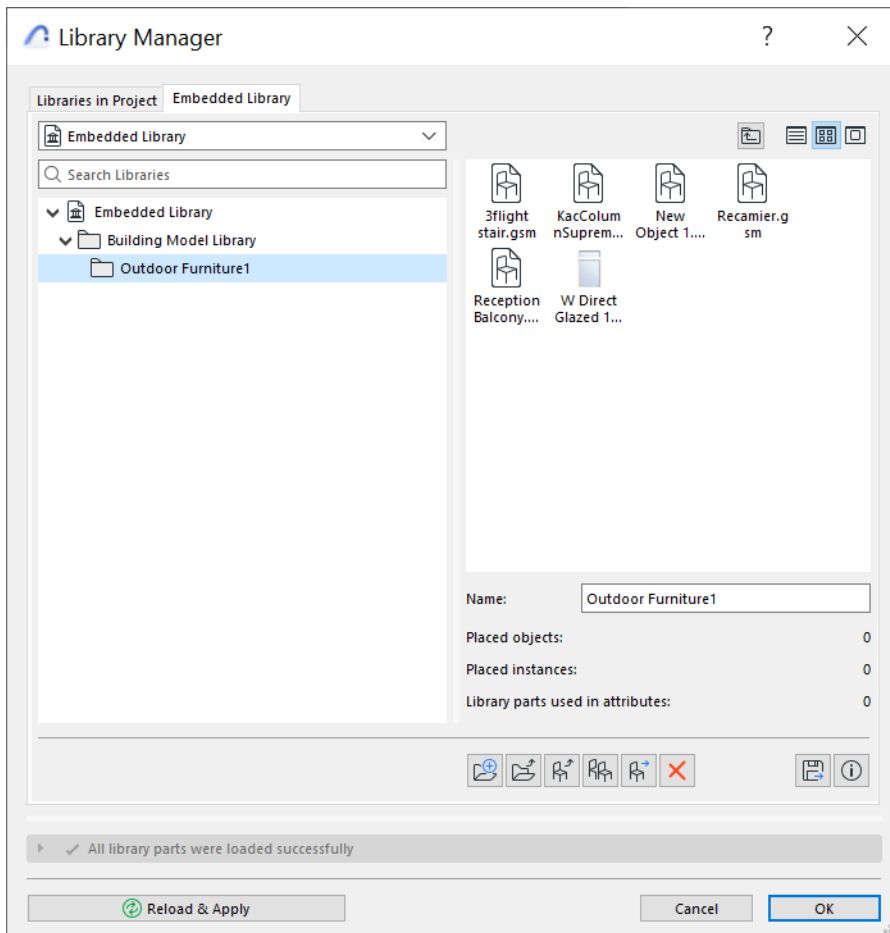
- Object files you create and save yourself (e.g. Patch, Stair, Trusses, Custom Object, Custom Door/Window components)
- Image files loaded for use as Surface textures or other purposes
- Any object file that you browse for, choose and add to this library, including Web Objects downloaded from BIMcomponents.com
- Loaded .txt files that function as macros

Note: List Schemes (from the Calculation function) will function in Teamwork projects only if you save them to the Embedded Library.

Restrictions:

- The Embedded Library cannot contain any .lcf file (Library Container File)
- An archive (.pla) folder used as a Linked Library will not be mounted unless the folder functions as a library root (i.e. at the top of the library hierarchy). Any .pla files at a deeper level of the hierarchy will not be mounted.

Click on any folder listed in the Embedded Library Content tab to see its contents on the right side of the Library Manager.



Embedded objects exist only in that particular project and can be used only in that project. (In contrast, objects of a linked or BIMcloud library can be used in many projects.) If you delete an object from the Embedded Library, it is gone. If you edit the script of an embedded object, that modification will occur right there, in the Embedded Library of that particular project.

For any selected object(s), view its information below (e.g. Name, number of placed instances).

Related Topics:

[Open and Edit Library Part](#)

[Add Objects to Embedded Library](#)

[Manage Embedded Objects](#)

[Embedded Objects in Teamwork](#)

[Recommendations for Using Libraries](#)

Add Objects to Embedded Library

An object can be added to the Embedded Library in several ways. These are described in the following sections:

[Embed Object Using Library Manager](#)

[Embed Placed Objects from a Library](#)

[Add Custom Object to Embedded Library](#)

[Load Object File to Embedded Library](#)

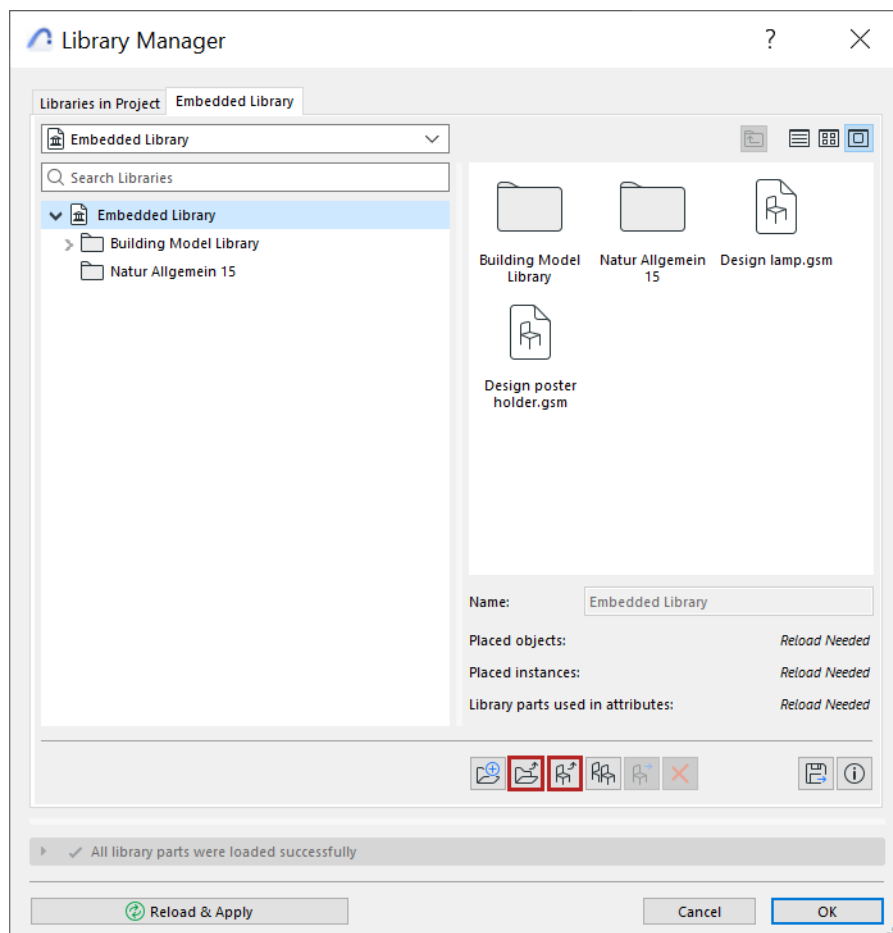
[Add Library of a Hotlinked Module](#)

Embed Object Using Library Manager

Use Library Manager to add any folder or library-type file to your Embedded Library.

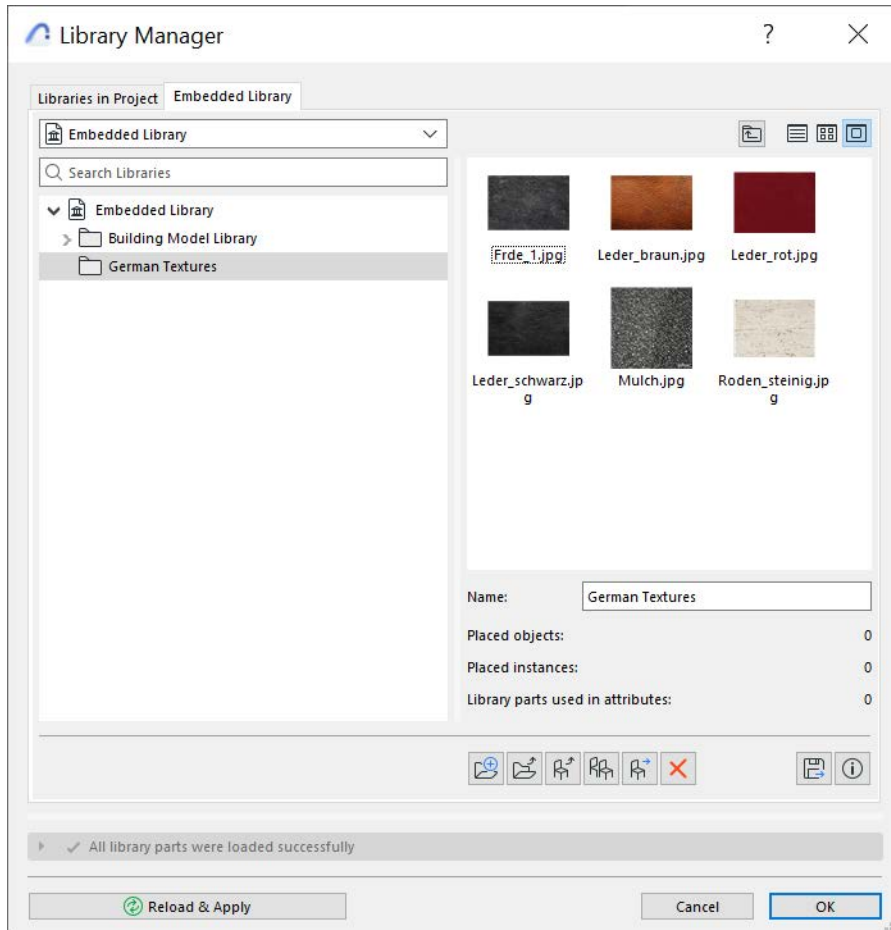
In a Solo Project:

1. Go to Library Manager.
2. Select the **Embedded Library** tab page.
3. Click “Add new file” or “Add new folder” from the command icons.



- Browse for the desired library folder or file, select it, then click **Choose**. (To view and choose items within a folder while browsing the directory dialog box in Windows, you must double-click the folder.)

The chosen library file or folder, with its contents, now appear in the Embedded Library in Library Manager.



Use Drag and Drop

From any location on your computer, drag and drop the file into the Embedded Objects folder of Library Manager.

Embed Placed Objects from a Library

Use this feature to embed all the objects placed from a linked library or BIMcloud Library into the project. This function is useful to ensure that the objects you need will always be available in this project, because they will be embedded.

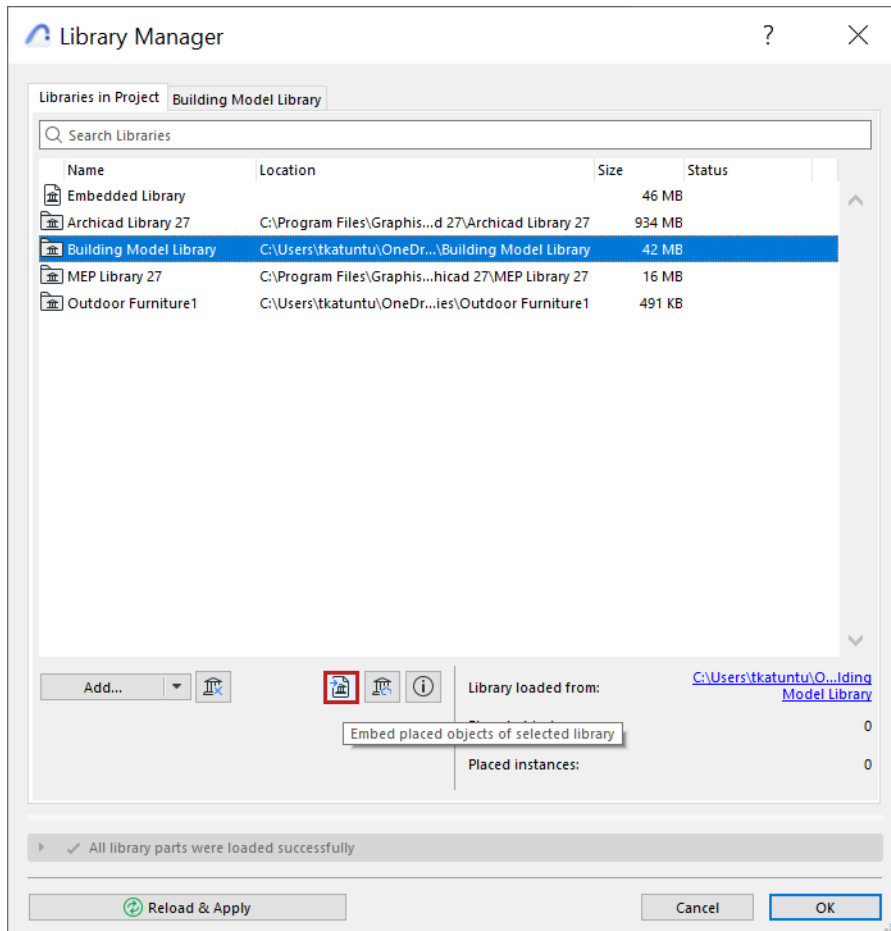
Note: Embedding an object will also automatically load any needed macros along with it.

If you use this command, the original linked library or BIMcloud Library will be *removed* from the project.

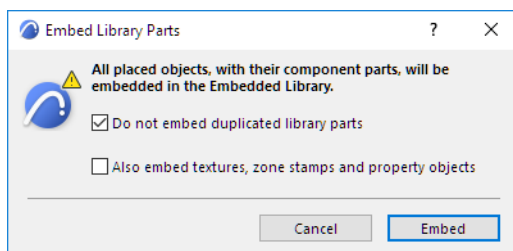
Important: Embedding a library does *not* embed the entire contents of the library; it will only embed those objects which have been placed into the project.

To embed a library's placed objects into your project:

1. Select the library folder from the **Libraries in Project** list in Library Manager.
2. Click the **Embed** icon. (If no objects from the selected library have been placed, you get an alert that there is nothing to embed.)



3. The appearing dialog box gives you two options associated with the embedding process:



- **Do Not Embed Duplicates:** Click this to avoid duplicates when embedding objects. Any objects that already exist in other linked libraries will not be embedded.
- **Also embed surfaces, zone stamps and property objects.** This is usually relevant only if you are trying to embed objects from an Archicad library. Surfaces, zone stamps and property objects are accessory GDL library parts that have been written into certain objects. Check this box to embed these parts as well.
Note: It is usually unnecessary to embed objects from an Archicad library; the **Consolidate** function is a better technique for streamlining your Archicad libraries.

[See Consolidate Multiple Archicad Libraries.](#)

4. Click **Embed**.

Notes for Teamwork Users

Only the objects in your current local copy of the project will be embedded.

It is possible that other users have placed objects from a BIMcloud library that you want to embed. If the other users have not yet sent these changes to the server, those placed objects will *not* be embedded, because they are not available to your local copy of the project. But since the library folder will be removed as part of the embed process, those objects will consequently be shown as “Missing.”

To avoid this situation, ensure that all users have sent in their changes before you embed a BIMcloud Library of a Teamwork Project.

Add Custom Object to Embedded Library

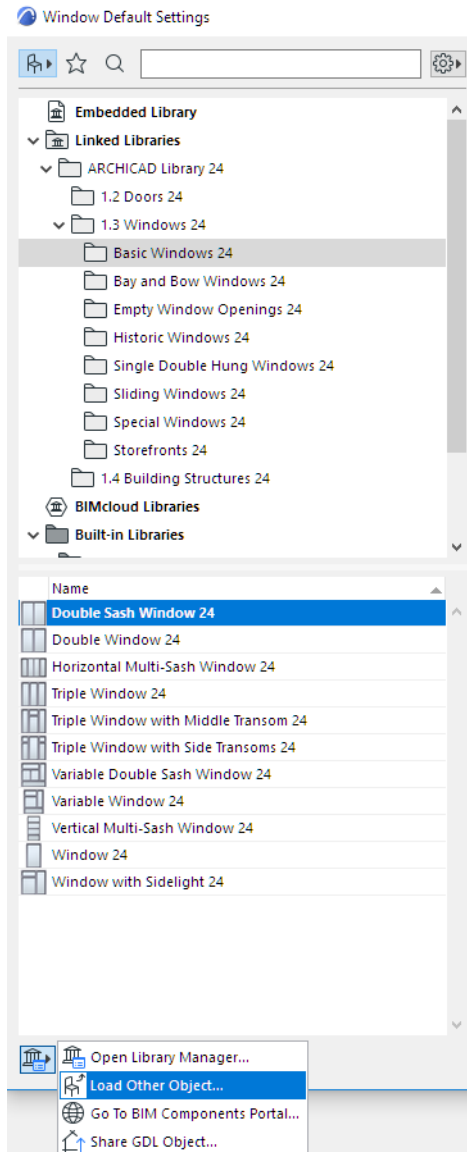
Objects saved using **File > Libraries and Objects > Save Selection as** will save the resulting files as embedded objects by default.

[See *Create Custom Library Parts and Components*](#).

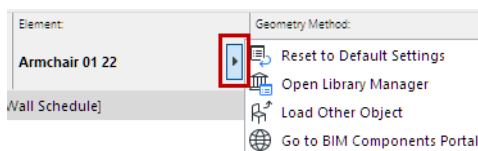
Similarly, objects created by you (e.g. using TrussMaker or RoofMaker) will be added to your project's Embedded library by default.

Load Object File to Embedded Library

Individual objects not part of the project libraries can be used in your projects via the **Load Other Object** commands in the corresponding tool settings dialog boxes. Such objects are automatically stored as Embedded Objects.



Note: The Load Other Object command is also available from the Info Box pop-up of Library Parts.



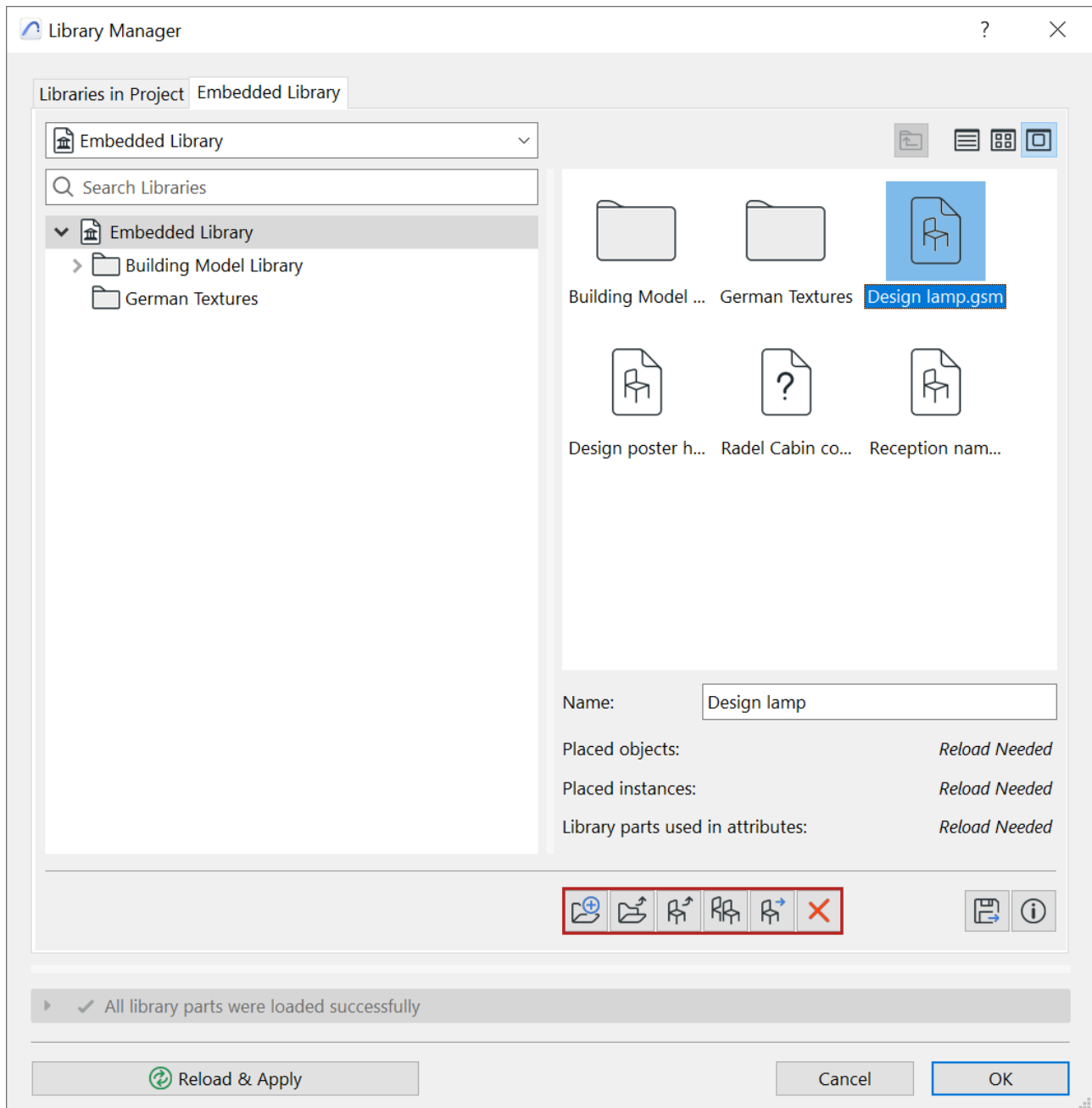
Add Library of a Hotlinked Module

Placing a Hotlink module (Archicad file formats) will add the source file's entire Embedded Library to your host file's Embedded Library. The newly added Embedded Library can be seen in your Library Manager, inside the automatically created "Hotlinks" folder of your Embedded library.

See also [Library Parts in Hotlink Modules](#).

Manage Embedded Objects

Open the Embedded Library folder of Library Manager to see the embedded objects.



Use the icons below to:

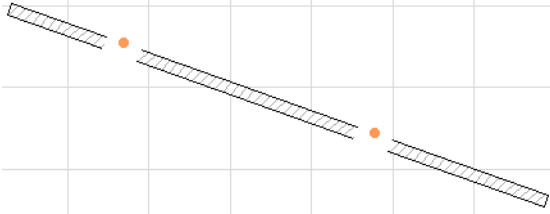
- **Create a new folder** within the Embedded Library folder.
- Add new folder to the Embedded Library (browse for an existing network or local folder)
- Add new object file to the selected Embedded Library folder

See [Embed Object Using Library Manager](#).

- **Duplicate** the object. The new object, with a new name and assigned a separate GUID, appears alongside the original object in the Embedded Library folder.

- **Move** the object from one folder to another in the Embedded Library. Select the object and click the Move icon. Use the appearing dialog to browse for a location to which to move the object.
- **Delete** the object.

Important: Deleting an object from the Embedded Library means that all placed instances of that object file will disappear from your project and be replaced by a dot. The object is deleted for good. You cannot retrieve it unless you earlier saved it to another location (in that case, you can Add it again to the Embedded Library)



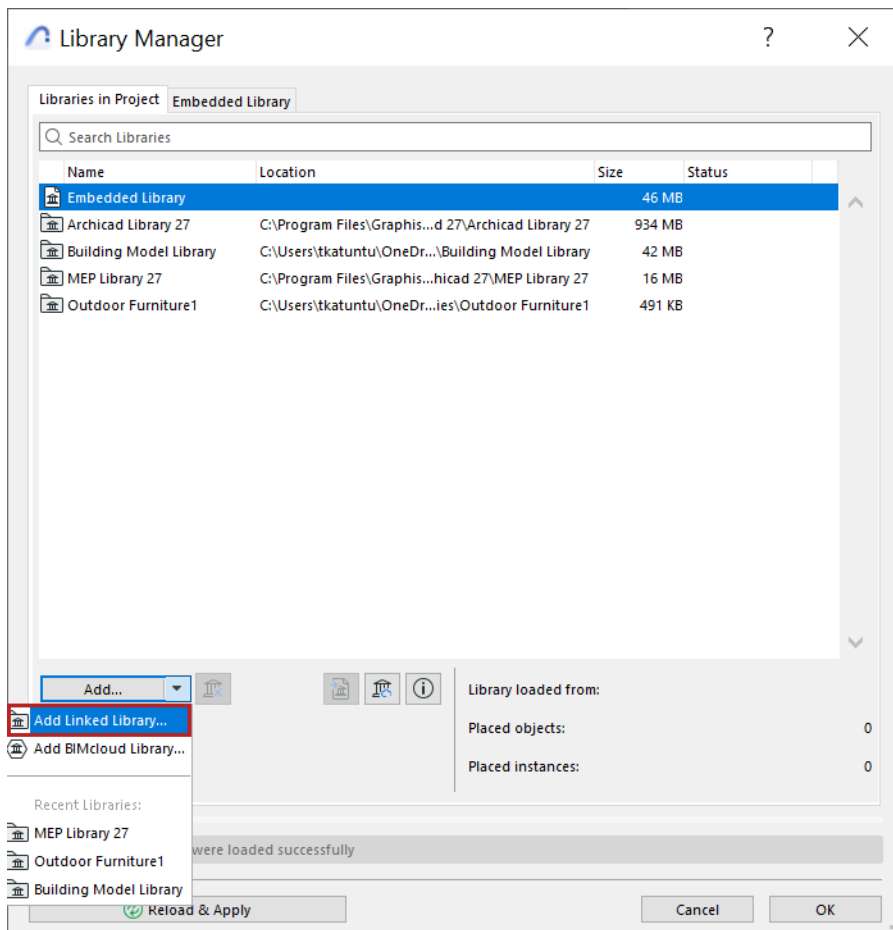
Linked Libraries

Linked Libraries include the standard Archicad library loaded by default, as well as any other libraries you add. In contrast to the objects of the Embedded Library, these libraries are linked folders, or .lcf or .pla files. Linked libraries are used in solo Archicad projects. (In contrast, Teamwork projects use BIMcloud libraries.)

Add a Linked Library

To add a linked library to your solo Archicad project:

1. Open Library Manager.
2. From the Libraries in Project tab page, click **Add**, or click **Add Linked Library** from the Add button's pop-up list. Recent Libraries are also listed: click to Add.



3. Browse for the desired folder, select it, then click **Choose**.

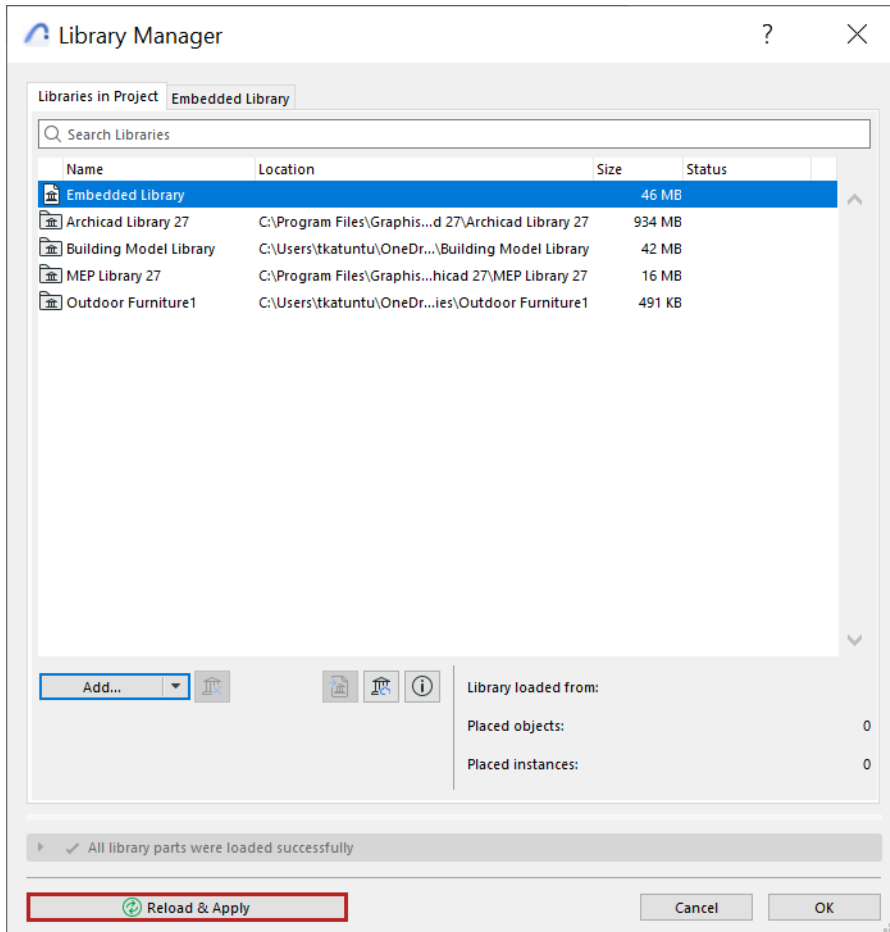
Note: If you add a folder containing a .pla file to your **Linked Libraries** folder, be aware that the library parts in the .pla file at the top of the hierarchy will be loaded. However, if the folder contains additional .pla files at a deeper level of the hierarchy, those .pla files will not be loaded. The solution is to add these .pla files to your project as separate files, rather than adding the folder that contains them.

- Use **Drag and Drop**. From any location on your computer, drag and drop a library file or folder into the **Linked Libraries** folder of Library Manager.

The chosen folder is now listed in the Library Manager.

When you are finished, click **OK** to reload project libraries and close the Library Manager.

Alternatively, you can reload libraries without closing Library manager: click **Reload and apply**.



Feedback at the bottom of the Library Manager provides

- the selected item's Path
- the number of placed objects from this library
- the number of placed instances.

If the selected linked library is unavailable, the number of the project's missing objects and instances which originate from this unavailable library are also listed.

BIMcloud Libraries

BIMcloud Libraries are located on the BIMcloud, but added to your project. You can add a BIMcloud Library to any project, either solo or Teamwork (provided that you can access the server).

To use a BIMcloud Library in either a Teamwork project or a solo project, do the following:

1. Upload the library to a BIMcloud
2. Add the library to the project using Archicad's Library Manager.

Advantages of using BIMcloud Libraries

- Whenever a BIMcloud library is updated and refreshed, this update occurs for all users, both solo (after reloading the library) and Teamwork (after doing a Send and Receive).
- Updates from the server occur with "delta logic," that is, only the changes are sent.
- A BIMcloud library's contents are cached on a Teamwork user's machine and are read locally. In contrast, a linked library is always read through a network.

Related Topics:

[Add BIMcloud Library](#)

[Manage BIMcloud Libraries](#)

[Upload a Library to BIMcloud](#)

[Upload Individual Objects to BIMcloud](#)

[Libraries in Teamwork](#)

Add BIMcloud Library

Once it has been uploaded to a BIMcloud, a library can be added to any Archicad project, whether solo or Teamwork.

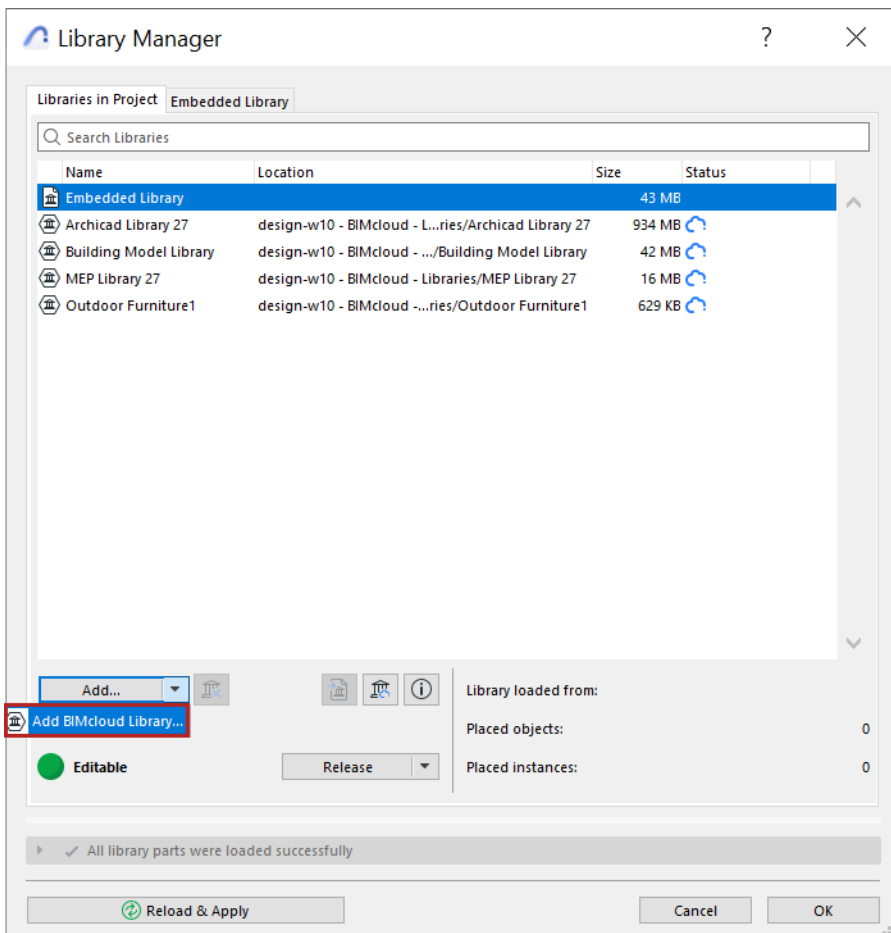
For information on uploading to the server, see [Upload a Library to BIMcloud](#).

You can access libraries on BIMcloud even if you are currently working on a solo project, as long as you have a valid user name and password enabling access to the server.

Note: In a Teamwork project, you must also have the access right of “Libraries - Add/Remove” in order to add or remove BIMcloud Libraries from the project.

To add a BIMcloud Library to your project, do the following:

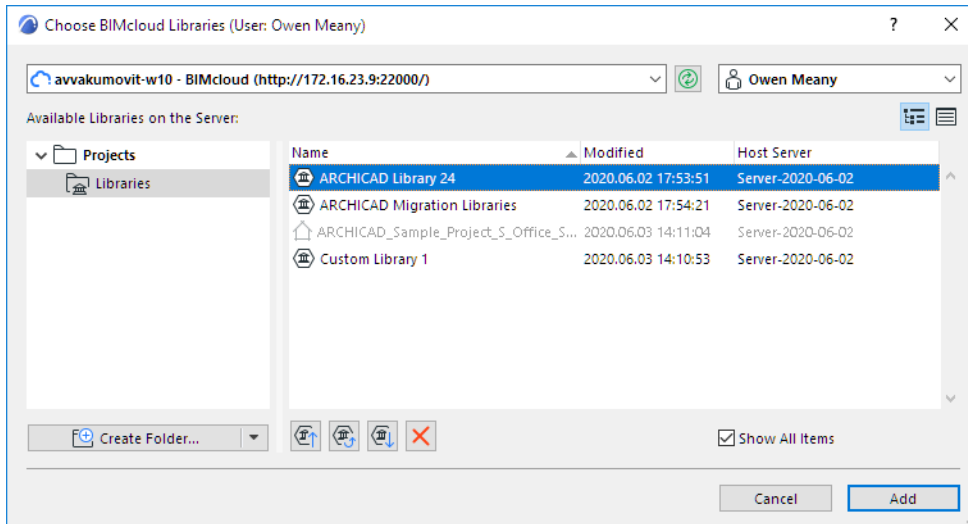
1. Open Library Manager (**File > Libraries and Objects > Library Manager**).
2. From the Libraries in Project tab page, click **Add BIMcloud Library** from the Add button’s pop-up list.



Note for Teamwork Users: To access the **Add** commands, you must first **Reserve** the Library List on the **Libraries in Project** tab page.

- At any time: use **File > Info > Action Center**

3. The Choose BIMcloud Libraries dialog box appears.



4. Log in to the BIMcloud if needed.

See [Log in to BIMcloud](#).

5. Select a library and click **Add**.

The library will now appear in the BIMcloud Libraries folder of the Library Manager.

6. In Library Manager, click **OK** to reload project libraries and close the Library Manager.

Alternatively, you can reload libraries without closing Library manager: click **Reload and Apply**.

See [Manage BIMcloud Libraries for information on managing the uploaded libraries](#).

Synchronizing with the Server Library

- When you add a library, a local copy of the library will be created on your computer (regardless of whether you are working in solo or Teamwork mode).
- If you are working in Teamwork mode, this copy is synchronized with the server data every time you Send or Receive changes.
- If you are working in solo mode, this copy is refreshed every time you click **Reload and Apply** in Library Manager. This reloading also occurs if you have made modifications in the Library Manager, then click **OK** to close the Library Manager.

Note: The Windows operating system has a limit of 260 on the number of characters that the library path name can contain. Consequently, you may encounter a warning that the program is unable to save a local copy of the BIMcloud library, because the library path is too long.

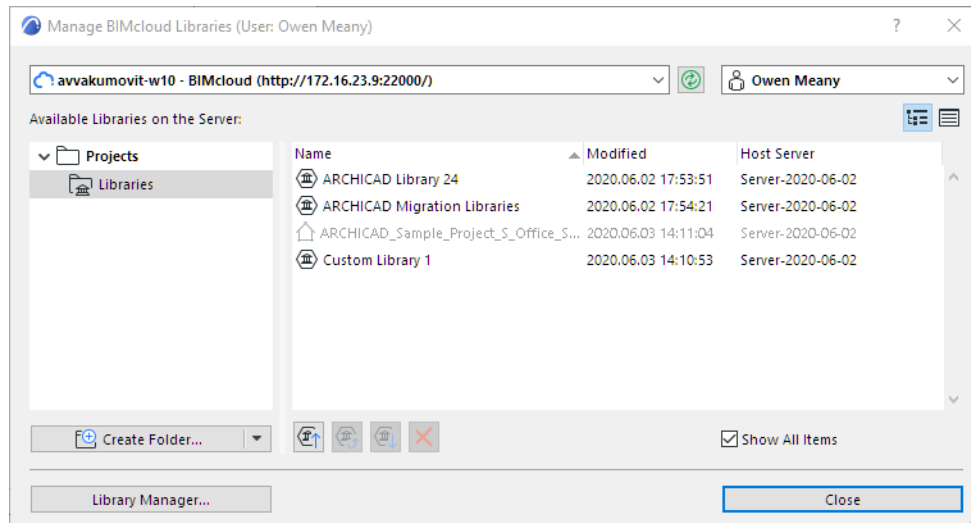
For details, see [Library Path Too Long](#).

See also [Libraries in Teamwork](#).

Manage BIMcloud Libraries

Use the **Manage BIMcloud Libraries** dialog box to view, upload and otherwise manage the libraries on any BIMcloud.

To use a BIMcloud Library in a project (either solo or Teamwork), that library must first be uploaded to the BIMcloud, using this dialog box.



Note: In a Teamwork project, you must be a Project or Server Administrator to use this dialog box.

To open this dialog box, do one of the following:

- use **File > Libraries and Objects > Manage BIMcloud Libraries**
- From Library Manager, click the **Manage BIMcloud Libraries** icon

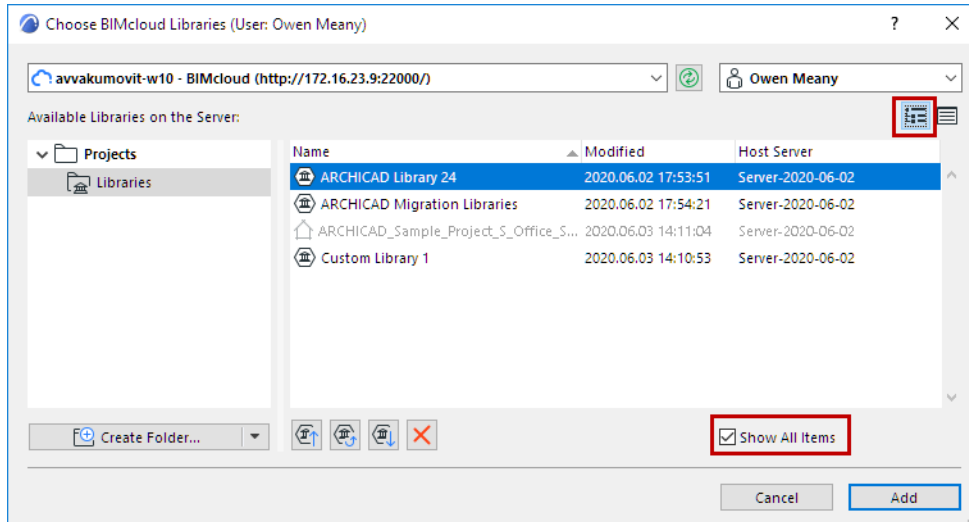
At the top of the dialog box, use the pop-ups to choose a BIMcloud and to log in if needed.

[See Log in to BIMcloud.](#)

Click the green **Refresh** button as needed to refresh the library list and user pop-up information shown in this dialog box.

Use either the **Tree view**, or switch to a **Flat list** using the icons at top right.

Tree view



The tree view displays not only libraries, but also their position within the hierarchy of BIMcloud Manager (Projects root).

Both projects and libraries are listed by default. To show the libraries only, uncheck **Show All Items**.

In Tree view, use **Create Folder** to create a new one on the Projects root.

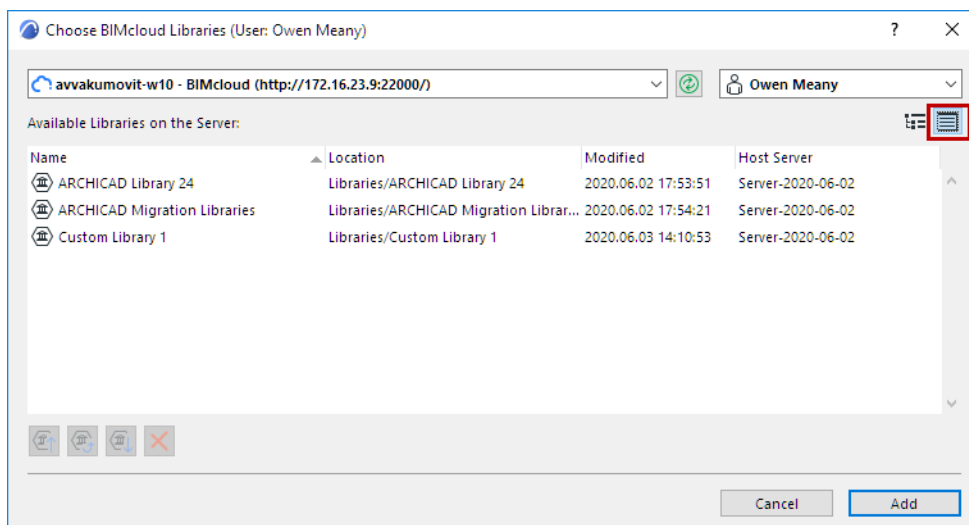
Required permission:

- Simplified Management: Project or Server Administrator
- Detailed Management: Modify details and Modify folder content for the destination folder

The folder is created immediately; you can delete or otherwise manage it only through the browser-based BIMcloud Manager. (Click the **Manage Folders** link from the New Folder drop-down button to access the Projects page of the BIMcloud Manager.)

If a library of the same name already exists within a folder on the BIMcloud, you cannot upload it: any given library can exist only once in a given BIMcloud location.

Flat View

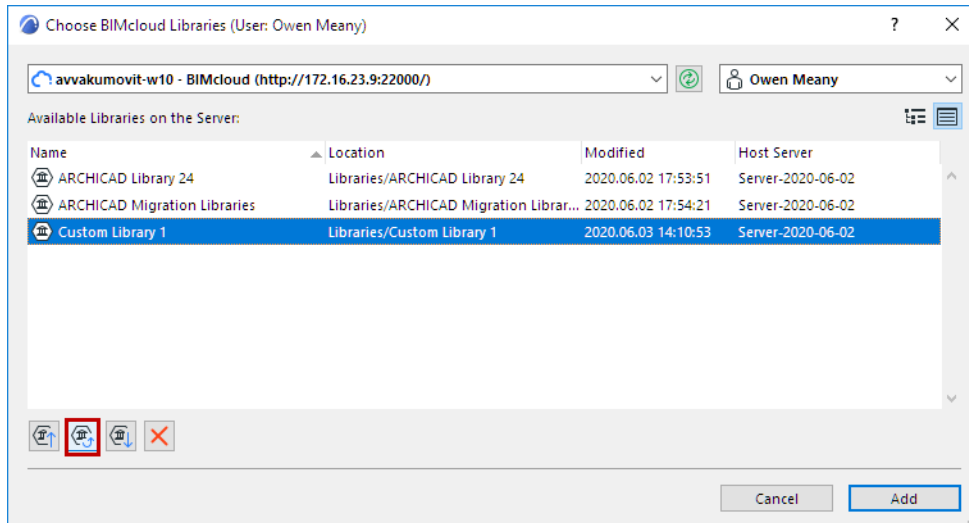


BIMcloud Libraries are listed with upload date and the address of the host server.

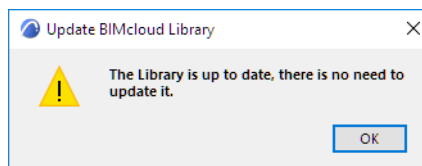
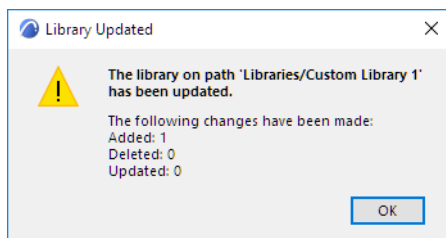
Update BIMcloud Library

Use this command to update a selected library (already uploaded to the BIMcloud) with a newer library of the same name.

1. Select the library to update.
2. Click **Update**.



3. Browse for the library with which to replace the selected library. It must be a newer library of the same name.
4. Click **Choose**.
5. The feedback tells you the result of the update.



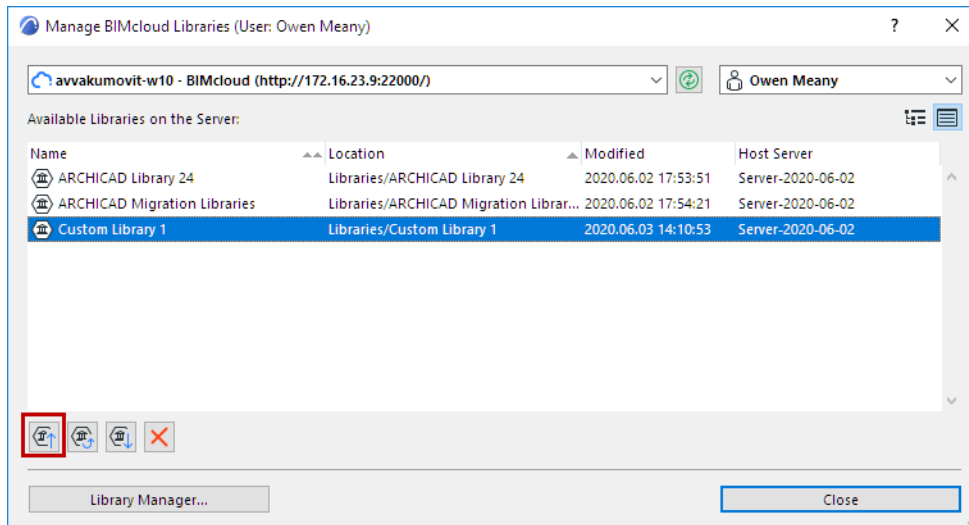
Upload a Library to BIMcloud

Required permissions:

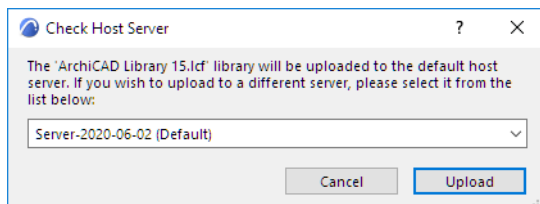
- Simplified management: Project or Server administrator
- Detailed management:
 - Modify folder content
 - Modify details for Libraries root

If you do not have permission to upload to a particular folder, you are prompted to choose the default folder.

1. Go to **File > Libraries and Objects > Manage BIMcloud Libraries**.
2. Click **Upload**.



3. Browse for the library you wish to upload. Uploaded libraries can include .pla or .lcf files, or any folders. You cannot upload individual gsm files; you can only upload .lcf or .pla files, and regular folders.
4. Click **Open**.
5. **Check Host Server:** The default host server is shown, where the library will be uploaded. To upload a library to a different BIMcloud Server (other than the default), click the pop-up.



Required permissions:

- Simplified management: Server administrator
- Detailed management: Modify hosted data on BIMcloud Server

6. Click **Upload**.

Once libraries are uploaded, individual users can **add** these BIMcloud Libraries to projects, either solo or Teamwork.

See also [Add BIMcloud Library](#).

Upload Individual Objects to BIMcloud

To upload individual objects to the BIMcloud, first place them into a folder, then upload the folder, as described above ([Upload a Library to BIMcloud](#)). The result is a BIMcloud Library.

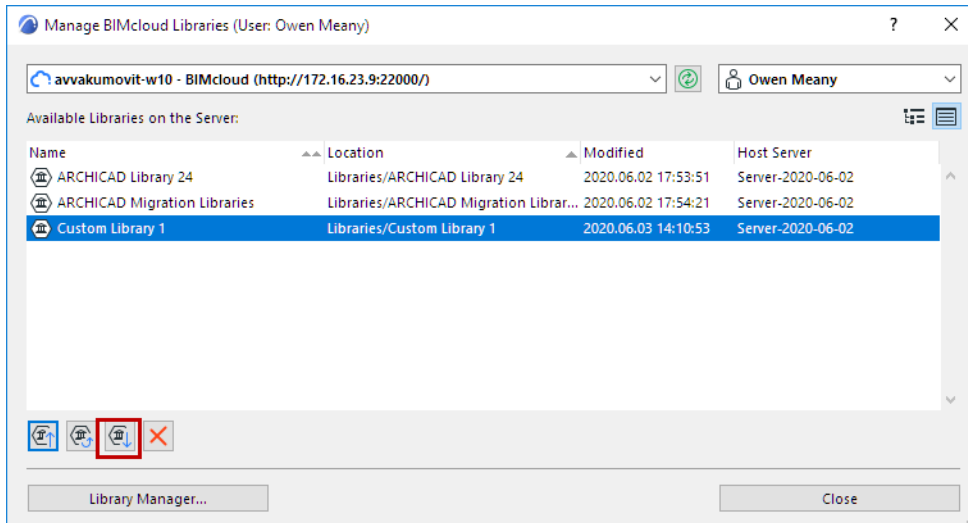
You can also upload individual object files and folders from the BIMcloud:

From the Projects Page of BIMcloud Manager, use the Upload Files/Upload Folders command. The result is individual files and folders uploaded to BIMcloud's file system.

See the [BIMcloud User Guide for details](#).

Download Library from BIMcloud

Use the **Download** command to download the selected library to the location of your choice.



Delete Library from BIMcloud

Click Delete from the **Manage BIMcloud Libraries** dialog.

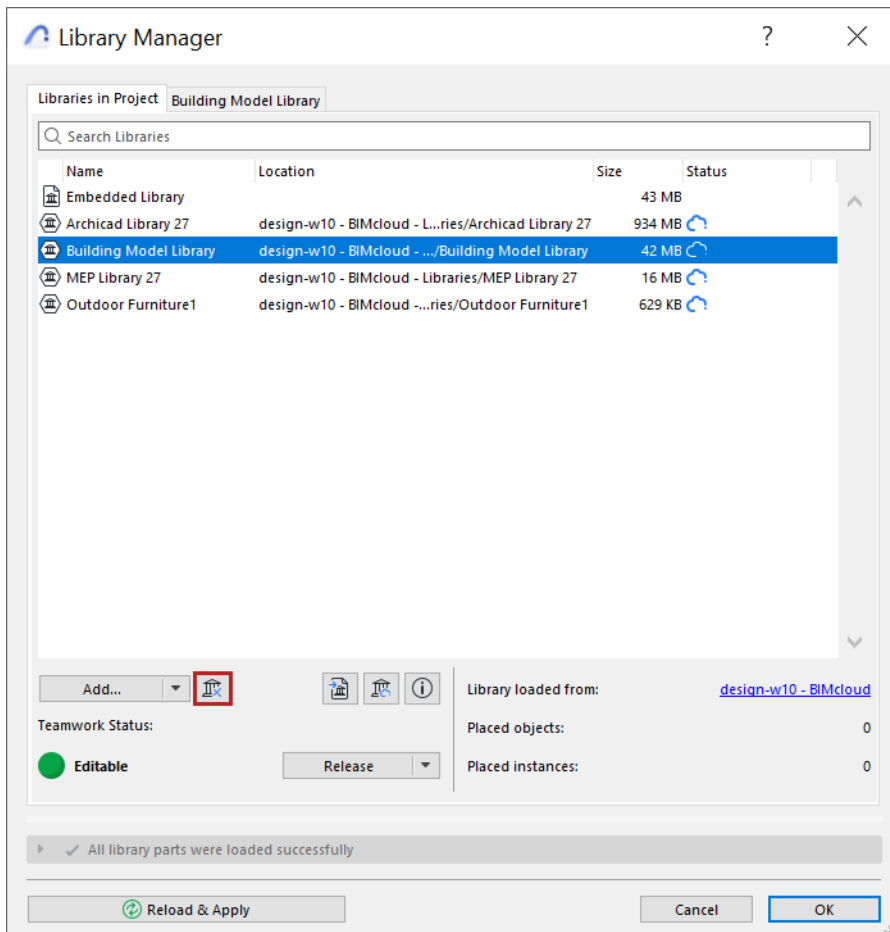
Required permissions:

- Simplified management: Project or Server administrator
- Detailed management: Delete/Remove content for library root

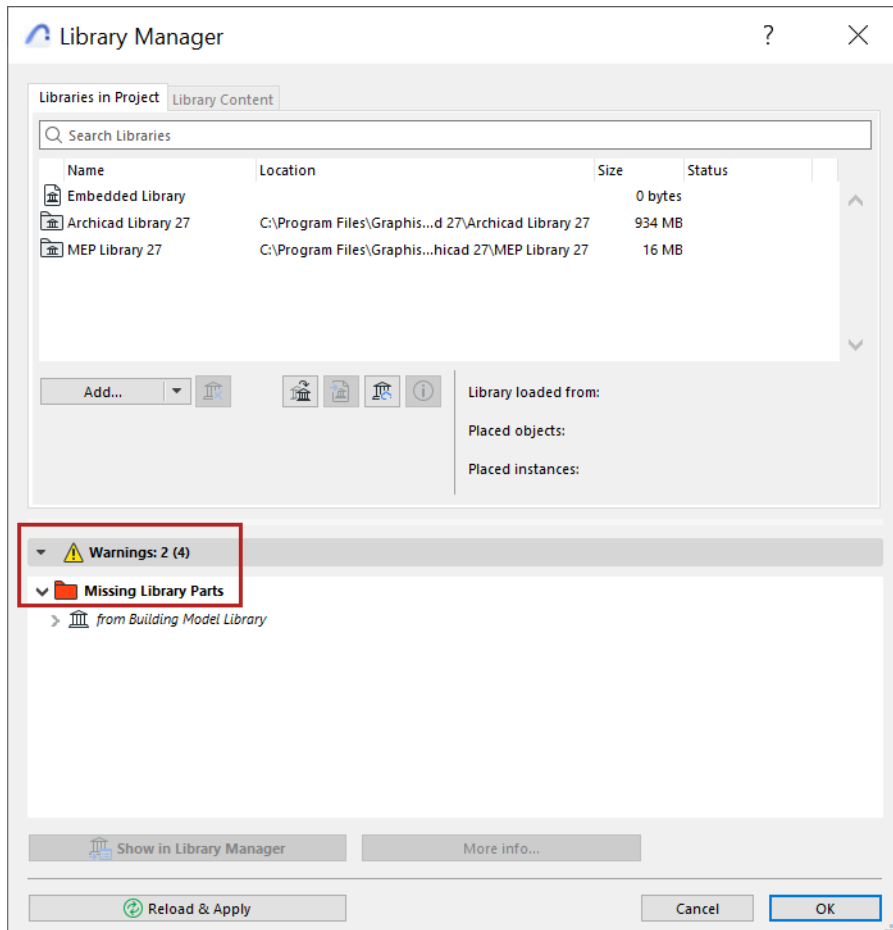
Remove Library from Project

Select any Linked Library or BIMcloud Library from the Library Manager. (**File > Libraries and Objects > Library Manager**).

Click the **Remove** icon to remove the library.



As a result of removing a library, you may end up with missing library parts. These are listed in the Warnings panel of Library Manager, in the “Missing Objects” folder. In the project model, they will be represented by fuzzy dots.



Removed libraries can be re-linked to your project at any time; then the missing objects will be found.

Note: The number of missing objects is minimized by automatic replacement with their updated counterparts, where available. For example: if both Archicad Libraries 16 and 17 are loaded, and Archicad Library 16 is then removed, then the Archicad 17 Library will automatically replace the missing placed objects from the AC Library 16 with their updated counterparts from AC Library 17. If you subsequently re-link AC Library 16, the placed instances which were automatically replaced with AC 17 counterparts will remain as-is.

See also [Troubleshooting Library Issues](#).

Recommendations for Using Libraries

It is recommended to organize your libraries as follows:

- Upload all your company's libraries to a dedicated BIMcloud. This enables one-step updates which are then available to all users, including solo and Teamwork project users.
- The Embedded Library should serve as your project library, containing project-specific objects that are always available and editable, and saved with the project.
- Any project-specific object that you modify frequently should be placed in the Embedded Library.
- Only embed those objects that are project-specific. Try to keep the Embedded Library small.
- If you want to use a custom object in several projects, it is better to place it into a Company Library stored on a BIMcloud that is accessible to all users, rather than save it to an Embedded Library.
- When you archive a project, use the PLA file format. It is NOT recommended to save all objects in the Embedded Library.

See also [Troubleshooting Library Issues](#).

Troubleshooting Library Issues

From the following list, click on the specific library-related issue for more information:

[Missing Library Parts](#)

[Objects Temporarily Missing](#)

[Unsupported Objects](#)

[Find and Select Missing Library Parts](#)

[Duplicated Library Parts](#)

[Substituted Objects](#)

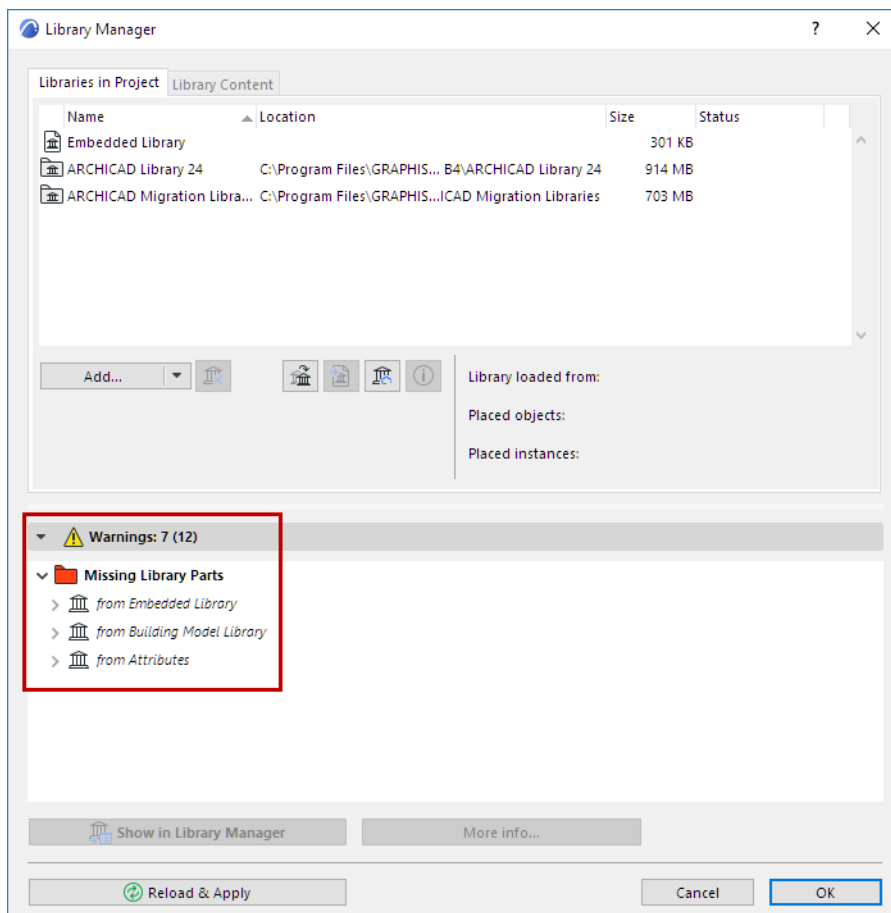
[Consolidate Multiple Archicad Libraries](#)

[Remove Unused Linked Libraries](#)

[Library Path Too Long](#)

Missing Library Parts

The Library Manager indicates any Missing Library Parts.



Missing library parts cannot be displayed in the project. (The place of missing library parts is indicated by fuzzy dots.)

Missing library parts are listed in the Warnings Panel of the Library Manager, together with:

- the name of the source library if any, with its last known location, from which the missing objects originate.

- number of missing object, plus the number of placed instances of each missing object. For each missing object, the number of its placed instances is shown in parentheses.

Items in the Missing Library Parts folder of the Library Manager are missing from the project

- either because they were embedded objects, and have been deleted (you cannot retrieve them, since they no longer exist)
- or because they were located in libraries that have been deleted from the project or moved to a different location.

To retrieve these objects, use the **Add** command in Library Manager to add the missing libraries, then click **Reload & Apply**.

Note: Some Favorites include a GDL object (such as a Favorite Zone that includes a Zone Stamp). If that GDL object becomes Missing (i.e., its library is not loaded), the Library Manager lists all missing objects from Favorites.

Objects listed under “Missing from unknown library” are missing objects about which no other information is available - their source library (or libraries) are not known.

Objects Temporarily Missing

In a Teamwork project, you may encounter a category in the Library Manager entitled “Temporarily Missing.” This may occur when a user makes changes in a BIMcloud Library that result in placed objects being updated in some way. If those objects are reserved by any user, then all users joined in to the project will encounter the “Temporarily Missing” category.

This will be resolved as soon as the user(s) who have reserved the objects do a Send & Receive; then the objects will no longer be missing.

Unsupported Objects

These files reference unsupported image formats.

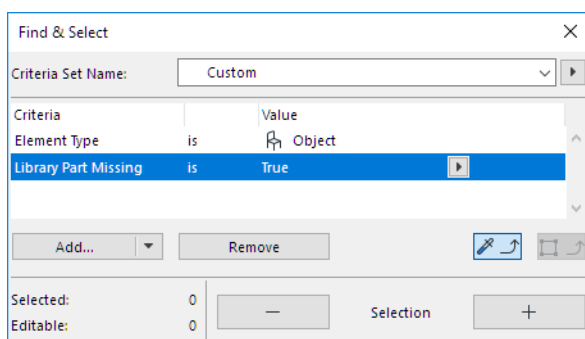
Find and Select Missing Library Parts

To find and select missing library parts in the active window, use the Find & Select function.

[See also *Find and Select Elements*.](#)

1. In the first “Criteria” row in the Find and Select dialog box, choose a GDL object type element.
2. In the next row, for the next criterion, choose “Library Part Missing.”
3. As the value, select “True”.

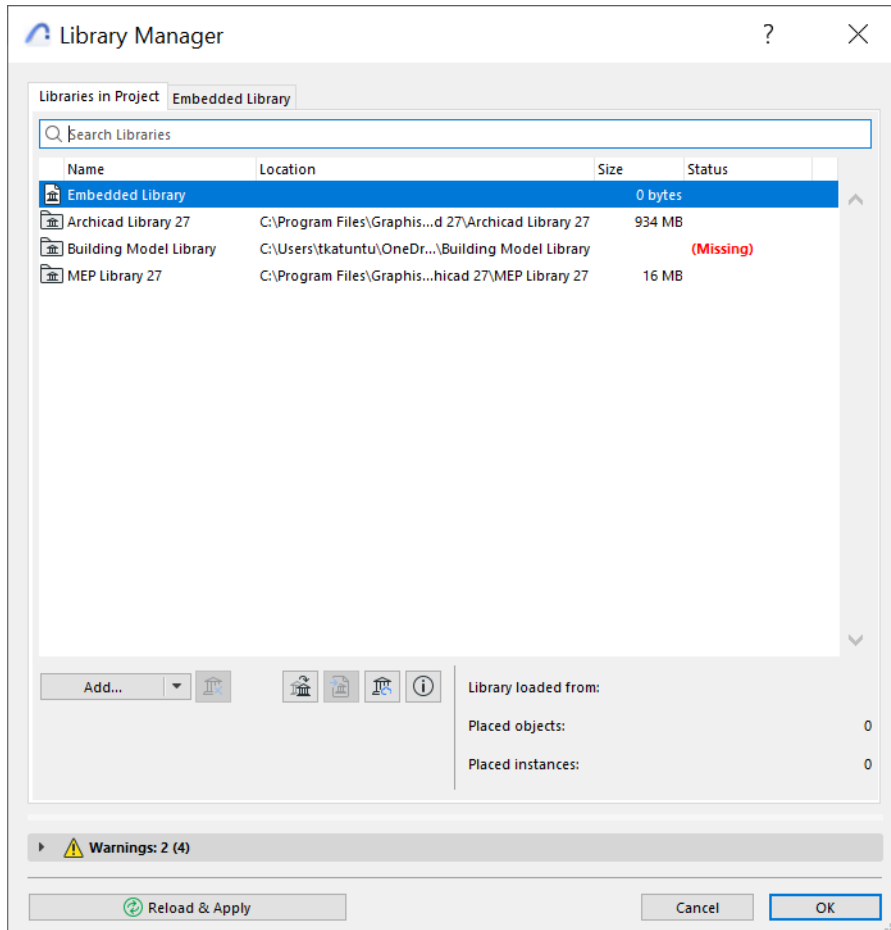
Note: You can also choose “False” as the value; this will select all instances of the defined GDL Object types, *except* for the ones whose library object is missing.



4. Click the Plus button to execute the Find & Select function.

Missing Libraries

If you have modified the name of a library folder, or changed its location in the file hierarchy, then the next time you open Archicad, the library folder listed in the **Library Manager** dialog box will be shown as Missing.



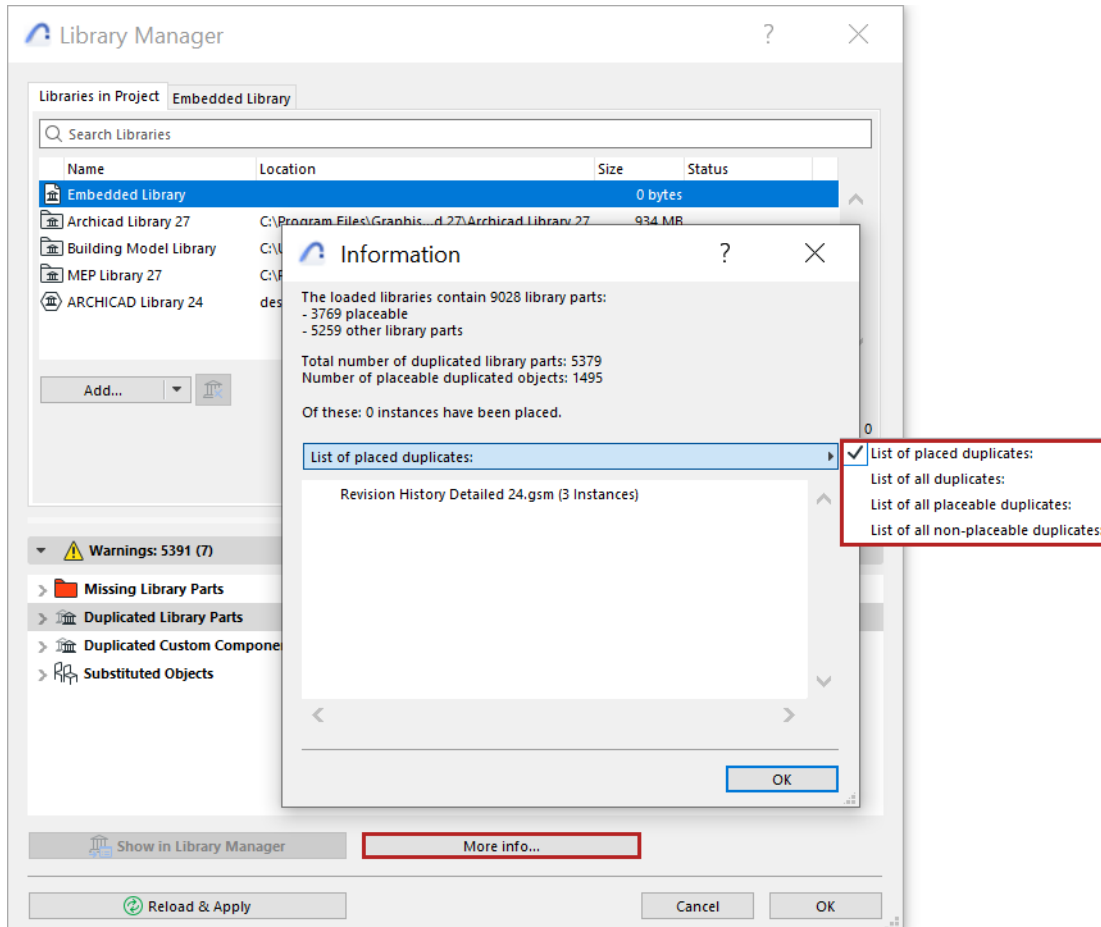
Placed objects from unavailable libraries are listed in the “Missing Library Parts” folder of Library Manager. In the project model, they are represented by fuzzy dots.

The solution is to click **Add Linked Library** from the Library Manager’s **Add** button, and browse for the library file under its new name or location.

Duplicated Library Parts

Any Duplicated Library Parts are listed in the Warnings Panel of the Library Manager. Each such library part is followed by the name and path of the project libraries that contain it.

Click **More Info** at the bottom of Library Manager for detailed information on the duplicates:



In the Information box, click the black arrow to choose the list format you need:

- List placed duplicates
- List all duplicates
- List all placeable duplicates
- List all non-placeable duplicates

[Watch the Video](#)

Avoid Duplicate Library Parts

The problem with having a lot of duplicate objects in your projects is twofold: every time you load your libraries, you will be loading many superfluous objects and/or libraries, causing unnecessary slowdowns; and the presence of duplicates can lead to further ambiguities if you should modify a duplicate library object or save it under a new name.

[See Substituted Objects.](#)

There are several possible strategies to resolve the issue of duplicates:

- If the Libraries with Duplicate Objects are Archicad Libraries, it is best to Consolidate.

[See Consolidate Multiple Archicad Libraries.](#)

- If all the placed objects from one of the libraries are duplicates (i.e. these objects also exist in the other library), you can remove the first library altogether.
- If the duplicated libraries are not Archicad libraries, you might consider comparing the two libraries' contents manually (open the library folder at its source and view the objects), and then delete the duplicates from one of the libraries.
- If the duplicate placed objects are project-specific, you should embed them into the project. This will also eliminate the library from the project.

See [Embed Placed Objects from a Library](#).

Substituted Objects

An object in this category has been placed in the project, but *several possible substitutes* for this object (that is, objects with the same GUID as the placed object, though they may be named differently) exist in at least two linked libraries. *Archicad randomly uses one or the other source for the object's placed instances.*

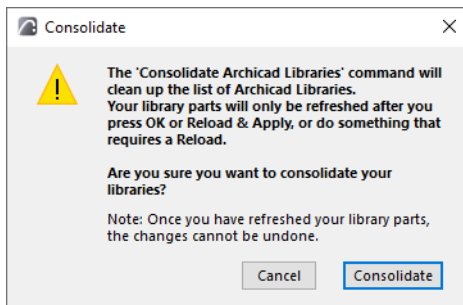
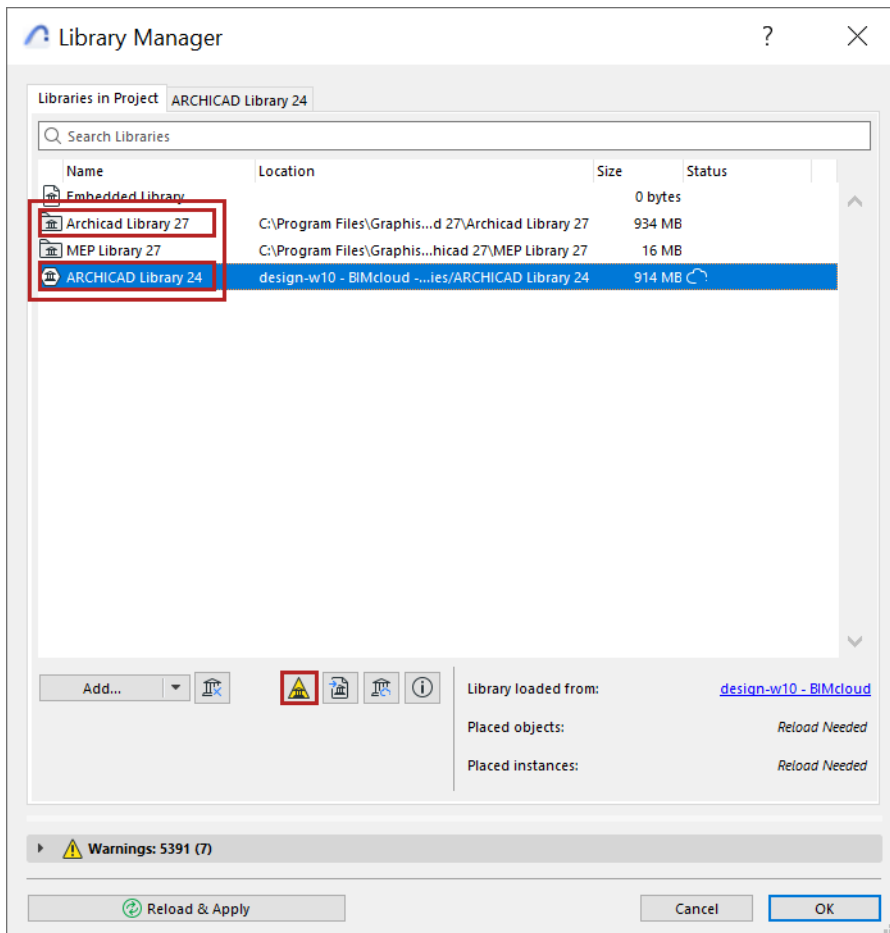
To avoid this situation, you should manually replace the instances of the "substituted object" with a single one of its possible duplicates from the loaded libraries.

(To do this, you can use Find & Select to locate the placed instances of the substituted object. Then choose a specific library object from the loaded libraries to replace it.)

Consolidate Multiple Archicad Libraries

Any time you add an additional Archicad library to a project that already has one, Archicad will recognize the presence of older-version Archicad libraries (versions 10 and later). In this case, the yellow Consolidate button appears in Library Manager.

Note: The Consolidate button appears in Library Manager any time that at least one outdated Archicad Library - that is, a Library which is not the latest one - has been added to the project.

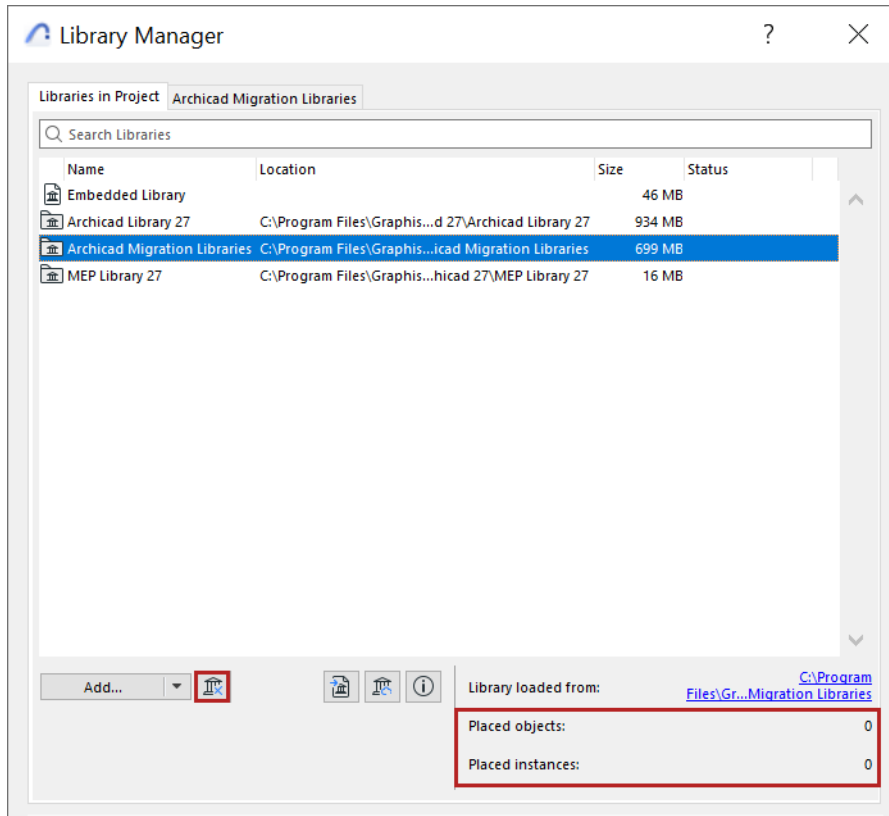


Consolidating Archicad libraries is recommended because:

- it replaces the old Archicad libraries with the latest library
- it loads the requisite Migration Libraries so that you still have access to any now-obsolete library parts that may still be placed in the project
- Migration Libraries are much smaller than the full Archicad libraries, so you will avoid superfluous objects
- you avoid errors that can result from placing identically named library parts that are present in several loaded libraries

Remove Unused Linked Libraries

Library Manager tells you whether you have actually placed any objects from your linked libraries. If not, you don't need that library and you can remove it from the project. In this example, there are no objects placed from the selected Archicad library; you can remove it.



To remove a library folder, select it from Library Manager and click the “Remove” icon with the blue X.

Library Path Too Long

The Windows operating system has a limit of 260 on the number of characters that the library path name can contain. Consequently, you may encounter a warning that the program is unable to save a local copy of the BIMcloud library, because the library path is too long.

To resolve this problem, try one of these solutions:

- **Simplify the library’s file structure.**
Go to **File > Libraries and Objects > Manage BIMcloud Libraries**.

Download the BIMcloud library to a folder on your hard drive.

In this folder, do one of the following:

- Create a container file (.lcf) out of the library folder, using **File > Libraries and Objects > Create Container**.

See also [Library Container File](#).

- Alternatively, create a different, simplified folder structure for the library (e.g. move the library folder to a higher position in the folder hierarchy so that the file path becomes shorter).

Go to **File > Libraries and Objects > Manage BIMcloud Libraries** and use **Update** to overwrite the server library with the .lcf file, or the restructured folder.

- **Move the Local Data folder.**

Your local copies of BIMcloud libraries are stored in your Local Data Folder on your hard drive.

If this location results in a file path that is too long, move the Local Data folder:

[See *Move Local Teamwork Data to a Different Folder*.](#)

The new location should be at a higher level of the folder hierarchy, thus making the path shorter.

Library Container File

The extension .lcf identifies a “Library Container file”. The default Archicad Library is stored in an.lcf, but you can also create your own Library Container file, or extract its contents, using the **File > Libraries and Objects > Create/Extract a Container** command.

Since the Library Container file is a single file, yet contains all the objects used in your project, it lets you keep all the objects used in your projects in one place, while keeping hierarchies intact within the.lcf.

Favorites

Use **Favorites** to save and apply tool settings configurations that you use frequently.

Favorites are saved with your project file, and can be imported and exported between projects.

The Archicad package includes some predefined tool Favorites.

You can access and apply Favorites from three places in the interface:

- Favorites Chooser in the Toolbox or Info Box
- Favorites Palette (Window > Palettes > Favorites)
- Tool Settings Dialog box

Which Parameters Are Applied with Favorites?

A Favorite's stored parameters include:

- All of the element's dimensions that can be entered in the settings dialog boxes for the element, e.g., wall thickness and height (but not length)
- Classification and Properties data
- for Fill Favorites: the style of its Fill Text, when "Show Area Text" is on
- for Library Parts: the name of the Library Part

Apart from the above, you can customize and save sets of additional parameters ("Transfer Sets") to be applied with Favorites.

For details, see [Element Transfer Settings](#).

Topics in this section:

[Apply a Favorite](#)

[Search Favorites](#)

[Create New Favorite](#)

[Redefine Favorite](#)

[Edit Settings of One or More Favorites](#)

[Missing Content in Favorite](#)

[Favorites Palette](#)

[Import/Export Favorites](#)

Related Topic:

[Element Transfer Settings](#)

[Watch the Video](#)

Apply a Favorite

Apply a stored Favorite to selected element(s), or apply it as default Tool settings (if nothing is selected).

Favorites can be applied from three different locations, each shown below:

- Favorites Chooser in the Toolbox or Info Box
- Favorites Palette (Window > Palettes > Favorites)
- Tool Settings Dialog box

For information on Favorites controls in Library Part tool dialog boxes, see [Settings of Library Part Elements](#).

In every case, select the Favorite, then double-click to apply to a selected element.

Or select a Favorite and click **Apply** (button or context menu).

Text-type Tools

Text tool Favorites can be applied to **any text-type items** when Dimensions, Labels, and Fills with Area Text are selected.

Line-Type Tools

Favorites of any of the four Line-type tools (Line, Arc, Polyline, Spline) can be applied to any of the other Line-type elements. For example, apply a Line Tool Favorite to a selected Arc.

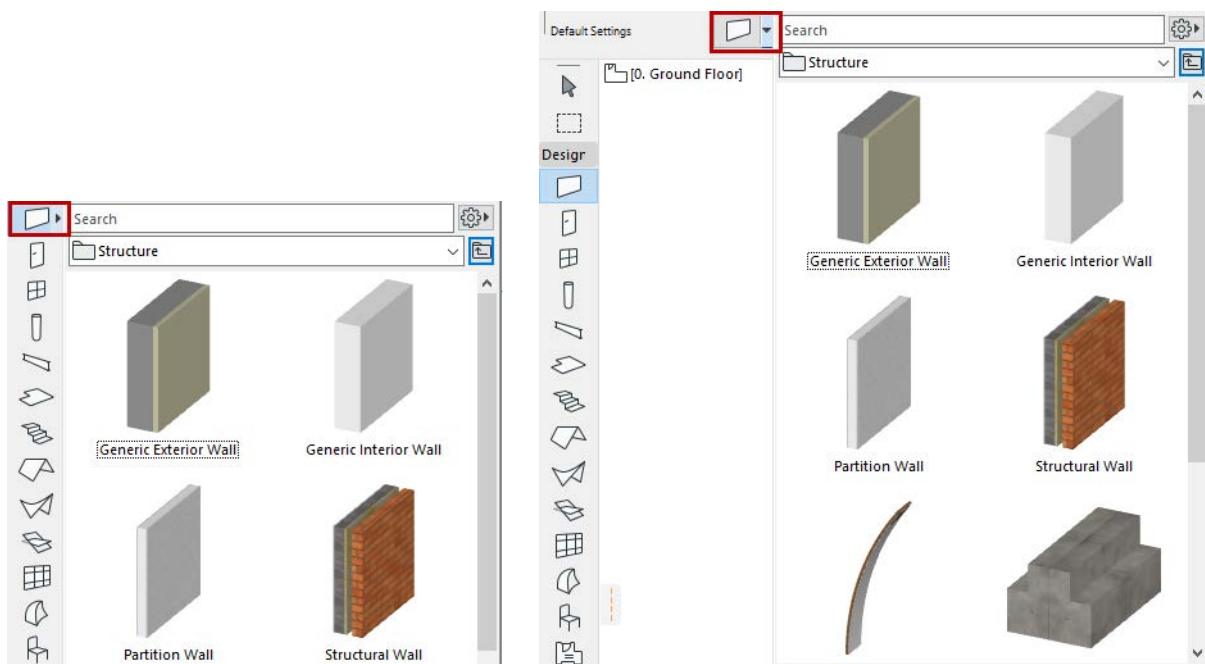
From Favorites Chooser (Toolbox/Info Box)

This is the fastest way to apply Favorites to a selected element as you work.

1. Access the Favorites Chooser by doing one of the following:

- Click and hold the Tool icon (Toolbox only)
- Click the arrow from the Toolbox or Info Box

Note: If no Favorites are defined for a tool, the Chooser will not appear from the Tool Box.



Note: To switch on the fly between the 3D and 2D preview of any Favorite, hover your cursor over the preview and press **Shift**.

2. Search or browse, then **double-click** the desired Favorite.

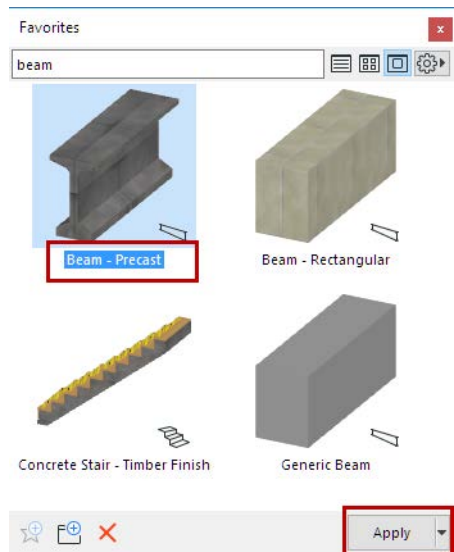
[See also Search Favorites.](#)

Note: The folder tree view is not available from the Favorites Chooser.

From Favorites Palette

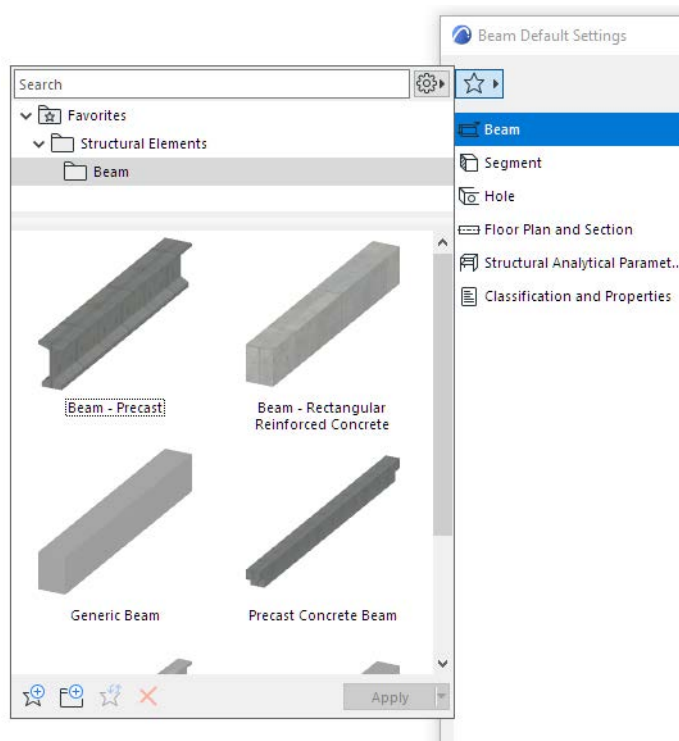
From the Favorites Palette (Window > Palette > Favorites), search and browse, then do one of the following:

- **Double-click** the desired Favorite, or
- Select a Favorite and click **Apply**



From Tool Settings

1. Open the Tool Settings dialog box.
2. Click the Favorites (star) button at top left.



3. Search and browse, then do one of the following:

- **Double-click** the desired Favorite, or
- Select a Favorite and click **Apply**

[See also Search Favorites.](#)

4. Click **OK** to close the Tool Settings dialog.

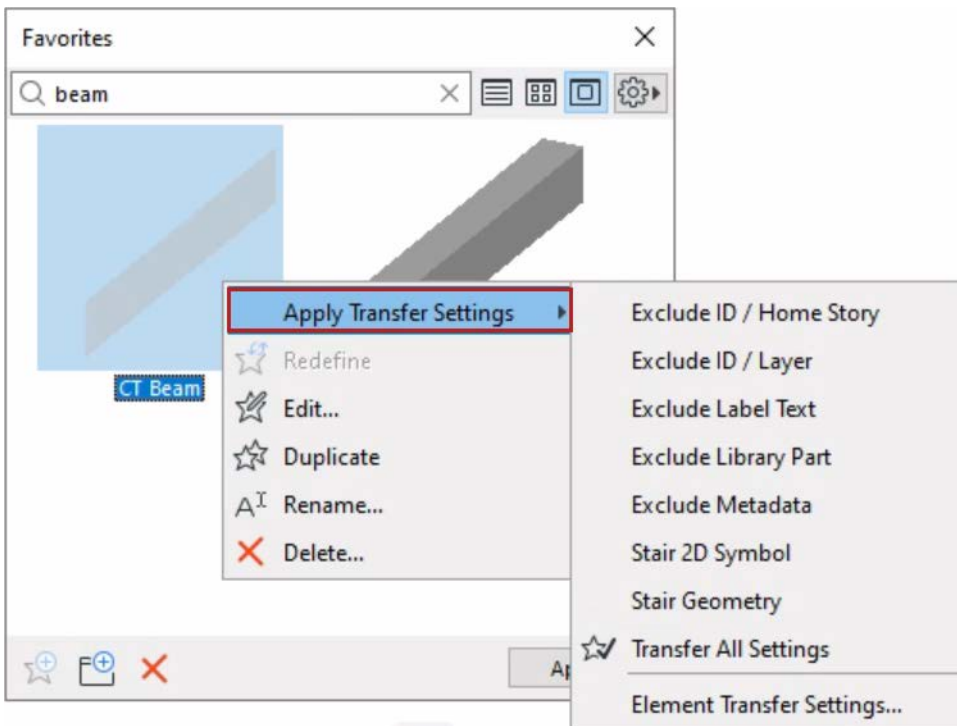
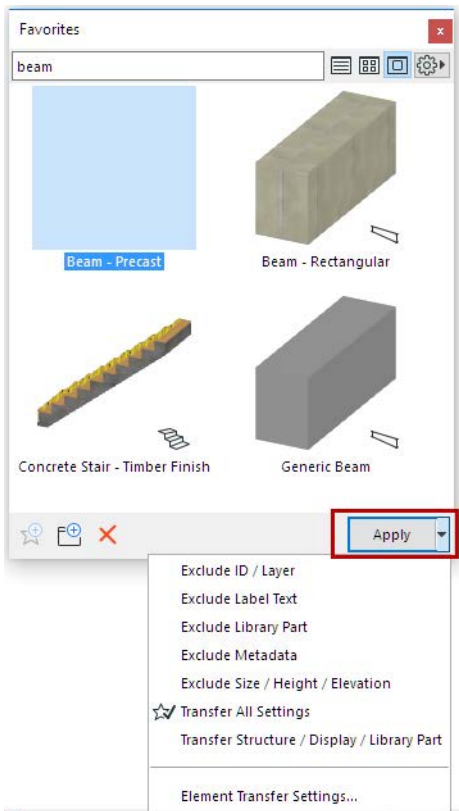
Apply a Different Transfer Set

To see the list of the project's stored Transfer Sets:

- Use the Apply button's pop-up, or
- Click **Apply Transfer Settings** from a selected Favorite's context menu

The Transfer Set defines the list of parameters applied with the Favorite.

Change to a different Transfer Set, if needed: click on any Set name to apply it.



- Define and manage Transfer Sets here: [Element Transfer Settings](#)
- One of the sets is tagged (checked star icon) as the default to use when applying Favorites.

Search Favorites

When using Favorites, you can use the search function to find the one you need. Search Favorites is available in all of the Favorites interface locations:

- Favorites Palette
- Favorites Chooser (Info Box)
- Tool Settings dialogs

Filter the list of displayed favorites as you type, by using the Search field at the top. A found item displays its folder location in the info tag.



To return to the previous view, delete the Search term or use the **Clear Search** command from the Favorites Additional Settings pop-up menu.

Use Search to instantly filter the list of Favorites, not just by name, but also by information contained within the Favorite, including:

- Favorite Name
- Library Part name - such as for Door, Window, Object Settings
- ID
- Text values from Properties (does not include numerical fields or name of Property)
- Default text stored in a Text label

Use quotation marks around a phrase to search for exactly that phrase, rather than the individual words.

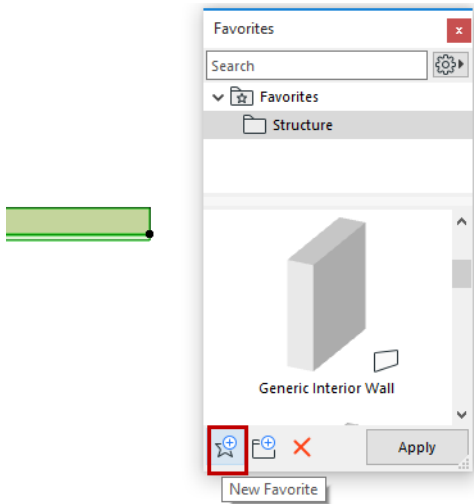
Folder names are ignored.

[See also Search for Library Part.](#)

Create New Favorite

Based on Selected Element:

1. Select an element that you want to store as a Favorite
2. Click the **New Favorite** icon at the bottom of the Favorites Palette

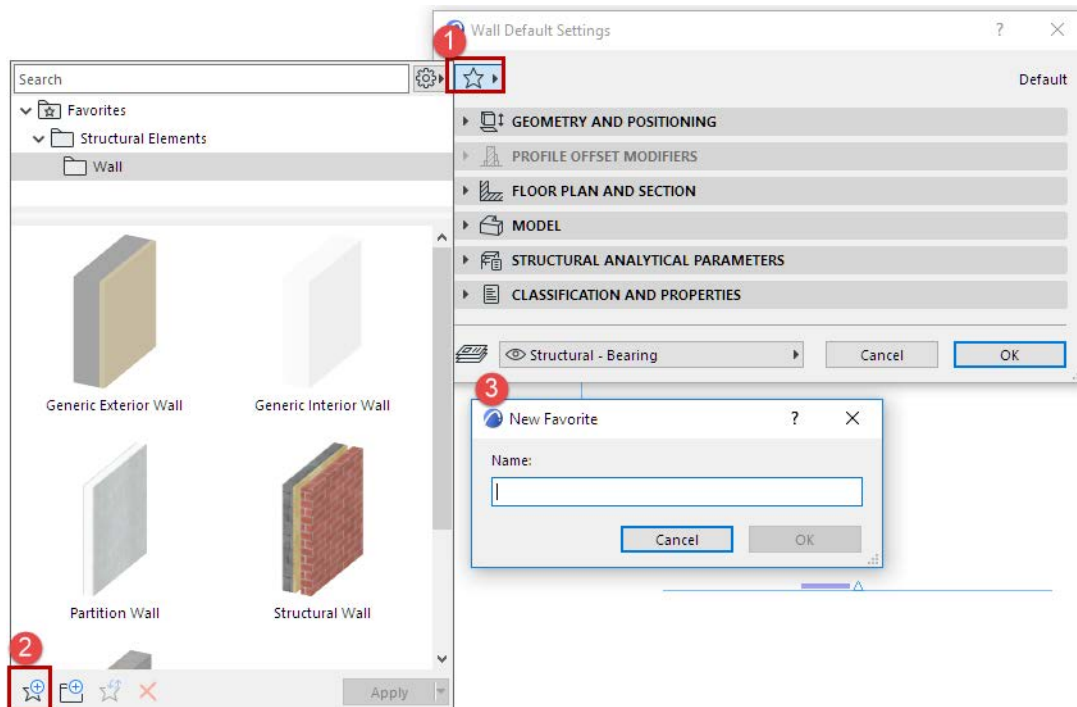


3. Enter a name in the New Favorite dialog box that appears and click OK.

In Tool Settings:

Use this method to store the current default or selected Tool Settings.

1. Click the **Favorites** (star) button to open the **Favorites pop-up**
2. Click the **New Favorite** icon



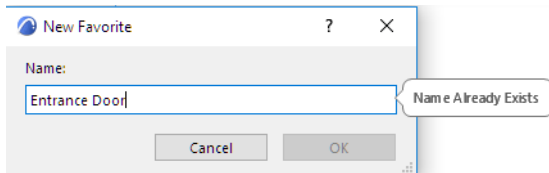
3. Enter a name in the New Favorite dialog box that appears and click OK.

Your new Favorite is now listed, as applicable, everywhere that Favorites are available.

Within the Tool Settings Favorites pop-up, you can drag and drop Favorites among folders.

Favorite Name Conflict

If the name you give already exists as a Favorite, you are alerted, and the OK button is grey. Enter a different name to immediately dismiss this tool tip warning.



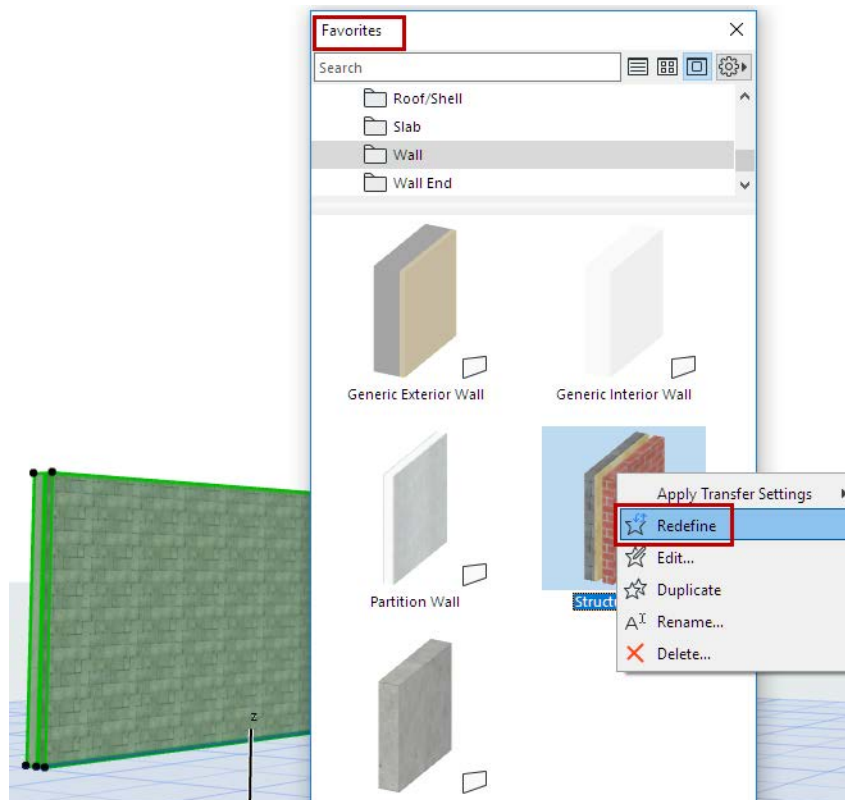
See also [Redefine Favorite](#).

For general information, see [Favorites](#).

Redefine Favorite

Based on Selected Element:

1. Select one element from the model.
2. Select a Favorite (whose settings you want to change) from the Favorites Palette. (The Favorite must be the same tool type as the selected element.)
3. Right-click the Favorite and choose **Redefine**.

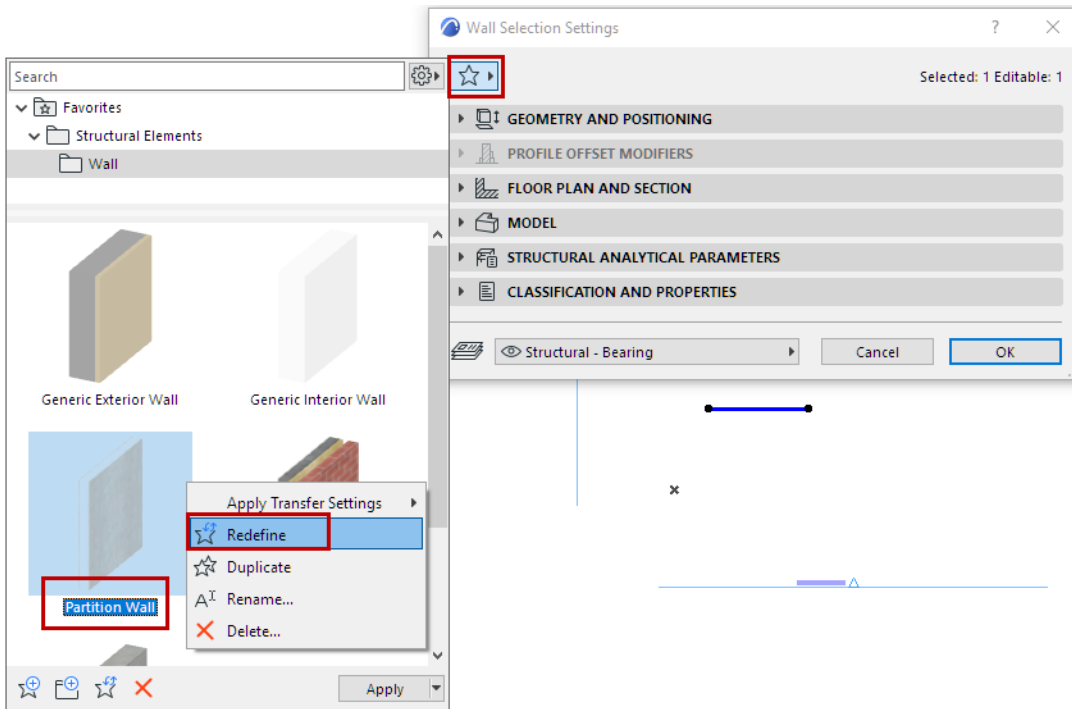


This Favorite is now redefined based on the selected element.

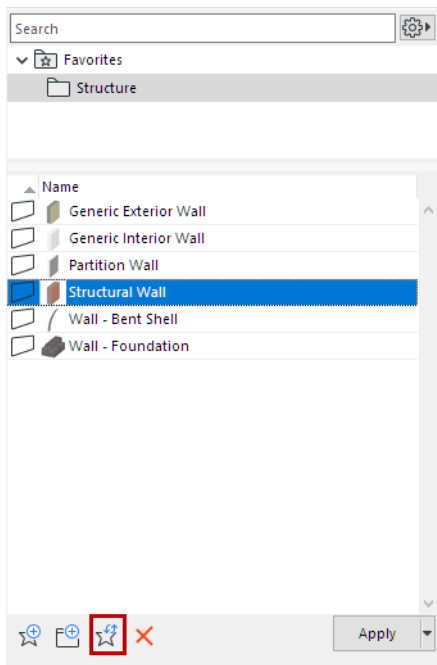
In Tool Settings:

Use this method to redefine the current default or selected Tool Settings.

1. Click the **Favorites** (star) button to open the **Favorites pop-up**.
2. Right-click a Favorite and use **Redefine** from the context menu.



With an element selected: Select a Favorite and click the **Redefine** command icon.



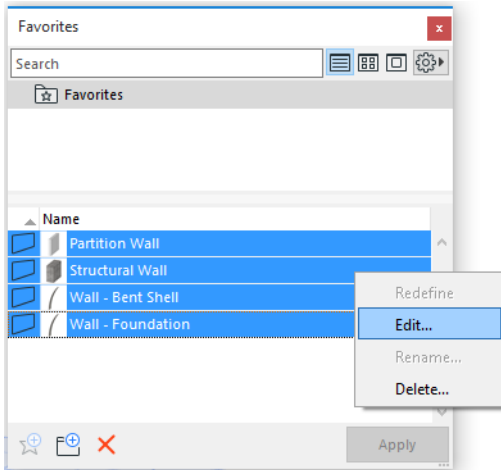
For general information, see [Favorites](#).

[Watch the Video](#)

Edit Settings of One or More Favorites

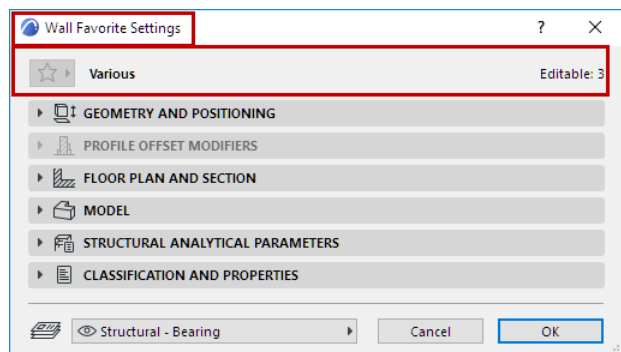
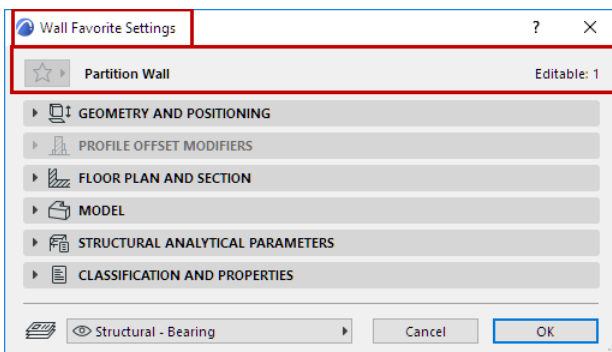
Use this workflow to adjust particular settings in multiple Favorites (e.g., add the same new Property definition to all of your Wall Favorites at once). This workflow is available only using the Favorites Palette.

1. In the Favorites Palette, select one or more Favorites, of the same Tool Type.
2. From the context menu, click Edit.

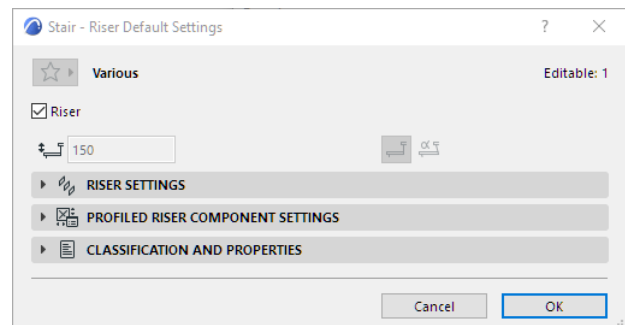
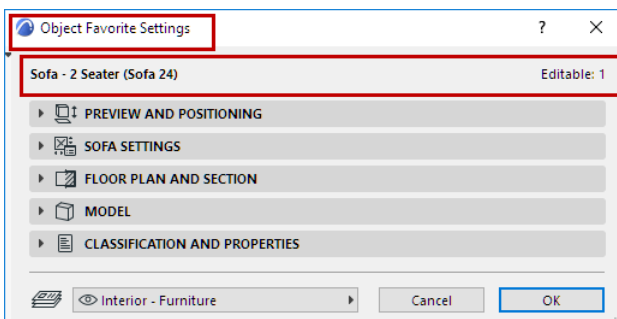


3. The Tool Settings dialog box opens. (In case of multi-selection, the settings of the last-selected element are shown.)

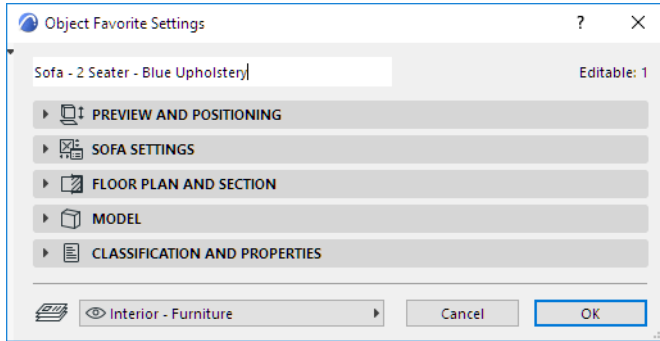
Note that the dialog box is named, e.g., “Wall Favorite Settings”, and shows the name (“Various” if multiple) and number of the Favorites being edited.



When editing a Favorite of a Library Part, the Favorite name (“Various” if multiple) is followed by the name of its Library Part in parentheses (or that of the last-selected Library Part, in case of multi-selection).



When editing a single Favorite, you can edit its name simply by clicking on it and typing into the field:



4. Adjust the settings as needed and click **OK**: the Favorites are redefined accordingly. Only the modified or new Settings definitions are changed, for each Favorite in the selection set.

For general information, see [Favorites](#).

Note: Favorites content and the Favorites pop-up are disabled in Tool Settings while you are editing Favorite(s).

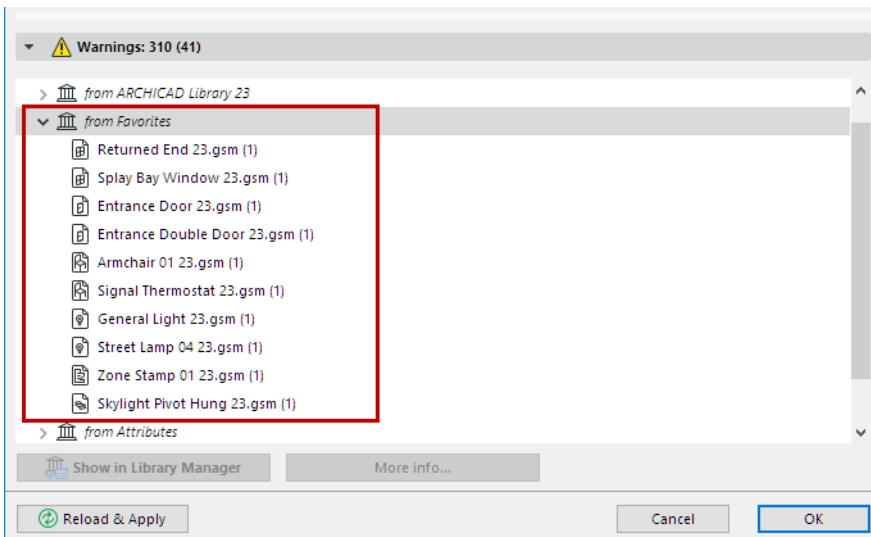
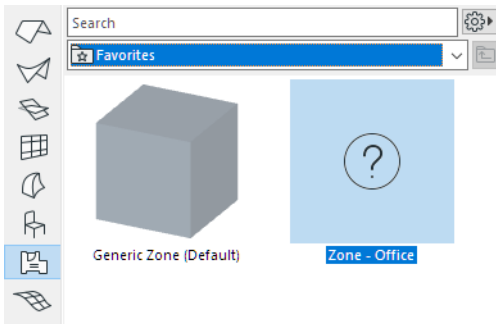
[Watch the Video](#)

Missing Content in Favorite

Some Favorites include a GDL object (such as a Favorite Zone that includes a Zone Stamp).

If that GDL object becomes Missing (i.e., its library is not loaded):

- the Favorite is shown in the project and Favorites Chooser with a question mark
- Library Manager lists all missing objects from Favorites



You cannot apply a Favorite that has missing content. However, you can still delete/rename/redefine or edit it, using the Favorites Palette or Tool Settings.

For general information, see [Favorites](#).

Favorites Palette

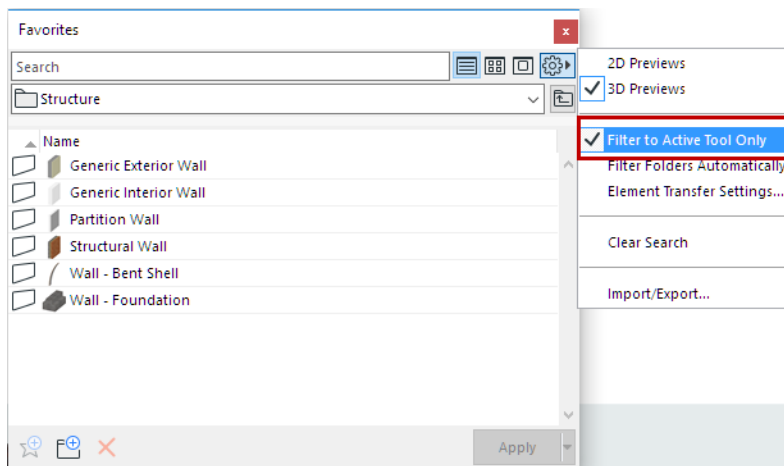
Open this Palette at **Windows > Palettes > Favorites**.

Use the Favorites Palette to

- Organize and manage Favorites ([see below](#))
- Apply Favorites ([see Apply a Favorite](#))
- Create a new Favorite ([see Create New Favorite](#))
- Import and Export Favorites between projects ([see Import/Export Favorites](#))
- Customize Favorite Parameters ([see Apply a Different Transfer Set](#))
- Access Favorite Parameter Settings
- Edit the settings of multiple Favorites simultaneously ([see Edit Settings of One or More Favorites](#))

Show All Favorites/Show Active Tool Favorites Only

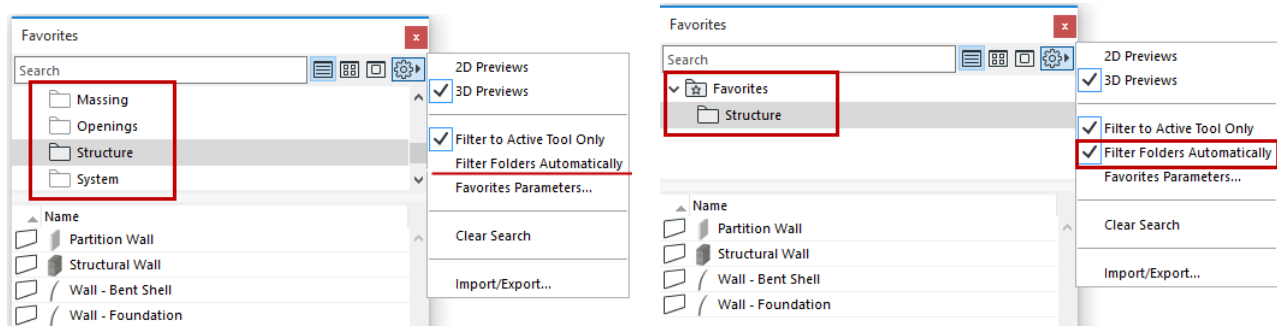
Choose **Filter to Active Tool Only** from the Settings pop-up (this option is active by default):



If the Arrow or Marquee tool is active, then all Favorites are listed, including those of sub-components such as Curtain Wall Frame and Panel.

Filter Folders

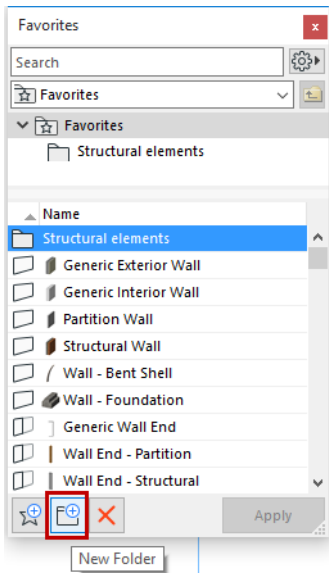
Enable **Filter Folders Automatically** to hide any folder that has no visible content.



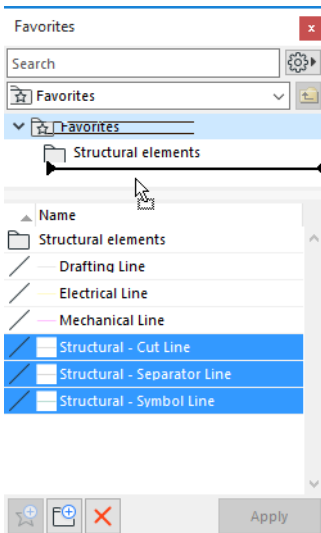
Organize Favorites

Add new folders using the **New Folder** button.

Also available from the context menu in the folder tree view.



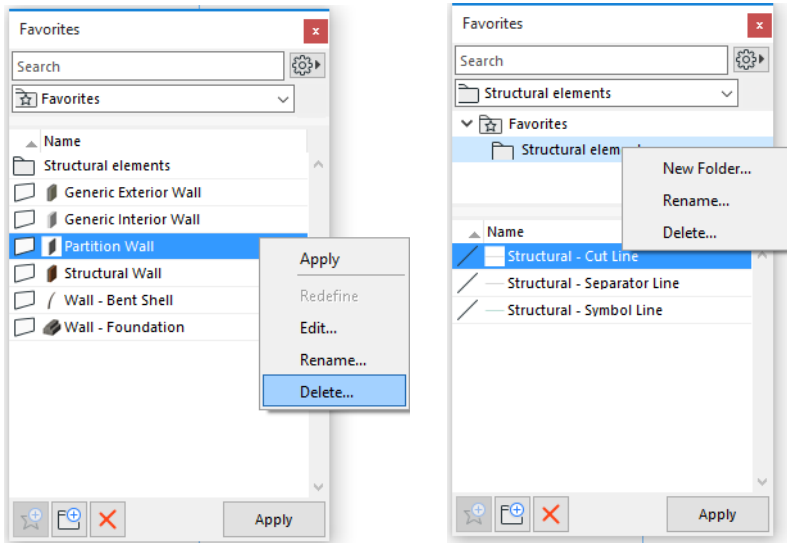
- **Drag and drop** Favorites or folders from the list into any folder. (Press Alt/Ctrl to drag a copy.)



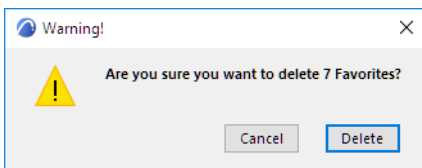
Rename or Delete a Favorite or Folder

1. From the Favorites Palette, select a Favorite, or a folder.
2. Use the context menu:
 - Rename
 - Delete (also available for multi-selection)

Delete is also available with the red X icon at the bottom of the palette.

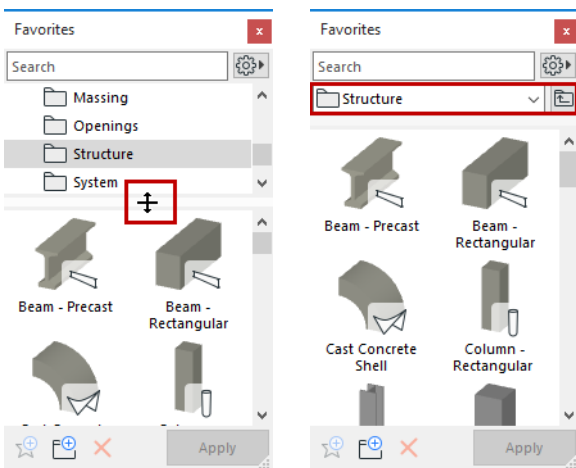


A warning appears, informing you of the number of Favorites being deleted, including Favorites that are not currently displayed (if display is limited to the currently active tool).



Resize Panels

Resize the two panels vertically as needed. Drag the splitter to the top to make the top panel disappear, or double-click the splitter to open/close panels:

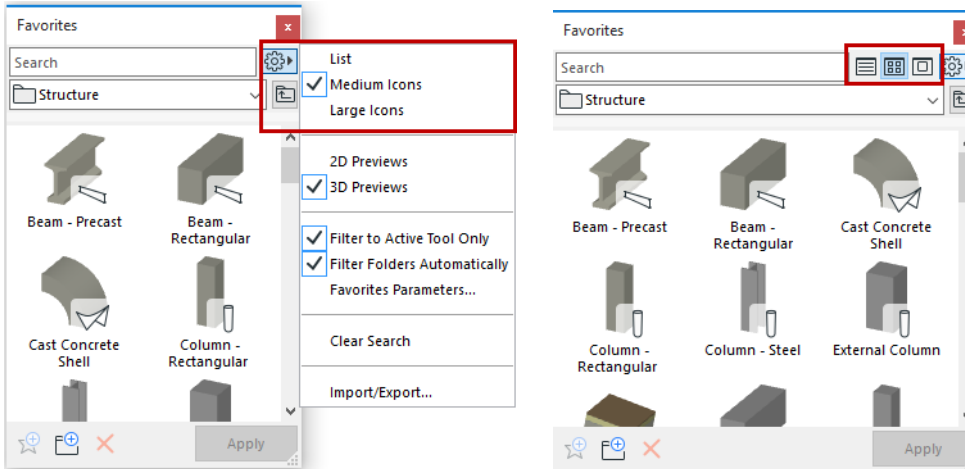


Display Preference

Choose **List View**, or **Medium** or **Large Icons**

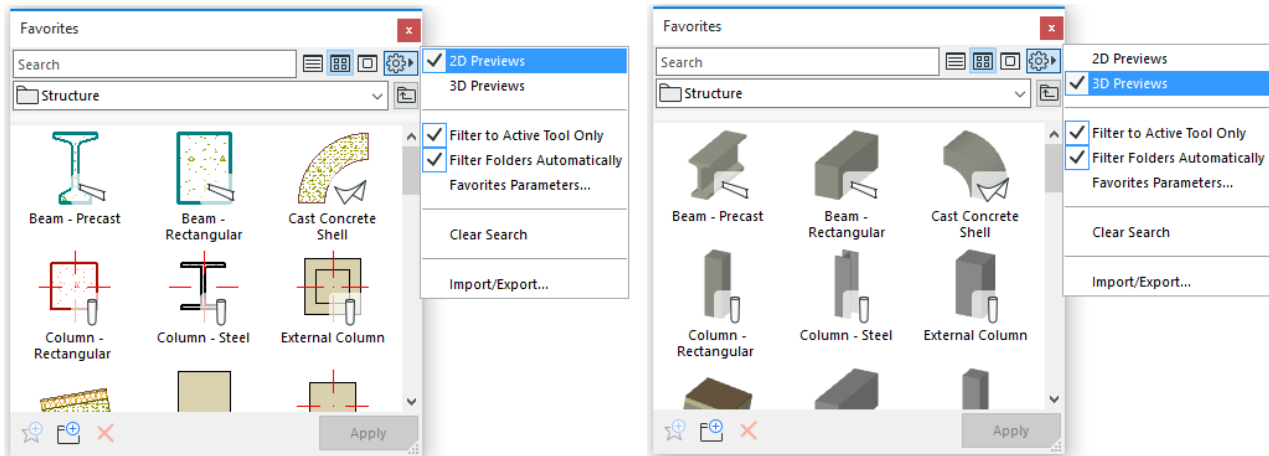
(Available from Additional Settings pop-up in every Favorites chooser interface.)

Expand the Palette horizontally to also access these options as buttons.



2D or 3D Previews

Use the Additional Settings pop-up (from any Favorites interface):



- To switch on the fly between the 3D and 2D preview of any Favorite, hover your cursor over the preview and click **Shift**.
- If no 3D content is available, then 2D Previews are used.

Notes:

- The 3D Previews are Hardware acceleration-based and automatically generated from generic elements. Some settings (e.g. Beam/Column/Wall slant angle) are ignored to improve clarity.
- To see contours on the 3D Preview, make sure the Work Environment option is checked: Show Contours on Favorites and Library Part Preview. (Options > Work Environment > On-Screen Options).

See also [Previews and Pictures in the Interactive Schedule](#).

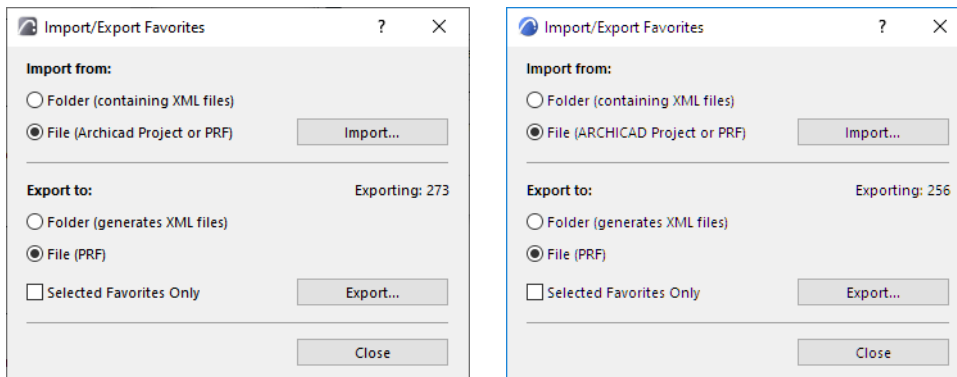
Search for Favorites

See [Search Favorites](#).

[Watch the Video](#)

Import/Export Favorites

Use the **Import/Export Favorites** dialog box, available from the Favorites Palette Additional Settings pop-up.

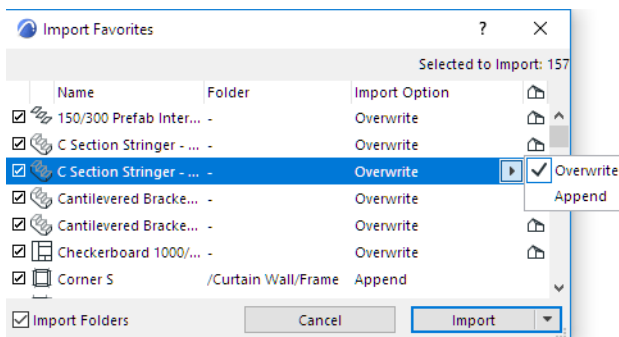


Importing Favorites

- If you choose **Folder**: browse for a folder containing Favorite XML files.
- If you choose **File**: browse for Favorites PRF file, or for any Archicad project (PLN Solo project, PLA archive, or TPL template) to import its Favorites.

Note: When importing from an earlier Archicad version, it is recommended to use an AC project as the source file. PRF and XML files from older-version Archicads may not be imported correctly.

The **Import Favorites** dialog box appears.



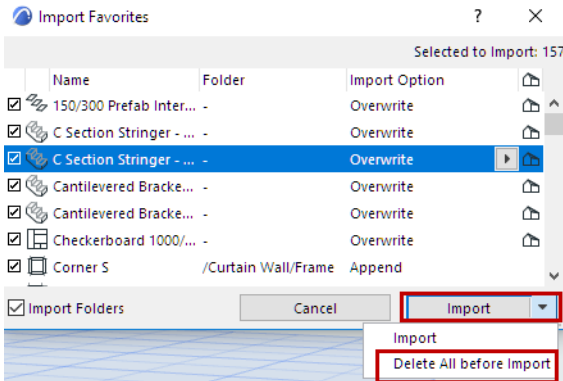
For general information, see [Favorites](#).

- For any Favorite you don't want to import, uncheck the box at the left. The "Selected to Import" number, shown at top right, is updated accordingly.
- Imported Favorites are compared (based on Favorite name) with the host project's list of Favorites.
 - If the imported Favorite does not exist in the project yet, it will be appended to the list of project Favorites.
 - If an imported Favorite's name already exists in the project, a house icon appears at the end of the row. You can either **Overwrite** the exiting Favorite with the imported Favorite, or **Append** the imported Favorite to the project's Favorites: in this case, the appended Favorite gets a name extension.
- Select multiple Favorites to check/uncheck or Overwrite/Append them simultaneously.

- **Import Folders:** Checked by default. If you uncheck it, then the import process will ignore any folder hierarchy within the imported file, and the imported Favorites will simply be added to the top of the project's Favorite list.

Click **Import**.

- **Delete Before Import...:** Under the Import button, optionally choose **Delete all before Import** to erase all Favorites and folders in the host project before a clean import of Favorites (as selected in the Import Favorites dialog). A warning appears before the delete.



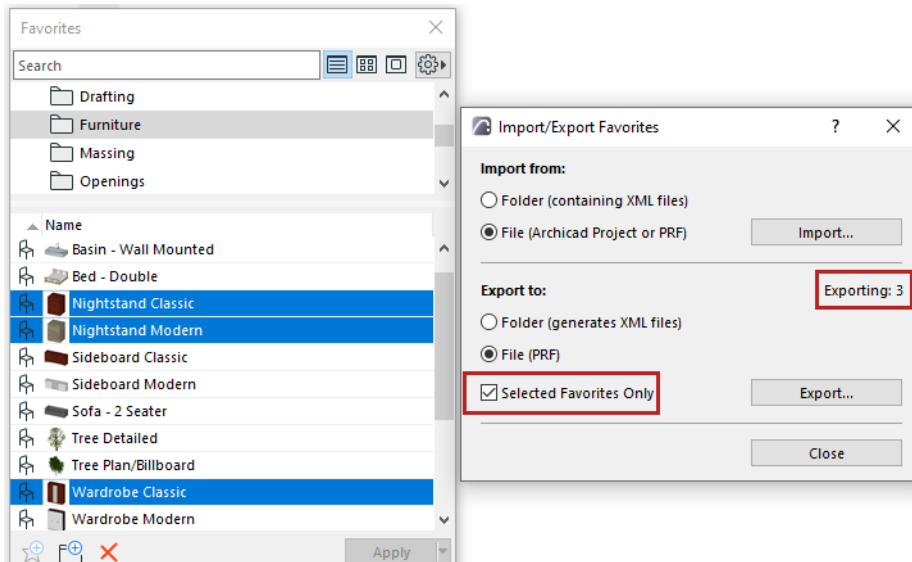
[Watch the Video](#)

Exporting Favorites

- If you choose **Folder:** a separate XML file will be created in this folder for each Favorite that you export
- If you choose **File:** a single binary file will be created out of the exported Favorites

Check **Selected Favorites only** to limit the export to the Favorites you have selected.

Note: Empty folders will not be exported.



[Watch the Video](#)

Licensing and sign-in

[Archicad license types](#)

[Sign in with Graphisoft ID](#)

[Log in to BIMcloud](#)

[License troubleshooting](#)

[License Information dialog](#)

[Using Archicad on multiple devices](#)

Archicad license types

This page explains the different types of Archicad Licenses, including Demo Mode.

[Commercial License - hardware or software key](#)

[Cloud license](#)

[Demo Mode](#)

[Education License](#)

Commercial License - hardware or software key

To run the commercial license of Archicad in full mode, you need either a hardware key (CodeMeter) or a software key.

If you purchased a single copy of Archicad, you received a “single license” key. If you purchased multiple licenses of Archicad, you received a “NET” key which holds all your licenses.

Full License

With a full license, you can use the fully functional Graphisoft product with unlimited usage. You will also be able to run older versions with some technical limitations.

For more details, see [Archicad Protection Key Compatibility](#).

Rental License (Selected Markets Only)

With the rental option, you can buy a license for a certain time period, such as one month. During the purchased time period, you can use Archicad without any limitation - e.g. running multiple sessions all day. The service period starts on the order date (the user gets 5 extra days that cover the delivery time of the physical key).

When starting up Archicad, a dialog box will let you know how many days remain available on your rental license. You can also check this number at **Help > License information**.

Cloud license

A cloud-based license is connected to a user’s Graphisoft account, so it works only if the user has signed in.

You can use a cloud-based license if

- your company has purchased a cloud-type subscription from the Graphisoft Store (selected markets only)
- you have a Graphisoft account, and you have signed in
- your company administrator has assigned a license to you

After purchasing a subscription, your company administrator assigns you a license using the cloud-based license site.

A cloud license can only be assigned to a user (not a group or team).

When you start Archicad, you are prompted to sign in to your account with your Graphisoft ID. Your license is then activated, and you can start working.

Make sure you sign in with the right account and the right credentials, otherwise you might not get your license.

To release your license, you must sign out of your account.

See also [Sign in with Graphisoft ID](#).

Demo Mode

When you start Archicad, the program automatically checks to see if you have a license available. If you don't have a license at all, you can continue in Demo Mode.

Demo Mode gives you full functionality except file saving and Teamwork. Use Demo mode to open, view and try out functions using any Archicad file.

The unavailable commands and menu items are disabled, and the Archicad title bar changes to

Archicad Demo.

Your Archicad program may also be restricted to Demo mode for the following reasons:

- no key is attached to your computer
- all of your Archicad licenses are in use
- your hardware protection key is defective
- CodeMeter service is not running

Education License

The Education license is available if you are a student, teacher or contact person at an educational institution.

A serial number can be obtained by registering at www.myarchicad.com. The first step of registration gives you a temporary serial number and a download link. This temporary serial number is valid for 30 days.

After you register, the credentials you provided go through an approval stage, after which Graphisoft's local representative sends you a final serial number.

Educational Serial numbers are issued for a one-year period. Once it expires, you can apply for yearly extensions until the end of your studies.

File Format and Compatibility

- The Archicad Education Version cannot be converted to a commercial version, because it is designed for use by students or for educational purposes only. The Education Version cannot be used for commercial purposes.
- EDU and commercial project files have the same file format (.PLN), but EDU files have an irremovable watermark at the top right corner. This watermark will be present on all printouts.
- EDU project files (including shared Teamwork projects) can be opened with a commercial version of Archicad. In this case, Archicad switches to EDU mode and the watermark will be present.
- Commercial project files can be opened with an EDU version of Archicad. The watermark is added, and cannot be removed.

Note: Teamwork Projects shared in the commercial version *cannot* be joined by users having the Education version.

- Apart from 'Open', no other data exchange (e.g. copy, paste, merge, hotlink) is allowed between EDU and commercial project files.

- If your serial number is that of an individual student rather than an educational institution, you will not have the option to create a custom Installation Package.

	Time Limit?	Save files?	Open file?	Teamwork?
DEMO	None	No	Open and view any full-version Archicad file	No
EDU	Initially 30 days (temporary serial number). After approval: one-year serial number (renewable)	Save as EDU version only: Permanent watermark appears on all output	Open commercial file with EDU (watermark appears) Open EDU file with commercial (watermark appears)	Yes, for Teamwork projects shared in EDU

[Read More](#)

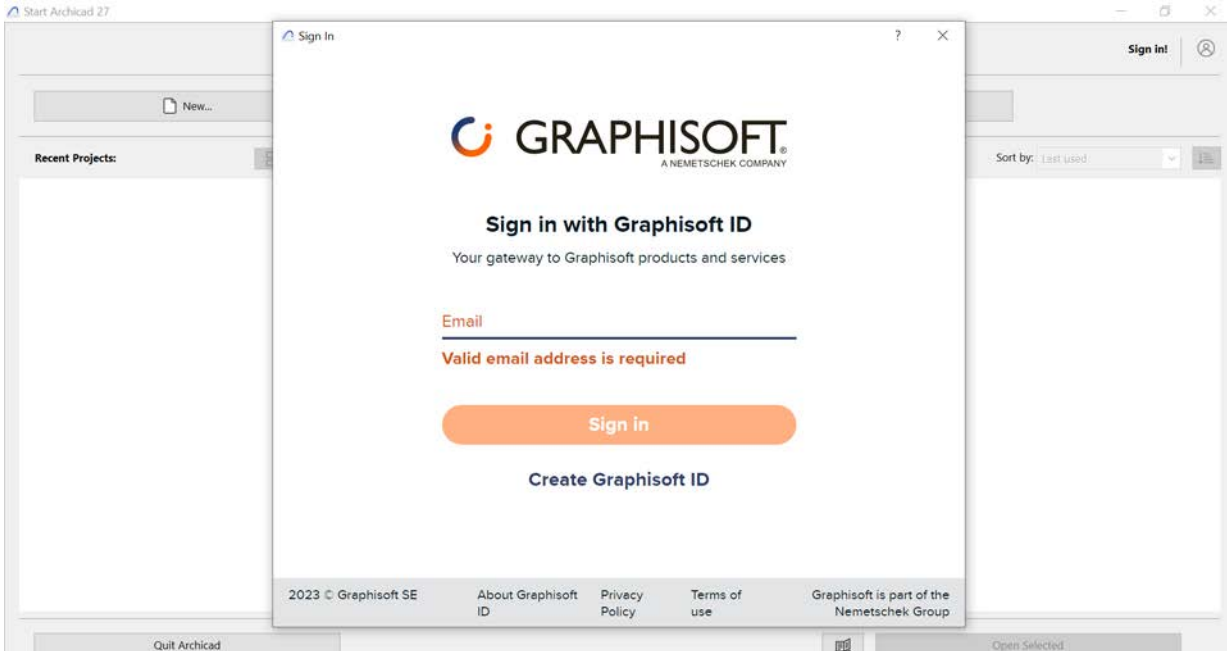
Related topics

[License troubleshooting](#)

Sign in with Graphisoft ID

The Start Archicad dialog prompts you to sign in with your Graphisoft ID.

If you don't have a Graphisoft ID, click **Create Graphisoft ID**.



By signing in, you help Graphisoft provide support and content that are better tailored to your specific needs, so you can get the most out of Archicad.

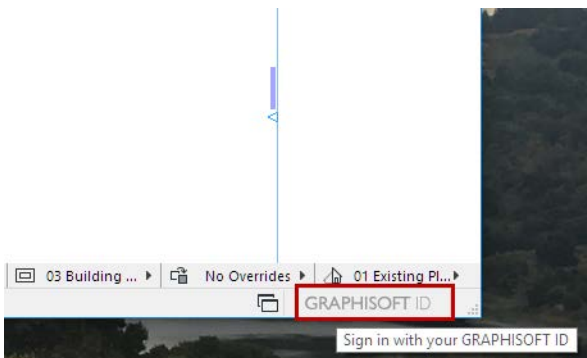
For more details, follow the link on the left side of the startup screen.

You can skip Graphisoft ID sign-in, but after the grace period expires, signing in is required in order to use the full version of Archicad. Otherwise, you can use Demo mode, then sign in later when you want to use Archicad's full version.

Other Ways to Sign In or Out with Graphisoft ID

In a running Archicad, do one of the following:

- Use the Archicad Help menu
- Right-click Graphisoft ID in the Status Bar (bottom right of your screen)



- Use the command in the **Select License Types** dialog. (This dialog appears during Archicad Startup if the required license is missing.)

Other Functions Requiring GSID

- when publishing a BIMx Hyper-model to the BIMx Model Transfer site
- when managing your Graphisoft licenses using License Manager
- when activating BIMcloud Basic
- when signing in as administrator on BIMcloud as a Service

Related topics

[License troubleshooting](#)

[Using Archicad on multiple devices](#)

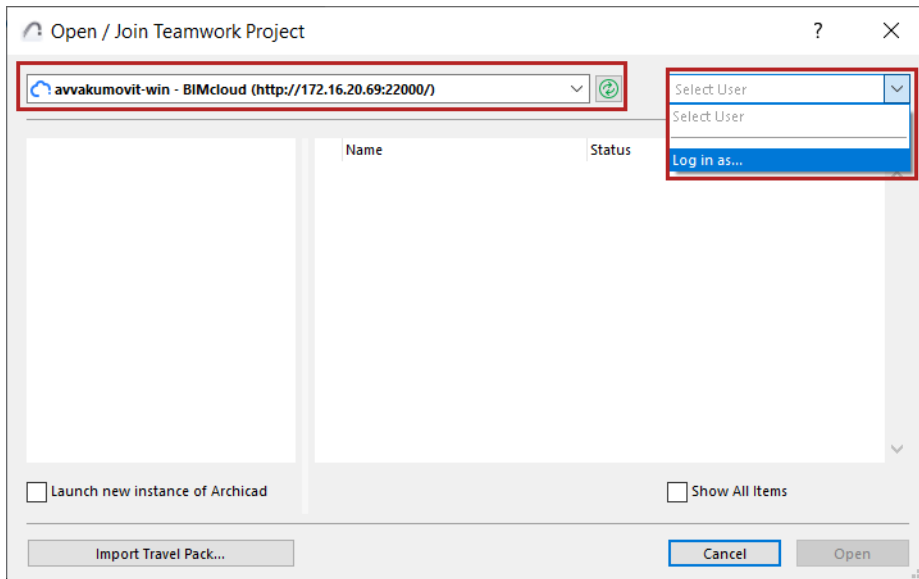
Log in to BIMcloud

The first time you access a particular BIMcloud server from Archicad, you have to log in to the selected BIMcloud.

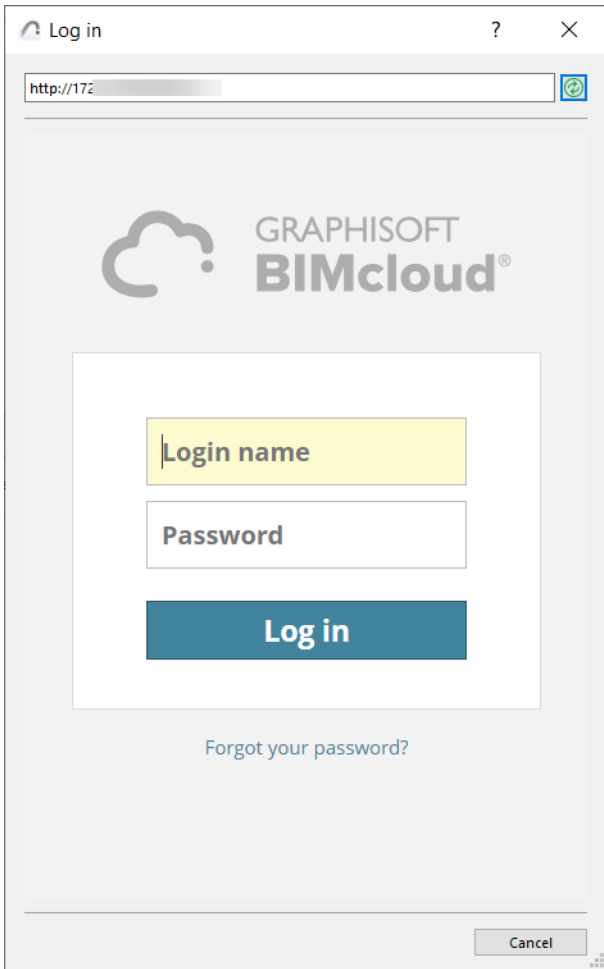
BIMcloud access is integrated in many Archicad functions. For example:

- To share or join a Teamwork project
- To open or merge an external file from BIMcloud (such as an external Drawing)
- To save, export, or publish an item to BIMcloud

In these dialogs, choose a BIMcloud server and a user. For example, in the Join Teamwork dialog:

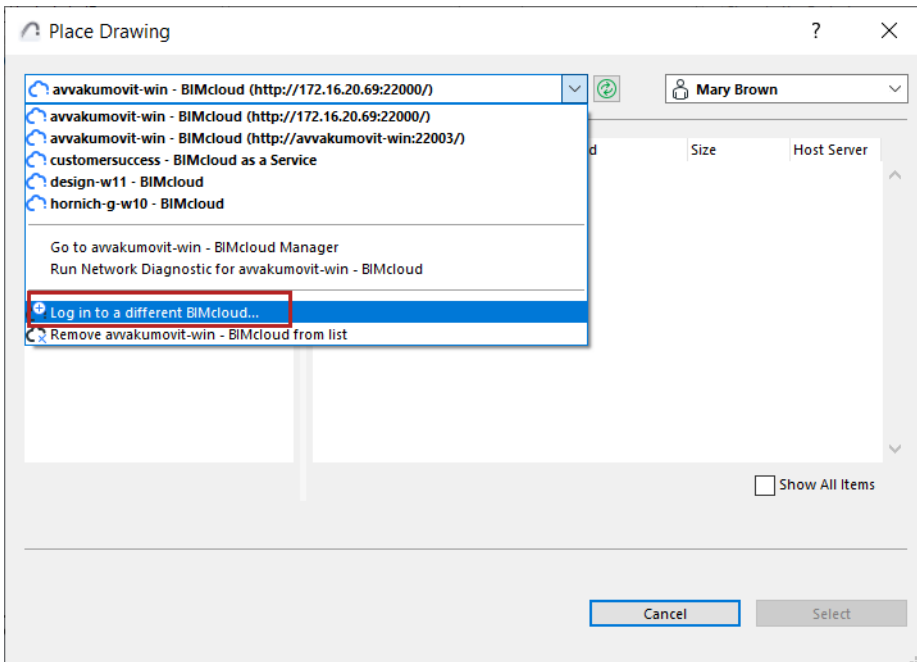


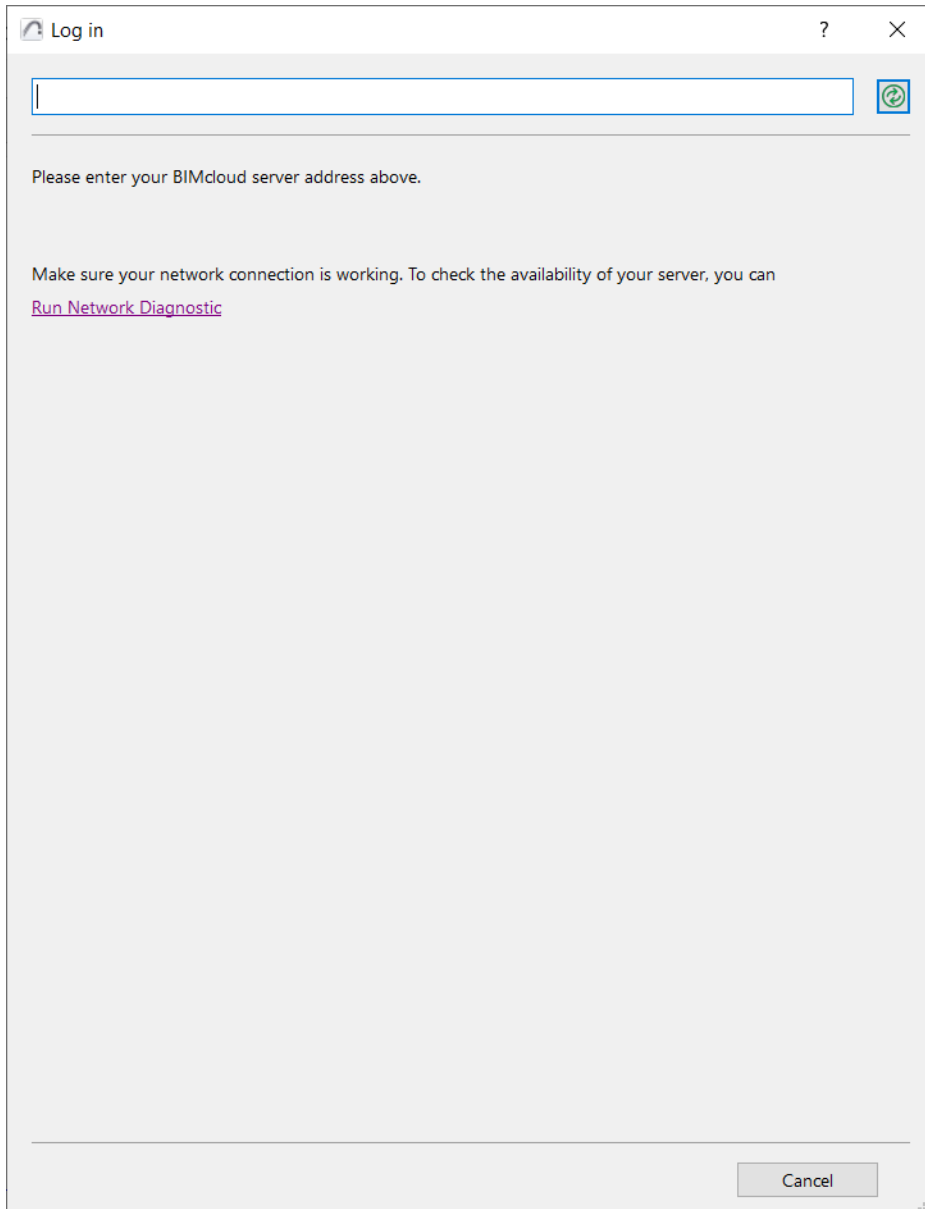
In the appearing dialog, use your BIMcloud user name and password to log in. This works if an administrator has already added you as a user to your company's BIMcloud.



Log in to a different BIMcloud

If you don't see the BIMcloud you need, you can add it with **Log in to a different BIMcloud**.





Enter the BIMcloud address as follows:

“http://<DNS name>:<manager port>” or “http://<IP address>:<manager port>”

- You can also see the BIMcloud address on the BIMcloud Manager’s landing page. Consult your server administrator.
- Depending on the server type, the address may begin with **https**:
- The manager port is the one selected during BIMcloud Manager Installation, e.g.: 1228.

If there is a problem with the server connection, you will get feedback on the issue.

Click **Run Network Diagnostic** to open a dialog box with details and help in solving the problem.

[See *Network Diagnostic Information \(BIMcloud\)*.](#)

License troubleshooting

Please click the topic that describes your issue:

[No license found - when starting Archicad](#)

[No license found - while using Archicad](#)

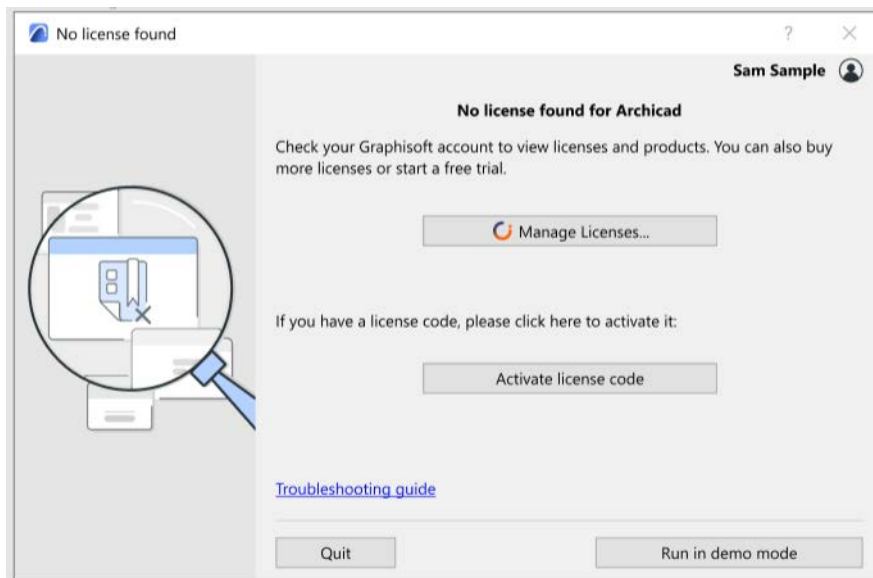
[Problem with serial number](#)

See also

[Archicad license types](#)

No license found - when starting Archicad

This dialog box appears when you start Archicad and you are signed in, but your license is not available.



Why don't I have a license?

Possible reasons:

- Your Company account administrator has not assigned a license to your Graphisoft account
- You had an assigned license, but it expired (and your contract manager did not renew it, e.g. due to issues with the payment)
- You had an assigned license, but your contract manager canceled the subscription
- You signed in to a Graphisoft account with the wrong Graphisoft ID, and that account does not have an assigned license

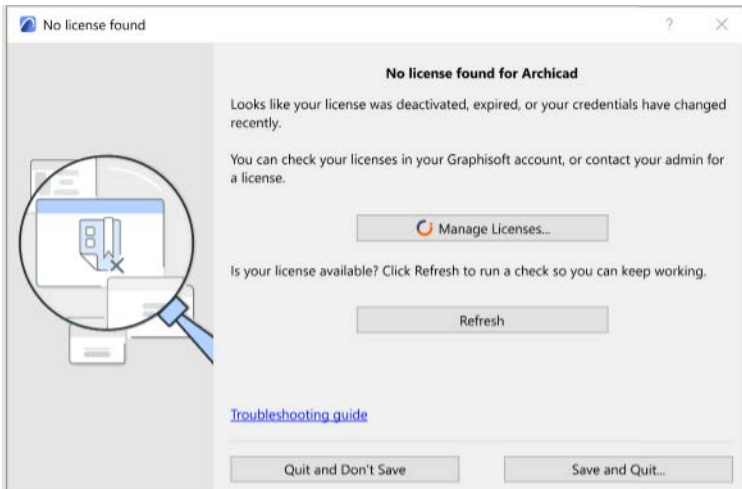
What should I do now?

One of the following:

- Click **Manage Licenses** to check you license status in your Graphisoft account, and try to fix it.
- Ask your Graphisoft account administrator or contract manager to assign or buy a license for you
- If you have an EDU or Emergency license, click **Activate license code** to enter it.
- Click the button to **Run in demo mode**

No license found - while using Archicad

This dialog appears if you are working in Archicad with a cloud license, but then something happens so your license is no longer available.



Why is my license missing?

Possible reasons:

- You had an assigned license, but your administrator revoked it and/or assigned it to another user
- You had an assigned license, but it expired (and your contract manager did not renew it, e.g. due to issues with the payment)
- You had an assigned license, but your contract manager canceled the subscription

What should I do now?

One of the following:

- Click **Manage Licenses** to check you license status in your Graphisoft account, and try to fix it. Then return to this dialog and click **Refresh**.
- Ask your Graphisoft account administrator or contract manager to assign or buy a license for you
- If you cannot solve the problem here, choose either **Save and Quit**, or **Quit and Don't Save**.

[See also Archicad license types.](#)

Problem with serial number

To use Archicad in Educational mode, you must enter an email address (User ID) & Serial number.

Get a Serial number by registering an account on myarchicad.com.

When you first launch Archicad, you are prompted to provide your email address (User ID) & Serial number. Archicad checks if your entered data match with the MyArchicad database.

If it detects an issue, Archicad displays an error message.

The following points outline the issues that can possibly occur, and explains solutions to each.

What's Covered

Serial Number is Invalid

Serial Number has Expired

Serial Number Does Not Match the Email Address

Serial Number is Not Valid for the Educational Version

Serial Number is Valid for Another Language of Archicad

Miscellaneous Issues

Serial Number is Invalid

It often happens that the Serial number contains characters that are visually similar, like 1-l-l, or O-o-0. Make sure that you have typed the exact same characters as you can see on your MyArchicad profile. A good way of eliminating this issue is to copy & paste the Serial number from your MyArchicad profile page to the Archicad dialog.

Serial Number has Expired

Educational Serial numbers are issued for a one-year period. Once it expires, you can apply for yearly extensions until the end of your studies.

Serial Number Does Not Match the Email Address

The entered Serial number is valid, but it is matched to a different email address/username than what is provided in the User ID field. Check your MyArchicad profile to see which User ID belongs to this Serial number, then try again.

Serial Number is Not Valid for the Educational Version

If you are an Educational user, you must obtain an Educational license. Re-register as a Student, as a Teacher, or as a School at MyArchicad.com, using a different email address.

Serial Number is Valid for Another Language of Archicad

The entered Serial number is valid, but it has been issued for a different language version of Archicad than the one you are currently running. Go to your MyArchicad.com page and check which language version of Archicad you applied for. Make sure you use the Serial number for the language version of your installed Archicad.

If you would like to use a different language version of Archicad, you need to change your Country on the My Account tab of your profile.

Miscellaneous Issues

If none of the above tips could resolve your issue, please contact our MyArchicad support at: myarchicad@graphisoft.com

In your email please send a screen capture of the issue you are experiencing and describe the steps you have taken so far to resolve this problem.

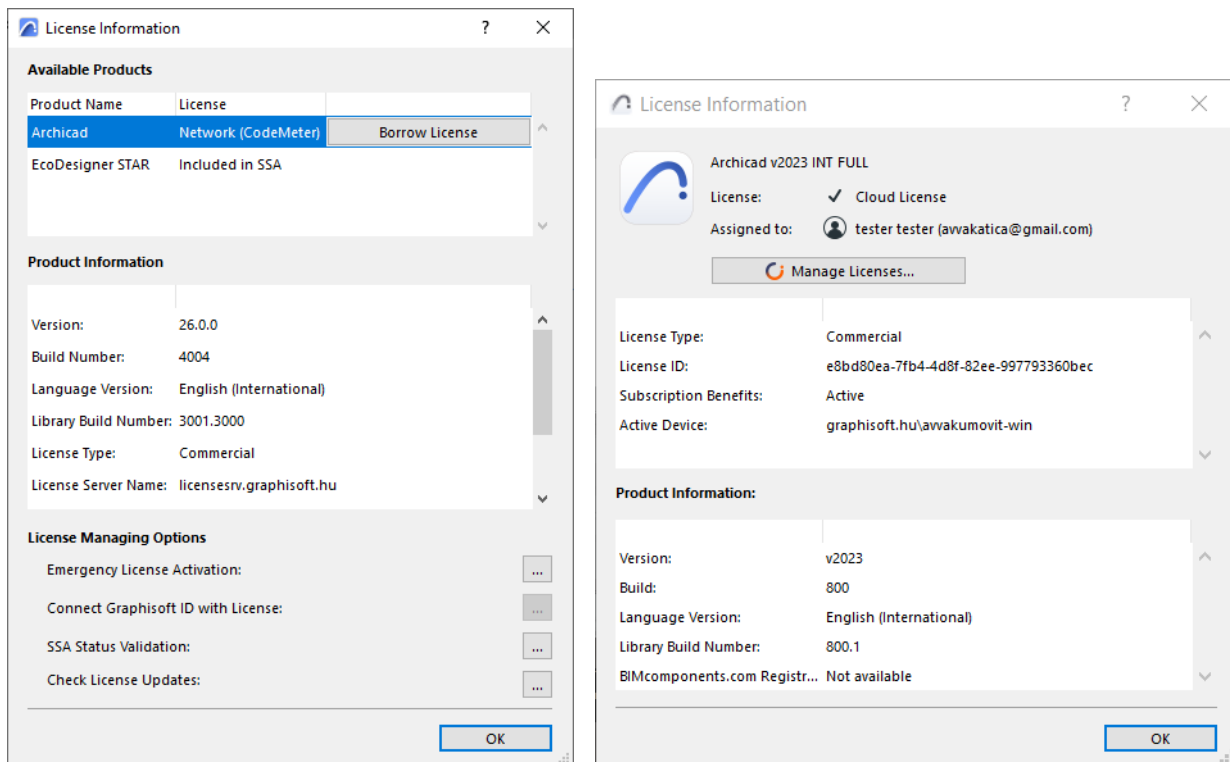
License Information dialog

Use the **License Information** dialog box to see license-related information for Archicad-related products available to you.

Do one of the following:

- Click on **Help > License Information**
- From the **Help > About Archicad** Dialog, click on License Information
- From the Action Center (**File > Info > Action Center**), select the Licenses item and click **License Information**

Available options depend on your license configuration. For cloud licenses, all license management functions are available on the Graphisoft account site (click **Manage Licenses**.)



License Managing Options (for non-cloud licenses)

See the following:

[License Borrowing \(Legacy feature\)](#)

[Connect Graphisoft ID with License](#)

[Emergency License Activation \(Not Available in All Markets\)](#)

[SSA Status Validation](#)

License Borrowing (Legacy feature)

Note: The license borrowing feature has been discontinued. But if you used it in the past, it remains available to you.

The License Borrowing feature allows individual users to “borrow” a license key temporarily if they will be going offsite, to enable use of the software even without being connected to the network.

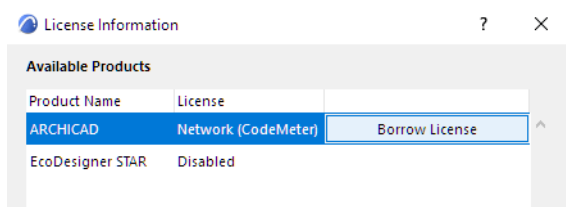
License borrowing is available if:

- Archicad is running on your machine with a license from a CodeMeter NET key (either hardware or software key)
- The CodeMeter NET key contains borrowable licenses to use with Archicad and/or other Graphisoft products

[For more information on CodeMeter NET, see this Community Site article.](#)

In any case, borrowing is available only if the selected product's License is "Network (CodeMeter)".

Click **Borrow License**.



If the process is successful, the program lets you know that you have borrowed a license for the product, with a particular expiration date.

In the License Information dialog box:

- The selected product's License information is listed as either "Borrowed (CodeMeter Stick)" or "Borrowed (License file)", depending on the form of protection used. (This can vary by country.)
- The selected product is now shown with a **Return** button: click here when you are ready to return the license. The expiration date of the borrowed license is shown in the Product Information section.
- If your license expires before you return it, the program will automatically return it for you, even if you are not online.

Emergency License Activation (Not Available in All Markets)

Enter the Activation Code to enable the Emergency License on a computer that is currently running Archicad in Demo mode.

This feature is relevant only if you have obtained an Emergency License (a temporary solution, for licensed users, if Archicad is launched with no Protection Key available). In this case, an Activation Code is issued via internet.

[For more information, contact your distributor, or see this Community Site article.](#)

Connect Graphisoft ID with License

Use this optional function to connect your personal Graphisoft ID to your Graphisoft ID company, based on the protection key running Archicad.

Once the personal Graphisoft ID is connected, users can access SSA-exclusive web pages.

[For more information, contact your distributor, or see this Community Site article.](#)

SSA Status Validation

This function is relevant only for Archicad users who have a Software Service Agreement (SSA).

Background: SSA's are available according to terms and conditions set by your local distributor. Some Archicad-related Add-ons can be used with full functionality only if you have a valid SSA.

Archicad *automatically* validates the status of your SSA at program startup if you are online. If the SSA status is up to date, your installed Add-ons (those which require an SSA) will function, and you don't have to do anything else.

When to Use this Function

If you install an Add-on or obtain a new SSA *after* starting Archicad, you must run the SSA status validation manually:

Click **Validate SSA Status**.

- If you are online: Archicad will validate your SSA status, and the affected Add-ons will now function.
- If you are offline: You are directed to access and install the needed SSA validation files manually.

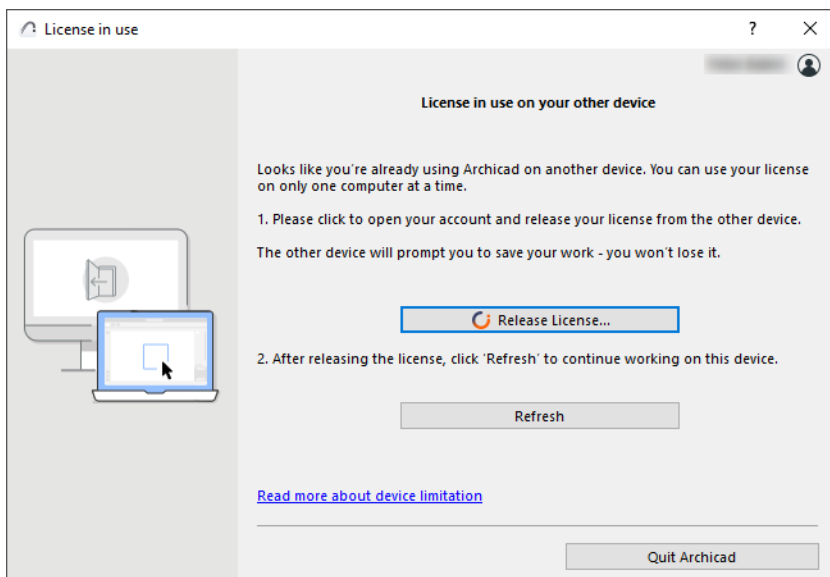
Using Archicad on multiple devices

Note: This workflow applies to Archicad users who have the new “Cloud license”. The cloud license is available with some subscriptions purchased from the Graphisoft Store. See also [Archicad license types](#).

Your Archicad cloud license is not tied to any specific device - but you can only use the license on one device at a time. When you are finished working on one device, **sign out**: this will free up your license for the other device.

If you stay signed in on the first device and start Archicad on a second device, you can't continue until you release the license from the first device. You can do this remotely - no need to physically access the first device.

When you try to start Archicad on the second device, you get this dialog. Click **Release License**.



This takes you to your Graphisoft account site. Use the Licenses page to release the license from the first device. Then you can use your license on the other device.

Attributes

Archicad attributes are groups of defined settings available to your project.

Edit and manage attributes in a dedicated dialog for each attribute type.

Layers

Lines

Fills

Building Materials

Composite Structures

Pen Sets

Surfaces

Zone Categories

MEP Systems

Operation Profiles (Energy Evaluation)

Profile Manager

In addition, the **Attributes palette** gives you direct, easy access to any attribute and its settings. In the Attributes palette, you can also import and save sets of attributes.

Note: In addition to the Attributes palette, the Attribute Manager dialog box (familiar from earlier Archicad versions) is available. In more complex import/export workflows, you might prefer its two-panel interface to compare two lists of attributes. [See *Attribute Manager*](#).

Related Topics

[Using the attribute dialogs](#)

[Attributes palette](#)

[Delete attributes](#)

[Missing attributes](#)

[Custom attributes of GDL objects](#)

[Attributes of XREF Files](#)

Layers

Layers are used to separate elements logically. Each tool has a default layer assignment: when you place an element using that tool, the new element is automatically placed on the corresponding layer.

An element can only belong to a single layer.

Note: Doors, Windows, Openings, Wall Ends and Corner Windows do not have separate layers; they are handled together with the layers of the wall into which they are placed. Cameras do not have layers.

Each Archicad project has a single layer set, managed in Layer Settings. However, the Layout book can have different settings than those used in Model views.

The Archicad Layer is a special layer that cannot be deleted, hidden or locked, since an Archicad project must always contain at least one layer. In case of a file error, any elements that may have lost their layer definitions will be placed on the Archicad Layer.

Related Topics

[Change element's layer](#)

[Layers dialog](#)

[Layer status](#)

[Layer Combinations](#)

[Quick Layers](#)

[Use layers to prevent intersections](#)

[Use separate layer settings for the Layout Book](#)

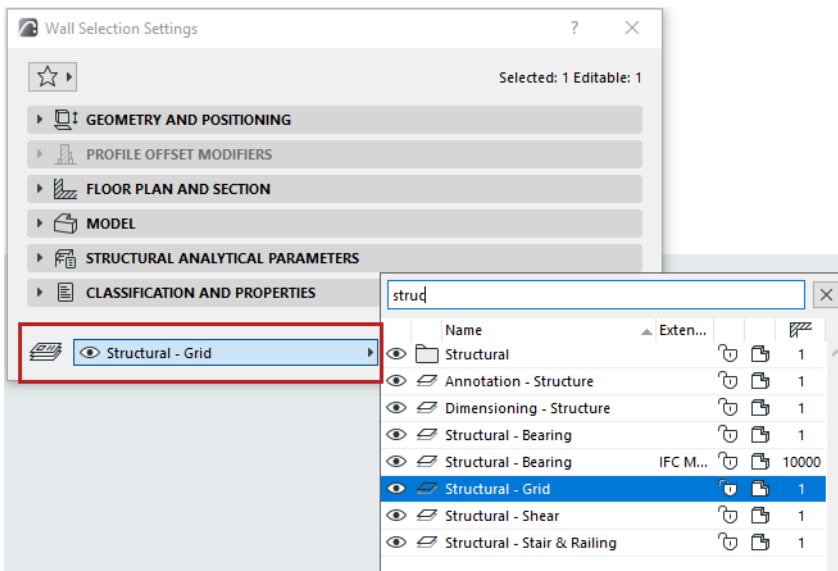
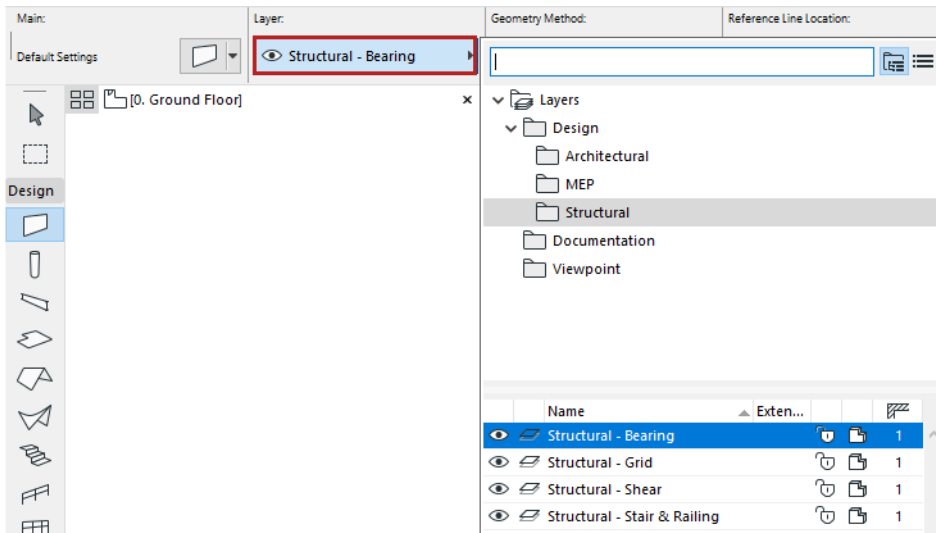
[Place all elements on a single "active layer" \(Simulate AutoCAD work methods\)](#)

For general information: see [Using the attribute dialogs](#).

Change element's layer

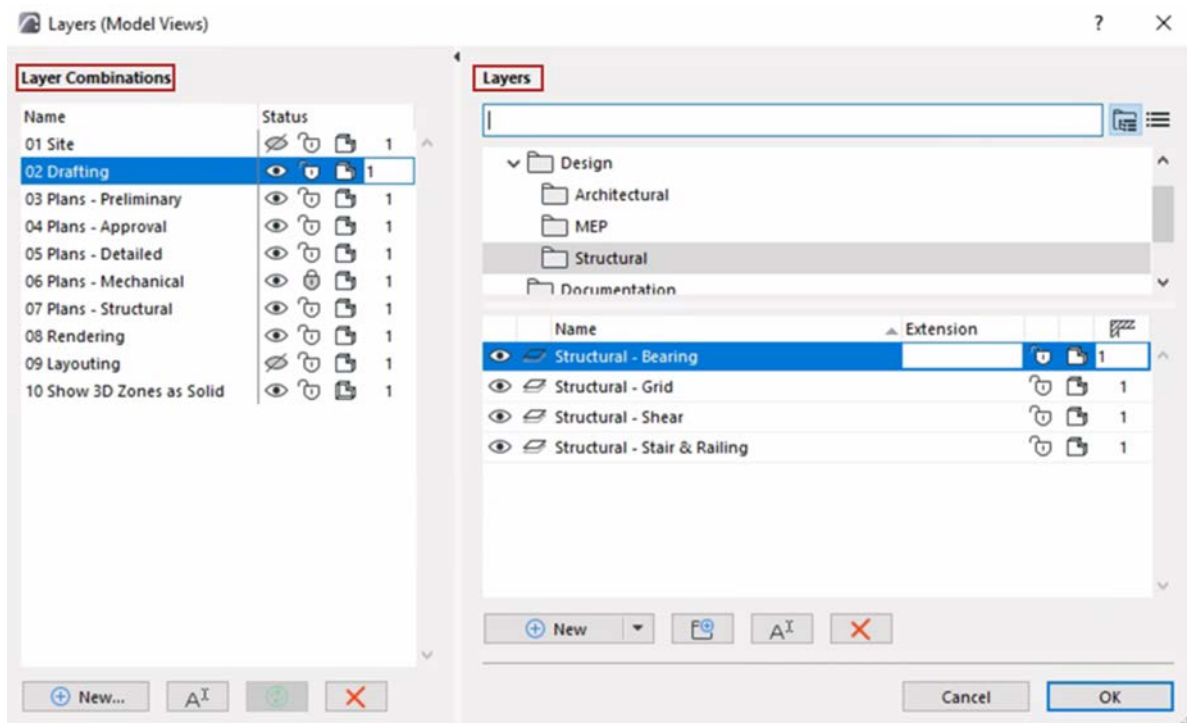
Each tool has a default layer assignment, defined in element settings. To change an element's layer:

1. Select the element
2. Open the layer picker drop-down from
 - the Info Box, or
 - the bottom of element Settings



3. Click the layer you need.

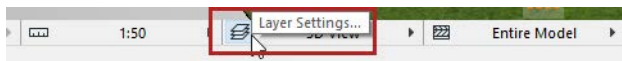
Layers dialog



For general information: see [Using the attribute dialogs](#).

To open the Layers dialog, do one of the following:

- shortcut: Ctrl+L
- Layer Settings icon in the Quick Options bar



- Options > Element Attributes > Layers
- Document > Layers > Layers
- Layers icon of Attributes toolbar (Windows > Toolbars > Attributes)



The left panel lists existing layer combinations. The right side lists all the layers defined in the project, in either list view or folder view.

Listed layers are displayed with icons indicating their status:

- Show/hide
- Lock/unlock
- 3D view mode

See [Layer status](#).

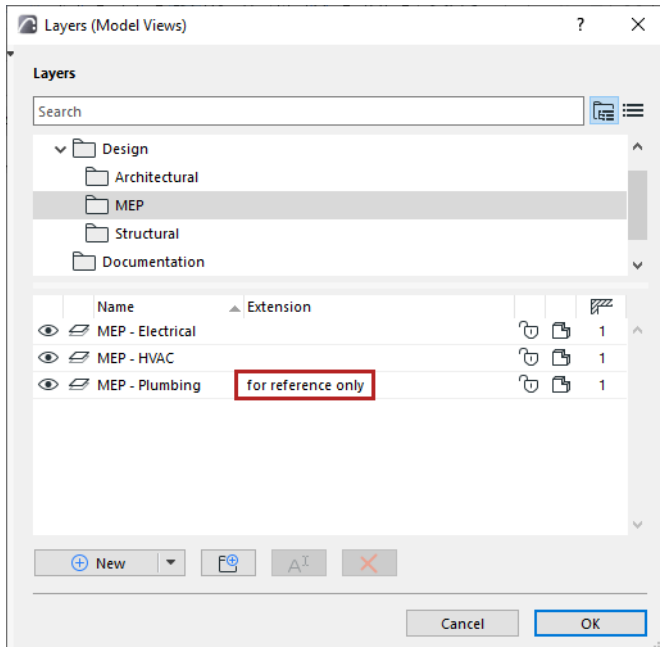
Intersection group number

For any layer: set an Intersection group number to control element intersections.

See [Use layers to prevent intersections](#).

Layer name extension

For any layer: add an optional **extension** to its name. This can act as an additional sorting control.



Layer status

For each layer, you can define the following statuses:

- lock/unlock
- show/hide
- 3D view mode
- layer intersection group

Throughout the Archicad interface, icons indicate each layer's status.

Edit these statuses in Layer Settings: select one or more layers or folders, and click the icons to change their status.

Note: The Archicad Layer cannot be deleted, hidden, or locked. This layer collects all those elements that may have lost their layer definitions.

Lock/unlock



Lock a layer to prevent its contents from accidental modification. Elements on a locked layer cannot be edited or deleted and no new elements can be placed on it.

The “mixed” status is shown on folders that contain items of varied status.

Show/hide



Set the layer's show or hide status. Elements on a hidden layer are not displayed.

The “mixed” status is shown on folders that contain items of varied status.

Show all layers

Do one of the following:

- Use **Document > Layers > Show All Layers** command or the same command from the **Arrange Elements** toolbar;
- In list view: select all the layers with Ctrl+A and toggle the eye icon
- In folder view: select the topmost “Layers” folder, multiselect everything in the list, then toggle the eye icon.

3D view mode



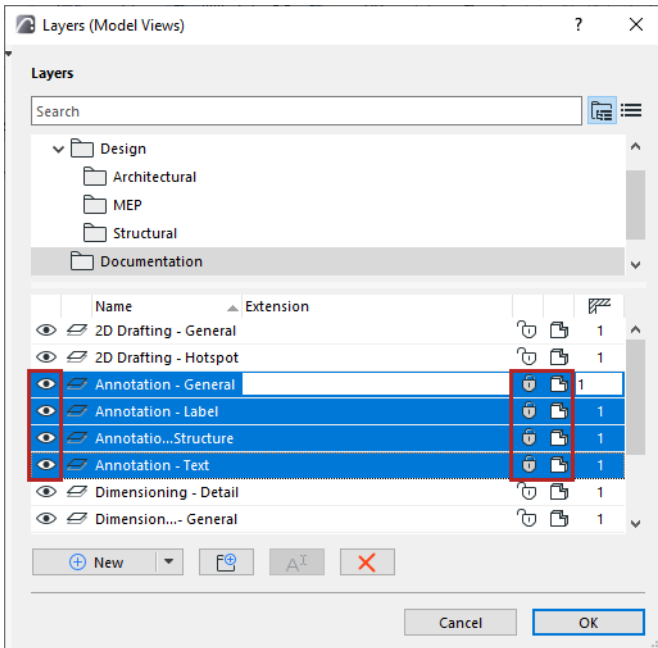
Choose Shaded vs. wireframe to display this layer's elements in 3D views.

For example, you can place operator elements of Solid Operations on wireframe layers, and their target elements on solid layers.

Note: This setting is independent of the current 3D Style in the **View > 3D View Options** menu.

Another way to change layer status: use the shortcuts in the Quick Layers palette.

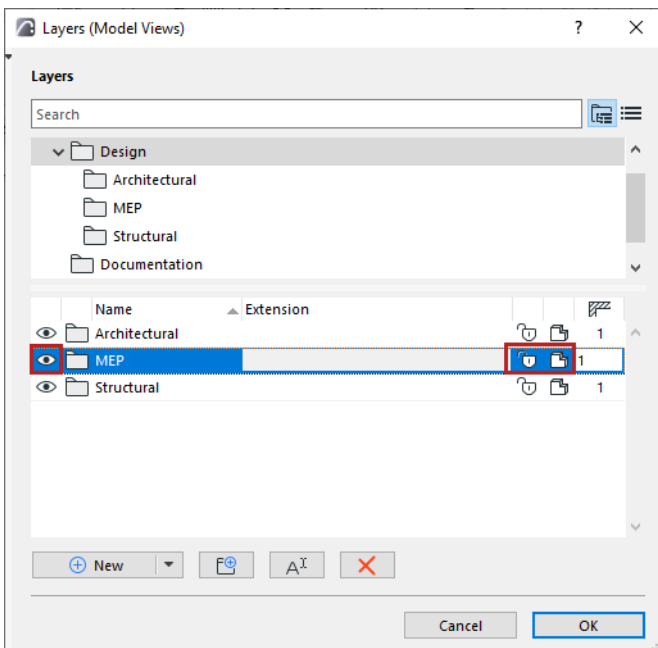
[See *Quick Layers*.](#)



Select and edit individual layers

Edit status of all layers in a folder

In folder view: you can select a sub-folder from the bottom list. This way, you can edit the status of all the layers inside that folder.



Select and edit layer status - by folder

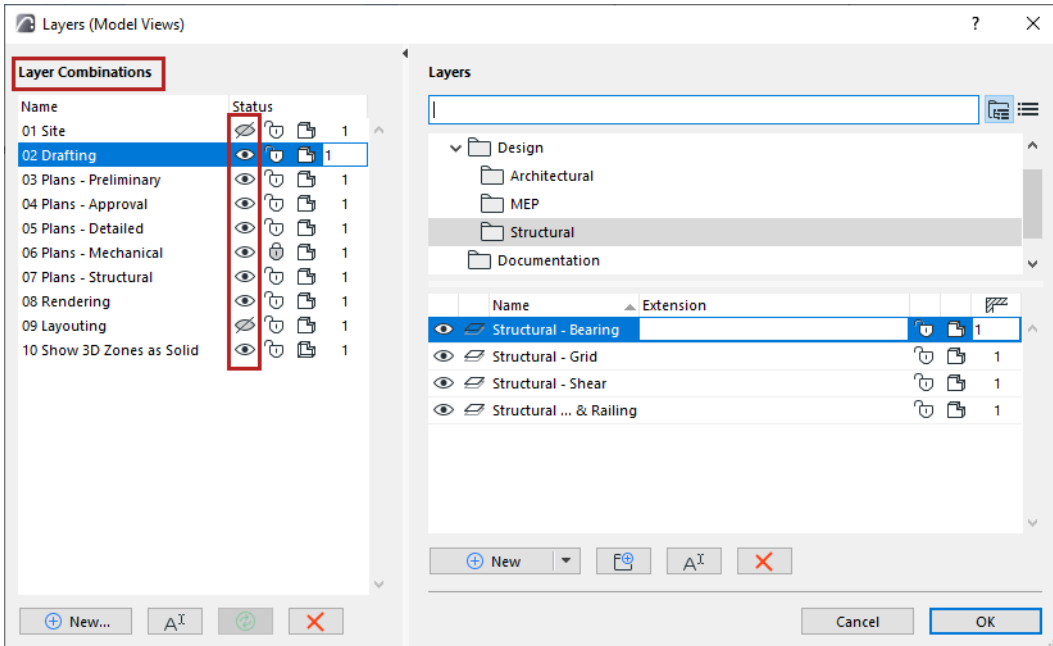
Layer Combinations

Layer Combinations are named sets of layers with varying statuses. Use these combinations to hide or lock layers that you don't need to see or edit in a view.

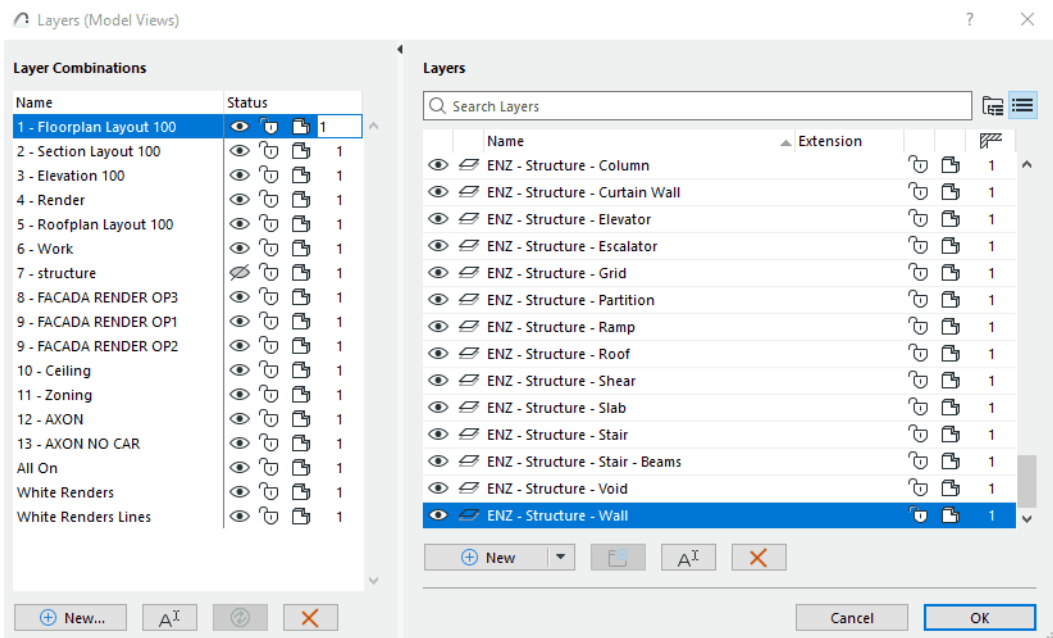
Each view is saved with a particular Layer Combination.

Archicad comes with a default set of Layer Combinations, listed on the left panel of Layer Settings. You can modify these and create new ones.

A given layer can have different statuses in different Layer Combinations. To check this, select any layer and see its status in each Layer Combination.



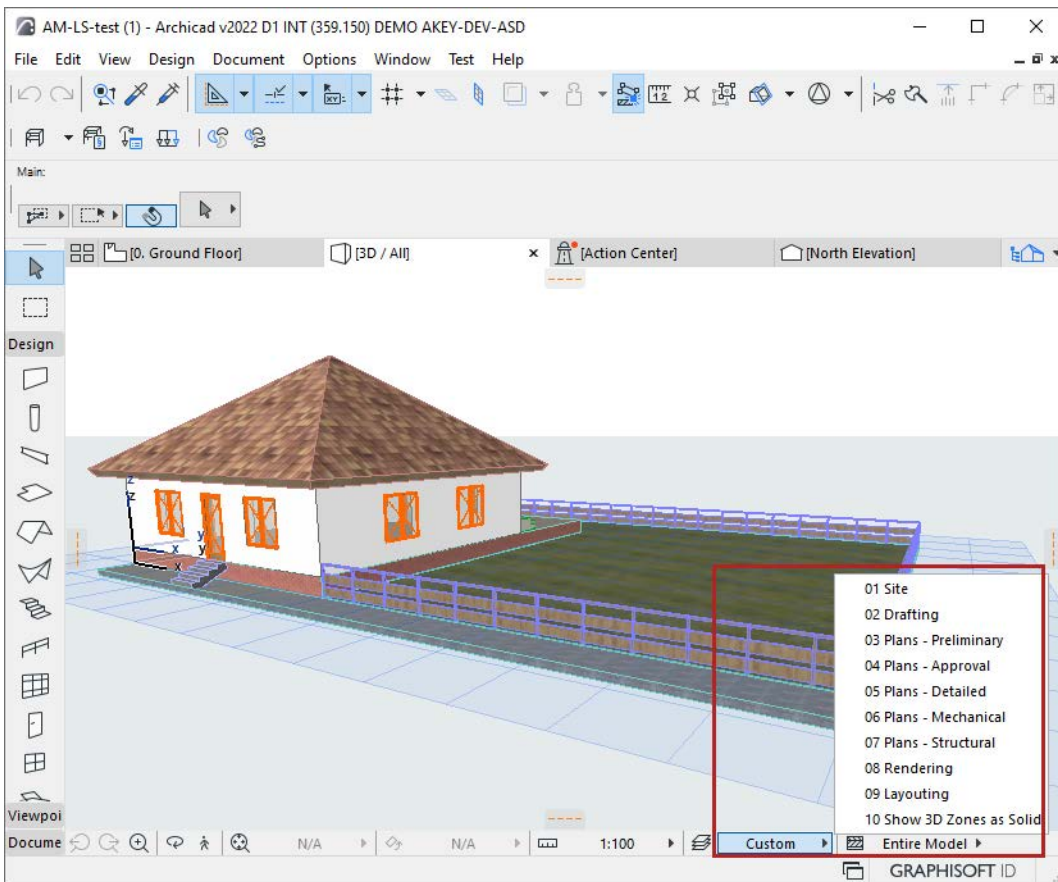
Note: You can customize the order of the columns to best fit your workflow.



Apply Layer Combination to current view

Do one of the following:

- Use the Layer Combination pop-up control from the Quick Options Bar at the bottom of the model window.



- Open the Layer Settings Dialog Box. Select a layer combination from the list in the left panel, then click OK
- Go to **Document > Layers** and click a listed layer combination.
- In View Settings, switch the Layer Combination.

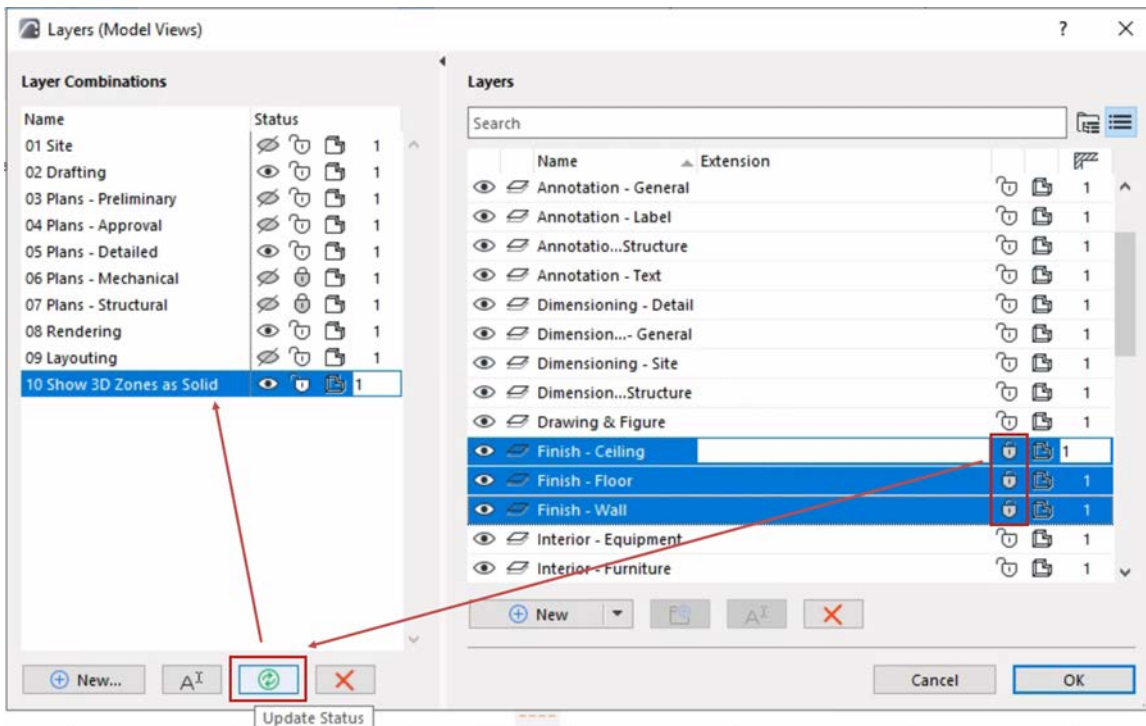
Edit Layer Combination

Do one of the following:

- Select Layers to edit.
- Change their status in any Layer Combination, using the Layer Combination's icons.

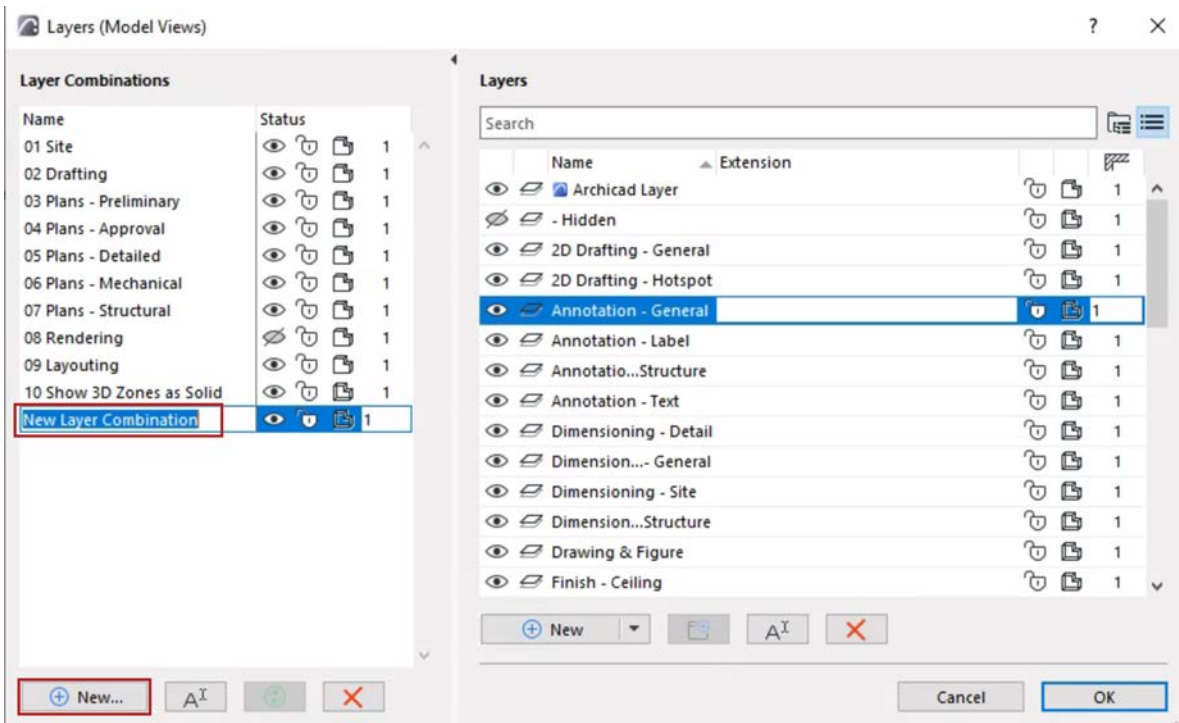
OR:

- From the list on the left: select one or more Layer Combinations to edit
- From the Layers list on the right: select one or more Layers
- As needed, change status of individual layers on the right (show or hide, lock, etc.)
- Click **Update**.



Create new Layer Combination

1. In Layer Settings: click New at the bottom of the Layer Combination list.
2. Enter a name.



3. In the Layers list, select layers and change their status as needed.
4. Click Update to apply these modifications to the new Layer Combination.

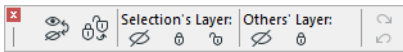
Quick Layers

Display this toolbar from:

Window > Palettes > Quick Layers

The Quick Layers toolbar lets you quickly change layer status without having to open Layer Settings - for selected elements' layers, or for all project layers.

The commands, from left to right:



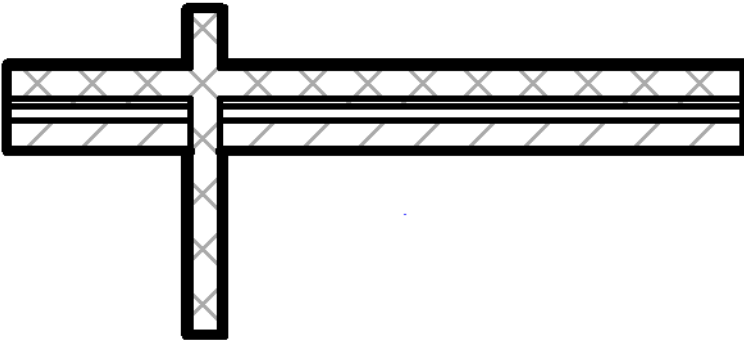
- **Show/Hide** toggle: Reverses the show/hide status of all layers in the project
- **Lock/Unlock** toggle: Reverses the lock/unlock status of all layers in the project
- **Hide Selection's Layer:** Hides layer of selected elements.
- **Lock Selection's Layers:** Locks layer of selected elements.
- **Unlock Selection's Layers:** Unlocks layer of selected elements.
- **Hide Others' Layers:** Hides the layer of the elements that are not selected. (If nothing is selected, all project layers will be hidden.)
- **Lock Others' Layers:** Locks the layer of the elements that are not selected. (If nothing is selected, all project layers will be locked.)
- **Undo/Redo Quick Layer Actions:** undo or redo Quick Layer actions (up to 10 actions).

Use layers to prevent intersections

Intersecting elements that belong to the *same* Layer Intersection Group will connect to each other, provided they are drawn correctly.

Exception: Two elements that are both set to intersection group 0 will NOT intersect.

By default, every layer is set to Group 1, so all intersecting elements will be joined according to the regular connection rules for Archicad.

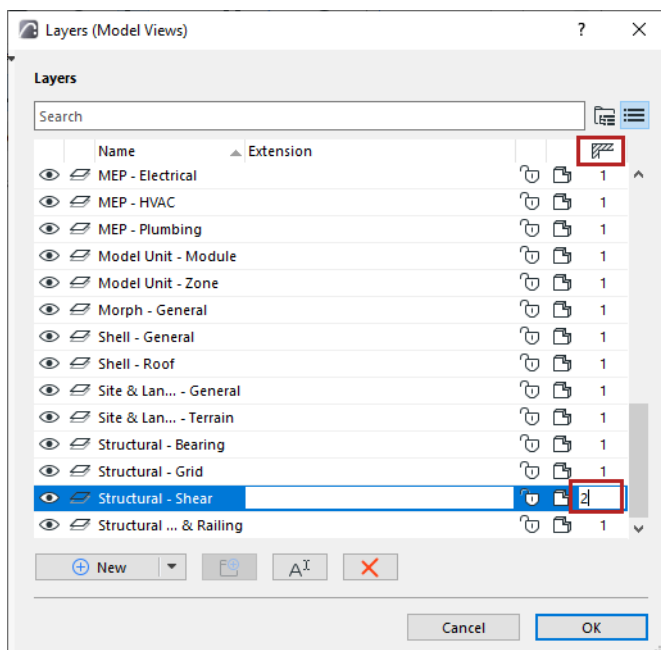


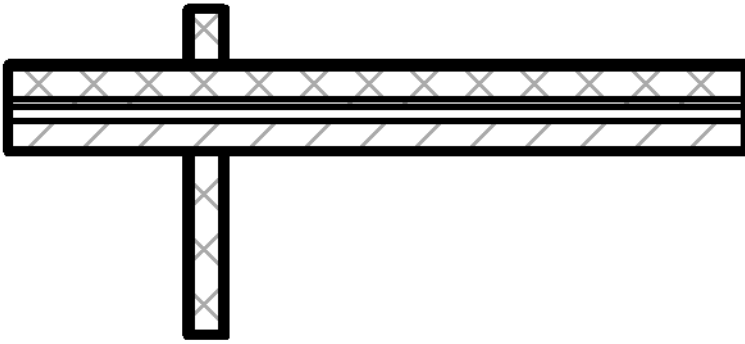
Wall Layers Have Identical Intersection Group Numbers: Walls Connect

Prevent element connections: Assign different layer intersection groups

You may want to prevent element connections - for example, to show the details of composite walls. In this case, place the intersecting walls on different layers, and then give one of these a layers a different Layer Intersection Group number. Elements on *different* Layer Intersection Groups will not connect.

1. Open Layer Settings (**Document > Layers > Layer Settings**)
2. Select the layer of one of the intersection elements.
3. Edit the value under the **intersection** icon.





Wall Layers Have Different Intersection Group Numbers: No Connections

Note on intersection with hidden layer: Elements having identical intersection groups will intersect even if one of the layers is hidden. This may result in “missing” lines, indicating an intersection with an element on a hidden layer. To avoid this, go to Layer Settings, select one of the layers, and assign it any different intersection group.

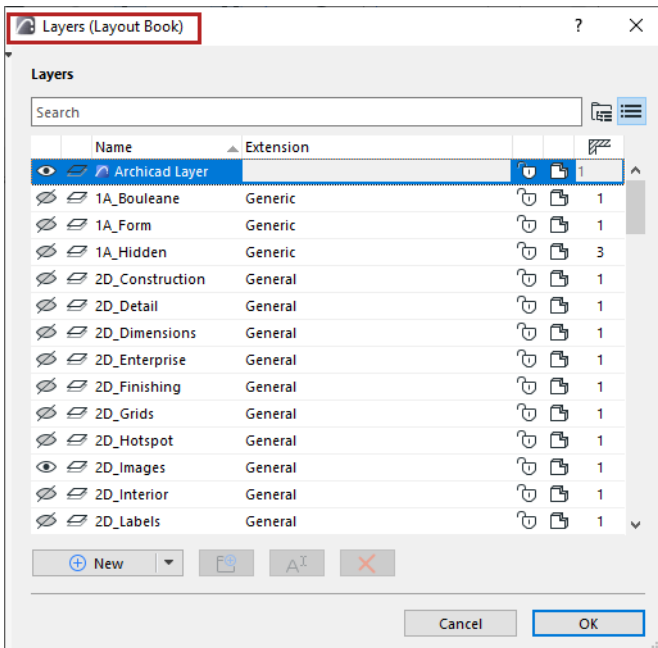
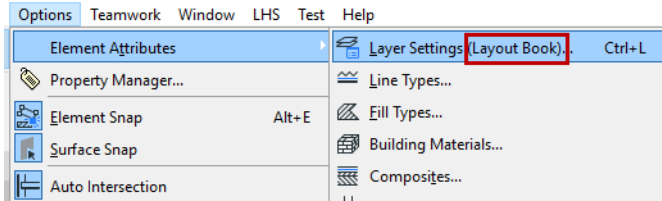
Note: Layer Intersection numbers have nothing to do with element priorities; they serve merely as tags for grouping elements.

Related Topic:

[Line Elimination Between Model Elements](#)

Use separate layer settings for the Layout Book

While each Archicad project has only one layer set, the settings of any particular layer (for example its show/hide or locked/unlocked status) can be different in the Layout Book and in Model Views.

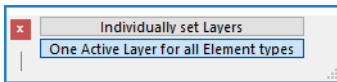


Important: Layer settings for the Layout Book serve to show/hide elements directly placed on layouts, such as lines, texts and drawings as a whole. Layout Book layers do not affect the drawing content, which is determined by the layers of its associated view.

Place all elements on a single “active layer” (Simulate AutoCAD work methods)

The **Active Layer** command assigns a single default layer to all tools, instead of using Archicad’s default layer set-up.

Choose **Document > Layers > Layer Extras > Active Layer** from the menu and choose the **One Active Layer for all Element Types** option from the appearing palette.



This will set the default layer of all element types to the layer of the *currently active tool* (regardless of any selection). All Tool Settings dialog boxes will now use the same default Layer definition and successive elements will all be placed on that layer, regardless of type.

You can override this setting manually for selected elements.

To return to the previous layer state, choose the **Individually Set Layers** option. The last set of manually defined layers will be used.

Lines

Lines are attributes, defined at Options > Element Attributes > Lines.

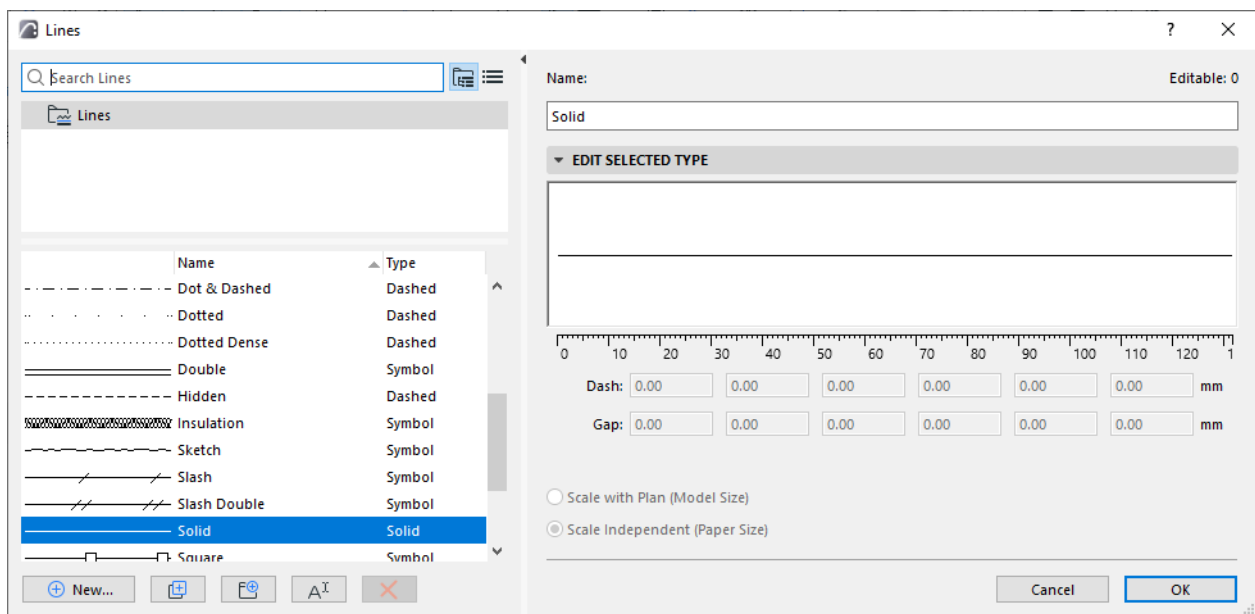
Lines types are assigned to Archicad elements in their settings dialogs.

For example, when defining how a Column should be displayed in a 2D window, you will apply different lines for its core outline, for its overhead display and for its crossing symbol.

Edit and manage line types

Use the Lines dialog (Options > Element Attributes > Lines) to modify the standard lines (solid, dotted, dashed, etc.) and define customized lines.

For general information: see [Using the attribute dialogs](#).

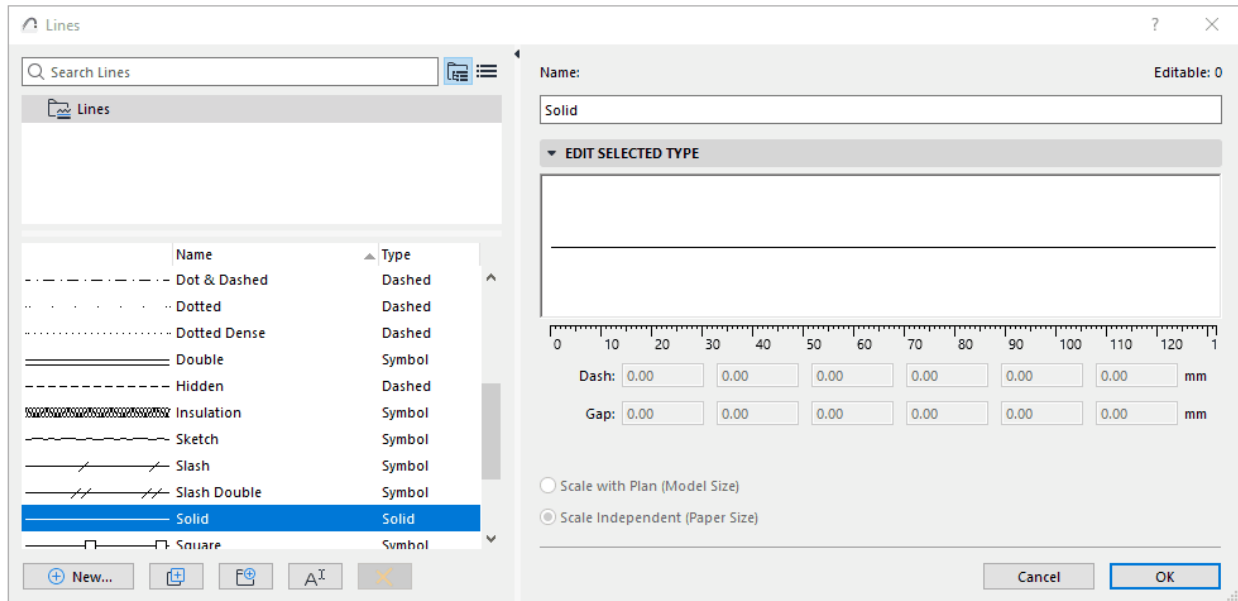


Choose a Line Type from the left side of the dialog to edit its parameters on the right.

Note: Solid lines are not editable.

Create new line type

1. Click **New**.
2. Choose a line type:



Set line parameters as described here:

[Edit Line Type \(Dashed\)](#)

[Edit Line Type \(Symbol\)](#)

Line Weight

Hairline (default)

By default, all lines are displayed at **Hairline** width, at one pixel wide.

- You can also enable **Bold Cut Lines (View > On-Screen View Options)**; this will show all Cut lines as bold (two pixels wide, regardless of the line's true pen weight). All other lines will be Hairline.

True Line Weight

Enable Line True Weight in **View > On-Screen View Options**. Each line's pen weight will be displayed based on its pen weight value (defined in the Pens and Colors dialog).

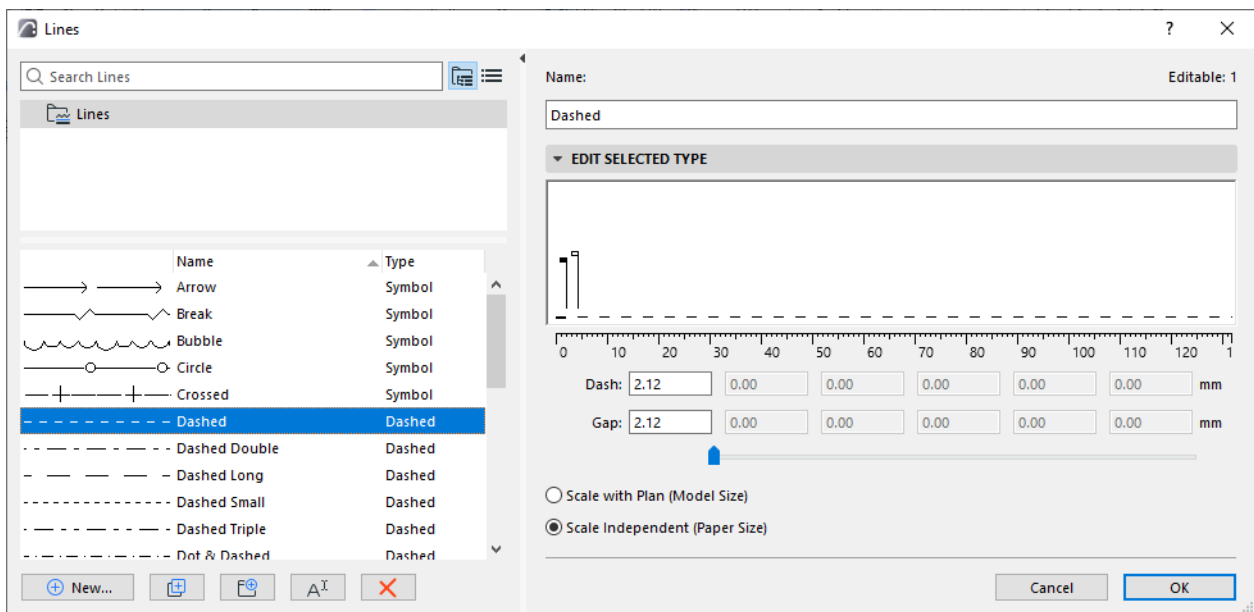
Line Scale

The appearance of the lines on the screen, printer or plotter depends on the following:

- Scale with plan (Model size):** Display the line type at the same scale as the model on every output.
Note: Avoid editing a scaled line type if the project scale is different than that of the line type, as it will be distorted.
- Scale-independent (Paper size):** Display, plot and print the Line type at a fixed size, regardless of output scale.

Edit Line Type (Dashed)

For a dashed line, you can graphically edit its first section.



Black flag: the end of the Dash segment. White flag: the gap between each dash

Edit the dash segment and the gap by dragging the flags horizontally, or edit their value in the **Dash** or **Gap** field below.

The values use the units defined at **Options > Project Preferences > Working Units**.

Edit Line Type (Symbol)

A symbol line type is based on a customized stencil of 2D drawing elements copied from an Archicad window.

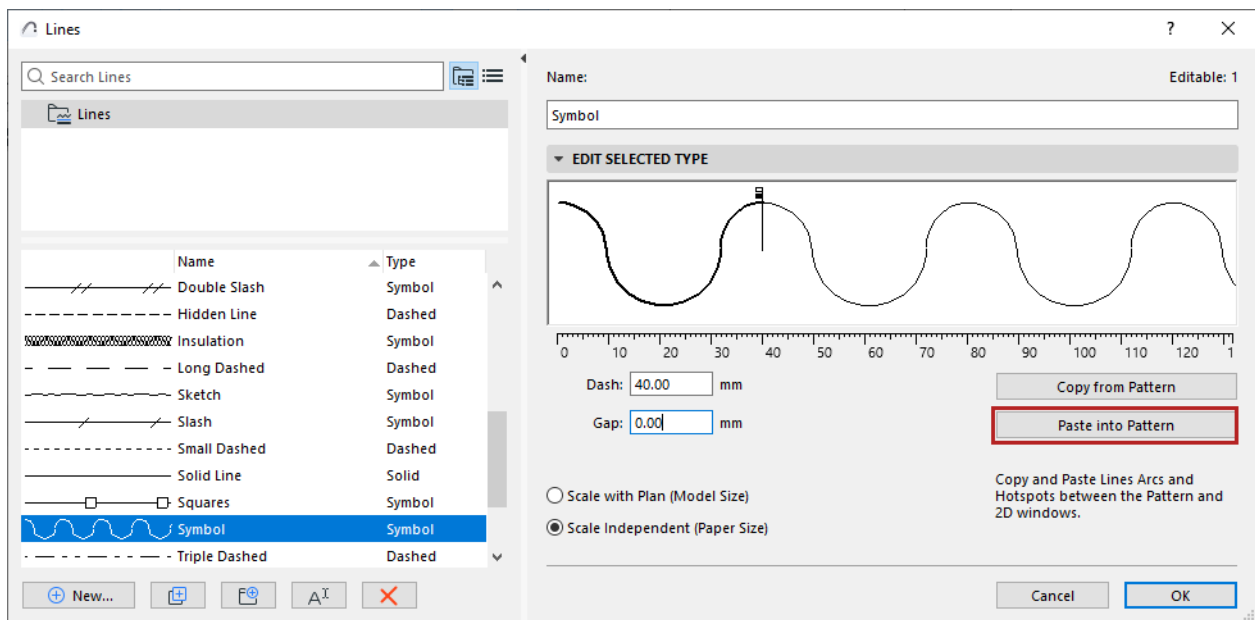
To create a new symbol line type:

1. Draw the stencil on the Floor Plan or in another window, using Lines, Arcs and Hotspots.
2. Select the 2D elements.
3. Choose **Edit > Copy**.

Note: Only Arcs, Lines and Hotspots may be used to define symbol lines. Splines, fills, text or other elements will be ignored.

4. Open the Lines dialog (**Options > Element Attributes > Lines**).
5. Click **New**.
6. From the New Line dialog box, select the **Symbol** option and click OK.
7. In the Lines dialog, click **Paste into Pattern** under the graphic editing section.

Note: Regular Copy-Paste operations do not work in this dialog.



The symbol appears in the editing window.

- Black flag: the end of the line segment.
- White flag: the gap between each segment

Edit the line segment and the gap by dragging the flags horizontally, or edit their value in the **Dash** or **Gap** field below.

The line component will be scaled as a whole.

In the resulting Symbol line, the bounding box of the pasted elements will be centered on the centerline.

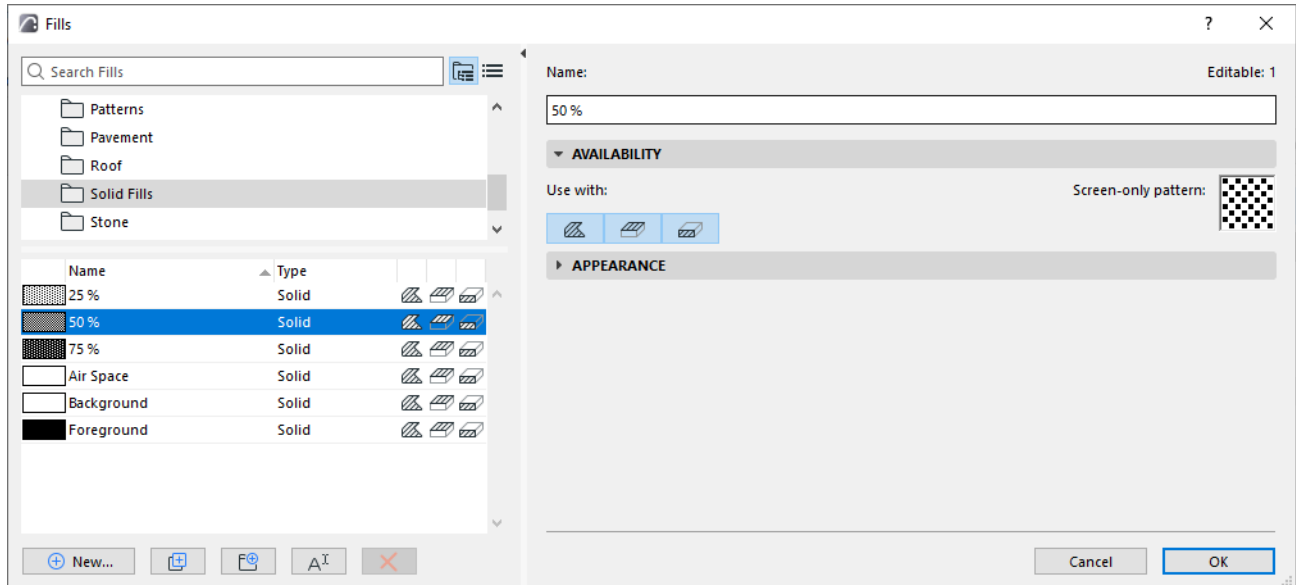
[Read More](#)

Fills

Archicad includes a predefined set of default fills.

Use the **Options > Element Attributes > Fills** dialog box to define and/or edit fills and patterns, and determine the category of each particular Fill (Drafting Fill, Cut Fill, and/or Cover Fill).

For general information: see [Using the attribute dialogs](#).



You can modify this set manually or by importing items with the Attributes palette.

See [Attributes palette](#).

Set Fill Availability by Category (Drafting, Cover, Cut)

Use the Availability Panel of the Fills dialog to define one or more categories for each Fill.

See [Fill Categories](#).

Related Topics

[Edit Vector Fill](#)

[Create/Edit Solid Fill](#)

[Create/Edit Symbol Fill](#)

[Create Image Fill](#)

Fill Categories

There are three Fill Categories in Archicad: Drafting, Cover, or Cut.

Fill Categories are used:

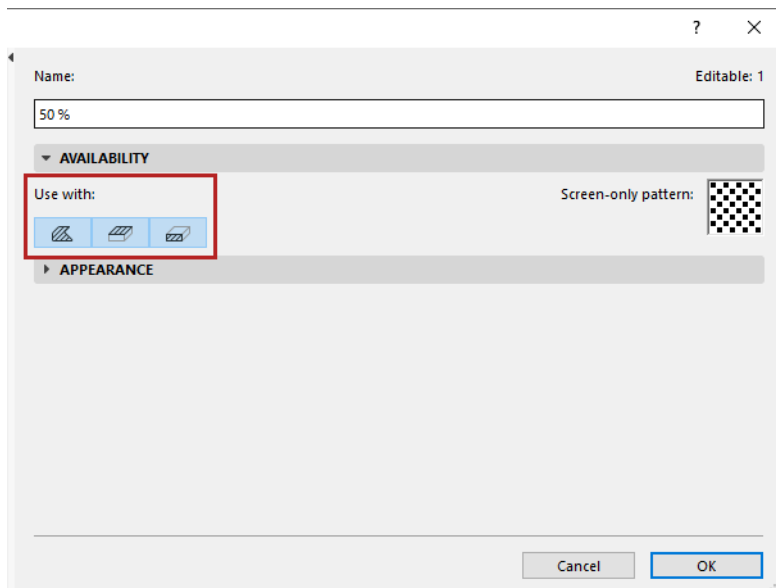
- to define which fills are available in Fill chooser pop-ups throughout the Archicad interface
- to choose and create freehand Fills using the Fill tool
- to apply Graphic Override Rules to certain Fill categories
- in criteria (e.g. Find and Select or Interactive Schedules)
- in handling exported cut fills (at DXF export)

Set Fill Availability by Category (Drafting, Cover, Cut)

Use the Availability Panel of the Fills dialog to define one or more categories for each Fill.

Not all project Fills are available in every fill pop-up. The Fill chooser pop-ups, located in dialog boxes throughout the Archicad interface, contain only the Fills which belong to a Fill Category that makes sense in that context.

1. Open the **Options > Element Attributes > Fills** dialog box.
2. Select the Fill(s) to edit.
3. Activate one or more of the **Use with** icons to define its Fill Category: Drafting, Cover, and/or Cut.



Drafting Fills

Drafting fills are drawn by hand using the Fill tool.

- A Fill type in the “Drafting” category is available in the Fill tool dialog box.

Cover Fills

A Fill type in the “Cover” category is available in

- Tool Settings dialogs of elements that can use a cover fill
- Cover Fill Foreground panel in the Surfaces dialog
- Override Cover Fills in Graphic Overrides

Cut Fills

A Fill type in the “Cut” category is available in

- Building Materials dialog
- Settings dialogs of GDL elements (which use cut fills)
- Override Cut Fills in Graphic Override Rules

Notes:

- Fill elements defined as “Cut Fill” can be optionally placed onto their own layer during DXF/DWG conversion. [See Custom Layers.](#)
- Image Fills cannot be used as cut fills.

Building Material-based Fills and Surface-based Fills

When drawing a freehand Fill using the Fill tool, two additional Fill categories make it easier to match cut fills of Building Materials, and fills based on a Surface (its cover fill, color or texture.)

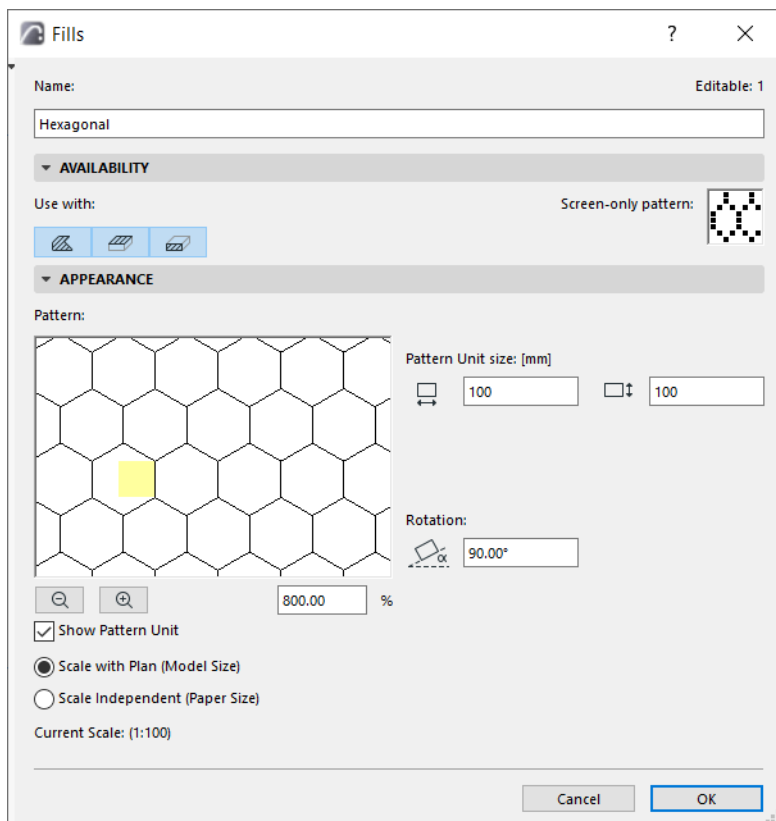
[See Draw a 2D Fill.](#)

See also:

[Note on Transparent Fills in Plotter Output](#)

Edit Vector Fill

Use the Appearance panel of the Fills Dialog Box (**Options > Element Attributes > Fills**) to edit vector fills.



Show Pattern Unit: Check this box to place a highlight over the symbol unit in the preview window.

Use the Zoom buttons to see the pattern close up.

Choose a Scaling option: [see Scale of Fills](#).

Pattern Unit size: Enter values for the horizontal and vertical pattern size.

Rotation: You can rotate the pattern by the value entered here.

Create/Edit Solid Fill

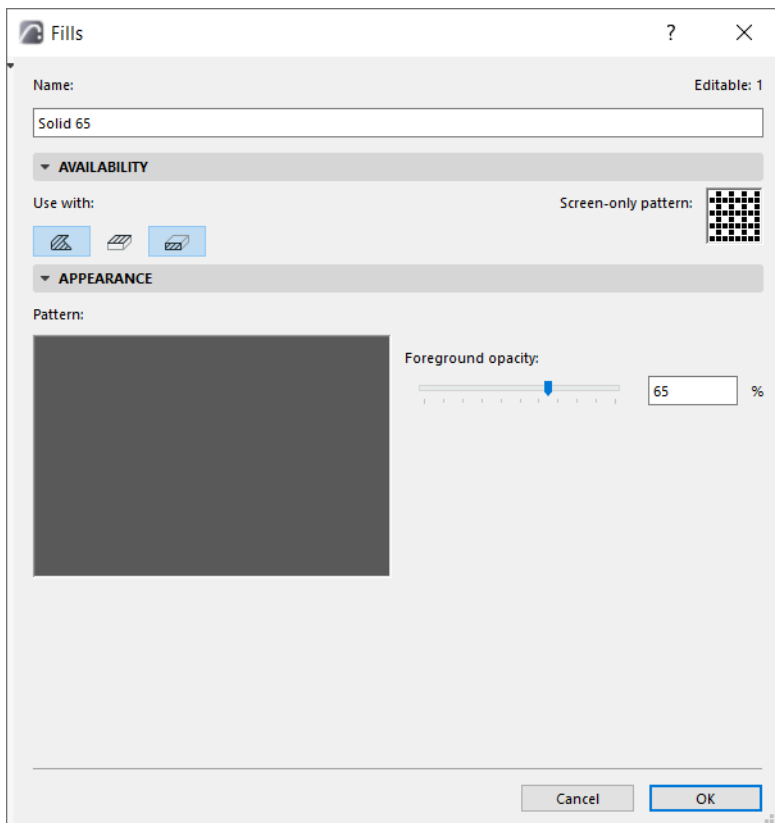
Solid Fills include

- the **Foreground** Fill: You see only the Foreground, because it covers up the Background.
- the **Background** Fill: You see only the Background, because the Foreground is set to zero.
- Three fills whose foregrounds are of predefined Opacity (25%, 50%, 75%).

In the Fills Dialog Box (**Options > Element Attributes > Fills**), click New, and choose Solid fill from the **New Fill** dialog box.

Enter a name.

In the Appearance panel, enter a foreground opacity percentage (or use the slider) - in this case, 65.



View the result in the Preview window.

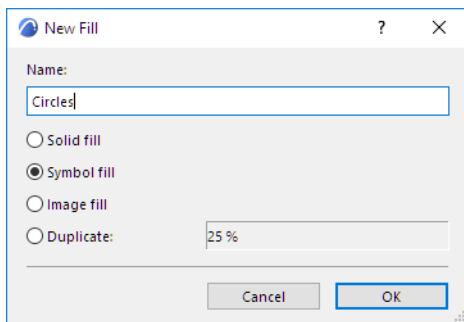
Create/Edit Symbol Fill

Create a symbol fill based on a sketch drawn in any 2D window, with the following steps:

1. Draw a pattern using Lines, Arcs and Hotspots.

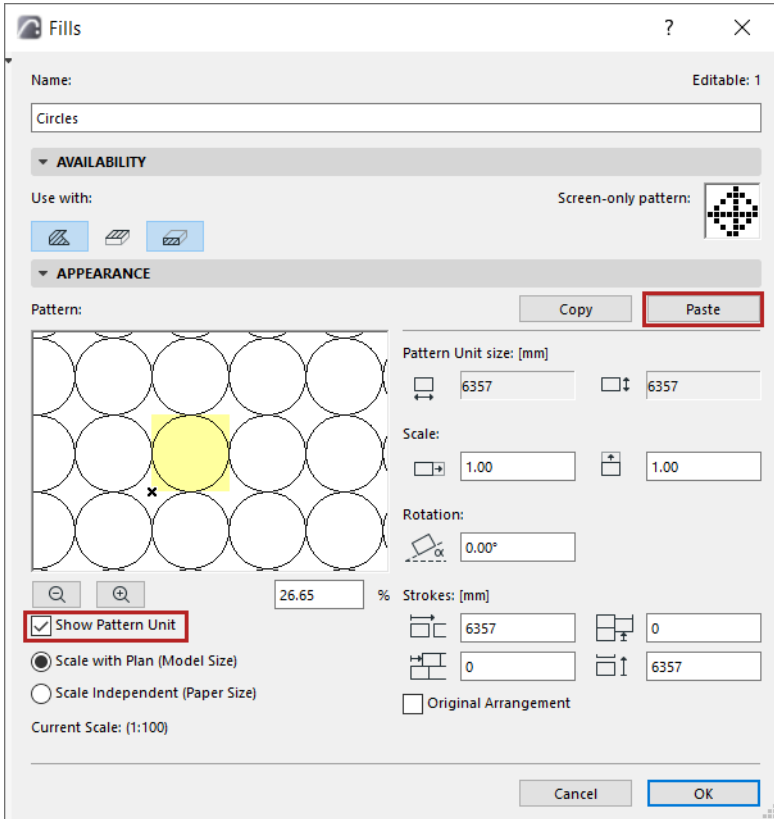


2. Select all the components of the pattern and choose **Edit > Copy**.
3. Go to **Options > Element Attributes > Fills**.
4. Click **New**.



5. Enter a name for the new fill.
6. Choose the **Symbol fill** radio button, then click OK.

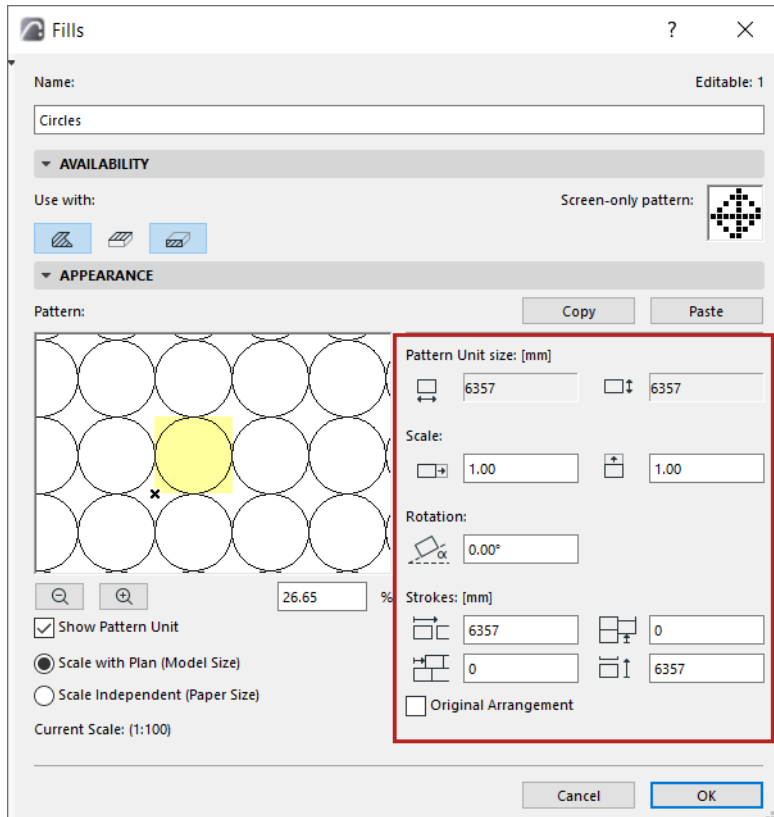
7. In the Appearance panel, click **Paste**. Your copied pattern appears in the Pattern window.



Note: The pasted selection can include only Lines, Arcs and Hotspots.

8. Check **Show Pattern Unit** to highlight a single unit of the pattern in the preview window of this dialog box.

9. Use the controls under Edit Symbol Pattern to set the size, pattern and rotation angle for the symbol fill.



Pattern unit size

This is the absolute size of the Symbol unit on the Floor Plan.

Use these four fields to set the vectors of rows which compose the symbol pattern.

Scale

Enter scale factors by which to “stretch” the Symbol unit both horizontally and vertically. (By default, this factor is 1.0, that is, the symbol unit is not stretched.)

The percentage field and the zoom buttons below the preview window refer to the image in the preview window only. The preview displays the symbol pattern at the given percentage as compared to its appearance on the Floor Plan.

Rotation

Enter an angle in degrees if you wish to skew the fill pattern.

Strokes

To change the pattern distribution, enter values for horizontal and vertical spacing between the pattern vectors.

You can also enter offsets, to create gaps in the pattern.

Check Original Arrangement to return to the saved values.

To determine the relationship between fills and project scale, use the two radio buttons.

[See also *Scale of Fills*.](#)

Related topics

[Symbol Fills as Skins of Composite Elements: Fit to Skin](#)

Radial Distortion (Symbol Fills only)

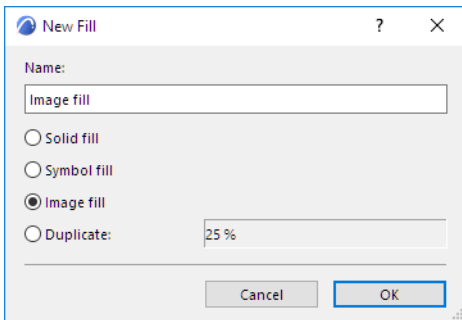
Create Image Fill

Image Fills can be used as the foreground part of Drafting fills or Cover fills.

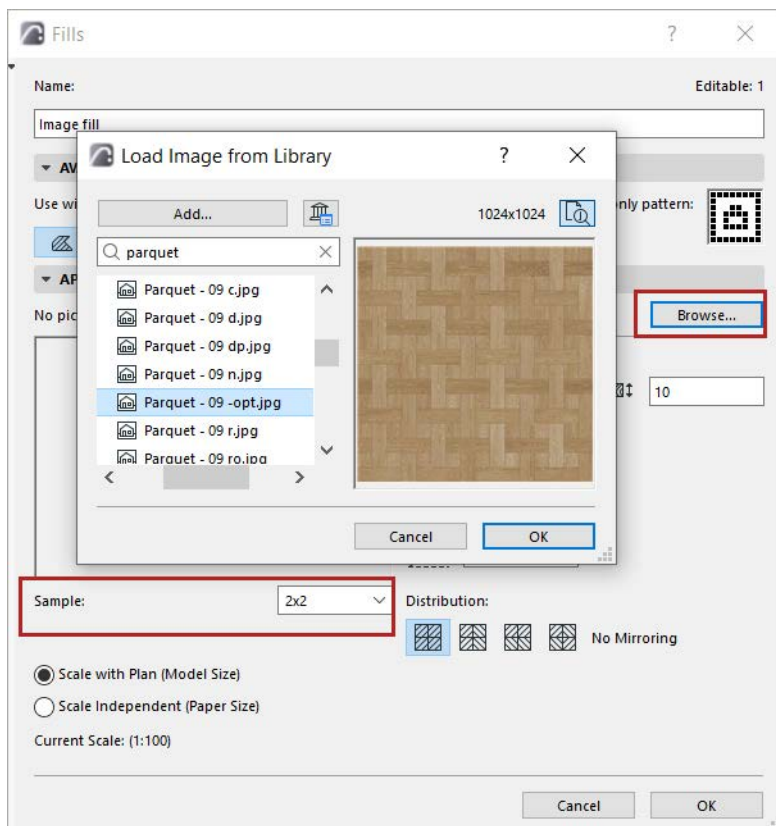
Your Archicad default template provides predefined Image fills, but you can define your own.

Create New Image Fill

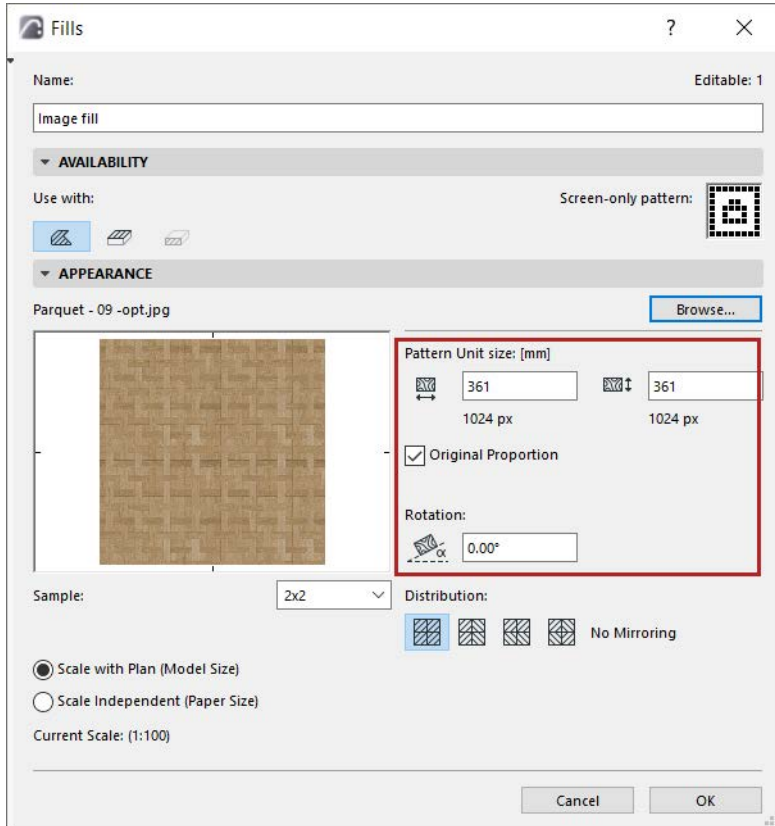
1. Go to **Options > Element Attributes > Fills**.
2. Click **New**.
3. In the **Add New Fill** dialog box, choose **Image fill**.



4. In the Availability panel, define the new Image fill as a Drafting and/or Cover fill.
5. In the Appearance panel, click **Browse** to bring up a library directory dialog box. Browse for the desired image and click **OK** to load it.
6. In the **Sample** field, set the number of texture units displayed in the Preview.



7. Use the **Image Size** controls to fine-tune the size and pattern of your Image fill.



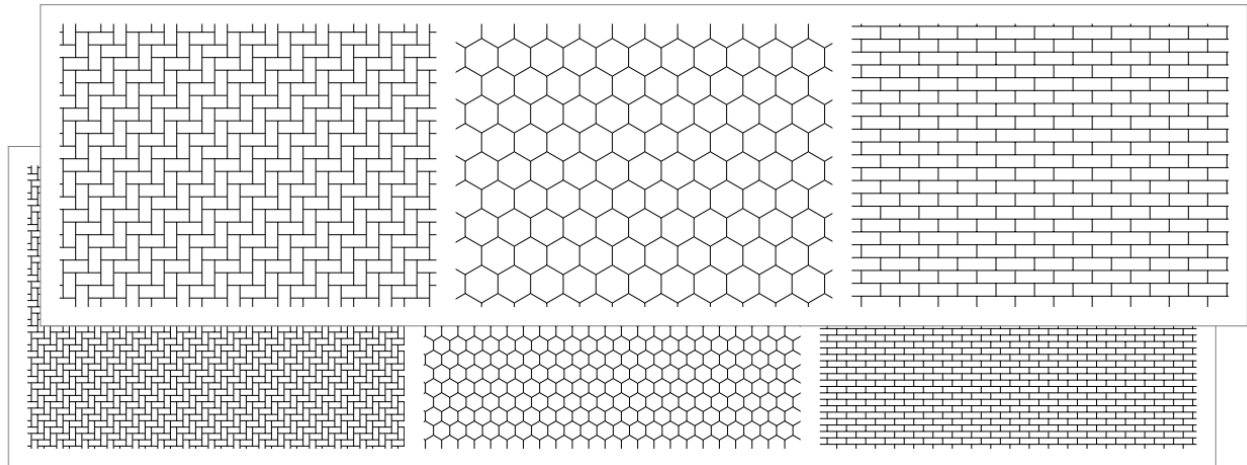
Original Proportion: Link the Horizontal and Vertical Size values so that the image is not distorted when editing its size.

Rotation: Enter an angle if you want to rotate the image pattern from the horizontal.

8. Choose a **Distribution** method to set up the repeating patterns within your Image fill: mirror the pattern vertically, horizontally, both, or none.
9. To determine the relationship between fills and project scale, use the two radio buttons.

Scale of Fills

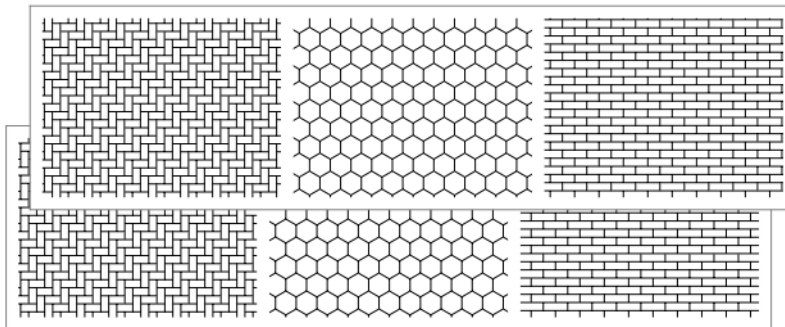
- **Scale with plan (Model size):** Display Fill at the same scale as the model on every output.



Scale with Plan

Note: Avoid editing a scaled fill type if the project scale is different than that of the fill type, as it will be distorted.

- **Scale-independent (Paper size):** Display, plot and print the current Fill type definition at a fixed size, regardless of output scale.



Scale Independent

10. Click OK to save the newly created fill as a project attribute.

Fill Display Mode: True Fill Appearance vs. Bitmap

Each fill type has two faces or display modes: True Fill Appearance and bitmap.

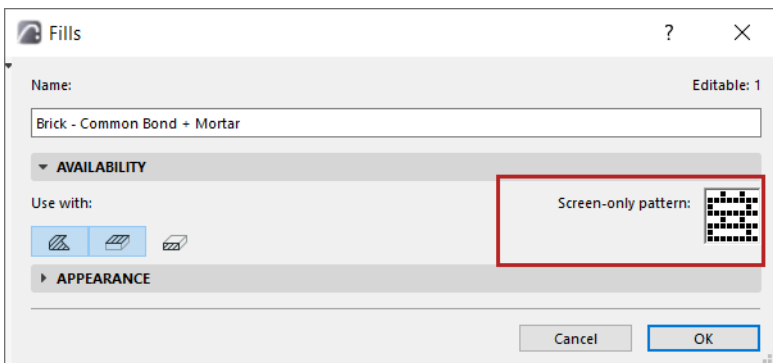
By default, all fills in your project are set to be displayed as True Fill, at **View > On-Screen View Options**.

If you turn this control off, fills will be displayed *on screen* in bitmap mode. However, when printing or plotting fills, the True Fill appearance will be used even if the on-screen display is set to bitmap.

The bitmap versions of all predefined fill patterns approximate their True Fill equivalents. If you change a True Fill pattern or generate a new symbol type pattern, you will need to create its bitmap version manually.

Screen-Only Fill Pattern

Use the **Availability Panel** of the Fills dialog box to view or edit the Fill's bitmap pattern.



This pattern is used:

- to identify the fill in the fill pop-up menus
- for on-screen fill display if you disable the “True Fill Appearance” on-screen view option (**View > On-Screen View Options**).

To edit this screen-only pattern, click in the pattern window to turn pixels on or off.

Building Materials

Building Materials are attributes, defined at **Options > Element Attributes > Building Materials**.

All Archicad model elements use Building Materials in their structures. Building Materials are also available for Stairs, Railings, profile elements, and certain Library Parts (in the INT Library: those in the Building Structures Folder and RoofMaker Library Folder).

Building Materials are also used as components of composite structures and profiles.

Each Building Material combines the following:

- Cut fill
- Cut fill pens (foreground/background)
- Intersection Priority
- Fill orientation (relevant for a composite or complex element)
- Surface
- Classifications and Properties, including Physical Properties

Related Topics:

[Edit and manage Building Materials](#)

[Intersection Priority \(Building Material\)](#)

[Building Material Classification and Properties](#)

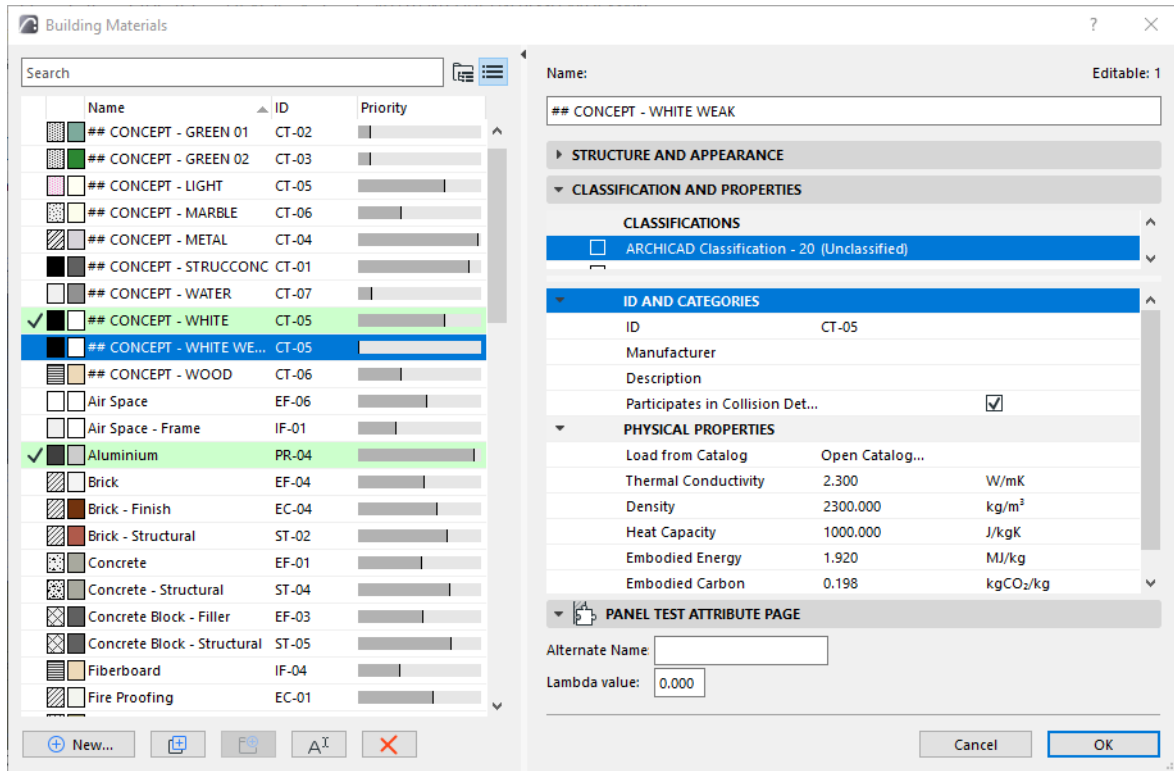
[Environmental Properties](#)

[Exempt Building Material from Collision Detection](#)

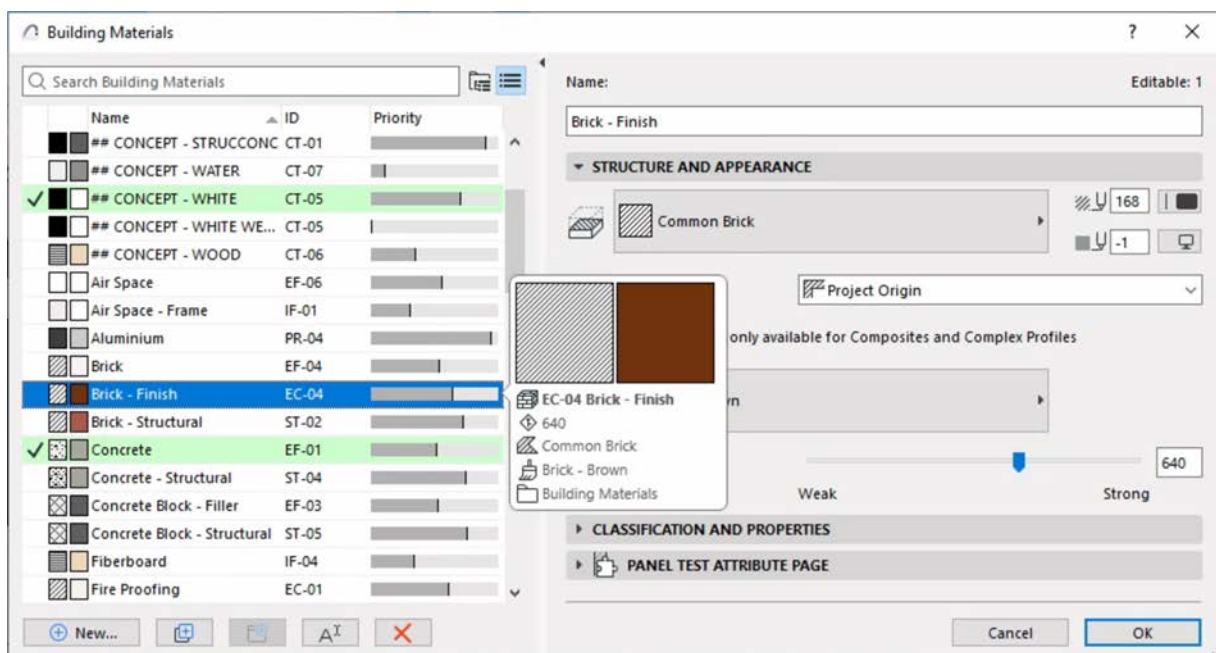
Edit and manage Building Materials

Open the Building Materials dialog:

- Options > Element Attributes > Building Materials
- Attributes toolbar (Windows > Toolbars > Attributes)



- The left panel lists the project's Building Materials, by **Name**, **ID**, and **Intersection Priority**.
- Green highlights indicate the Building Materials of currently selected elements.



Note: Hover over a material to see more information on it.

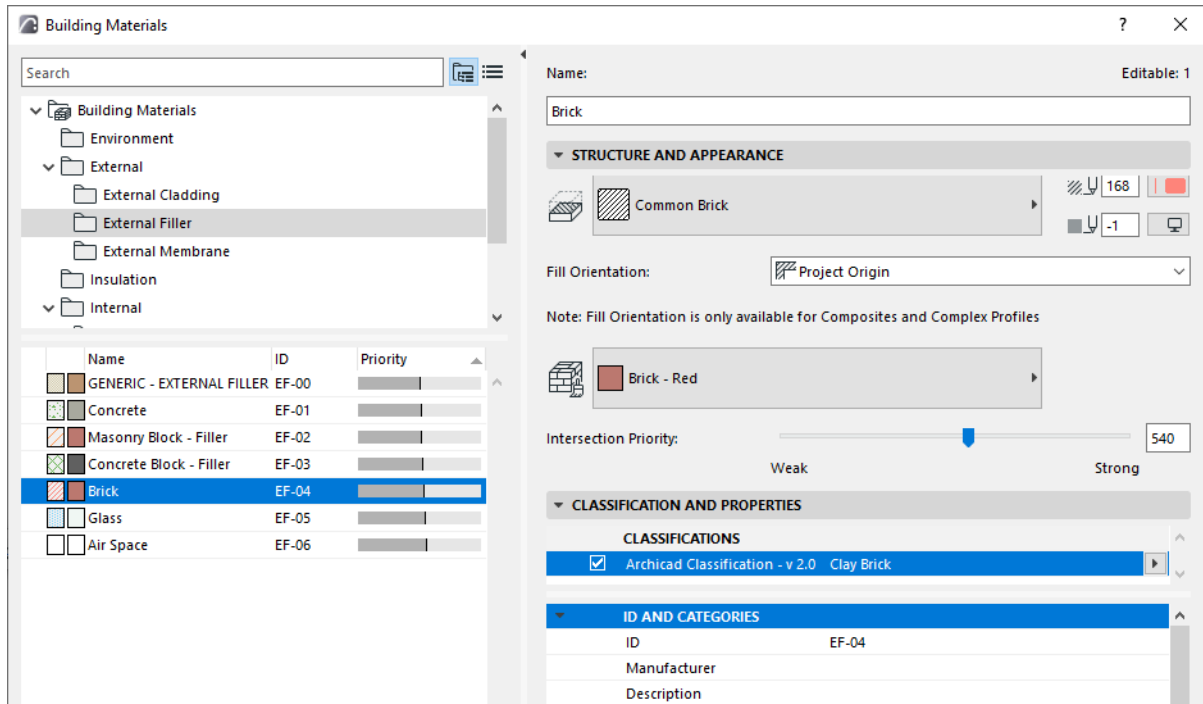
For general information: see [Using the attribute dialogs](#).

Edit Building Material

1. Select one or more Building Material(s) to edit.

Hint: In folder view, select a sub-folder to edit all its contents at once.

2. Use the fields on the right side of the dialog box to edit properties



Cut Fill

Choose a cut fill for this Building Material. This pop-up lists only fills defined as cut fills: see [Fill Categories](#)

Cut Fill Foreground Pen/Fill Background Pen: Choose the foreground and background pen for this Building Material.

See also [Line Elimination Between Model Elements](#).

Fill Orientation

Affects the fill pattern of Building Materials applied to *composite elements and Complex Profiles only*. Choose a fill orientation option:

- **Project Origin:** The fill pattern originates at Project Origin.
- **Element Origin:** The fill pattern is aligned with the element's direction.
- **Fit to Skin:** (Available only for symbol fills.) The pattern fits within the limits of the Skin Thickness.

Related Topics:

[Set Orientation of Vector or Symbol Fill Patterns](#)

[Symbol Fills as Skins of Composite Elements: Fit to Skin](#)

Surface

Choose a surface for displaying this Building Material in 3D.

Note: At the element level, you can override the surface in its Settings dialog (Model Panel) or Info Box.

See also:

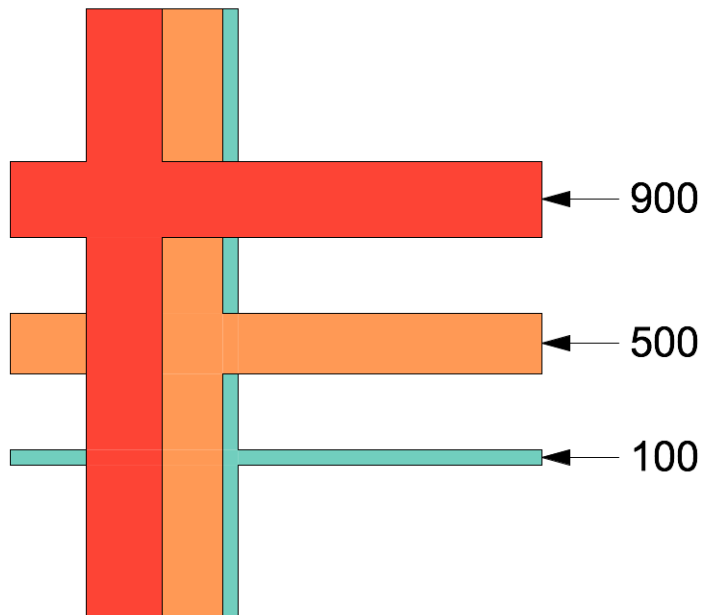
[Intersection Priority \(Building Material\)](#)

[Building Material Classification and Properties](#)

Intersection Priority (Building Material)

Building Material intersection priority affects the way elements intersect and display in model views.

Expressed in numbers, an intersection priority can run from weak - a minimum of 0 - to strong - maximum of 999.



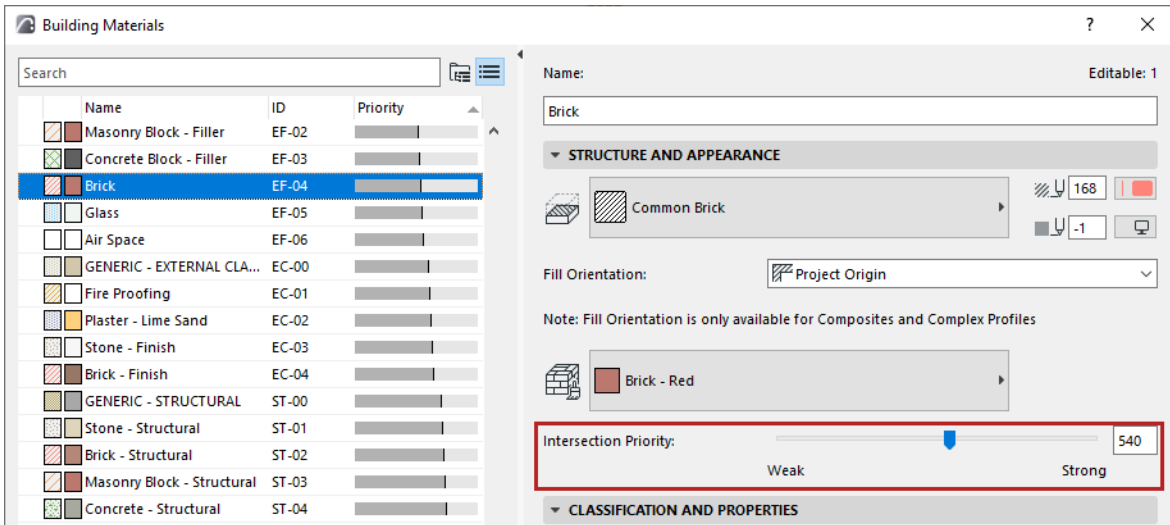
To fine-tune element intersections, you can adjust the Building Material priority.

Note: Any changes you make here will affect all project elements which use that Building Material! To adjust the priority in just one element of the project, create a duplicate Building Material with a different priority.

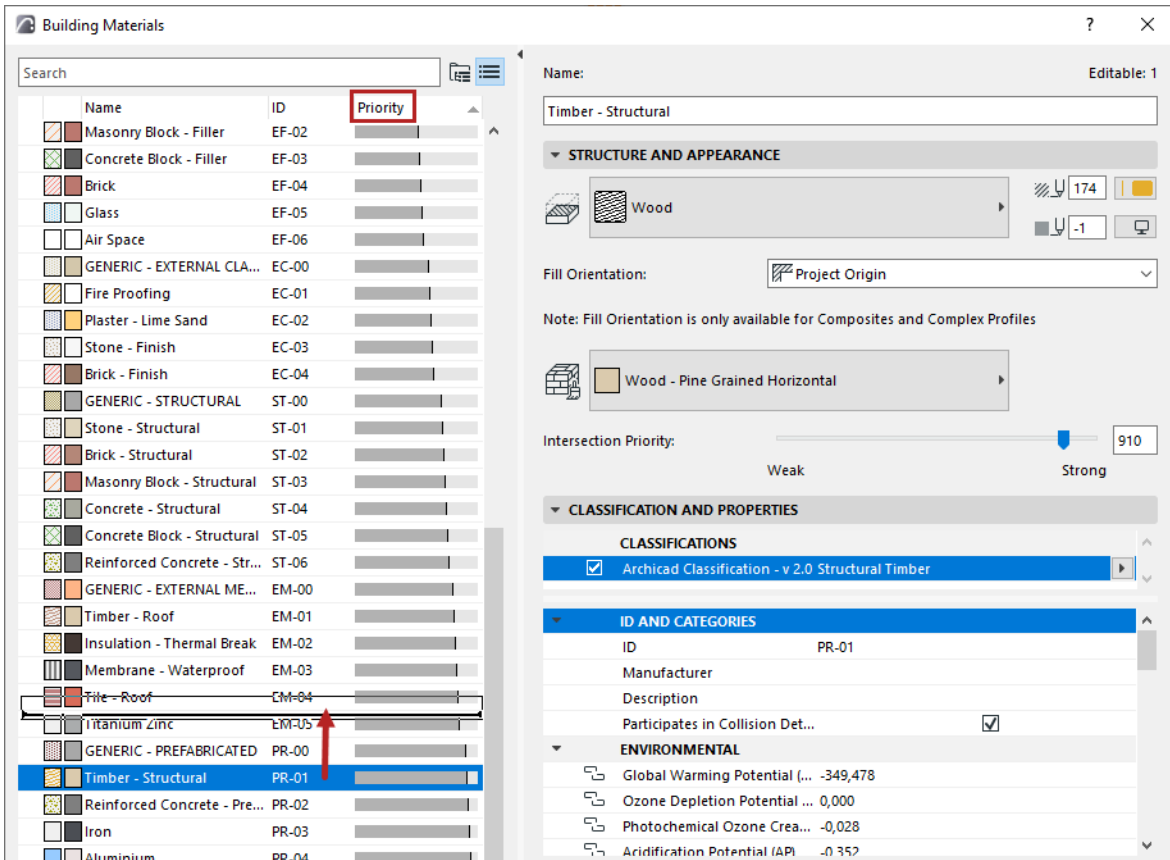
[See also *Basic Intersection Principles and Intersecting Elements with Identical Priorities.*](#)

Edit Intersection Priority

1. Open the Building Materials dialog:
 - Options > Element Attributes > Building Materials
2. Do one of the following:
 - Use the slider to adjust a Building Material's relative priority in its intersections with other construction elements



- Click the Priority header to sort by priority, then drag a selected Building Material up or down in the list. Its priority changes depending on its list position.



Building Material Classification and Properties

Options > Element Attributes > Building Materials

The screenshot shows the 'Building Materials' dialog box with the following content:

Name: Editable: 1

▶ STRUCTURE AND APPEARANCE

▼ CLASSIFICATION AND PROPERTIES

CLASSIFICATIONS

Archicad Classification - v 2.0 Mineral Thermal Insulation

▼ ID AND CATEGORIES

ID	IN-04
Manufacturer	
Description	
Participates in Collision Dete...	<input checked="" type="checkbox"/>

▼ ENVIRONMENTAL

Global Warming Potential (G...	-6.785
Ozone Depletion Potential (...)	0.000
Photochemical Ozone Creati...	-0.001
Acidification Potential (AP)	-0.008
Eutrophication Potential (EP)	-0.001
Abiotic Depletion Potential (...)	0.000
Abiotic Depletion Potential (...)	-95.184
Data Source	https://www.oekobaudat.de/OEKOBAU.DAT/r...

▶ COMMON (Materials)

▶ COMBUSTION (Materials)

▶ THERMAL (Materials)

▶ STRUCTURAL ANALYSIS DATA

▶ PHYSICAL PROPERTIES

Load from Catalog	Open Catalog...
Thermal Conductivity	0.036 W/mK
Density	115.000 kg/m ³
Heat Capacity	840.000 J/kgK
Embodied Energy	21.200 MJ/kg
Embodied Carbon	1.650 kgCO ₂ /kg

ID and Categories

- **ID:** Assign an optional ID to this Building Material. Any character can be used, with no character number limit.
The ID field serves as a way to identify Building Materials in lists and labels. You can also Find & Select elements by using **Building Material ID** as a search criterion.
- **Manufacturer:** Optionally, enter a manufacturer name for this Building Material. You can add Manufacturer as a field to your Interactive Schedule, e.g. to list all Building Materials by manufacturer.
- **Description:** Optionally, enter a Description for this Building Material. You can add Description as a field to your Interactive Schedule.
- **Participates in Collision Detection:** This data is used in exporting elements to IFC format, and in Archicad's Collision Detection feature. [See Collision Detection.](#)

[Watch the Video](#)

Physical Properties

These data are predefined for Archicad Building Materials, but you can edit them directly, or load the values via a material from the catalog.

Physical property	Unit
Thermal conductivity	W/mK
Density	lb/cu ft (imperial) kg/m3 (metric)
Heat capacity	J/kgK
Embodied Energy	lbCO2/lb (imperial) kgCO2/kg (metric)
Embodied Carbon	MJ/kg
Note: Set Volume unit at Project Preferences > Calculation Units.	

Load from Catalog: Access the built-in Material Catalog (click the **Open Catalog** button), and click the name of a material having the characteristics you need; its physical properties will then be applied to the Building Material you are editing.

For more information, see [Material Catalog](#).

Environmental Properties

Certain Archicad templates include this set of Building Material properties, corresponding to standard environmental indicators.

See [Environmental Properties](#).

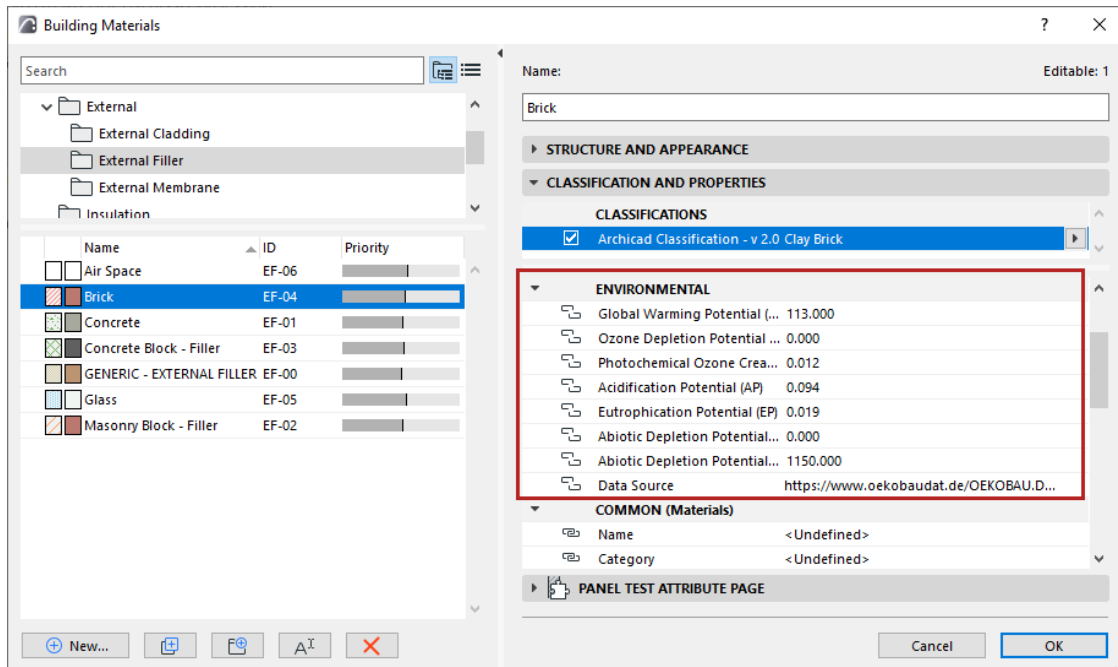
See also [Properties and Classification Systems](#).

Environmental Properties

Building Materials in certain Archicad templates include built-in Environmental properties, corresponding to standard environmental indicators.

Note: Available out-of-the-box for the INT template, plus in language versions based on the INT template. Property values are based on data of the German ÖKOBAUDAT database.

Use these validated properties to document embedded energy and CO2 footprint data for building lifecycle analysis and sustainability reports.



Name	Description
Global Warming Potential (GWP)	<p>Global Warming Potential: the insulating effect of greenhouse gases (GHG) - CO2 and methane - in the atmosphere preventing the earth losing heat gained from the sun. As global temperature rises, it is expected to cause climatic disturbance, desertification, rising sea levels and spread of disease.</p> <p>Unit: kg CO2 eq.</p>

Name	Description
Ozone Depletion Potential (ODP)	<p>Ozone Depletion Potential: depletion of the ozone layer (O₃) in the atmosphere caused by the emission of chemical foaming and cleaning agents allows the passage of greater levels of UV from the sun, causing skin cancer, damage to the immune system and reducing crop yields.</p> <p>Unit: kg R11 eq.</p>
Photochemical Ozone Creation Potential (POCP)	<p>Photochemical Ozone Creation Potential: Creation of ozone in the presence of sunlight, nitrogen oxides and volatile organic compounds. Ozone leads to chemical smog that affect human health, food crops and the ecosystem in general. The effects vary according to geography and climate and are especially problematic in heavily urbanized areas with existing pollution.</p> <p>Unit: kg Ethene eq.</p>
Acidification Potential (AP)	<p>Acidification Potential: emissions, such as sulphur dioxide and nitrogen oxides from manufacturing processes, result in acid rain which harms soil, water supplies, human and animal organisms, and the ecosystem.</p> <p>Unit: kg SO₂ eq.</p>
Eutrophication Potential (EP)	<p>Eutrophication Potential: increased concentrations of nitrates and phosphates can encourage excessive growth of algae and reduce oxygen levels. This increases mortality in aquatic fauna and flora, leads to loss of species dependent on low-nutrient environments, reduces biodiversity and has knock-on effects on non-aquatic animals and humans.</p> <p>Unit: kg Phosphate eq.</p>
Abiotic Depletion Potential (for non-fossil resources) (ADPE)	<p>Abiotic Depletion Potential for non-fossil resources: the over-extraction of minerals and other non-living, non-renewable materials which can lead to exhaustion of natural resources.</p> <p>Unit: kg Sb eq.</p>

Name	Description
Abiotic Depletion Potential (for fossil resources) (ADPF)	Abiotic Depletion Potential for fossil resources: The over-extraction of fossil fuels including all fossil resources. Unit: MJ
Data source	Building life cycle assessment data sheet in the ÖKOBAUDAT database https://www.oekobaudat.de/en.html

Composite Structures

Composite structures are attributes, defined at **Options > Element Attributes > Composites**.

Use composite structures to create walls, slabs, roofs, and shells.

Each composite consists of Building Materials, plus separator lines. Thus, each composite attribute itself is composed of associated attributes: cut fills, surfaces, and lines.

Related Topics

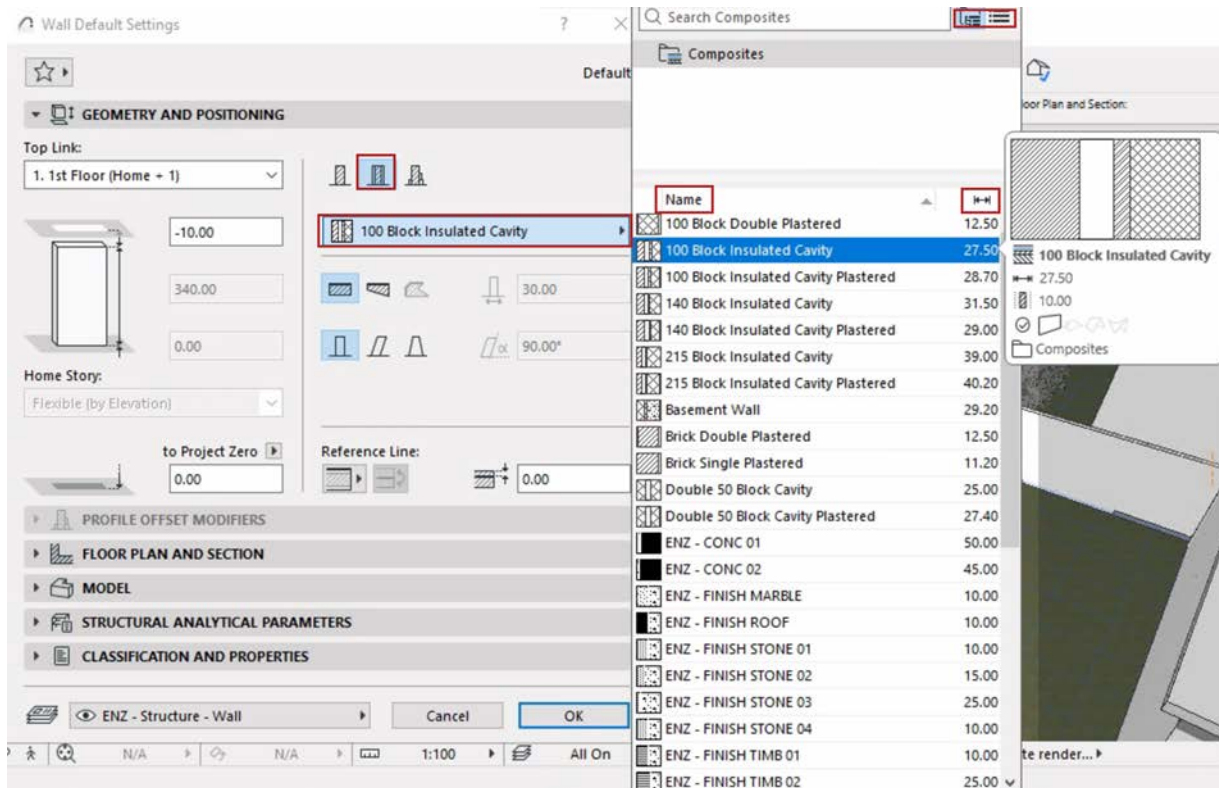
[Assign composite to wall, slab, roof or shell](#)

[Edit and manage composites](#)

[Component Data in Archicad](#)

Assign composite to wall, slab, roof or shell

1. Open the element settings (wall, slab, roof, or shell).
2. In the Geometry and Positioning Panel, click the “Composite” icon.
3. Use the pop-up to select a composite structure.



Note: This pop-up lists composites whose “use with” setting includes your element type: see [“Use With” - define composite availability](#)

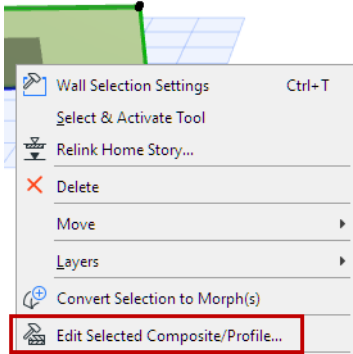
4. The composite is applied to the currently selected element.

Note: Display of composites is affected by the view’s Partial Structure Display.

Edit and manage composites

Open the Composites dialog:

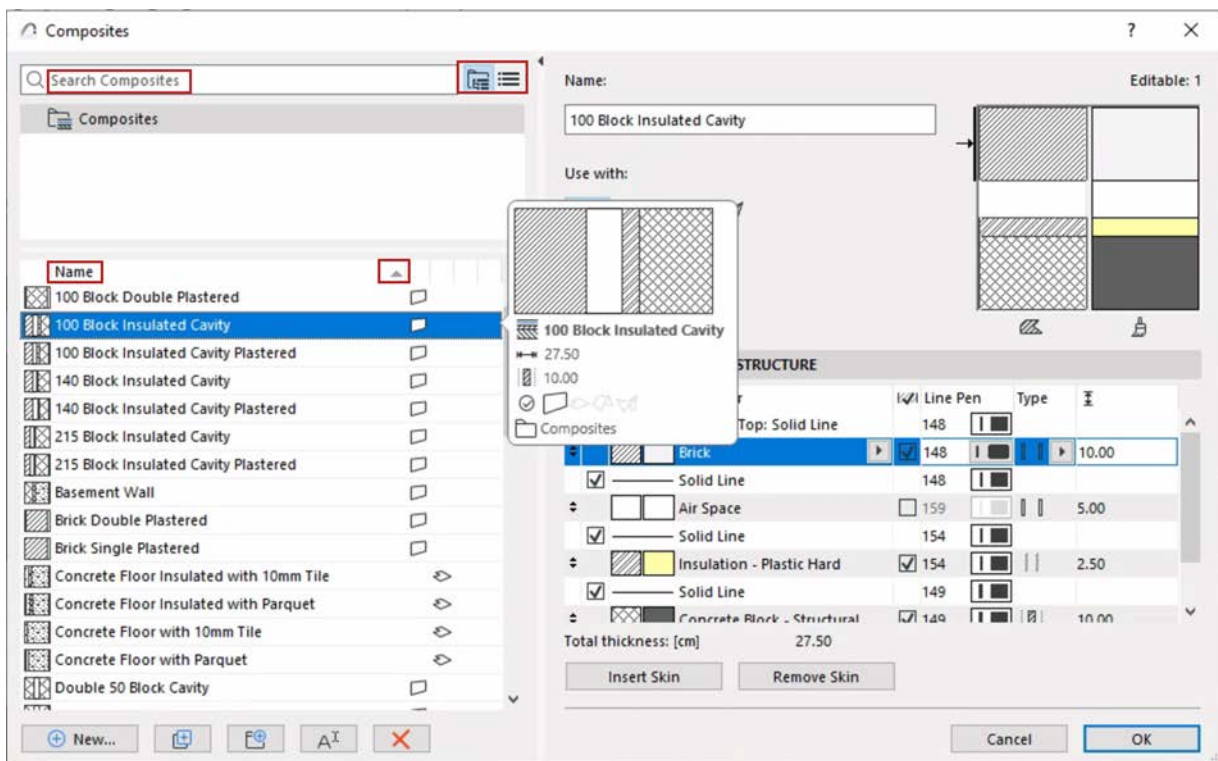
- Options > Element Attributes > Composites
- Attributes toolbar (Windows > Toolbars > Attributes)
- Context menu of a selected composite element:



Use this dialog to add, edit, and manage the project's composite attributes.

Hint: In folder view, select a sub-folder to edit all its contents at once.

For general information: see [Using the attribute dialogs](#).

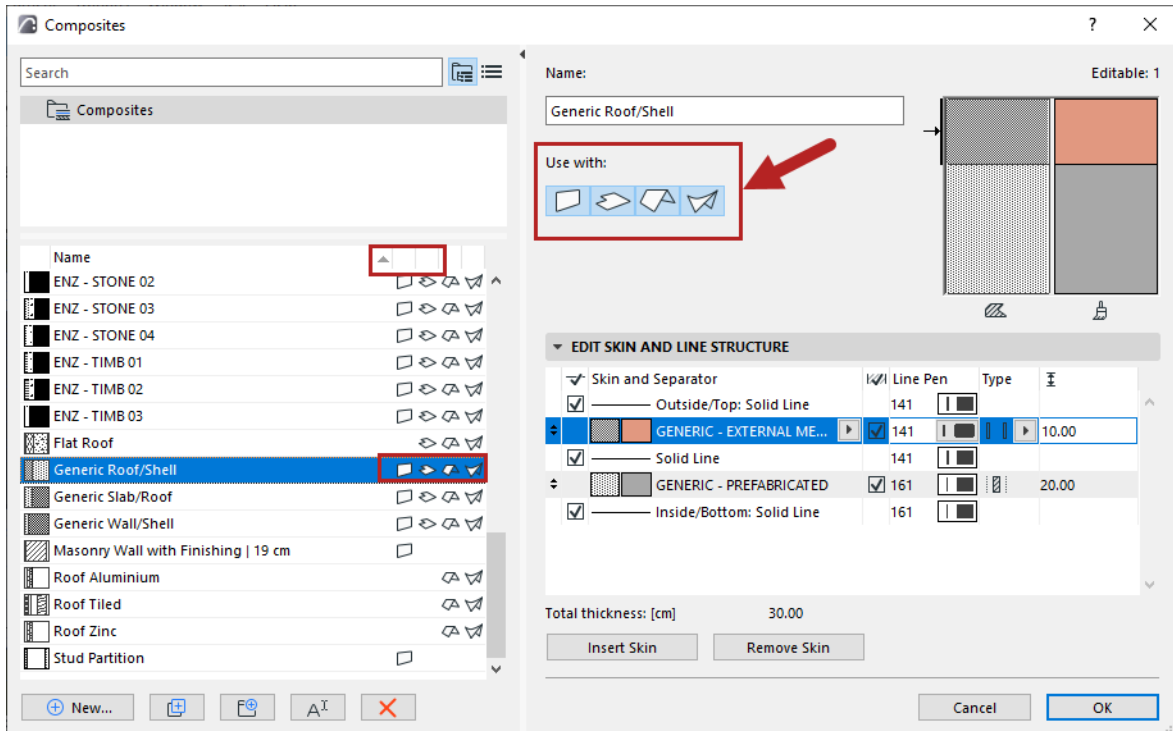


“Use With” - define composite availability

For one or more selected composites:

Click one or more icons - wall, slab, roof and/or shell - to make the selected composite(s) available for particular tools.

- For example, you can define a composite structure to be used with roof and shell. So this composite will appear in the composite pop-ups of roof and shell settings.
- In the left-hand list of composites, click on column headers to sort the list by status

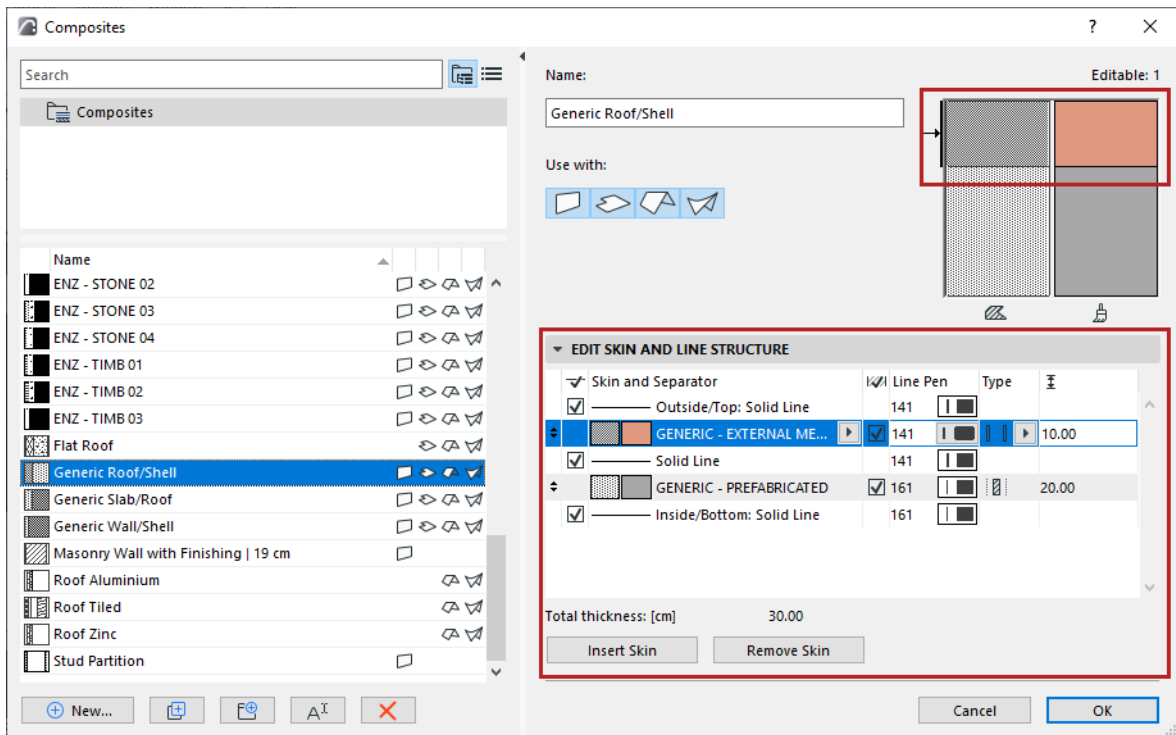


Edit skin and line structure

For a selected composite: the “Edit skin and line structure” panel lists its skins and separator lines.

Each skin consists of a Building Material, plus skin end lines that you can turn on or off

Check the preview to see the selected composite’s cut and surface appearance or line type. (The arrow indicates which part is currently selected.)



Insert skin

Select an existing skin, then click **Insert skin** to insert a new skin plus a separator above it.

Note: A composite structure can have up to 48 skins.

Remove skin

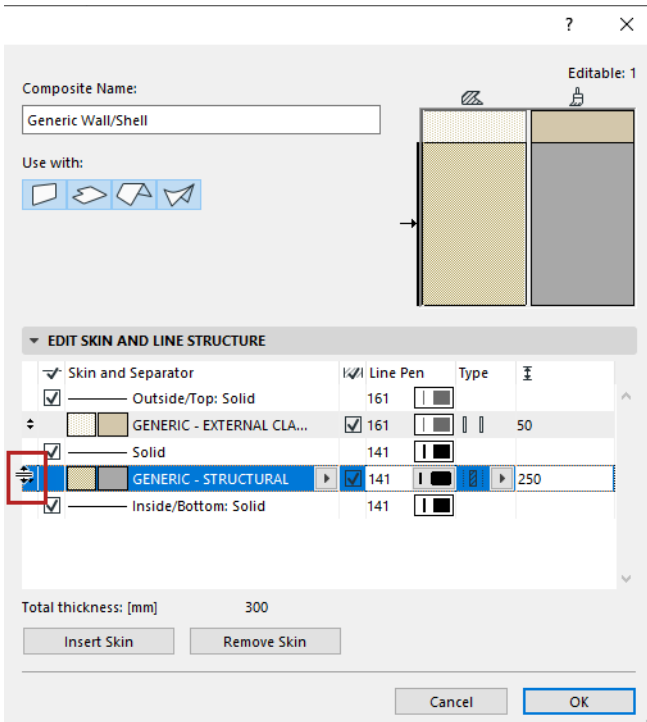
Click to delete the selected skin. The separator line above is also deleted.

Change skin order

The order of skins in a composite can be significant. The **first listed skin** is considered

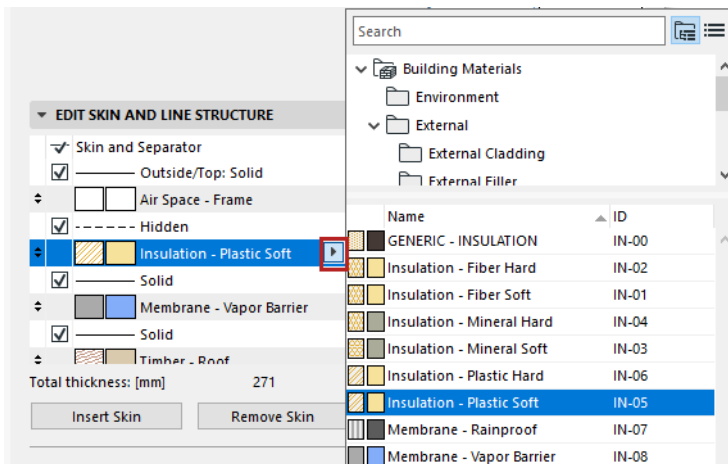
- “Outside” when used with Walls
- “Top” for Roof and Slab
- “Reference Side” for Shells.

Drag a skin to a new position using the up/down arrows at the left edge of the selected row.



Edit skin's Building Material

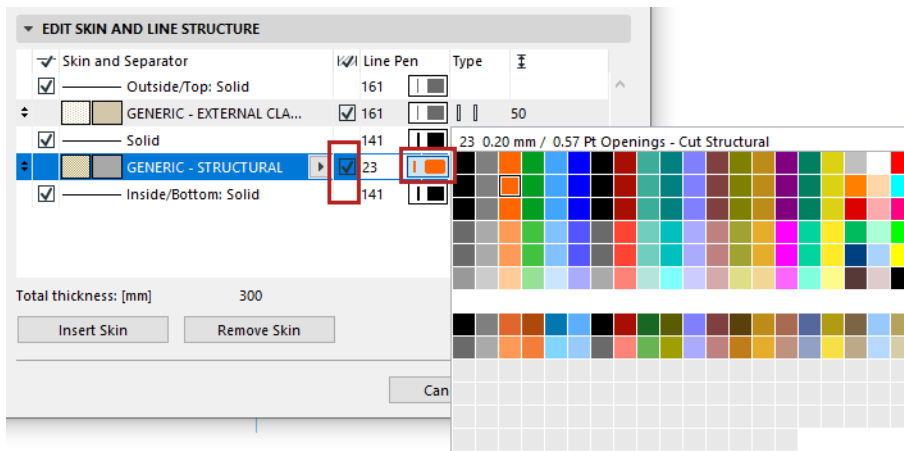
For a selected skin: click the pop-up to choose a different Building Material.



Show/hide skin end lines

For a selected skin: check the box to display its end lines.

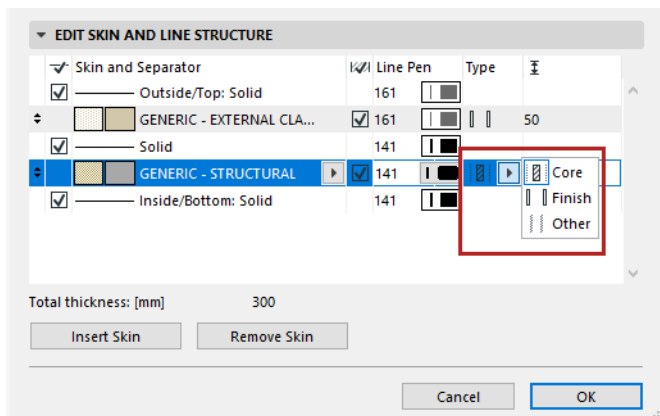
Use the pop-up to choose a pen for these lines.



Define skin structure (core/finish/other)

Use the pop-up to define the selected skin as Core, Finish or Other. This will affect the Partial Structure Display of the composite.

See [Partial Structure Display](#).

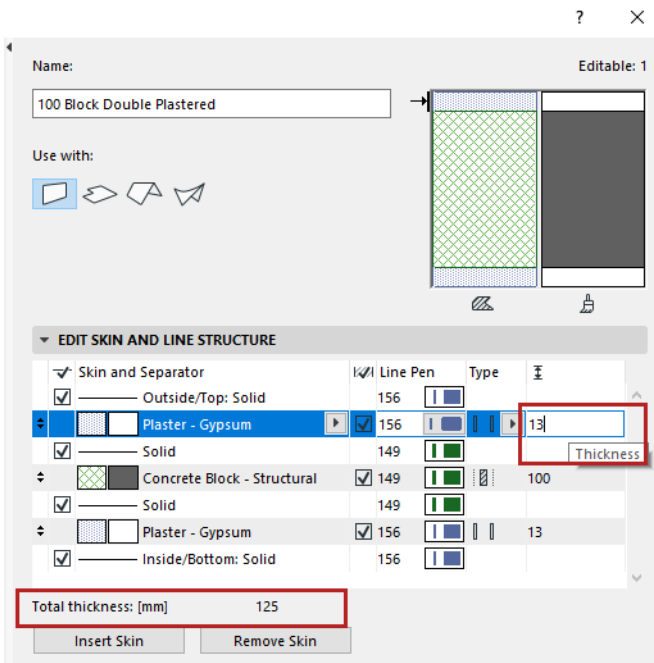


- You can define multiple skins as core, but these skins must be adjacent to each other.
- You can define multiple adjacent skins as finish, but they must include one or both of the outermost skins.
- If you change a skin type from core to finish, then every skin between the changed skin and the closest face will also change to a finish.

Skin thickness and Total thickness

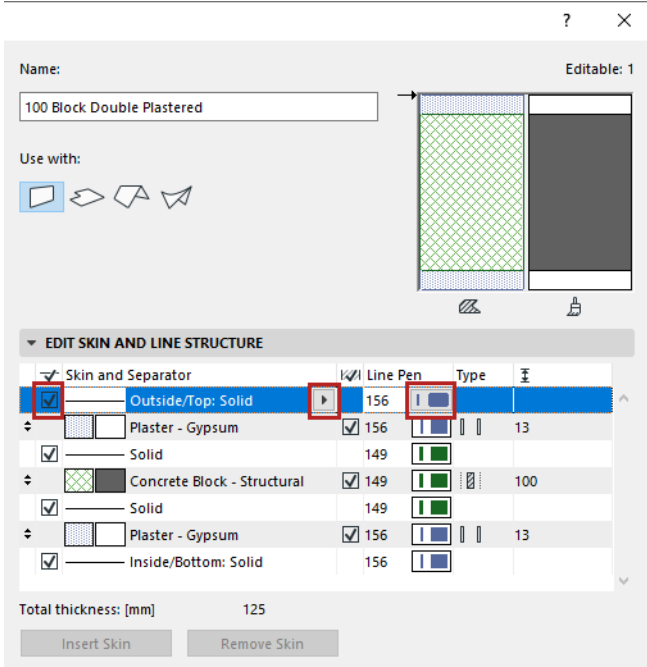
For a selected skin: enter its thickness value.

The composite structure's **total thickness** is shown below.



Show/hide separator line

For a selected line: check the box to display it.
Use the pop-ups to choose its line type and pen.



Pen Sets

Archicad comes with several predefined Pen Sets. These are listed and managed **Options > Element Attributes > Pens & Color**.

or

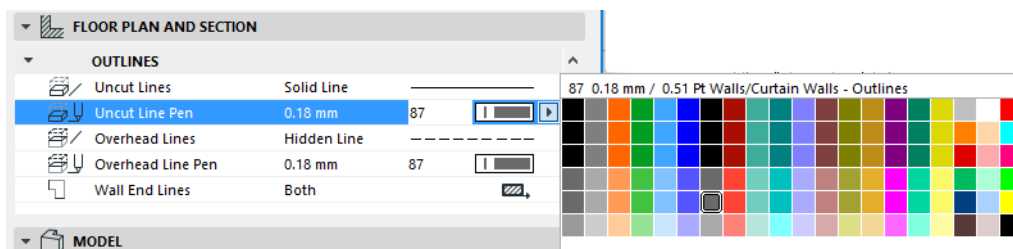
Document > Pen Sets > Pens & Color.

You can just use the default pen set, or you can apply a different Pen Set for your project, depending on the purpose of the current output.

The Layout and Model can refer to different pen tables. This means that you can define a separate set of pens to use on Layouts for the best printing and plotting results.

Note: The *Layout's* Pen Set affects elements placed directly on the Layout (such as Master Layout items like the Title Block). The *Drawing's* pen set affects the content of placed drawings and is assigned in Drawing Settings.

To **assign a pen** to an element, open the pencolor pop-up (in the element settings dialog box or in the Info Box of a selected element) and choose the desired pen for the element or one of its components - such as the cut line pen assigned to the slab in the following image:



The available pens in these pop-ups are derived from the Pen Set currently applied for this project.

[Apply Pen Set](#)

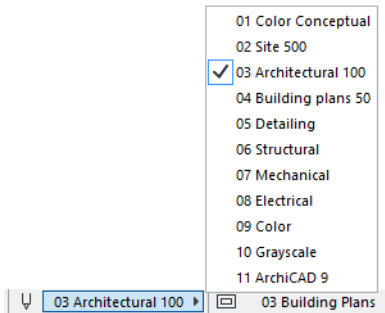
[Redefine Pen Set](#)

[Read More](#)

Apply Pen Set

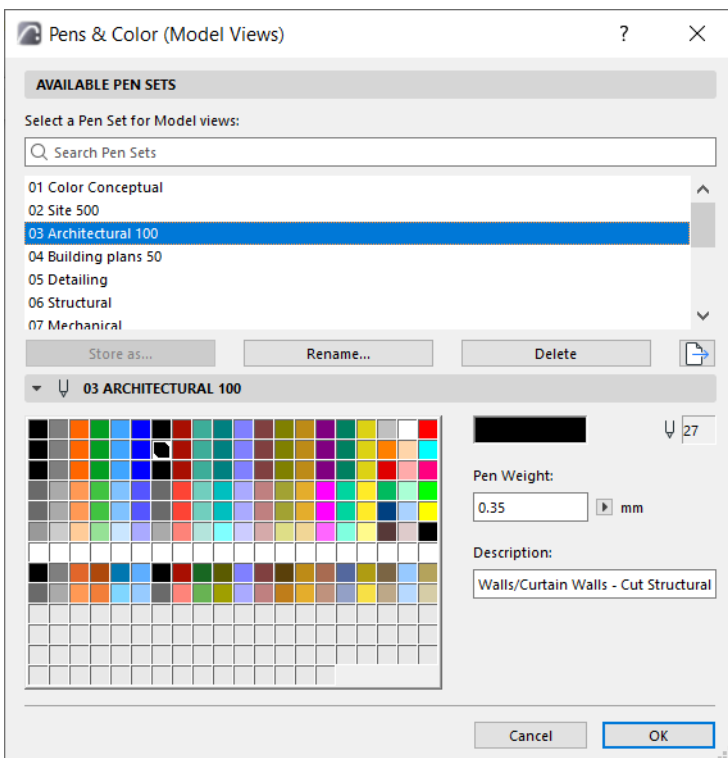
Do one of the following:

- Use the Pen Set pop-up from Quick Options:



- Options > Element Attributes > Pens & Colors
- Document > Pen Sets > Pens & Colors

Select a pen set from the “Available Pen Set” list and click OK.



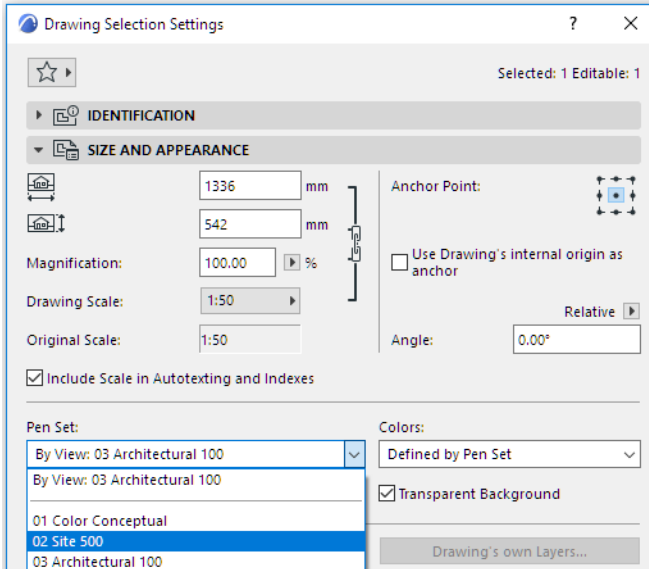
Apply Separate Pen Set for the Layout Book

You can use a separate pen set for your Model Views and your Layout Book: the name of the dialog box includes either “Model Views” or “Layout Book”, depending on which type of window is active.

The Pens & Colors (Layout Book) settings are applied only to items placed onto the Layout (such as Autotext and Master Layout items), but not to the content of placed Drawings.

Apply Pen Set to a Drawing

When you place a view onto a Layout, it becomes a Drawing. By default, the Drawing placed in the Layout Book uses its own pen set (the one saved in the source view's View Settings - by default, the pen set defined for the project's Model Views). However, you can override this pen set in the Size and Appearance Panel of Drawing Settings. Click the Pen Set drop-down menu and choose a different pen set, for this drawing only.

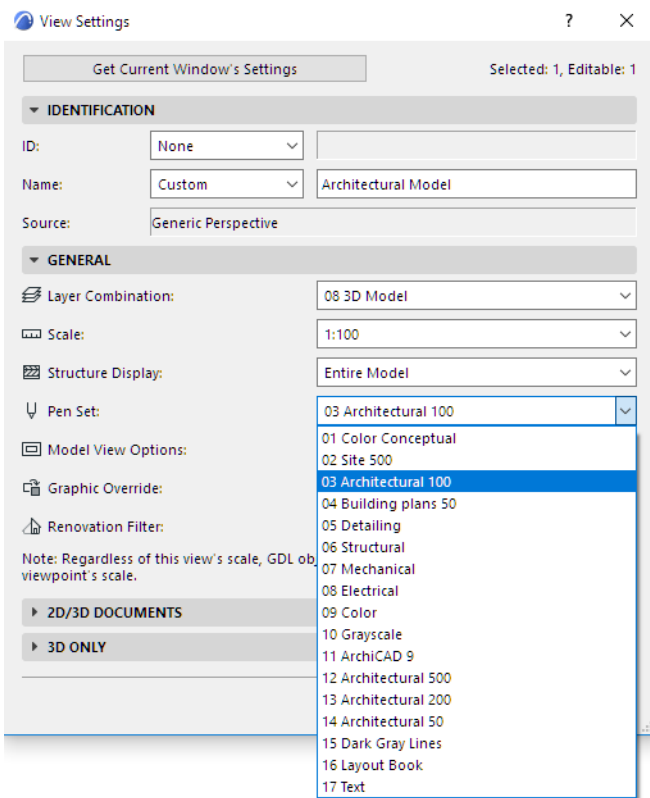


See [Drawing Size and Appearance Panel](#).

Drawings that have their source in external files - such as DWG/DXF/PDF files placed using Archicad's Drawing tool - can be assigned pen sets the same way: click the Pen Set drop-down menu and choose a different pen set, for this drawing only.

Alternatively, you can change the source view's pen set: go to its View Settings Dialog Box and choose a different Pen Set from the pop-up.

See [View Settings Dialog Box](#).



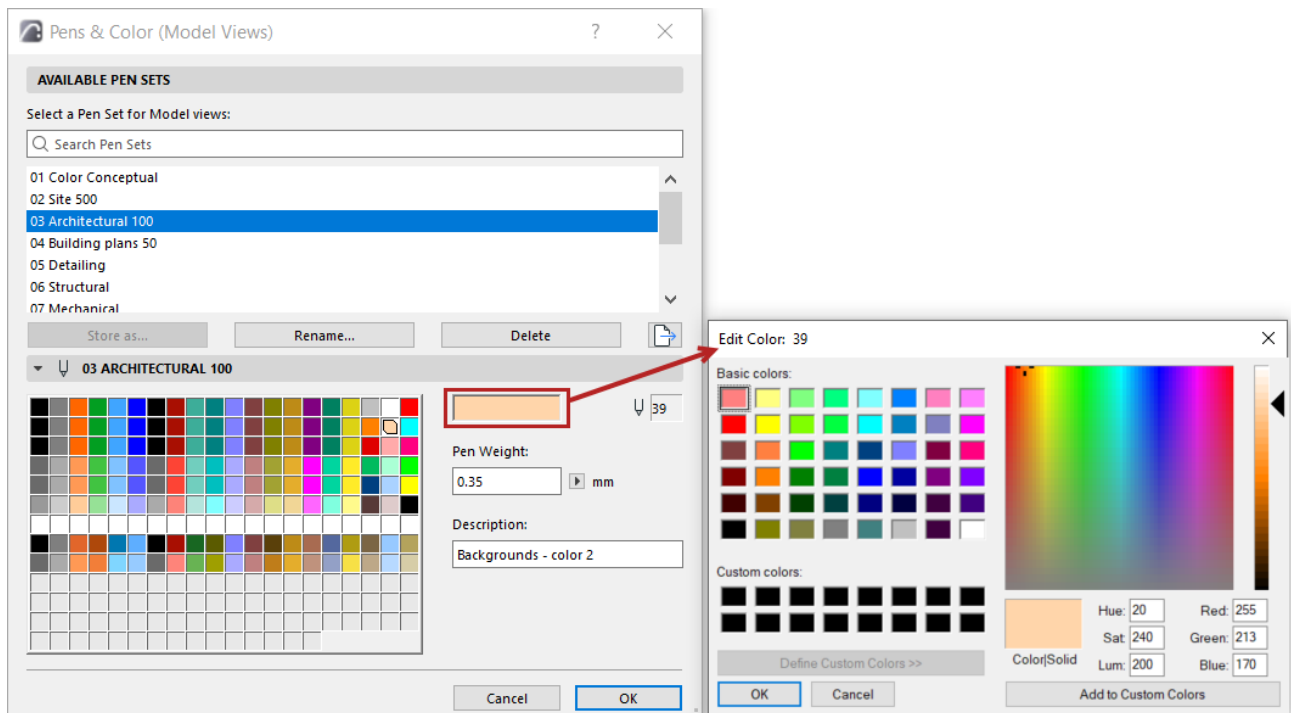
Redefine Pen Set

The predefined Pen Sets shipped with Archicad are based on common workflows.

See [Pen Sets](#) for an overview of the logic behind predefined pen sets.

However, you can use the Pens & Colors Dialog Box to

- redefine or rename any pen set together with its description
- redefine any pen's line weight or color, using the **Edit Color** control



See [Edit and manage Pen Sets](#).

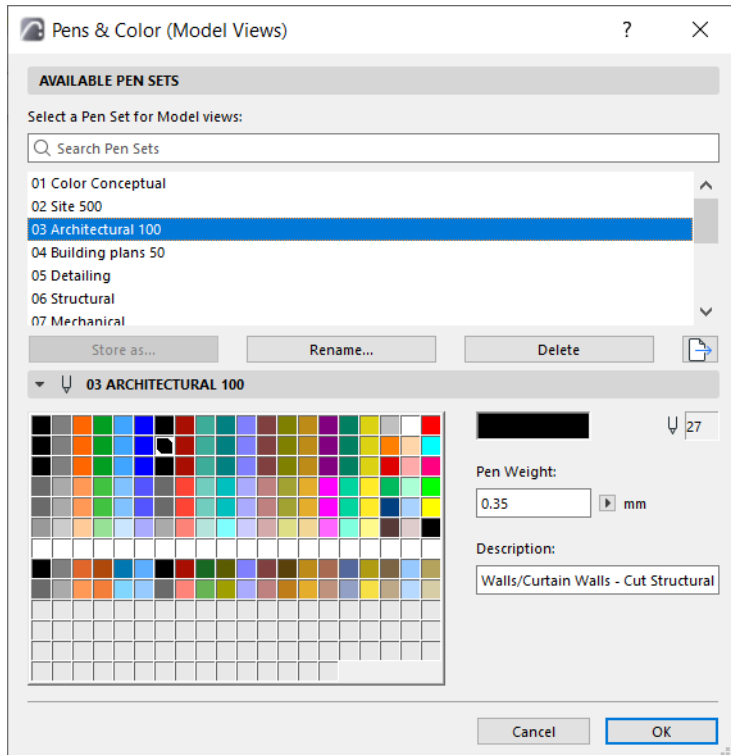
Pen numbers remain constant even if you change pen sets.

After redefining a color or changing the Pen Set, the Archicad construction elements immediately change to the new colors on the Floor Plan. The 3D window, 3D Document and Section/Elevation/IE/Worksheet windows may require that you rebuild the view.

Edit and manage Pen Sets

Open the Pens & Color dialog from:

- The Pen Sets icon in the Quick Options bar
- **Document > Pen Sets > Pens & Color** or
- Options > Element Attributes > Pens & Color

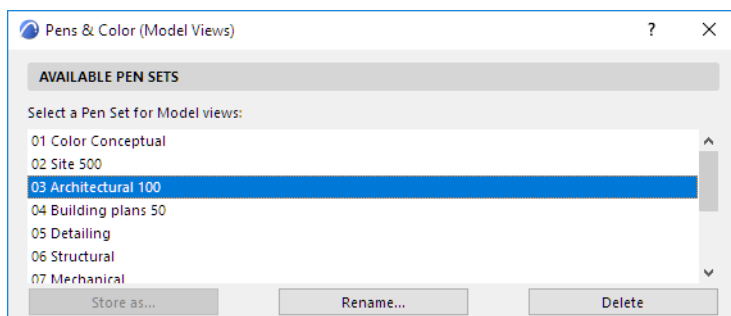


If a model view window is active, the name of the dialog box is Pens & Color (Model Views); the pen set you choose here will be applied to the model views.

If the Layout Book is active, the name of the dialog box changes to Pens & Color (Layout Book), and the pen set chosen here is applied to the Layout Book.

Available Pen Sets Panel

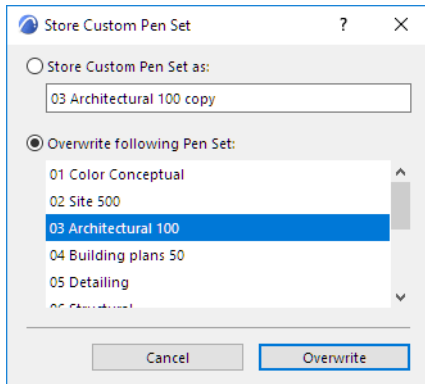
Select a Pen Set for either Model Views or the Layout Book.



- **Rename:** Click this button to give a new name to the selected pen set.

Note: The Rename button will not create another copy of the selected pen set; the new name will replace the old one.

- **Store as:** If you have edited any controls of the selected pen set, the active pen set appears as “Custom” and the Store as button becomes active. Click it to save the Custom pen set:



- **Store** the pen set under a new name (enter any name into the field); or
- **Overwrite** a selected Pen Set with your custom Pen Set

- **Delete:** Click this button to delete the selected pen set.

See [Delete attributes](#).

Selected Pen Set Editing Panel

There are 255 pens represented in the tablet. Their index numbers are indicated by their position in the tablet. The top row contains pens 1 to 20 from left to right, the row below, pens 21 to 40, and so on.

There is a default color assignment for each pen, but you can edit any color for a specific pen by double-clicking its color box or clicking the **Edit Color** button.

Note: To select continuous or discrete ranges of pens for simultaneous editing, use Shift-click and Control-click (Mac: Cmd-click), then click Edit Color.

The **Edit Color** dialog box appears, in which you can select your color graphically or numerically. This is a system-level control. (In case of multi-selection, the dialog box is called **Edit Selected Colors**.)

You can edit pen weight numerically using the fields in the **Pens and Color** dialog box. If you click on any color in a pen set, the **Description** field indicates the purpose for which the pen is optimized in this pen set.

If you have edited any of the selected pen’s controls, the name of the pen set changes to **Custom**. To save these changes, you must click **Store As** and name the pen set with a unique name, or overwrite an existing pen set.

Archicad offers user-defined **Pen Weights** associated with each of the 255 Pen Colors. Pen Weights are measured in points or millimeters, depending on the current drawing units set in the pop-up menu next to the Pen Weight edit field.

- Pen Weights will be displayed accurately only if you choose True Line Weight in **View > On-Screen View Options > True Line Weight**. (Disabling True Line Weight here means that Line Weight will be shown as Hairlines: all lines are one pixel wide only. If you also enable Bold Cut Lines, then all Cut Lines will be 2 pixels wide.)

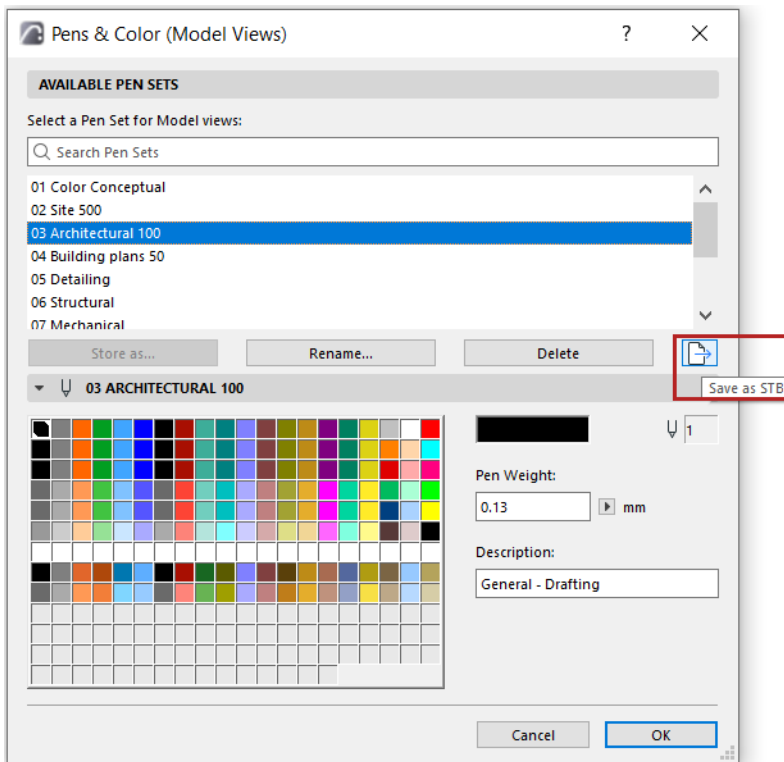
Note: The Hairline option (achieved by *disabling* True Line Weight in On-Screen View Options) affects display only. For hairline output, use the Hairline checkbox in the Print or Plot dialog box.

- Separate pens can be assigned to line arrowheads, various dimensioning components and vectored fill pattern strokes through the corresponding Tool and attribute dialog boxes.

Note: A Pen Weight of 0.00 always produces the thinnest weight possible (i.e., one pixel on the screen or a hairline on a laser printer, depending on the printer's resolution).

Save Pen Set in STB Format

Use **Save as STB** to save any single selected pen set in the format of AutoCAD Plot Style Table (*.stb).



The Plot Style Table is used by AutoCAD to map each color in your drawing to line widths on printed/plotted output.

Surfaces

Surfaces are attributes, defined at Options > Element Attributes > Surfaces.

Surfaces include a color, cover fill, and optional texture.

Surfaces defined here are assigned to construction elements through Building Materials.

Surfaces are displayed in the 3D Window, Section/Elevation/IE and 3D Documents, and in PhotoRenderings.

You can also use their color and texture in 2D Fills.

You can override the Surface at the element level: use either Surface Painter, or element settings.

Related Topics

[Edit and manage surfaces](#)

[Get more surfaces](#)

[Match surface settings: Cineware and Basic](#)

[Align 3D Texture](#)

[Override surface in element settings](#)

Edit and manage surfaces

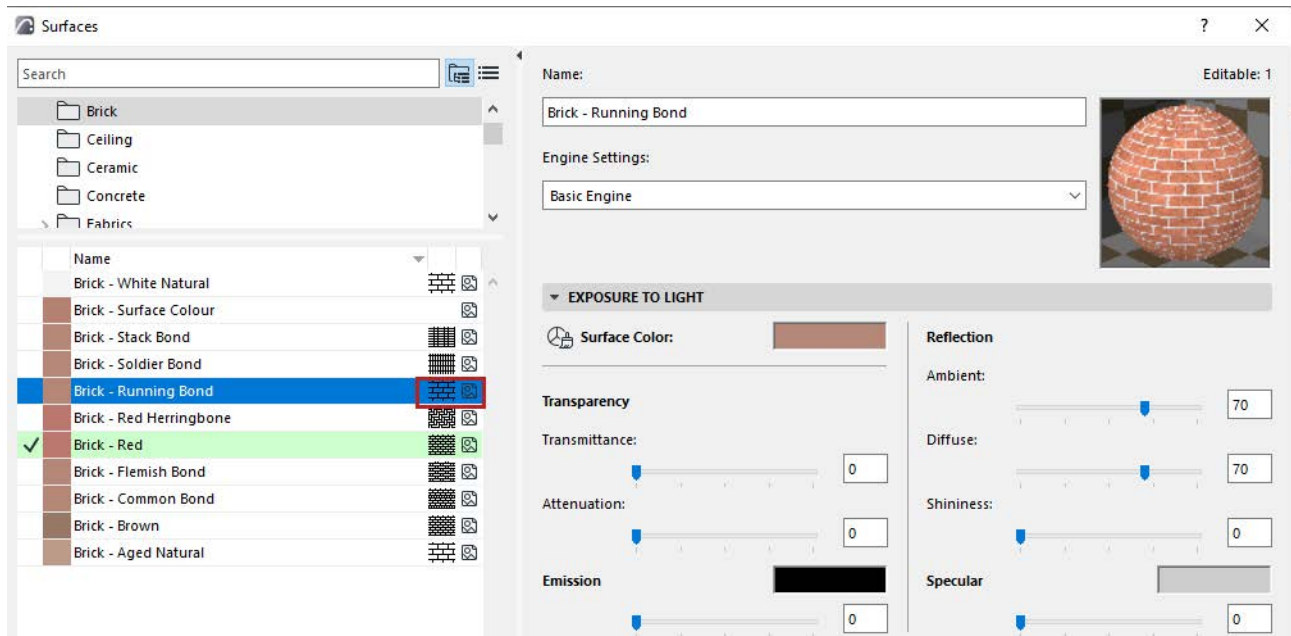
Open the Surfaces dialog:

- Options > Element Attributes > Surfaces
- Attributes toolbar (Windows > Toolbars > Attributes)

1. Select one or more surfaces to edit.

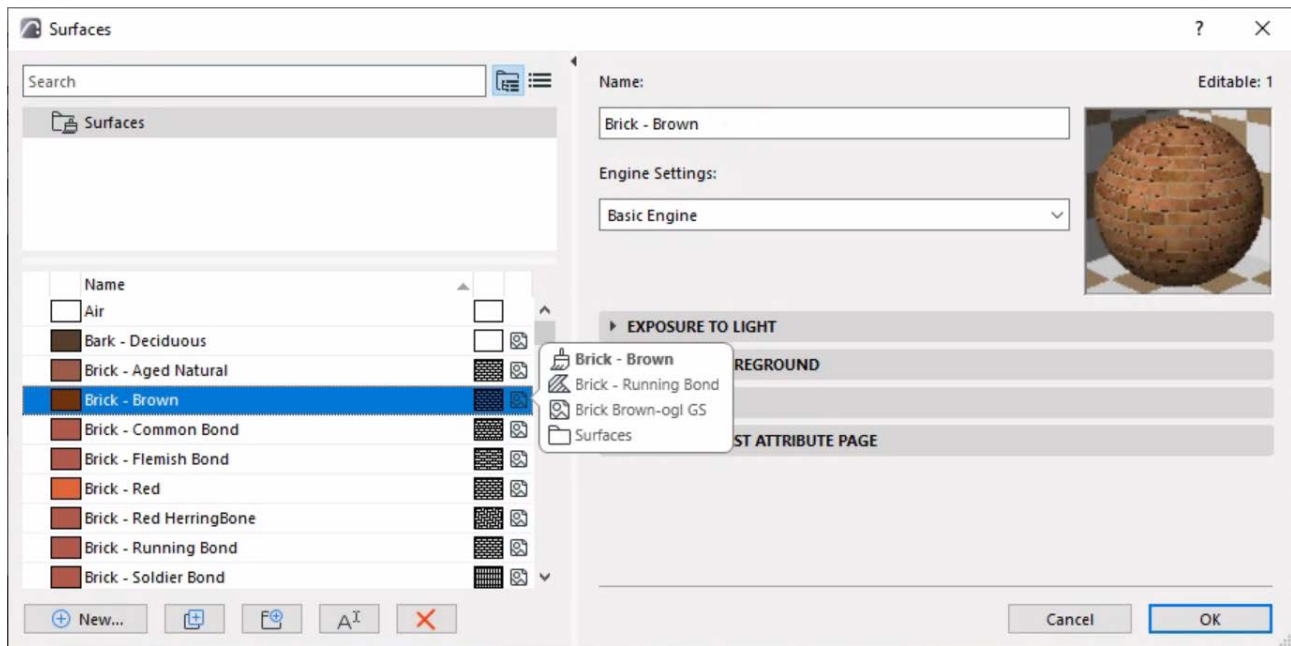
Hint: In folder view, select a sub-folder to edit all its contents at once.

2. Use the fields on the right side of the dialog box to edit the surface properties.



- Surfaces can have a cover fill and/or textures as part of their definition, as indicated by icons.
- If a texture is missing, the texture icon is displayed in red.
- Surfaces used by currently selected elements are listed with a checkmark and highlighted in green.

The **Preview** picture changes dynamically as you work in the Surfaces dialog box, so that you can see the effects of your settings.



Note: Hover over a surface to see more information on it.

For general information: see [Using the attribute dialogs](#).

Note on the GENERAL Surface

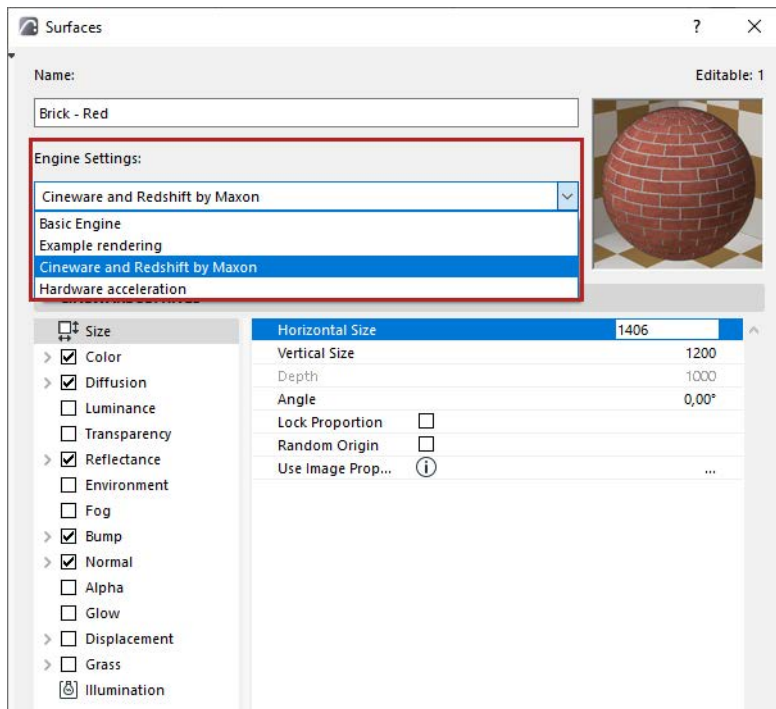
The GENERAL Surface is available for Building Materials, element-level surface overrides, and GDL. When a surface is defined as “GENERAL”, it will display a solid color based on the element’s uncut line pen.

By default, this General Surface will be assigned to new Building Materials and to new Surface parameters added within GDL. It is used to avoid assigning to an incorrect surface or creating a missing attribute. It is hard coded into Archicad and is not available in the Surfaces dialogue or the Attributes palette.

Engine Settings

Choose a rendering engine from the list.

Important: All surfaces are specific to the engine for which they are set up. If you change a surface parameter for one engine, those modifications will not take effect for the same surface parameter if you switch to another engine.



For more information on rendering engines, see [PhotoRendering Settings](#).

The panels of the Surfaces dialog are different for Basic and Cineware/Redshift engines.

They are described in the following sections.

[Surface Exposure to Light settings \(Basic Engine\)](#)

[Surface Cover Fill \(Basic Engine\)](#)

[Texture settings \(Basic Engine\)](#)

[Surfaces \(Cineware\)](#)

Surface Exposure to Light settings (Basic Engine)

(Options > Element Attributes > Surfaces)

The controls in this panel will affect renderings created by the Basic Rendering Engine and the Hardware acceleration engine.

Surface color: Define the surface's color by double-clicking the box. This opens the **Edit Color** dialog box where you can select your color graphically or numerically by mixing the HSL (Hue-Saturation-Luminosity) or RGB (Red-Green-Blue) components.

The rest of the attributes in this panel are set as a percentage of their theoretical limits. You can edit the values either numerically or with a sliding switch.

Transparency attributes control how light penetrates the selected surface. Use the following two controls to adjust transparency:

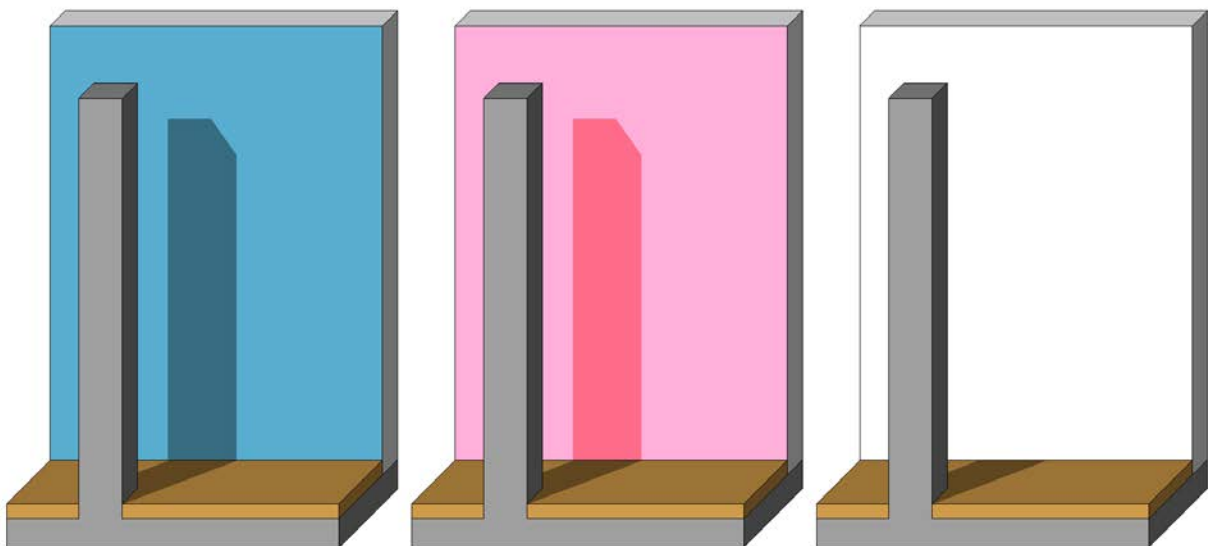
- **Transmittance:** The higher the value, the greater percentage of light is transmitted through the surface. Glass is at the top of the scale and opaque surfaces are at the bottom.
- **Attenuation** controls how sharply transmittance diminishes as the object's surface becomes less perpendicular to the viewing direction. Any number in a continuous range from zero to 100 can be set.

Emission is a measure of a surface's fluorescence (i.e., the light emitted by its surface).

- Use the **Attenuation** control to adjust the emission intensity of the selected surface. Emission intensity decreases as attenuation increases (i.e., the viewpoint is moved farther from the object).
- Click the **Emission color** control to bring up the Edit Color dialog box. Select your color graphically or numerically by mixing the HSL (Hue-Saturation-Lightness) or RGB (Red-Green-Blue) components.

Note: If shadows don't appear on a surface where they should, check the Emission color. If a surface's Emission color is white, then shadows will not appear. Changing the Emission color also affects the color and intensity of shadows.

In the image below, the first two surfaces have Emission color set to black and red; the third surface has an Emission color of white.



The Surface's **Reflection** has three attributes:

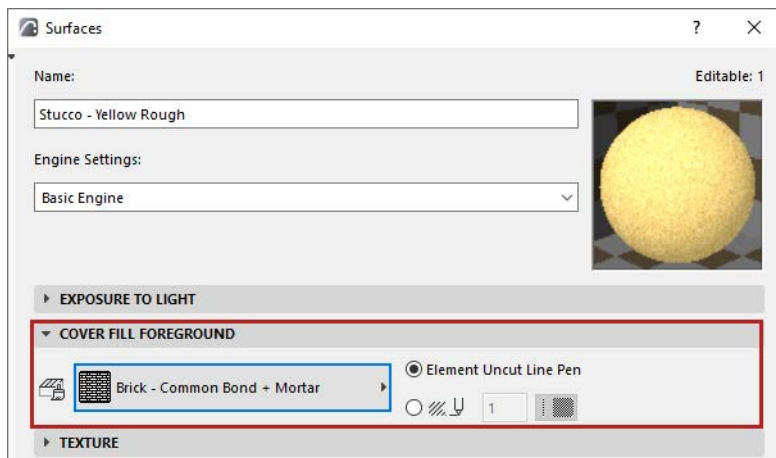
- The **Ambient** control sets the percentage of ambient light the surface is able to reflect. This determines how much a surface is subject to brightening effects (as opposed to shades or highlights caused by directional light).
- The **Diffuse** control describes the surface quality of the surface. Uneven or rough surfaces tend to reflect incident, directional light in a non-directional, diffuse way. Diffuse reflection carries the color of the surface rather than the color of light.
- **Shininess** is the ability of a curved surface to reflect directional light from a point-like source (sun, flashlight, spots, etc.), in a more or less focused fashion. The more focused the reflection, the shinier the surface appears. Highlights are confined to a small area if this control is set close to 100, while a wider highlight area will result if the control is set to a low value.

The Surface's **Glowing** is determined by its Specular setting.

- The **Specular** setting is the opposite of Diffuse. It is a directional reflection carrying the color of the incident light rather than that of the surface. An extremely smooth and saturated hard surface will produce strong specular reflections (set at close to 100%), while a soft and light surface will produce hardly any (set at close to 0%).
- **Specular Color** is theoretically identical to the color of directional light, but the program lets you define it independently. Double-click the sample color here to display the system-level **Edit Color** subdialog box, in which you can establish your definitions. This color, mixed with the Surface color and the color of the light, will determine the color of the surface where directional light is reflected.

Surface Cover Fill (Basic Engine)

(Options > Element Attributes > Surfaces)



Choose a Cover Fill foreground for the chosen surface.

Notes:

- Only Fills defined as Cover Fills are available. ([See Fill Categories.](#))
- This Cover Fill does not affect PhotoRenderings, so the Preview window (generated by the chosen PhotoRendering engine) will not reflect the modifications you make in this panel.

Element Uncut Line Pen

Click to display the vector pattern in the color of the fill's uncut line pen.

Alternatively, choose any other color from the pencolor pop-up.

Display Surface cover fill in 3D Window

1. Use the Vectorial Engine (View > 3D View Options > 3D Styles)
2. the Vectorial Hatching checkbox in 3D Styles must be On

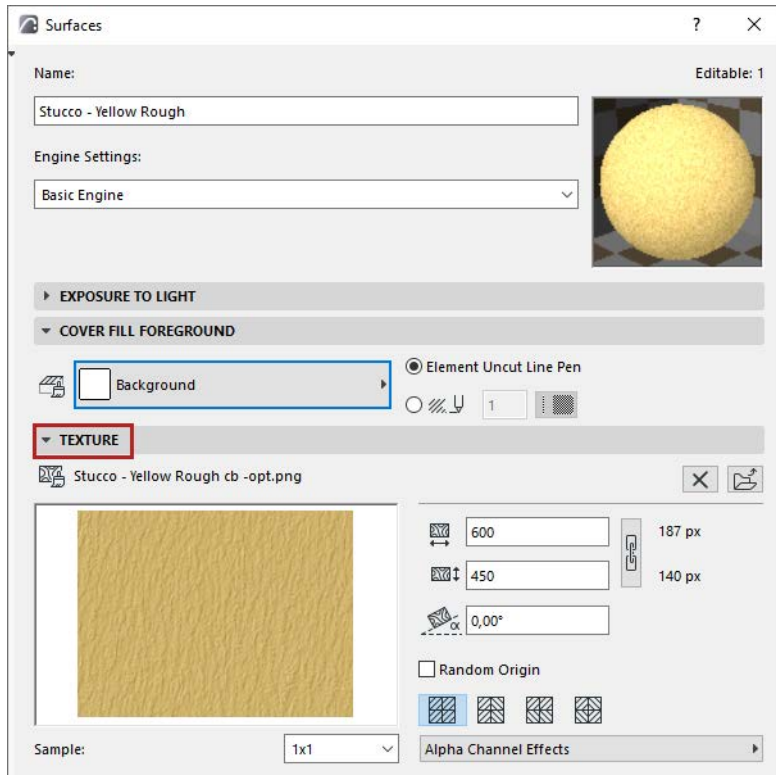
Texture settings (Basic Engine)

(Options > Element Attributes > Surfaces)

Use the **Texture** panel to link a texture to a surface attribute.

These surface settings are specific to the Basic rendering engine.

For Cineware texture settings, see: [Textures \(Cineware Surfaces\)](#).



Topics in this section:

[Choosing and editing a Texture](#)

[Define texture display](#)

[Alpha Channel effects](#)

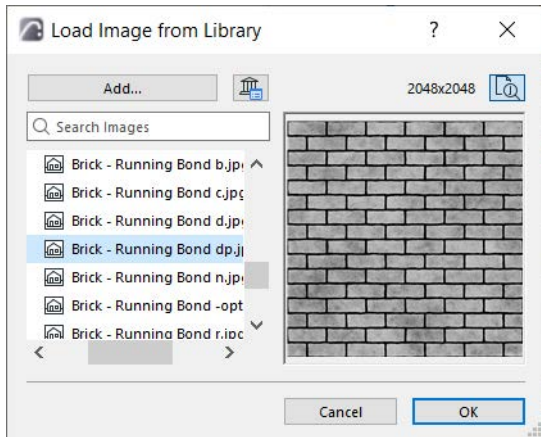
[Display Surface textures in 3D Window](#)

Choosing and editing a Texture

If the selected Surface already has a texture linked to it, the texture's file name and preview picture are displayed, along with the image size in pixels.

To assign a texture to a Surface:

1. Click the **Browse** button to open the **Load Image from Library** dialog box and find the needed texture map.
2. Click **OK** to assign the texture.

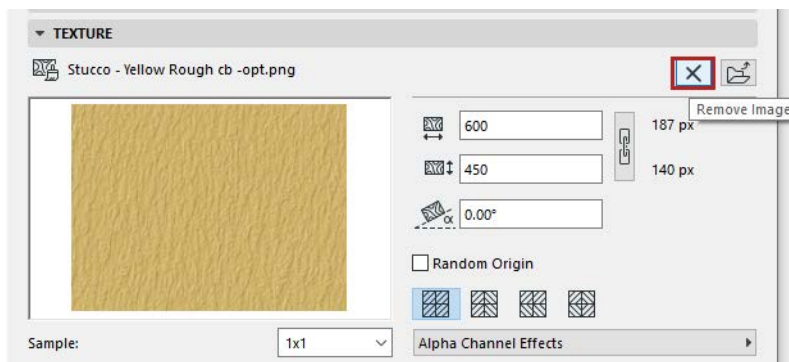


See also [Texture files - naming conventions](#).

Notes:

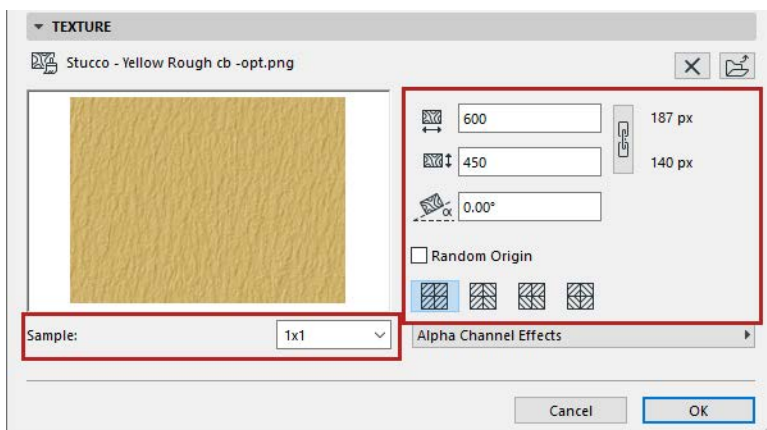
- You can only browse files from project libraries.
- If you save your Archicad project in archive format, you can use the **Include Linked Textures** checkbox (see [Archive Projects](#)).

Click **Remove Image** to unlink the Texture from the Surface.



Define texture display

- Define the vertical/horizontal size of the texture as applied to 3D surfaces.



- Check **Keep Original Proportion** to keep the proportions of the original texture file for all elements.
- Define the **Rotation angle** at which the texture is applied to elements.
- Use the four **Distribution** radio buttons to arrange the texture sequence.

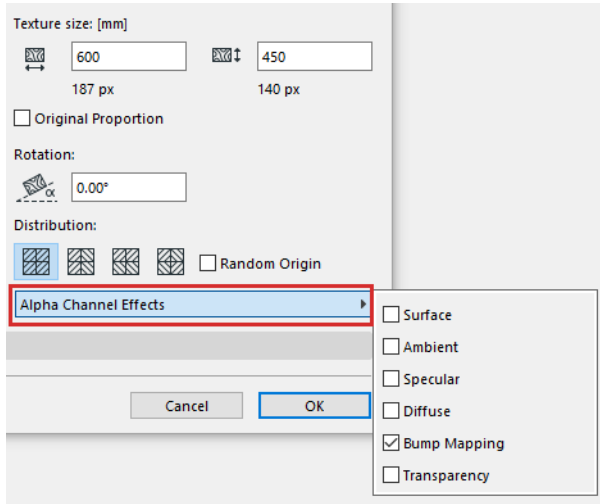
Note: For good results, you need textures prepared for seamless tiling. These are images that can be placed in a matrix with no apparent line at connecting edges.

- Activate the **Random Origin** checkbox to use a random starting point for applying the texture. This option is useful if there are several identical elements in the model that should differ in appearance.

Use the **Sample** pop-up list to adjust the number of texture units displayed in the Preview box - this helps you get a better idea of what the texture will look like.

Alpha Channel effects

Use this pop-up in the Texture panel for **Alpha Channel** effects.



In 32-bit color images, the Alpha Channel is an extra channel (in addition to RGB components) that can be edited in image editing applications such as Adobe Photoshop. Archicad allows you to take advantage of the information placed in the Alpha Channel in several ways:

- for masking elements in PhotoRenderings.
- for transparency or bump mapping.
- to model certain surface properties of the texture.

Surface

If the alpha value is white, then you get the color of the texture, while if it is black, then the surface's original color will be valid in the given pixel of the given surface. If the value is a shade of gray, then the two colors will be mixed and the result will appear in the PhotoRendering.

Ambient

The same as above, but applied to the Ambient color instead of the Surface color.

Specular

Controls to what extent the Specular effect is taken into account. If the alpha value is white, the Specular effect is entirely taken into account at the given pixel. If the alpha value is black, the Specular effect is entirely disregarded at the given pixel. Here again, intermediate values are allowed.

Diffuse

Controls the effect of diffuse light the same way as with Specular light.

Bump Mapping

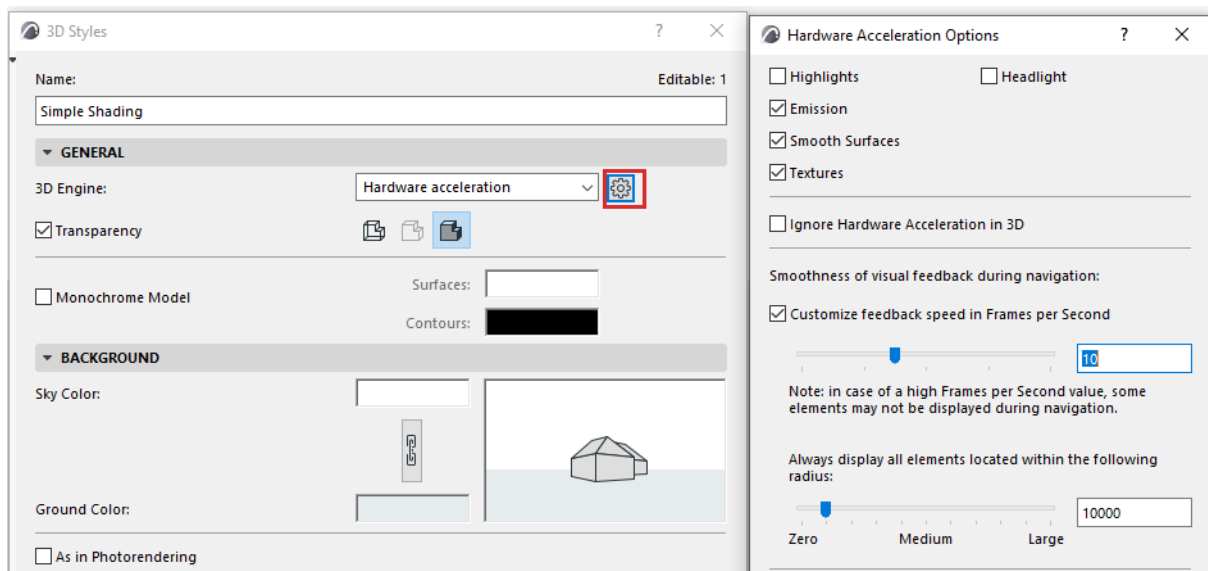
Controls the surface normal at the given pixel, allowing for a convex effect on the surface. White values represent bumps, while black values represent holes.

Transparency

If the value is white, then the color of the texture is visible; if it is black, the texture will be perfectly transparent at the given pixel. Intermediate values are allowed.

Display Surface textures in 3D Window

- Use the Hardware acceleration engine (View > 3D View Options > 3D Styles)
- Make sure the Textures box is checked in Hardware Acceleration Options (click Advanced Options from 3D Styles)



Note: You can customize the orientation and origin of textures in 3D views.

See [Align 3D Texture](#).

Get more surfaces

Add additional surface attributes to your project from a catalog or from BIM Components.

[Add new surface from catalog](#)

[Replace surface settings from catalog](#)

[Texture files - naming conventions](#)

[Search for surfaces in catalog](#)

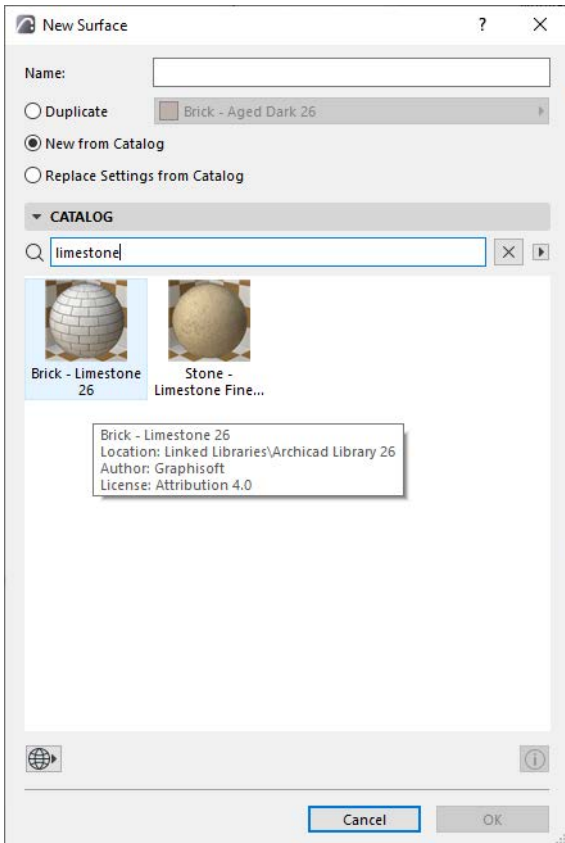
[Additional Surface Catalog for SSA/Forward Users](#)

[Load Cinema 4D material](#)

Add new surface from catalog

Use this to replace the current surface with a surface imported from the Catalog.

1. Open **Options > Element Attributes > Surfaces**
2. Click the **New...** button
3. Choose **New from Catalog**



4. Browse or search the catalogs to choose a surface.

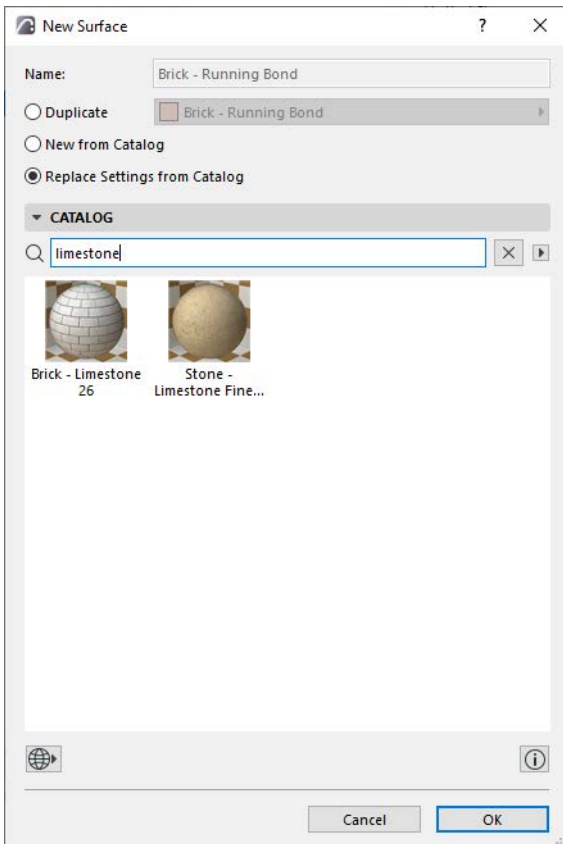
See [Search for surfaces in catalog](#).

5. Click **OK**.
 - The new surface attribute is added to your project
 - Any associated texture file is added to your Embedded Library, and any fill (vectorial hatch) is added to the project's fill attributes.

Replace surface settings from catalog

Use this to replace the *settings only* of the current surface with those of a surface selected in the catalog. The original surface's name, index number, and vectorial hatch remain unchanged.

1. Open **Options > Element Attributes > Surfaces**
2. Click the **New...** button
3. Choose **Replace Settings from Catalog**



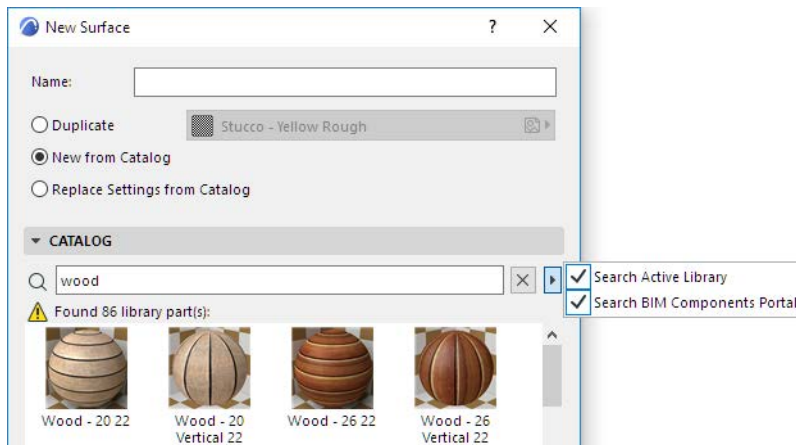
4. Browse or search the catalogs to choose a surface.

See [Search for surfaces in catalog](#).

5. Click **OK**.

Search for surfaces in catalog

1. Choose where to search:



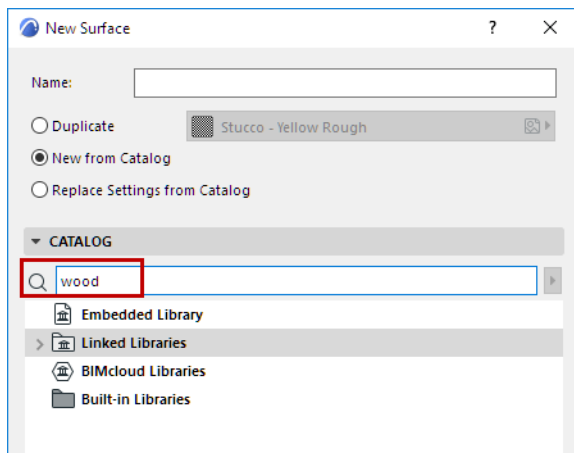
- **Search Active Library:** Includes linked libraries, the Embedded Library, and BIMcloud Libraries you have downloaded
- **Search BIM Components Portal:** Searches on BIM Components, a searchable and interactive depository of objects contributed by Archicad users or uploaded by Graphisoft.

For more information, see [BIM Components](#).

2. Enter any string of characters in the Catalog's search field.

The search will look in the following strings of surfaces:

- Library Part name
- Library name
- Description
- Subtype
- Name of author of Library Part
- Keywords

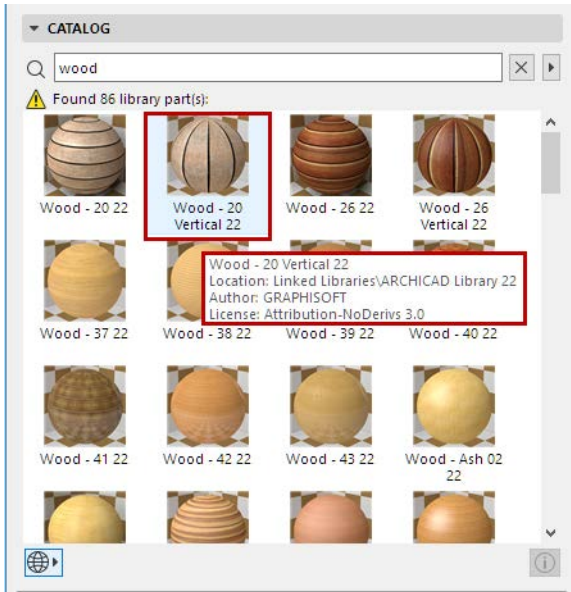


3. Click **Enter**, or just stop input.

Search results

Surfaces found in the active libraries are shown first. The dialog box will display a maximum of 50 of found items; if more are found, see the remaining hits by clicking on the **More Surfaces on BIM Components.com** button at the bottom of the dialog box.

Hover the cursor over any item (or select it, then click the info button at the bottom right of the dialog box) to see an information tag.

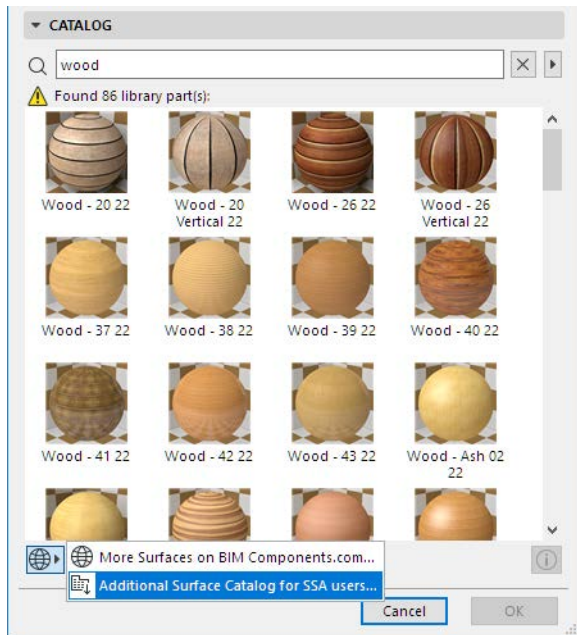


- BIM Components-based hits are distinguished from active library hits with a globe icon.
- BIM Components-based hits marked with a key are available only to users with some type of premium service agreement. (This may vary by market; contact your distributor.)

Additional Surface Catalog for SSA/Forward Users

Archicad users with an active SSA/Forward subscription can access additional surfaces, available as one-click downloads to the current project.

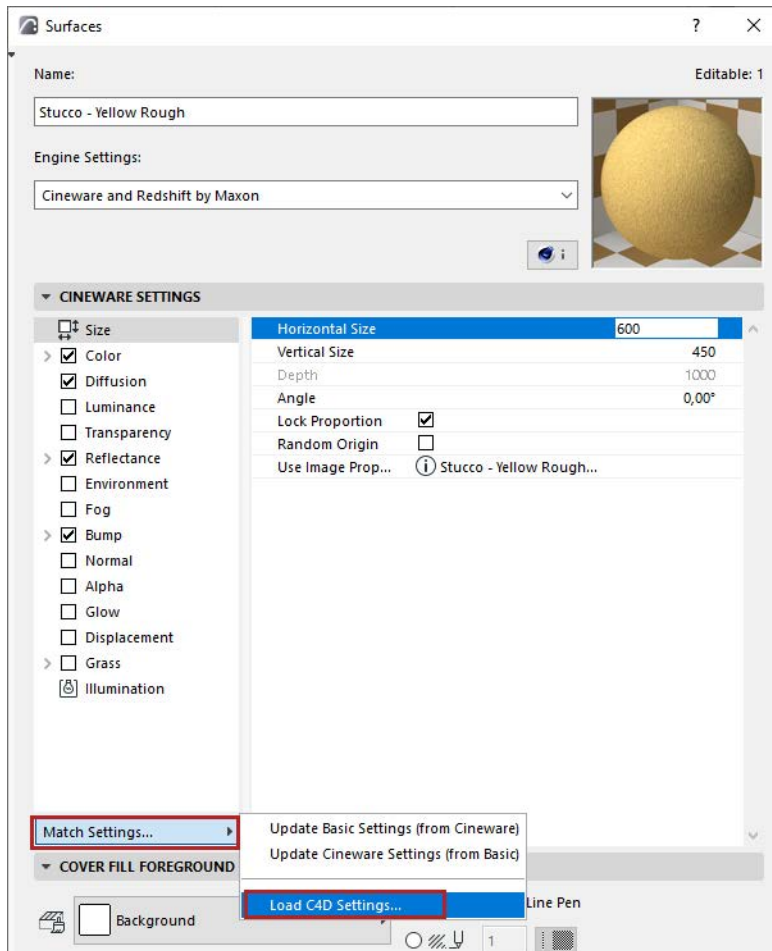
This premium surface catalog is available to SSA/Forward users directly from the Add New Surface dialog box: click the **Additional Surface Catalog** command from the pop-up button at the bottom of the dialog box. This command will check your license eligibility, then redirect you to the expanded surface catalog (if you are eligible) or to more information on the required license (if you are not).



Load Cinema 4D material

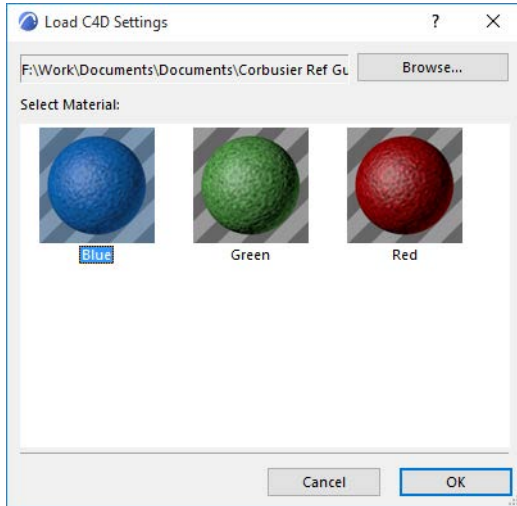
Use this command to import the settings of a specific Cinema4D material from an external file; these settings will be loaded for the current surface in Archicad.

From the Match Settings pop-up in Surfaces, choose **Load C4D Settings**.



In the appearing dialog box, browse for a Cinema 4D file. This can be either a single material or an entire model in Cinema 4D format.

If you choose a model file, a chooser dialog box appears: select the Cinema4D Material you need.



Click **OK** to load the chosen material. Its settings override those of your current surface.

Notes:

- The maximum number of parameters that can be used in a surface is 1024. ([See *Limit on Number of Surface Parameters*](#)).
- Node-based Materials from CINEMA 4D are not supported in Archicad. Such imported materials will be displayed with a specially generated material instead.

Match surface settings: Cineware and Basic

All surfaces are specific to the engine for which they are set up:

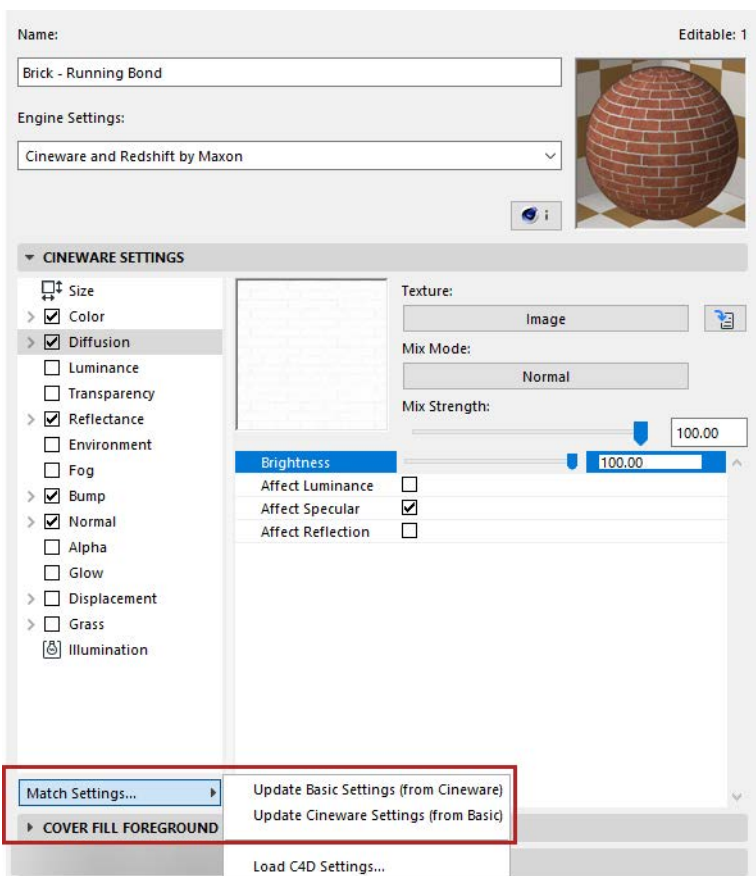
- Basic (including Hardware acceleration)
- Cineware and Redshift by Maxon

Generally, Cineware contains a larger number of more sophisticated parameters than the Basic or Hardware acceleration engines (which are very similar to each other).

If you change a surface parameter for one engine (e.g. Basic), those modifications will not take effect for the same surface parameter if you switch to another engine (e.g. Cineware).

Two commands are available to help synchronize the appearance of surface defined in Cineware with its appearance for the Basic (or Hardware acceleration) engine. Note that due to the difference in parameters, the result might only be an approximation. Generally, the basic parameters of simple surfaces will be matched accurately.

From the Match Settings pop-up in Surfaces:



Update Basic Settings (from Cineware)

This will apply the Cineware surface settings to its surface counterpart in Basic/Hardware acceleration.

Note: Because the Basic/Hardware acceleration engine uses relatively few parameters, the effect of this command may be minimal. In particular, the parameters of procedural surfaces (in Cineware) cannot be replicated in Basic/Hardware acceleration.

Update Cineware Settings (from Basic)

This will apply the Basic/Hardware acceleration surface settings to its surface counterpart in Cineware.

Limit on Number of Surface Parameters

Because of the many layers of shaders available for Cineware surfaces, you could, in theory, set an unlimited number of parameters for a given surface.

However, in Archicad, the maximum number of parameters that can be used in a surface is 1024. (Archicad uses the surface attribute in GDL objects, too, and the GDL programming language sets a limit of 1024 parameters.)

You may encounter this limit

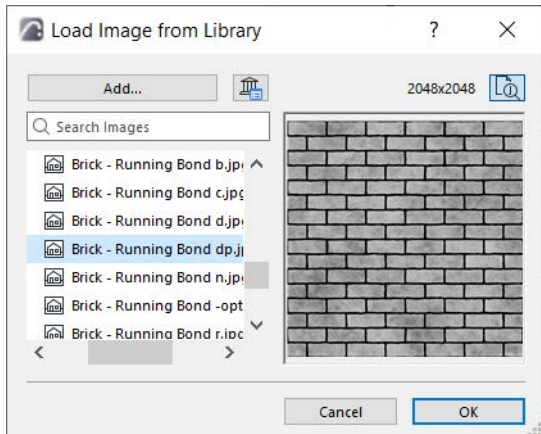
- when loading a surface from a Cinema 4D file (see Load Cinema 4D Settings, above)
- when configuring your own custom surface

The solution is to limit the number of shaders used in the surface, and/or to use a more simple shader structure (limiting the number of embedded layers).

Texture files - naming conventions

Surfaces contain references to texture files in the library. These texture files are differentiated by name only, disregarding their file extension.

The file names of many textures in the Surface Catalog contain postfixes which help to distinguish their usage:



- c for images in the Color channel
- d for Diffusion
- l for Luminance
- t for Transparency
- r for Reflection
- ro for Roughness
- e for Environment
- f for Fog
- b for Bump
- n for Normal
- a for Alpha
- s for Specular
- gl for Glow
- dp for Displacement
- gr for Grass
- -opt for images used by the Hardware acceleration and Basic Rendering Engine

If an image is used by multiple channels, then the postfixes are extended, e.g.: cs for color and specular channel usage.

Align 3D Texture

Use the commands at **Document > Creative Imaging > Align 3D Texture** to edit the origin and direction of construction element textures in 3D views.

These commands are only available in the 3D Window on selected construction elements whose Surface attribute includes a Texture assignment (**Options > Element Attributes > Surfaces**).

Notes:

- Not available for Shells.
- Texture realignment on elements with a Complex Profile might produce unexpected results.

For extra texture editing options on Morph faces, see [Morph Texture Mapping and Alignment](#).

See also [Default Texture Alignments/Reset Texture](#)

Related Topics

[Set Texture Origin](#)

[Set Texture Direction](#)

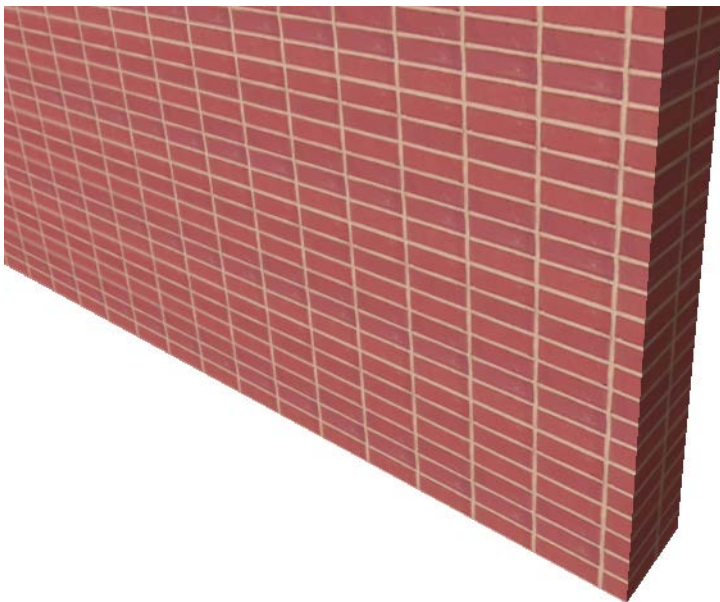
[Reset Texture](#)

[Align Edge Texture \(Walls Only\)](#)

[Default Texture Alignments/Reset Texture](#)

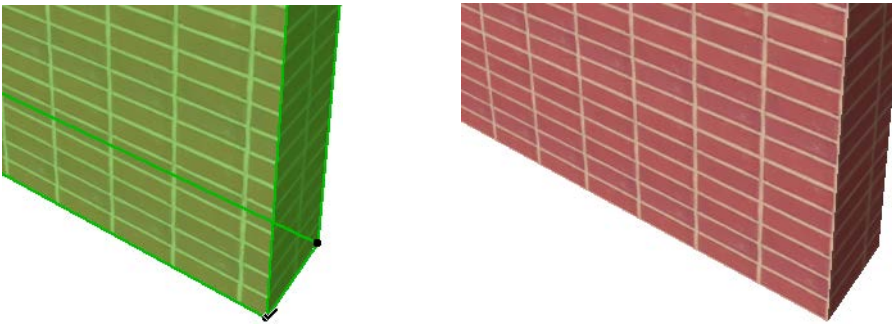
Set Texture Origin

In this example, the brick wall texture originally looks like this:



To reset the texture origin to the corner of the wall - so that the brick pattern starts from the corner - do the following:

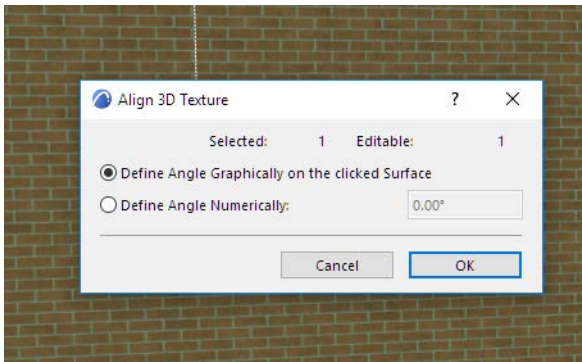
1. Select the wall
2. Use **Document > Creative Imaging > Align 3D Texture > Set Origin**.
3. Click on the corner and view the result.



The texture origin will affect all of the elements' surfaces. You cannot, for example, use a different origin for the inside and outside surfaces of a wall.

Set Texture Direction

1. Select one or several elements.
2. Use **Document > Creative Imaging > Align 3D Texture > Set Direction**.
3. Click on a surface of a selected element. You must click a “main” surface, not an edge. (For Columns, the editable “main” surface are the top and bottom surfaces.)
4. In the dialog box that opens, choose to either define the texture's angle graphically or numerically.
5. Click OK.

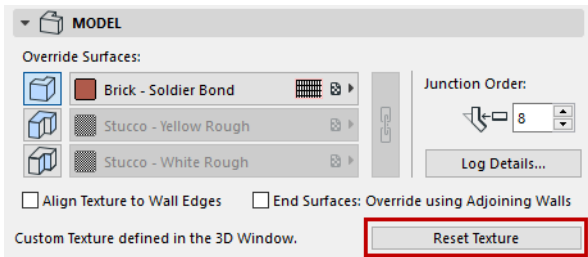


6. If you chose to define the angle graphically, draw the desired vector for the texture orientation. You can snap to element hotspots.
7. If you define the angle numerically, it will be applied relative to the default orientation of the Texture and measured counterclockwise.

Note: For Slabs and Roofs, texture direction will change automatically if you edit the element shape.

Reset Texture

To return to the default Texture settings, select the element, open its Settings dialog box and go to the **Model** panel. The **Custom Texture** information text is active. Click the **Reset Texture** button to discard the customization.



You can also select several elements and choose the **Reset** command in the **Document > Creative Imaging > Align 3D Texture** hierarchical menu.

To remove all texture customization of the elements of the 3D Window, do not select any of them and choose **Document > Creative Imaging > Align 3D Texture > Reset All**.

[See also *Default Texture Alignments/Reset Texture*.](#)

Align Edge Texture (Walls Only)

You can align the texture of a Wall edge only: Use the “Align Texture to Wall Edges” checkbox in the Model panel of Wall Settings.

[See *Wall Model Panel*.](#)

Note: Using the Align Texture command (Document > Creative Imaging) will override any other Wall texture settings, including the Align Texture to Wall Edges checkbox.

Default Texture Alignments/Reset Texture

Default texture alignments vary by element.

The “Reset Texture” command is available for selected elements if you have changed their texture alignments, either by using the Align 3D Texture command, or (as with Slabs and Roofs) by editing element shape.

The following table indicates how textures are aligned (by default and after reset).

Element	Default Texture Alignment	“Reset Texture” Result
Wall/Beam	Parallel to Reference Line	Parallel to Reference Line
Column	Perpendicular to Reference Line	Perpendicular to Reference Line
Slab	Aligned with longest edge	Realigns with longest edge
Mesh	Aligned with longest edge	Realigns with X Axis
Single-plane Roof	<ul style="list-style-type: none"> Parallel to Pivot Line The tile texture on a slanted roof always faces downward (as you draw the pivot line from A to B, the right-hand side is “downward”). Realigns automatically after editing Roof contour 	For angled roof: Realigns parallel to Pivot Line For flat roof: Realigns to longest edge
Zone	<ul style="list-style-type: none"> Aligned with longest edge Realigns automatically after editing Roof contour 	N/A

Related Topics:

[Align 3D Texture](#)

[Display Surface cover fill in 3D Window](#)

Override surface in element settings

For a model element, the surface setting depends on its Building Material, which includes a surface.

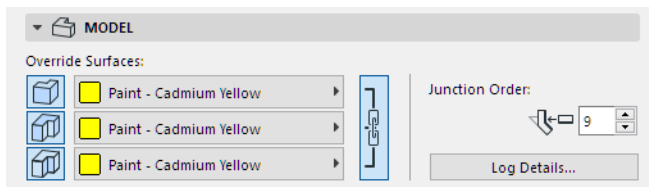
See [Building Materials](#).

However, for any model element, you can override the Building Material surface definition, and assign a different surface (or surfaces).

GDL objects do not use Building Materials, but you can override their surface definitions.

1. Select a placed element, or open the Default Settings dialog box for the element type.
2. In the Model panel, under Override Surfaces, click the toggle button of any or all of the surface pop-ups. Each pop-up corresponds to a face or edge of the element.

Note: Clicking the Chain icon (“Link Surfaces”) means that all of faces/edges will use a single surface override. To set a separate surface for any single face/edge, unlink the surface definitions by clicking the chain again.



To remove an override, just click the toggle button again. The surface defined in the Building Material is now used again.

See also [Override Surfaces With Surface Painter](#).

Override Surfaces With Surface Painter

For a model element, the surface setting depends on its Building Material, which includes a surface.

See [Building Materials](#).

You can override a construction element's Building Material surface definition, and assign a different surface (or surfaces).

Although GDL objects do not use Building Materials, you can still change their surface definitions.

There are two workflows for overriding an element surface:

- Use **Element Settings** (see [Override surface in element settings](#))
- Use the **Surface Painter** palette, described below.

Note: Before using the Surface Painter feature, make sure you are *not* working in “legacy” intersection mode. (Uncheck this option at **Options > Project Preferences > Legacy**.)

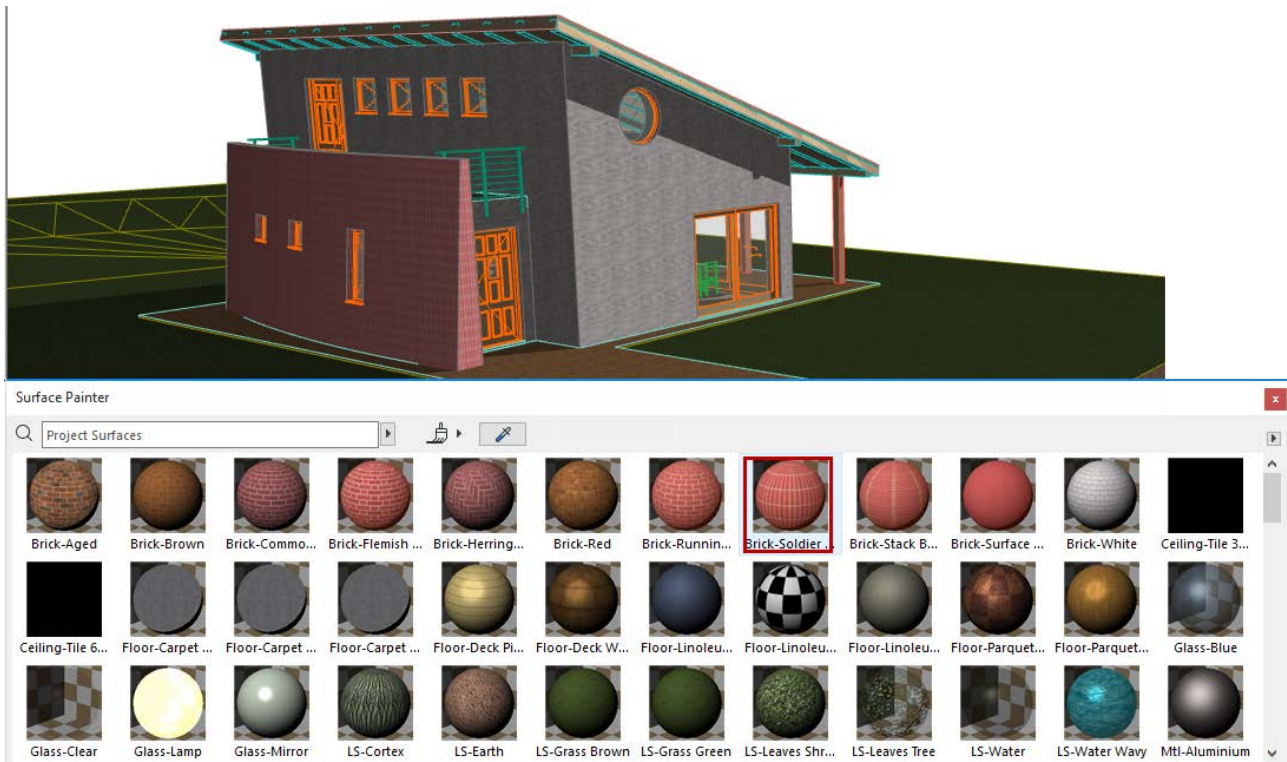
[Watch the Video](#)

Override Single Element with Surface Painter

1. Activate the 3D window.
2. Open the Surface Painter palette (**Window > Palettes > Surface Painter**).

For more information on the palette and its controls, see [Surface Painter Palette](#).

3. Choose the surface you want to use as an override. Do one of the following:
 - In the content pane of Surface Painter, click on a surface to select it

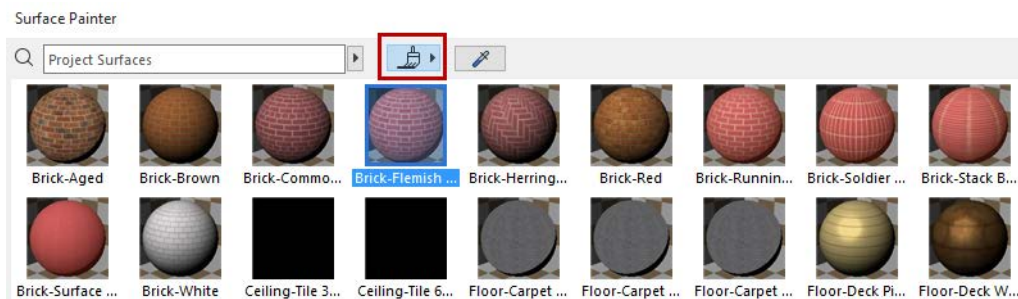


- Alternatively, click the **Pick up Surface** icon in Surface Painter, then click on any element surface in the 3D window to select it in Surface Painter.



- By selecting the surface to use as override, you automatically enter Paint mode (the paintbrush icon at the top of the palette is enabled).

Note: The default Paint Mode means that you will exit Paint Mode automatically after a single painting interaction. If you wish to paint multiple items without exiting Paint Mode, use the Infinite Paint mode: [See “Infinite” Paint Mode, below.](#)



Note: While you are in Paint mode, click **Alt** at any time to enable the **Pick up Surface** function. This way, you can paint using a picked-up surface.

[See also Search for Surface \(Surface Painter\).](#)

- Hover your cursor over the surface you wish to override. Notice the feedback:
 - The hovered face of the element whose surface will be overridden is highlighted.
 - Next, the highlighted face changes temporarily, to give you a preview of what it will look like after you click.

- The Info Tag shows which surface will override which. As needed, click **Tab** to cycle among the faces, or to select all faces/edges.

Note: If the clicked element has faces using several different surfaces, you can choose “All Similar Faces”: that is, you will override not just the clicked face, but all element faces which use the same surface.



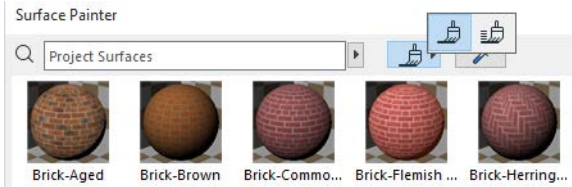
6. Click with the paint cursor to override the surface of the clicked element (or one of its faces).
The result is also shown in the PhotoRendering Palette preview (if the preview is set to Auto Refresh).
7. You now automatically exit Paint Mode.
Note: If want to keep painting without exiting automatically, you should use the “infinite” Paint Mode, described below.

Further Notes on Surface Painter Overrides

- If you override edges, *all* the edges are overridden uniformly, including any custom edges of a roof or slab.
- If you change any surface in a GDL object, *all* parts of the GDL object which use that surface will be changed uniformly. In other words, you cannot paint only a particular face of a GDL object; only the “All Similar Faces” option applies.
- GDL objects whose surface depends on a custom Model View Options definition cannot be overridden using Surface Painter.
- For Profile elements: The same override options are available as in the Model panel of element Settings: “Extrusion surface” and “Ends surface.” Varying extrusion or end surfaces cannot be overridden individually.

“Infinite” Paint Mode

Before beginning to paint, set the Paint Mode icon to its “infinite” mode.



Use this Paint Mode if you want to paint a whole series of elements using Surface Painter. This way, you will not exit Paint Mode until you do so explicitly.

Exit Paint Mode

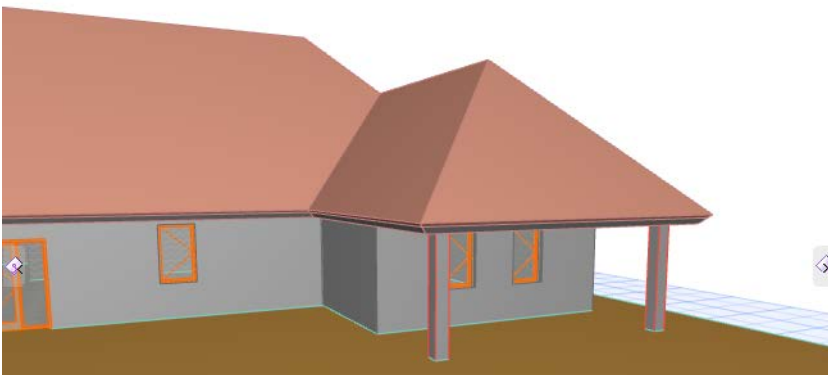
Do one of the following:

- press **Esc**
- press **Cancel** from the context menu
- click the **Paint mode toggle** in Surface Painter

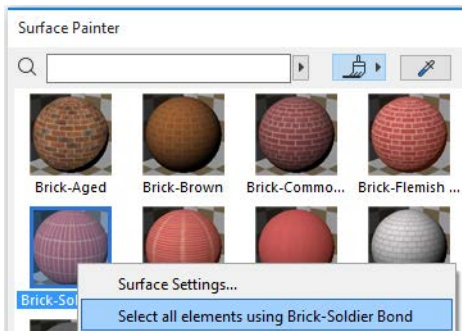
You only need to exit if you are in “infinite” Paint Mode. If you use the default Paint Mode, you will exit automatically.

Paint mode is temporarily suspended while you navigate (zoom, pan, orbit) in the window.

Override Multiple Selected Elements with Surface Painter

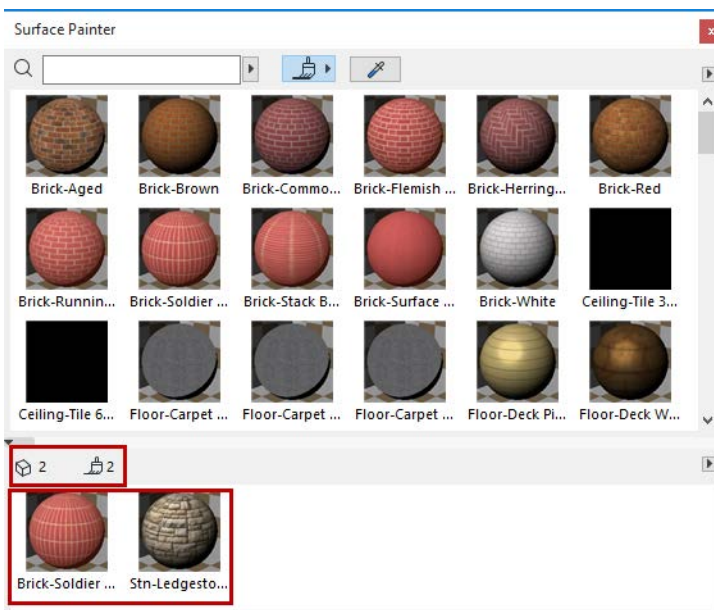


1. Activate the 3D window.
2. Open the Surface Painter palette.
3. **Select the elements whose surface you wish to override.** Do one of the following:
 - From the 3D window, select each element you need, one after the other, or
 - Alternatively, select *all* project elements that use a particular surface: right-click the surface from the content list and use the **Select all elements using...** command.

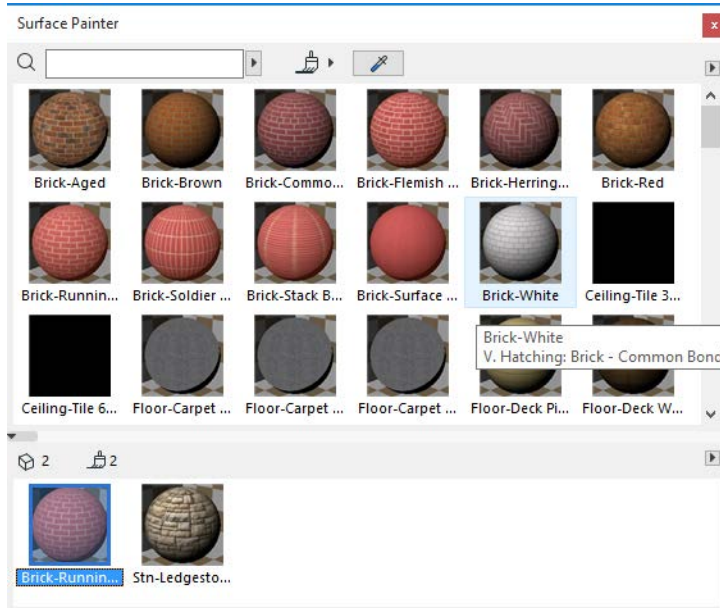


Note: To find a particular surface in the Surface Painter content list, you can use the **Pick Up Surface** function on an element which uses that surface.

- All surfaces of all selected elements are now listed in the Selection Panel of Surface Painter. (The number of selected elements, and the number of surfaces, is shown.)



- From the Content Panel (listing all project surfaces), choose the desired new surface.
- In the Selection Panel, click the surface that you wish to override.



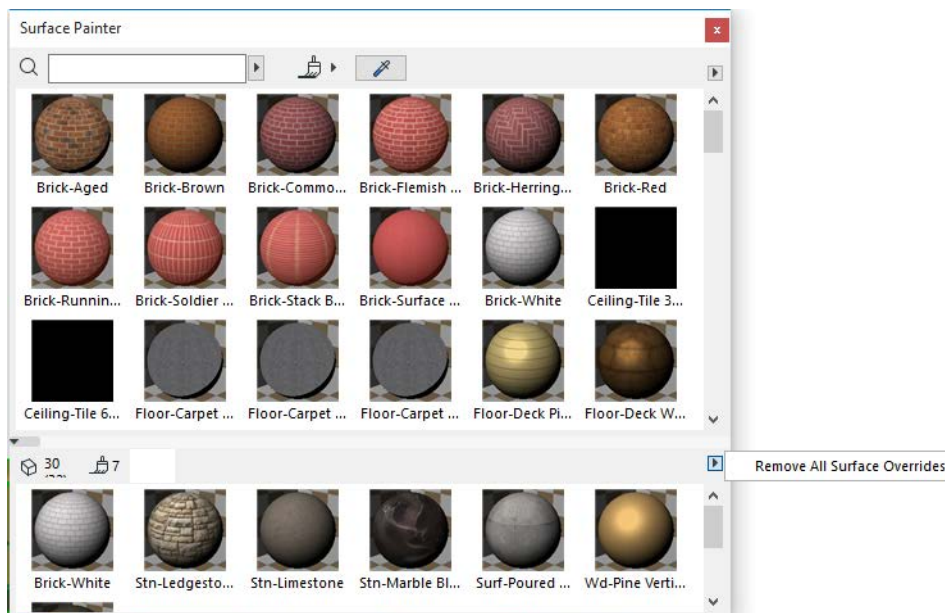
That surface is now overridden on all selected elements.

7. Exit Paint mode.

Remove Surface Overrides from Selected Elements

To remove surface overrides and return the elements to using the surfaces defined in their Building Materials:

1. In the 3D window, select all the elements from which you want to remove overrides.
2. From the Surface Painter's Selection Panel, click the black arrow and use the **Remove All Surface Overrides** command.



Note: GDL-object type elements do not use Building Materials. Consequently, any overrides applied to those objects are not affected by the **Remove All Surface Overrides** command. To change their surfaces, use Object Settings.

Surface Painter Palette

Use the Surface Painter as an easy way to override the surfaces of your project elements.

For a step-by-step workflow, see [Override Surfaces With Surface Painter](#).

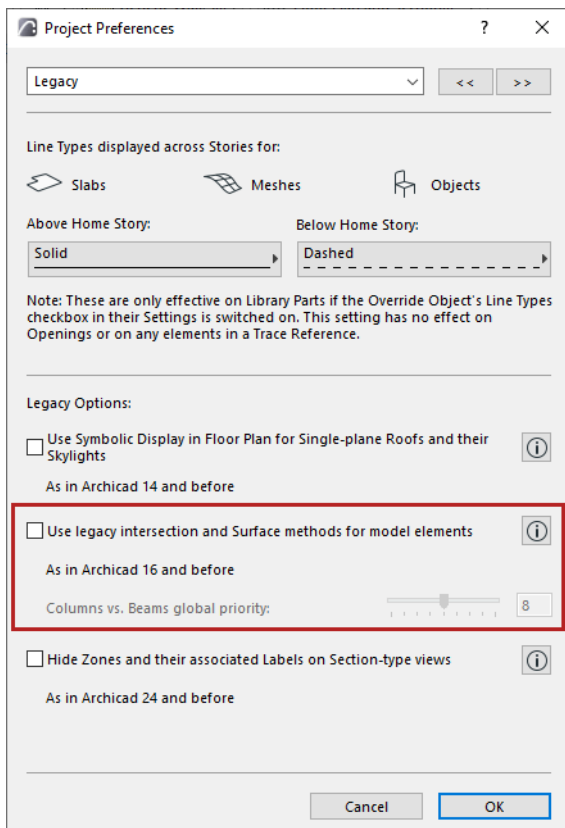
[Watch the Video](#)

Open Surface Painter

Open the Surface Painter palette from one of the following locations:

- Window > Palettes > Surface Painter
- Document > Creative Imaging > Surface Painter
- the Surface Painter command of the 3D Visualization toolbar

Note: Before using the Surface Painter feature, make sure you are *not* working in “legacy” intersection mode. (Uncheck this option at **Options > Project Preferences > Legacy**.)

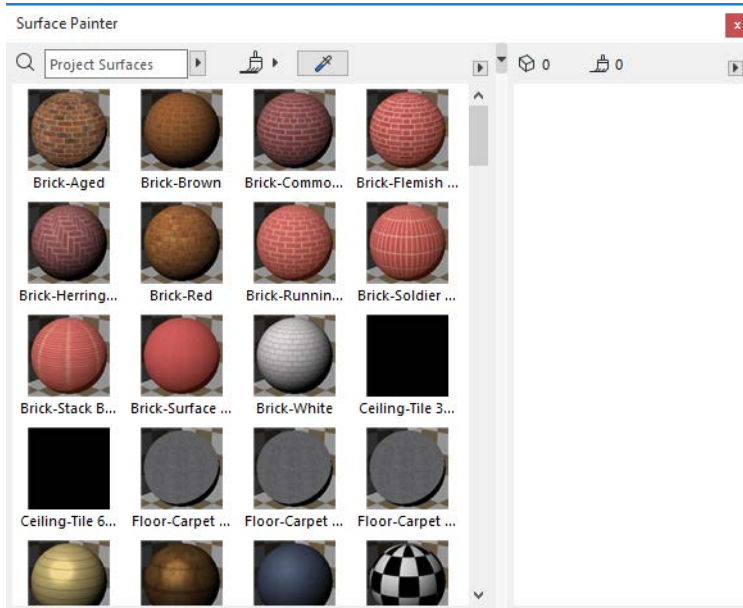


Content Panel of Surface Painter

By default, the **Content Panel** of Surface Painter (at left in the image below) lists all the surface attributes of the current project.

See also [Search for Surface \(Surface Painter\)](#).

Double-click any surface to open its Surfaces dialog box.



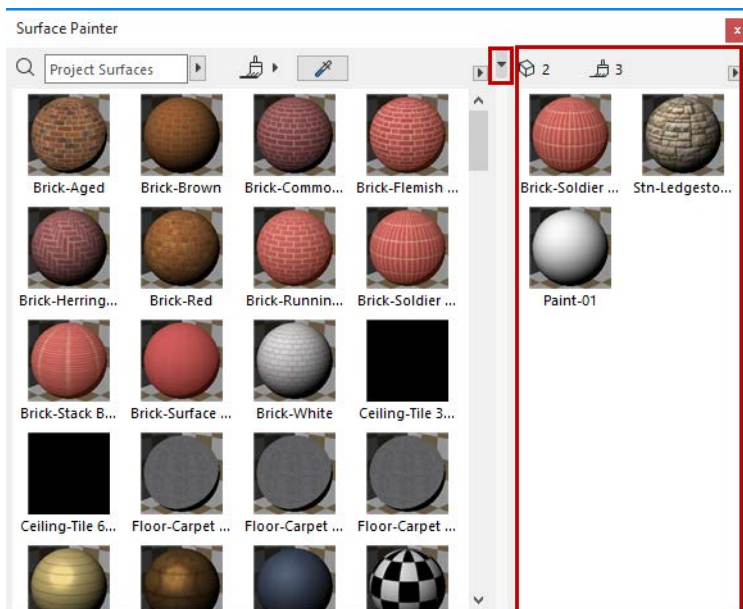
Content Panel Lists All Surfaces of Current Project

Selection Panel of Surface Painter

When any element is selected in the project, all surfaces of all selected elements are displayed in the Selection Panel of Surface Painter (at right, in the image below).

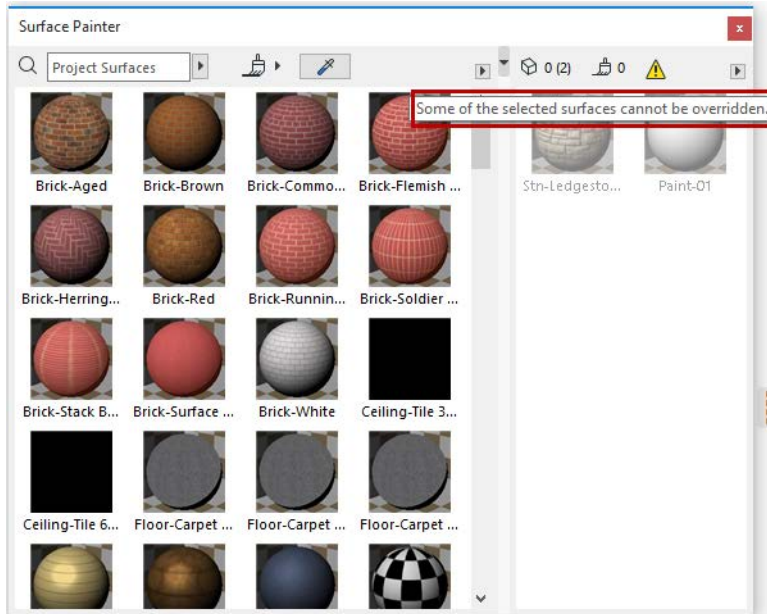
The number of selected elements and the total number of their overridable surfaces is also shown.

Double-click any surface to open its Surfaces dialog box.



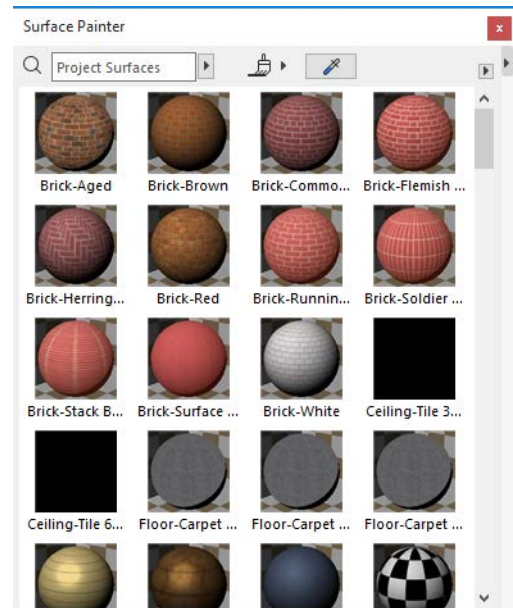
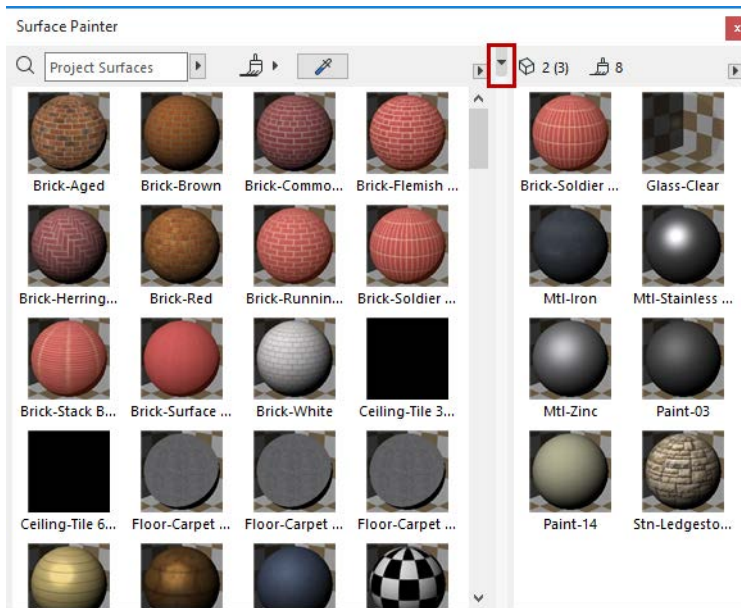
Selection Panel Lists All Surfaces of All Selected Elements

You may not be able to override a particular selected surface: for example, if the element or its layer is locked. In this case, a yellow info warning appears.



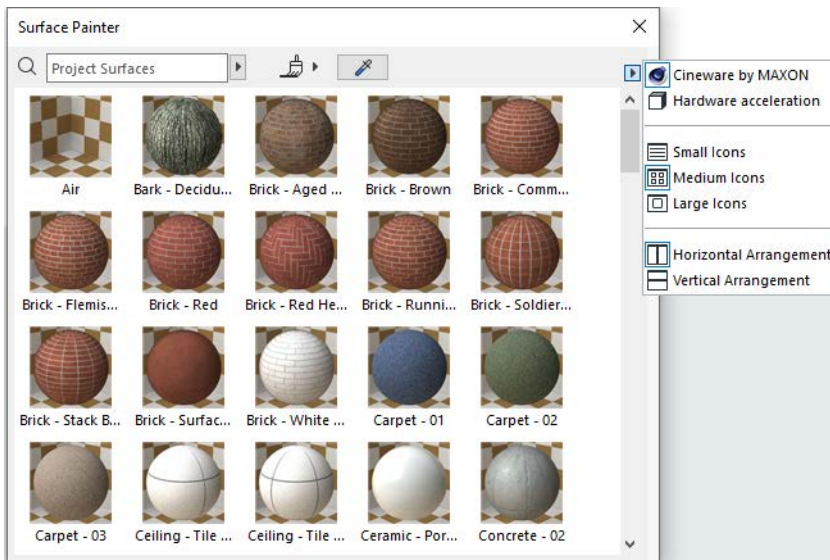
Open/Close Selection Panel

Double-click the splitter bar between the two panels to open/close the selection panel (or click the black arrow at the top of the splitter bar).



Customize the Surface Painter Palette

Use the options in the pop-up to customize the display of the Surface Painter:

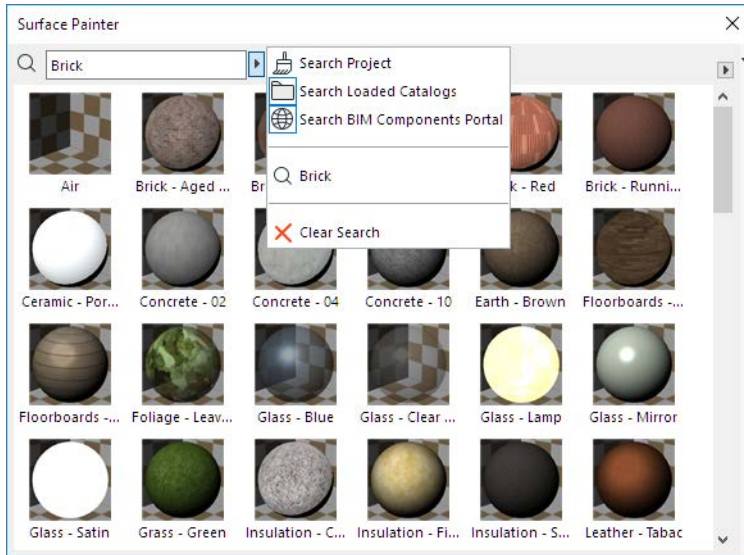


- Display surface previews for either **Cineware** or **Hardware acceleration** engine
- Choose preferred size of surface icons
- Choose how to arrange the two panels: horizontal or vertical

Search for Surface (Surface Painter)

Use the search field at the top of the palette.

Choose where to search:



- **Search Project:** Searches among all the surface attributes of this project
- **Search Loaded Catalogs:** Searches in surface catalogs added to the Library Manager
- **Search BIM Components** - Searches only on BIM Components, an interactive depository of objects and surfaces contributed by Archicad users or uploaded by Graphisoft.

For more information, see [BIM Components](#).

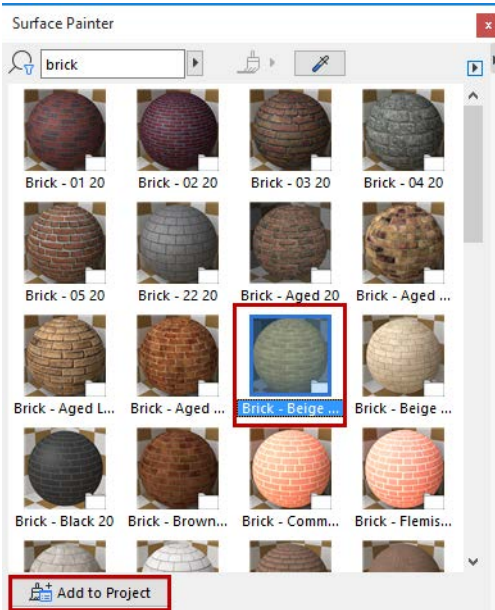
The last four searched items are listed in the search history.

Click **Clear Search** to return to the full content panel.

Add Surface to Project (Surface Painter)

To add a surface from a catalog: click the **Add to Project** button.

Note: In Teamwork, this function is available only if you have the “Create Surface” permission.



This action does the following:

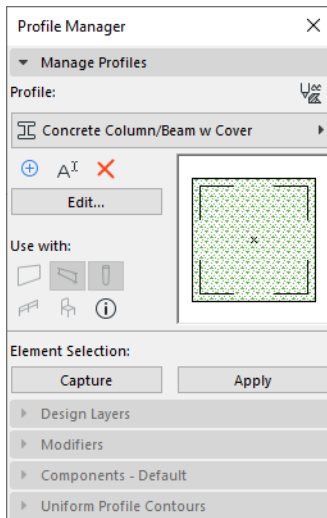
- Adds the surface to your project's surface attributes
 - Note:** Since a surface also includes a cover fill, adding a new surface to the project might result in adding a new fill attribute, if the project does not already contain a fill of the same name.
- Enters Paint mode

To add a surface found from the BIM Components Portal (marked with a globe icon), click the appearing **Download and Embed** button.

Profile Manager

Open from:

- Options > Complex Profiles > Profile Manager
- Options > Element Attributes > Profile Manager



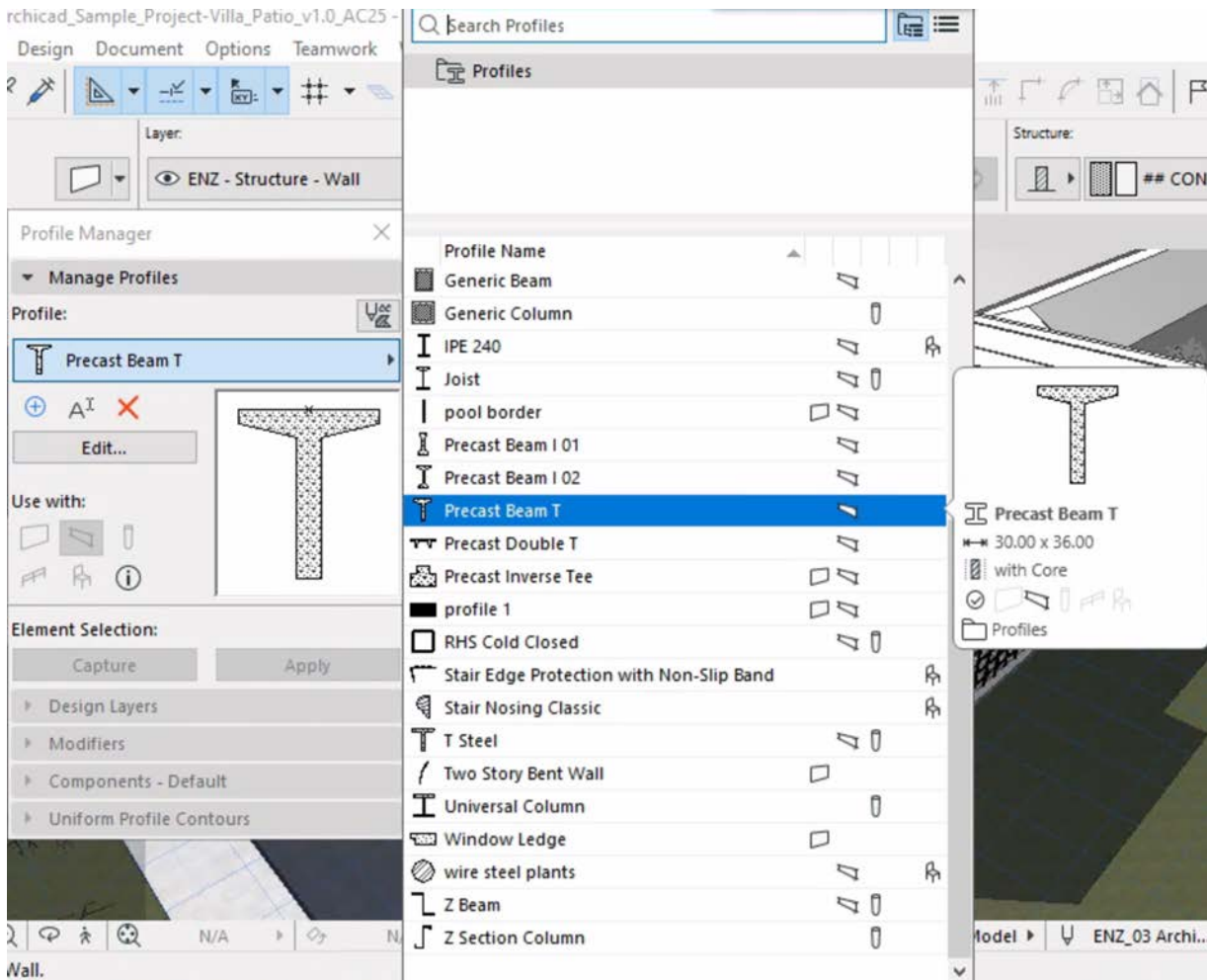
Use Profile Manager to

- Edit or create profiles, in the dedicated Profile Editor window (see [Edit Complex Profile](#))
- Manage the project's profile attributes

Select profile

Click the Profile selector pop-up and find the one you need.

Like other attribute types, profiles can be organized into folders. View and search available profiles in flat list or folder view.

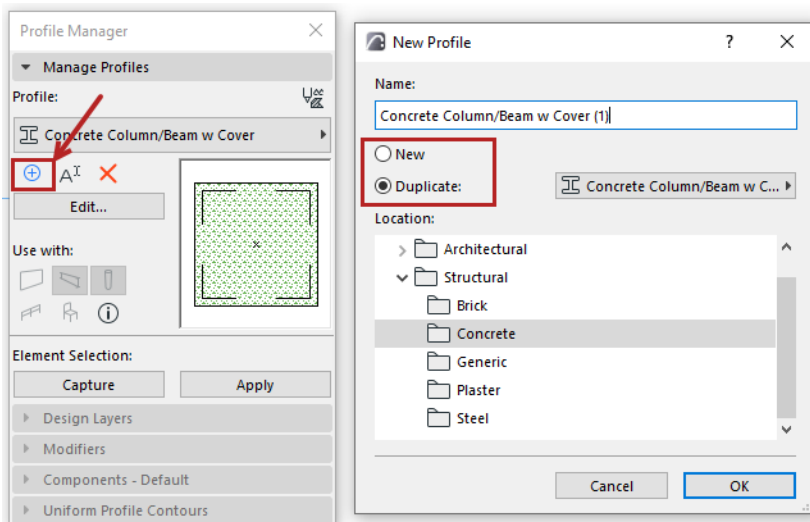


Note: You cannot create and manage folders in Profile Manager: use the Attributes palette instead. From the top of Profile Manager, click the Attributes shortcut to go there.

[See Attributes palette.](#)

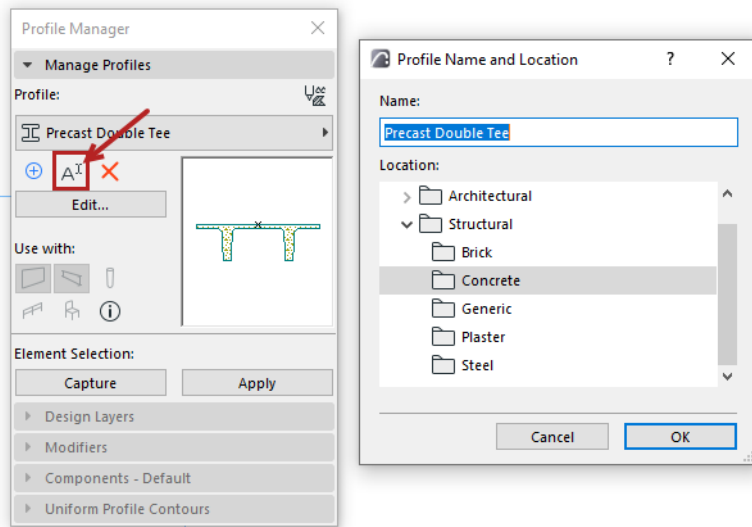
New or duplicate profile

Click the New button to create a new profile, or **duplicate** a current one under a new name.



Rename or move profile

Click to rename the profile and/or to choose a different folder.



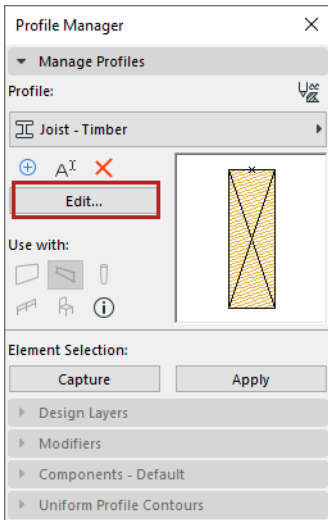
Delete profile

See [Delete attributes](#).

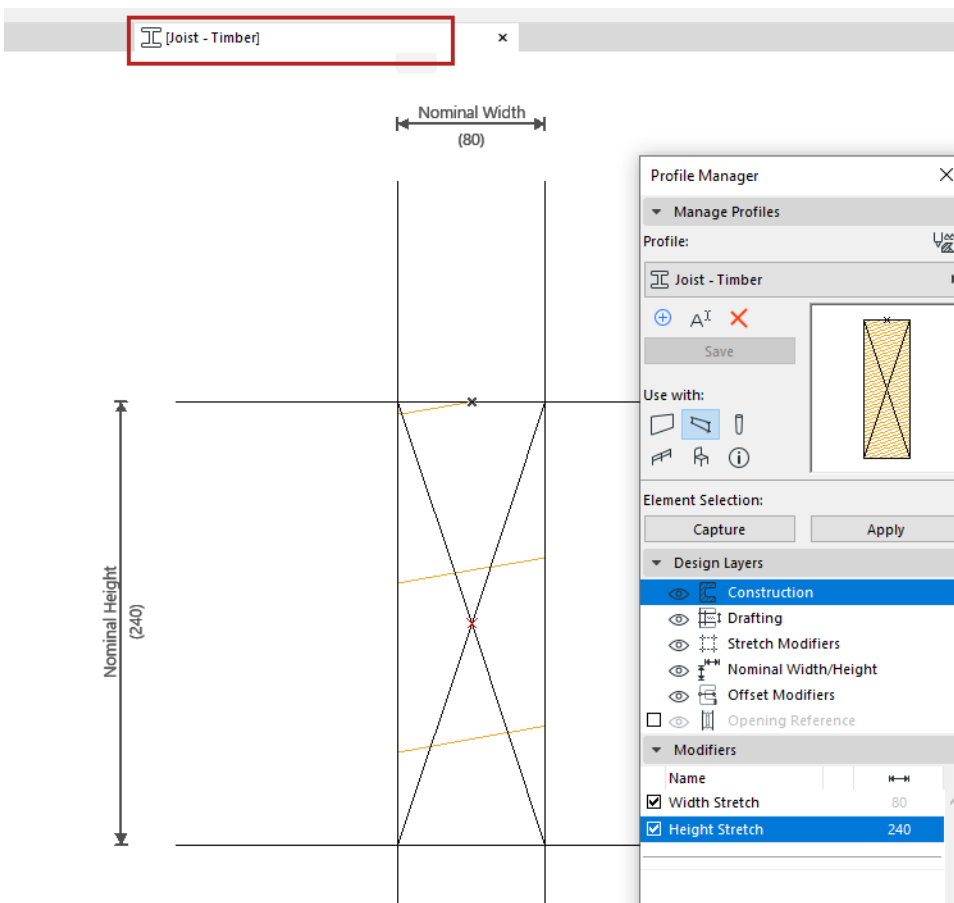
Edit Complex Profile

To edit a Complex Profile, you will use Profile Editor - a separate 2D editing window which opens automatically whenever you open a Profile to edit.

1. Open Profile Manager and select a profile to edit.
2. Click Edit.



The selected Profile opens in Profile Editor.

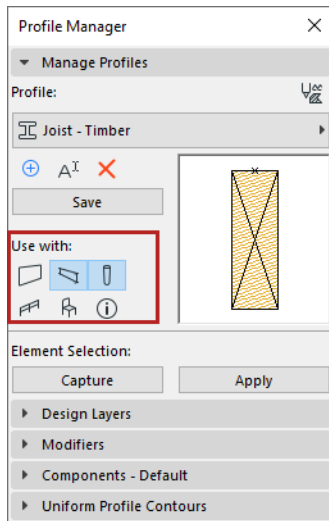


“Use With” - define profile availability

In Profile Editor:

1. Click Edit
2. Click one or more icons to make this profile available for particular tools: walls, columns, beams, railing components, or Library Parts.

Note: Profiles for Library Parts are available only for GDL elements that include a profile selection option - such as Curtain Wall and Stair components.

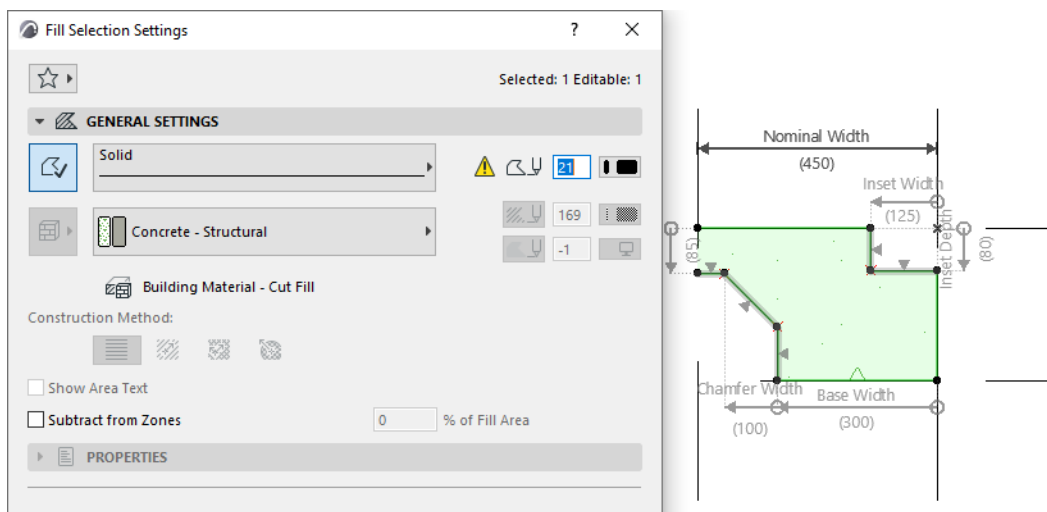


Create/edit profile components

In Profile Editor:

Use the Fill tool to draw or edit the cross-section of the complex profile. Only 2D drawing tools are available.

Select the fill element, then use Fill Selection Settings to assign Building Material and Cut Line type.



- Only Hotspots, and fills drawn with the Fill tool, will be part of the saved profile.
- Other 2D elements added to Profile Editor (lines, dimensioning, etc.) can be used as local drafting aids, but will have no effect on the final appearance of the complex profile. These drafting aids are located on the “Drafting” Design Layer of Profile Manager.

- Overlapping fills are supported, and will be cut out based on their Building Material Priority at the placed element level.
- The Profile Origin is a significant reference point. [See *Profile Origin in Profile Editor Window*](#).

Use the other controls of Profile Manager to edit the Profile geometry and behavior:

[Design Layers Panel](#)

[Modifiers Panel](#)

[Components Panel](#)

[Uniform Contours Panel](#)

See also:

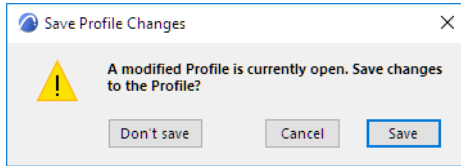
[Stretch Modifiers](#)

[Offset Modifiers](#)

Save Named Profile

Do one of the following:

- Click the **Save** button in the Profile Manager, or
- Switch to another profile and choose “Save” from the appearing dialogue.

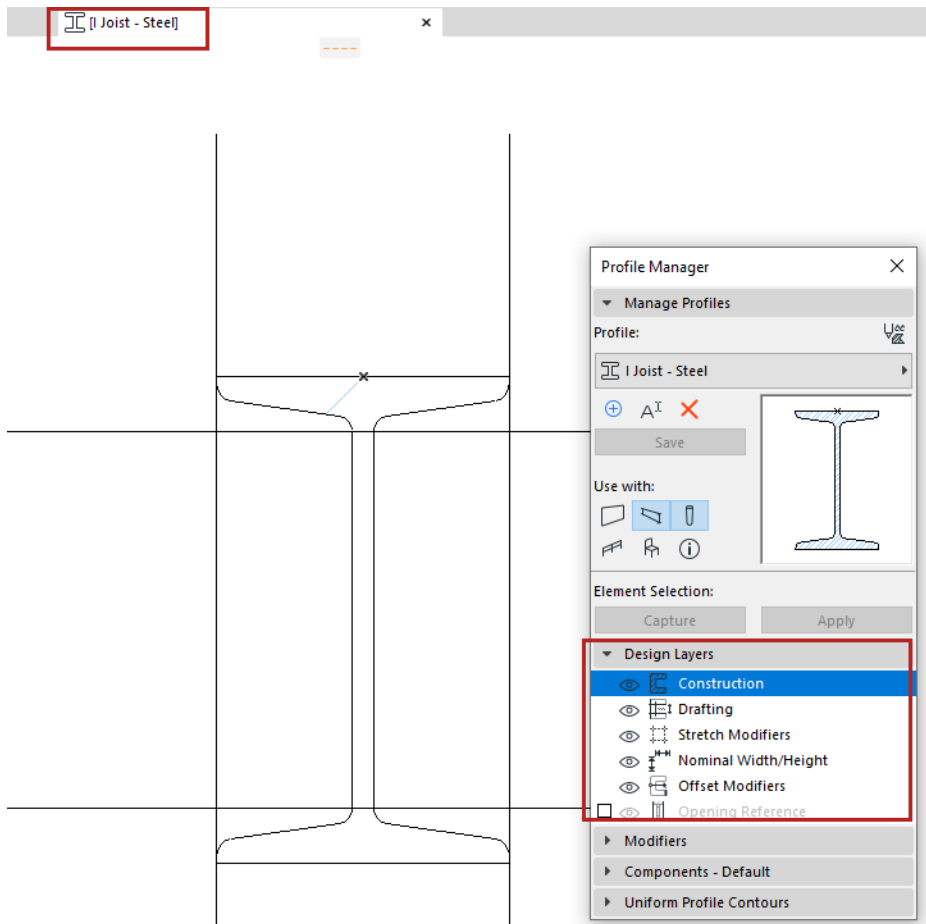


See also [Save Custom Profile](#).

Design Layers Panel

Use this panel to show or hide elements in Profile Editor.

This panel is active only if Profile Editor is open.



Construction

Show/hide fill or hotspot elements in Profile Editor.

Drafting

Show/hide elements drawn with any other drafting tool (e.g. lines, arcs, polylines) in Profile Editor.

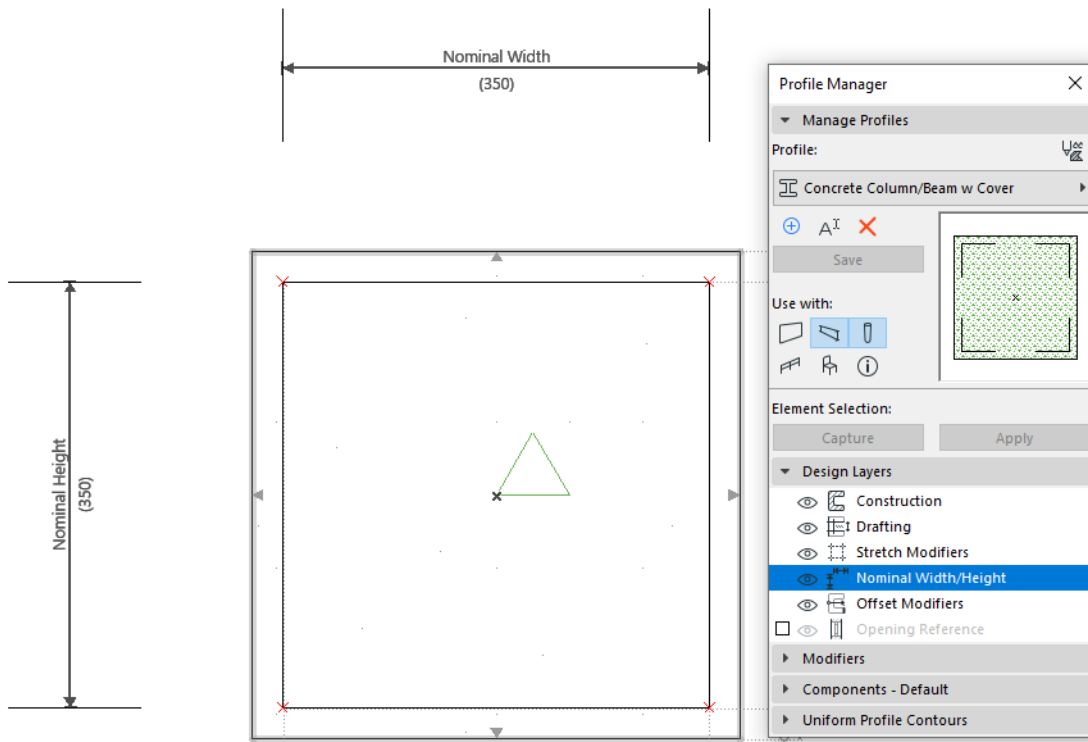
Stretch Modifiers

Show/hide Stretch Modifiers in Profile Editor.

See [Stretch Modifiers](#).

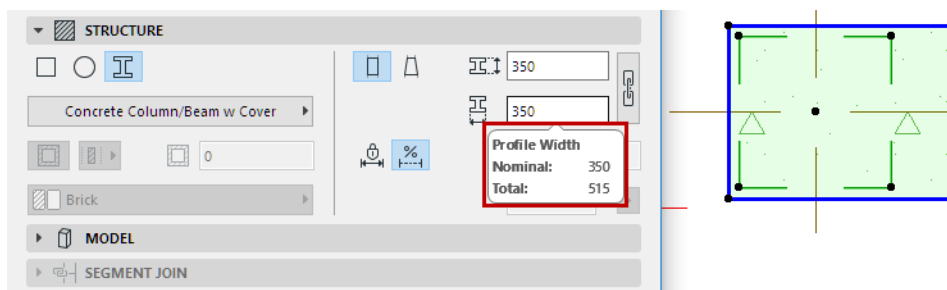
Nominal Width/Height

Show/hide the dimensions indicating the profile's Nominal Width and Height, in Profile Editor.



The **Nominal Width/Height** - listable element parameters - are shown in the element's Settings dialog. These values are editable in element Settings, if the profile's Width and/or Height Stretches are enabled in the Modifiers panel.

[See Nominal Width/Height.](#)



Nominal Width and Height dimensions can be repositioned by clicking and dragging on their dimension lines. Their lengths can be modified and aligned to another part of the profile - including outside of the Profile bounding box, by clicking on the ends of the nominal dimensions. If the lengths are not modified, their lengths will automatically resize to follow the bounding box of the fills as defined in Profile Editor.

Nominal Dimension ends, like Offset Modifiers, can be associated to fill nodes, but this may cause an over-constraint. In this case, a warning will appear.

In contrast, the **Total Width/Height** listing parameters also include element-level adjustments via Offset Modifiers, and reflect the overall bounding box size of the element. The Total Size is not editable in the element's Settings dialog, due to the constraints of Offset Modifiers.

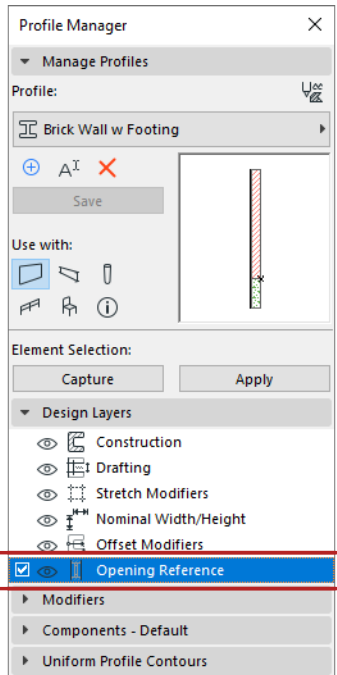
Offset Modifiers

Show/hide the dimensions and assigned edge/node highlights of Offset Modifiers.

Even if this is set to “hide”, an Offset Modifier currently selected in the Modifiers list will always be shown.

[See Offset Modifiers.](#)

Opening Reference (for Profile Walls)



- Check the box to display Opening Reference lines in the placed wall profile (the reference line where doors/windows will be placed).
- Click the eye icon to show/hide Opening Reference lines in Profile Editor.
- If necessary, edit, extend or break the lines to conform to the shape of the profile, so that openings will be placed and oriented in the correct location.

If you do not switch on Opening Reference, doors/windows will be placed along the wall’s bounding box.

Modifiers Panel

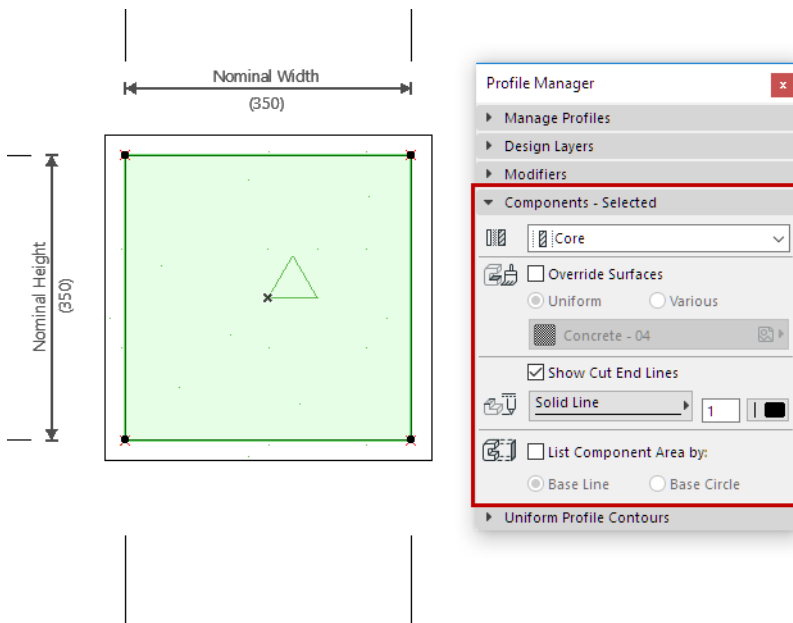
See the following sections:

[Stretch Modifiers](#)

[Offset Modifiers](#)

Components Panel

Select a fill component in Profile Editor.



Use the Components Panel of Profile Manager to edit the following parameters:

Component Type

Define the selected component as core, finish, or other. This definition will affect the Partial Structure Display of the complex element.

[See details at *Partial Structure Display*.](#)

Override Surfaces

Check **Override Surfaces** to choose a new surface from the pop-up. Otherwise, the selected component will use the surface defined in the fill's Building Material.

If you override an individual fill edge with a custom surface in Profile Editor, this will be reflected as "Various" in this panel.

Show Cut End Lines

Check this to display Cut End Lines.

Define line type and pen of the selected Profile Fill component. This line type and pen are used for end lines when the element is cut.

List Component Area by

Choose either **Base Line** or **Base Circle** to define how Archicad should calculate surface area for this component, in an Interactive Schedule of Components.

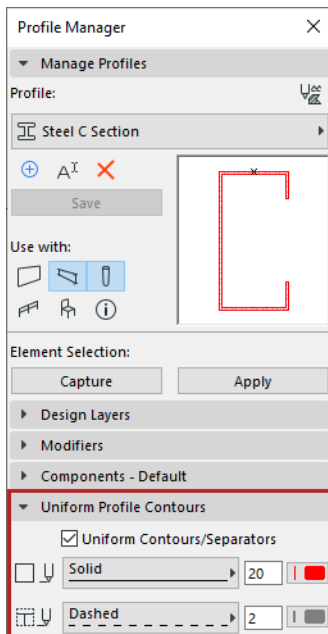
To view and edit the Base Line or Base Circle in Profile Editor, make sure you have turned on **View > On-Screen View Options > Component Area Baseline/Circle**.

[For more information, see *Component Data in Archicad*.](#)

Uniform Contours Panel

Apply uniform lines and pen colors to cut lines and to all separator lines at once. (This panel is active only if Profile Editor is open.)

Click Save, then view the feedback in the Preview window of Profile Manager.



Contours: Choose a uniform line type and pen color for all contour lines (outer lines) of the profile element.

Separators: Choose a uniform line type and pen color for all of the separator cut lines (inner lines) of the profile element.

Notes:

- Lines that separate two fills having the same Building Material will not be eliminated.
- Edges customized to Off (Show Cut Line is Off; see below) will not apply uniform contour settings.

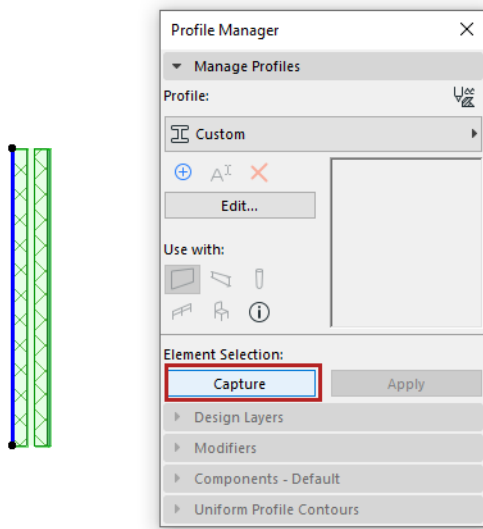
See also [Apply Custom Surface or Line to Profile Edge](#).

Capture Selected Element as Custom Profile

To create a new custom profile based on selected elements (walls, columns, or beams) in the active model window:

Select the elements, then do one of the following:

- Click **Capture** button in Profile Manager
- Options > Complex Profiles > Capture Profile of Selection



The new profile is called “Custom”.

Save Custom Profile

After editing a captured custom profile in Profile Editor, make sure you click **Save Profile** to save your edits and name the new Profile, before closing the window.

Alternatively, click Apply to apply a one-off custom to a selected wall, beam or column.

Note: Custom profiles are not directly assigned to elements from the Profile Manager, but locally stored in the element.

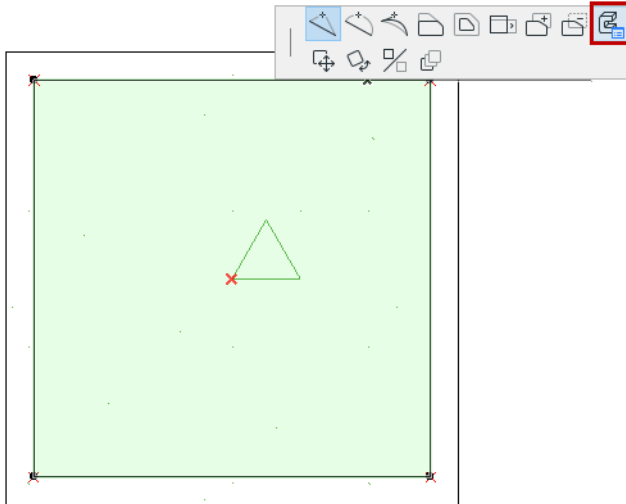
See also [Save Named Profile](#).

Apply Custom Surface or Line to Profile Edge

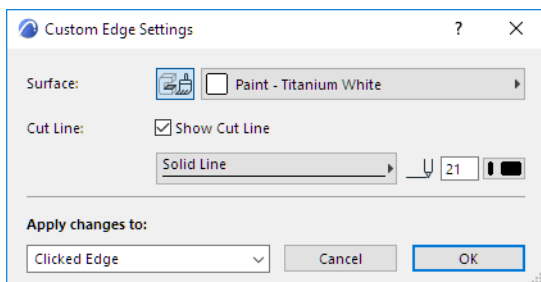
By default, all sides of a Profile element use the surface defined in the fill's Building Material.

To apply a custom surface or line type to one or more edges of a Profile component:

1. Select a profile component fill in Profile Editor.
2. Choose the **Custom Edge Settings** pet palette icon.



3. Edit the controls in the appearing Custom Edge Settings dialog box.



To switch off the contour line of the selected edge, uncheck **Show Cut Line**.

These custom edge settings will be applied to the edge from which you opened the Pet Palette (Clicked Edge), or to all edges.

You can apply a different custom edge surface/contour line to each individual edge.

Fill Selection Settings indicates custom edge or surface with the “custom” icon.

Profile Origin in Profile Editor Window

The Origin shown in Profile Editor is a significant reference point.

You cannot move the Profile Origin, but you can move the profile shape itself, so that it correctly positioned with respect to the origin.

Wall Profile

- If the profile is a wall, the origin represents the location of the wall's reference line.

Column or Beam Profile

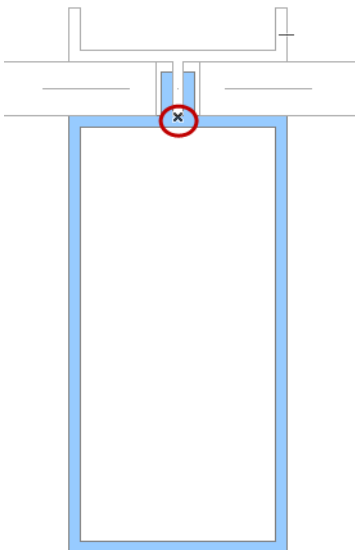
- For a column or beam, the origin represents the location of the element's axis.
- In case of profiled beams used in a Stair structure: for correct connections, make sure that the Profile Origin is defined at the *top* of the profile. (Alternatively, choose a top location for the Anchor Point).

See also [Beam Anchor Point](#).

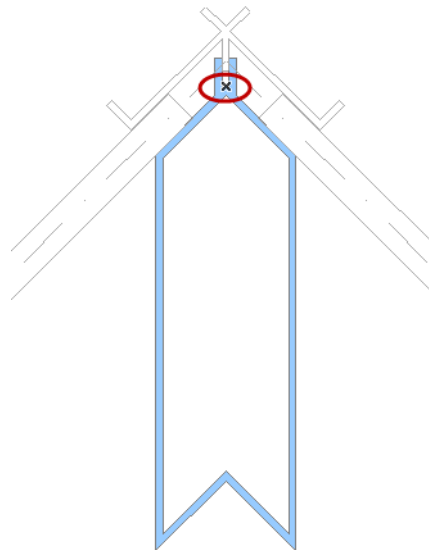
Curtain Wall Components

This applies if the Profile is set to be “used with” Library Parts, and if the Library Parts for Curtain Wall components are set up to use Profiles (as in the AC Library).

- If the profile is a **Curtain Wall Frame**, the origin is located on the **inside** of the glazed surface. The downward direction in the profile represents the inside direction of the Frame. Generally, the Profile Origin will be located at the upper midpoint of the profile.
- If the profile is a **Curtain Wall Corner Frame**, the origin is located on the **inside** of the glazed surfaces at their intersection.



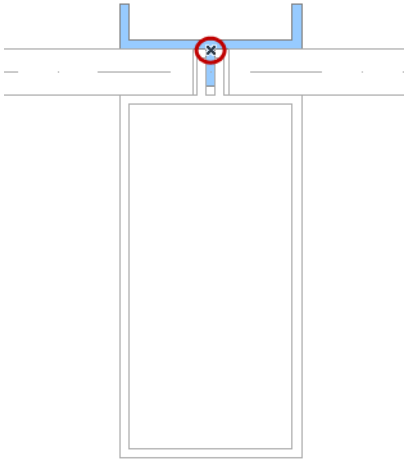
Frame



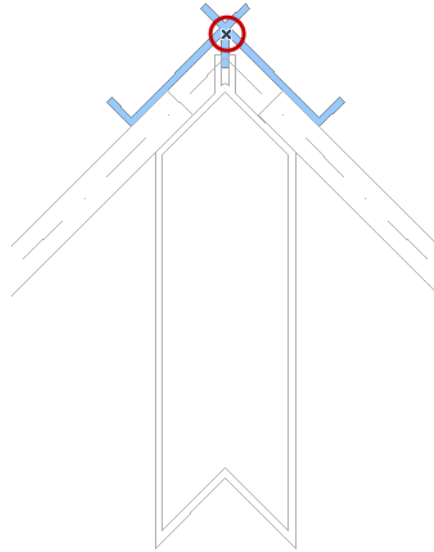
Corner Frame

- If the profile is a **Curtain Wall Frame Cap**, the origin is located on the **inside** of the glazed surfaces at their intersection.

- If the profile is a **Curtain Wall Corner Cap**, the origin is located on the **outside** of the glazed surface. The upward direction in the profile represents the outside direction of the Frame.



Frame Cap



Corner Cap

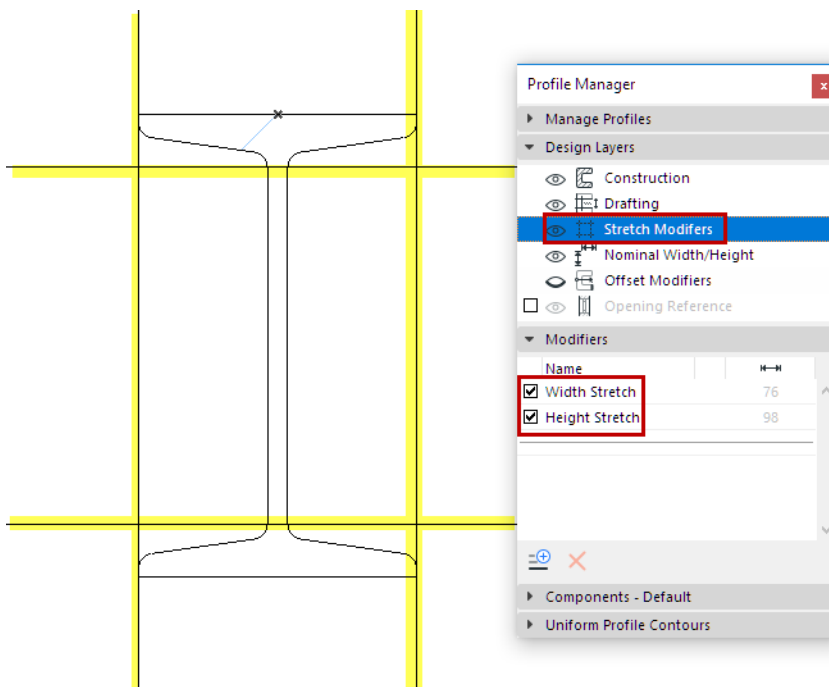
See also [Curtain Wall Corner Frames](#).

Stretch Modifiers

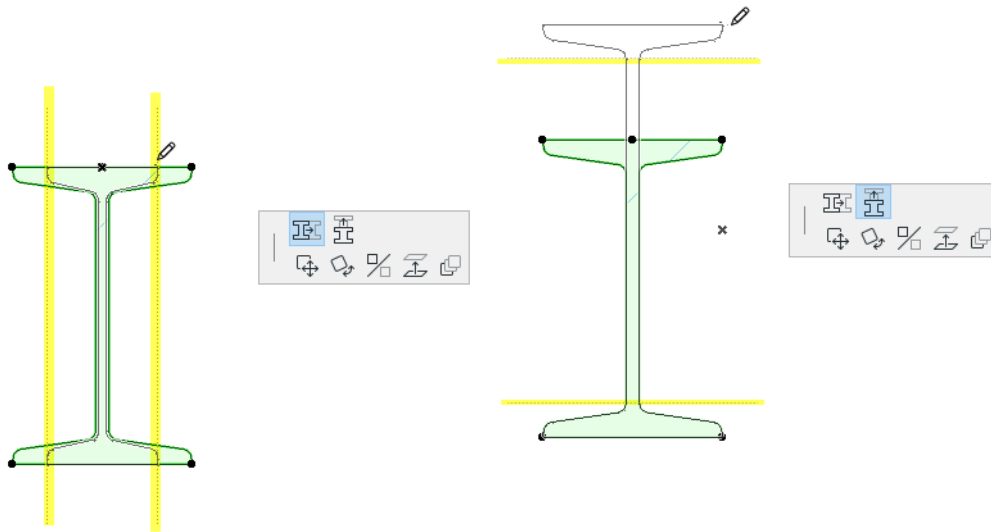
These controls are in Profile Manager (Options > Element Attributes).

In the Modifiers Panel, switch on Width Stretch and/or Height Stretch. This enables you to resize (increase or decrease the width/height) of the cross-section of the placed profile element.

- To show/hide Stretch lines in Profile Editor, use the Stretch Modifiers control in the Design Layers panel. In this image, they are highlighted in yellow.
- The Stretch Modifiers mark the boundaries of the geometry that will stretch or contract.
 - For a wall, its height or thickness
 - for a beam, its height or width
 - for a column, its cross-section height or width

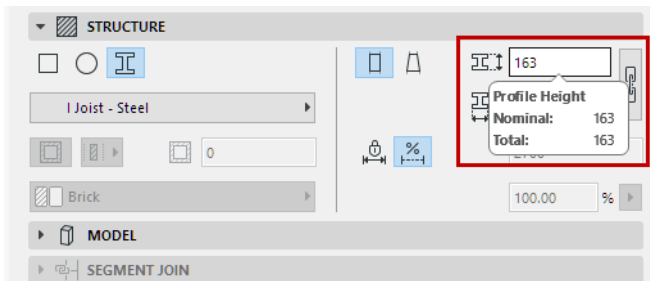


When you graphically stretch the placed element in either direction, the Stretch Modifiers appear in the model window as feedback.



Stretch Feedback on Stretched Part of Placed Profile

The element's Settings reflect the stretch in its Height/Width parameter values (hover to see the Nominal and Total values):



See also [Nominal Width/Height](#).

Note: Stretch Modifiers only support profiled walls, beams and columns.

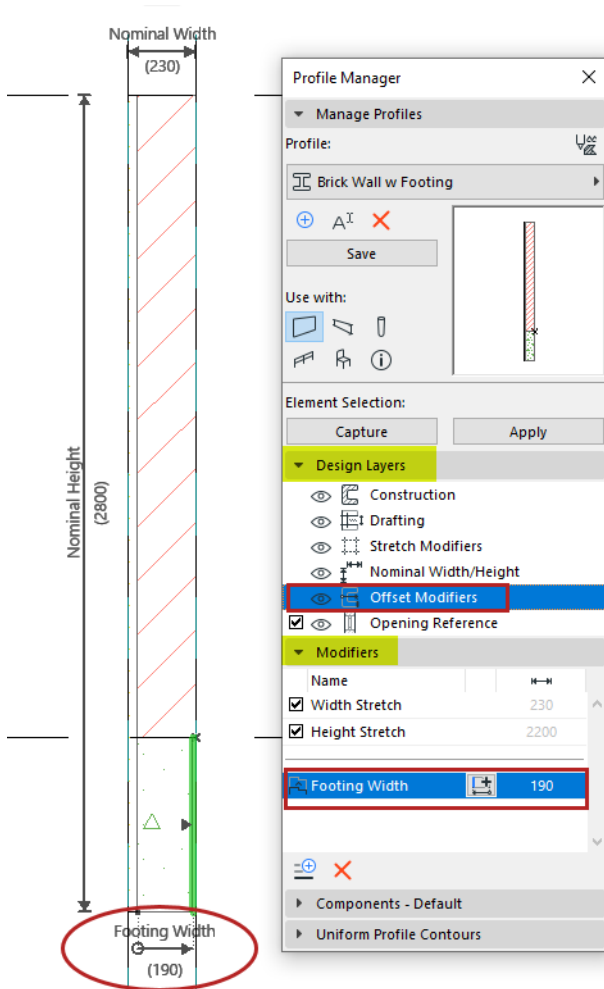
Offset Modifiers

These controls are in Profile Manager (Options > Element Attributes).

Offset Modifiers are optional controls added to a Profile, used to offset individual profile edges at the element instance level - so you don't have to create and manage many similar Profiles. They are available for wall, beam and column profiles. Parametric modifiers can be labeled, scheduled and used in Expression-type properties.

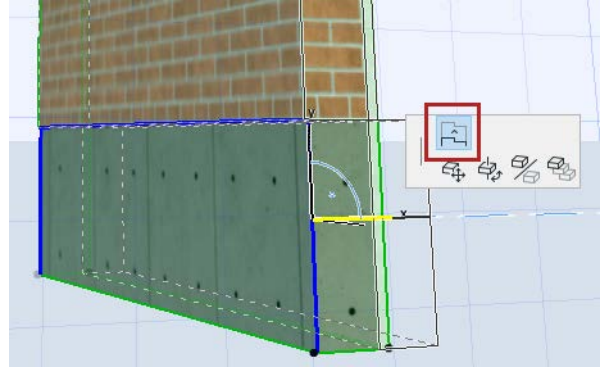
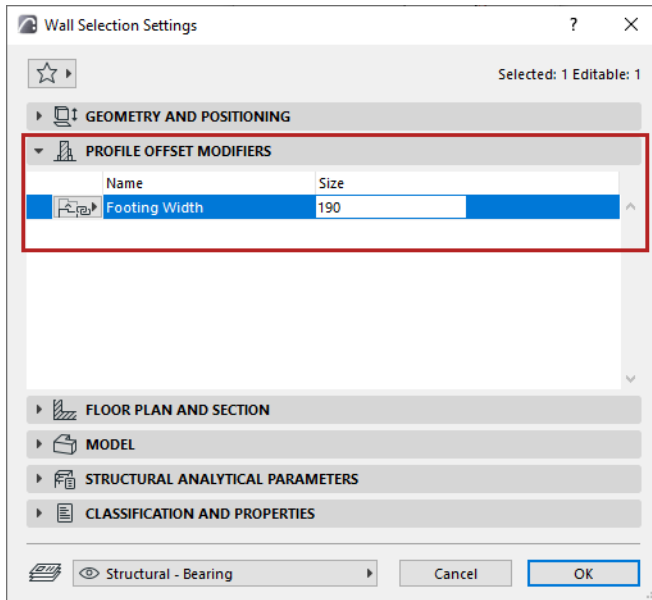
With Profile Editor open:

- Turn on Offset Modifier visibility in the Design Layers panel
- Add modifiers in the Modifiers panel



Profile Editor: Element with Offset Modifier

Edit the profile using its offset modifiers in its settings dialog, or graphically in a model window.



Related Topics

[Create New Offset Modifier - Example](#)

[Edit Offset Modifier](#)

[Offset Modifiers: Tips and Tricks](#)

Create New Offset Modifier - Example

A profile can include multiple Offset Modifiers, and one Offset Modifier can include one or multiple dimensions (of the same length). Also, a single dimension can control multiple edges.

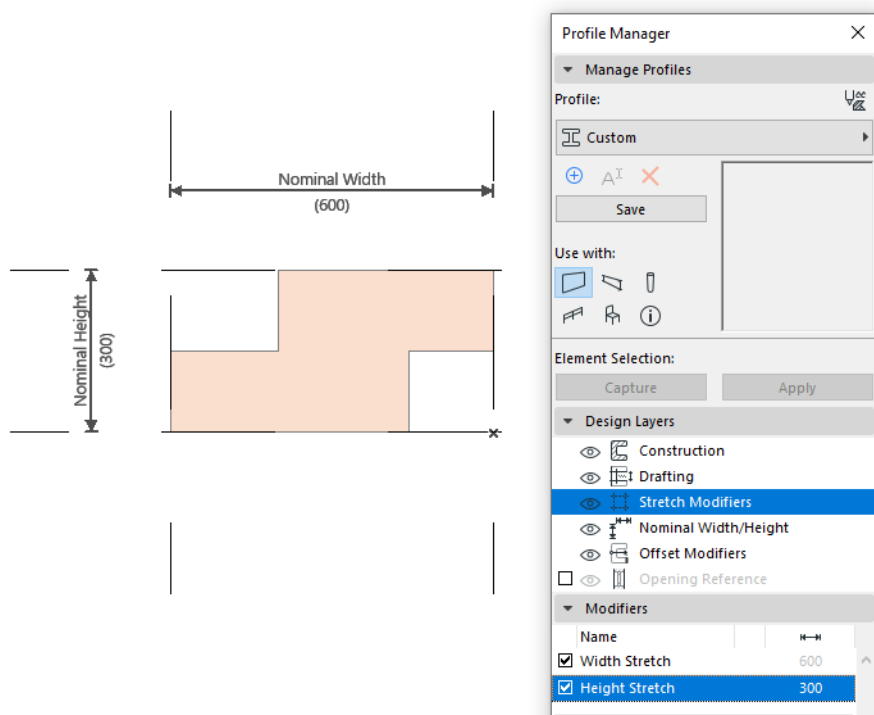
The following examples illustrate these features.

In this example, create a profile called Concrete Slab Edge Footing. You will add a modifier called “Base Width” to certain edges of this profile: after you place the element, you can graphically modify the Base Width edge.

1. In Profile Editor, draw the cross-section using the Fill Tool.

The **Nominal Height and Nominal Width** is shown. (The Nominal Width/Height display is turned on in the Design Layers panel.)

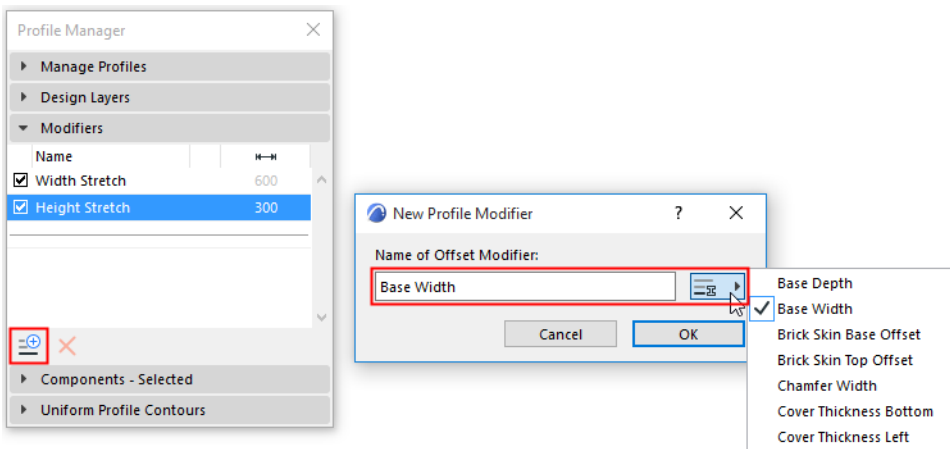
The **Stretch Modifiers** are also shown. (The Stretch Modifiers display is turned on in the Design Layers panel.) By default, the Stretch Modifiers coincide with the Nominal Height/Width dimensions.



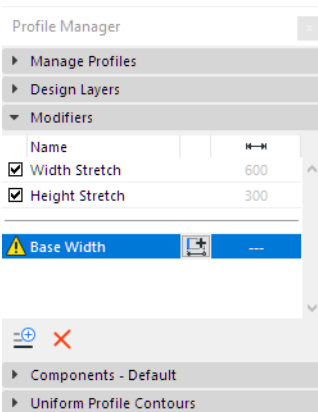
Now, create an Offset Modifier called Base Width on particular profile edges.

2. In the Modifiers panel, click **New Modifier**.
3. In the New Profile Modifier dialog, give it a name: **Base Width**.

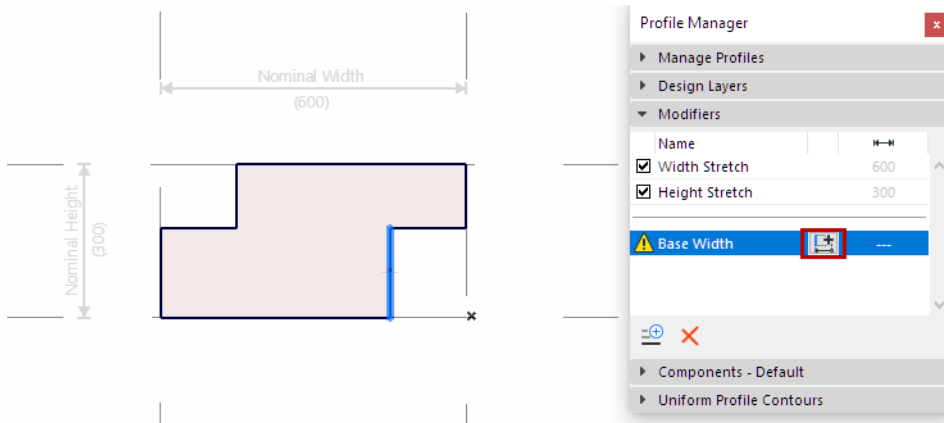
Or choose an existing name from the popup list, if this parameter has already been defined in another profile in the same project.



The newly created modifier appears in the Modifiers list. The initial warning triangle indicates that this modifier has no definition yet.



4. In Profile Editor, you are in dimension input mode. (If you are not in dimension input mode, click the **Add Offset Dimension** button in the Modifier panel as shown here.)



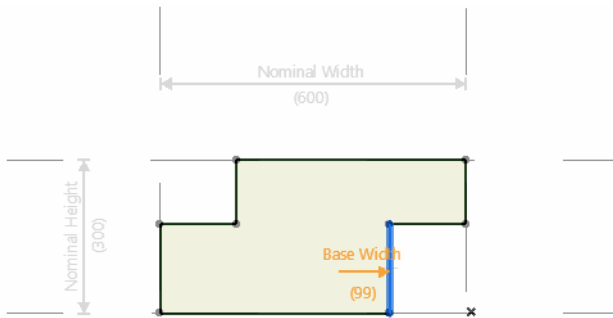
You have several options for defining the Dimension and anchoring it:

- Edge with Node Anchor
- Two Edges to Two Nodes (Anchored by Automatic End Nodes)
- Node and Node (with Neighboring Edge)

During input, available edges that can be offset are shown in black; when hovering, they will be shown with blue highlight.

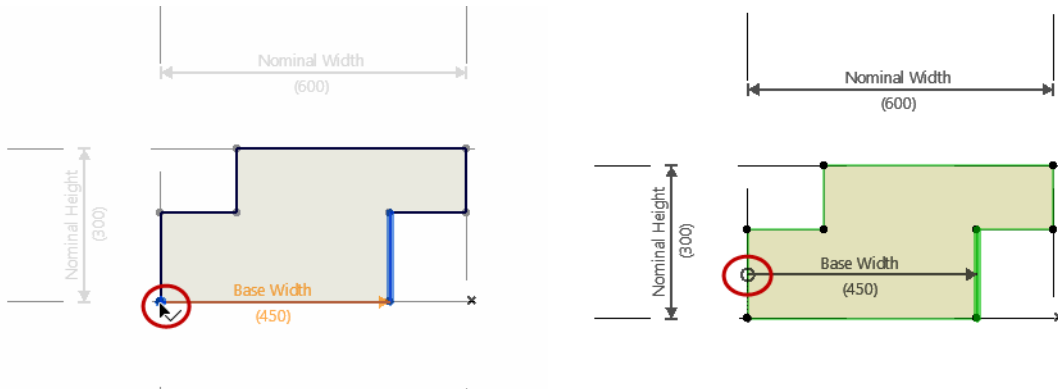
Edge with Static Node Anchor

1. Click the edge to be offset.
2. Now drag the cursor: the modifier's parameter name appears, and you are dragging its dimension line.

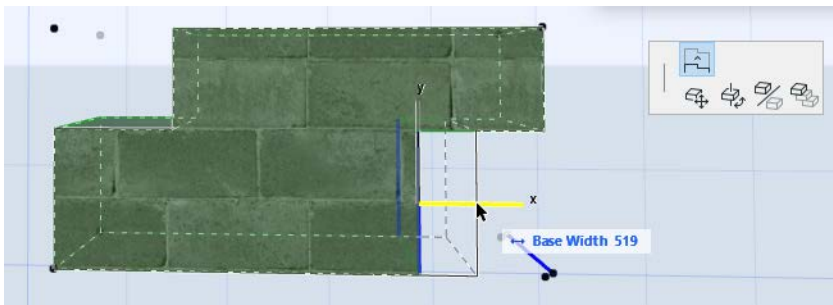


3. Click a fill node (this is the anchor), then drag the resulting dimension line and click again to place it. The result: You created an edge offset parameter that can be offset in the given direction, anchored from the clicked node.

The dimension end is a circle, indicating that its anchor is a node (rather than a static point).



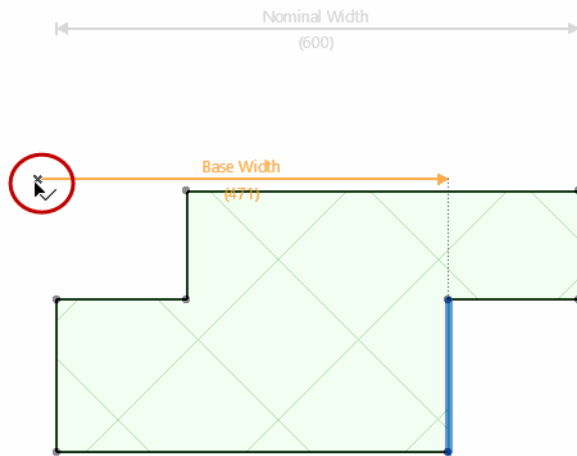
Modifier: Offset Single Edge, Anchored to Fill Node



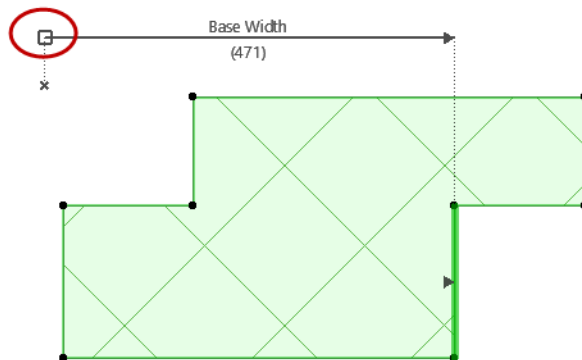
The anchor is associated to the fill node, so if the profile's fill geometry changes, the anchor point (and thereby the modifier) will follow these changes.

Edge with Static Anchor

Alternatively, click anywhere in Profile Editor to create a static point anchor (here, to the profile origin). Then drag the resulting dimension line and click again to place it.



The dimension end is a square, indicating that its anchor is a static point (rather than a node).

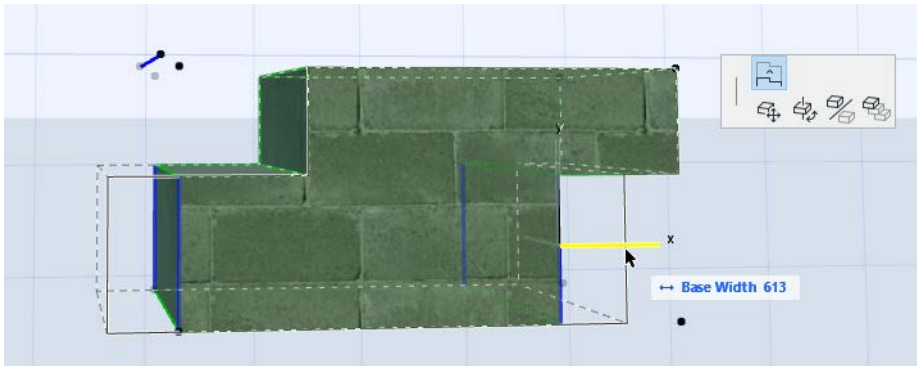
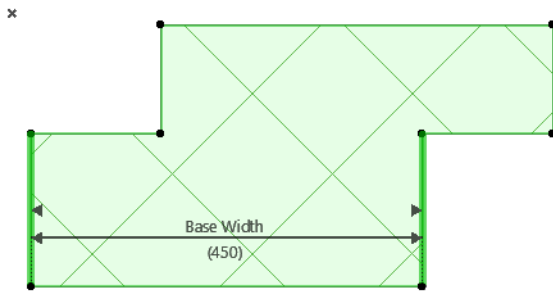
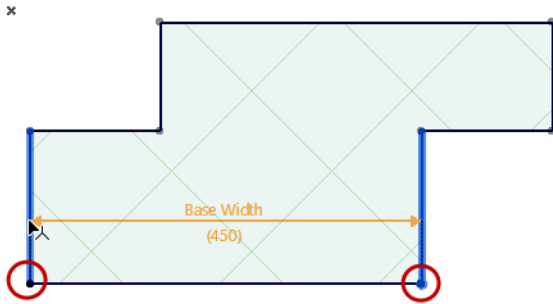


Two Edges to Two Nodes

Click the edge to be offset, then click a second edge. As you move the cursor, the dimension automatically finds the nearest end nodes to anchor to.

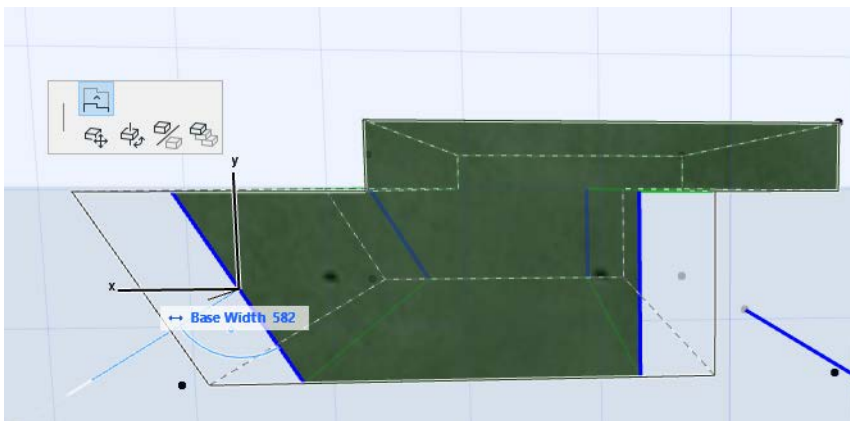
Then drag the resulting dimension line and click again to place it.

The result: You created an offset parameter with two edges. Modifying this parameter modifies both edges.



Modifier Offsets Two Opposite Edges

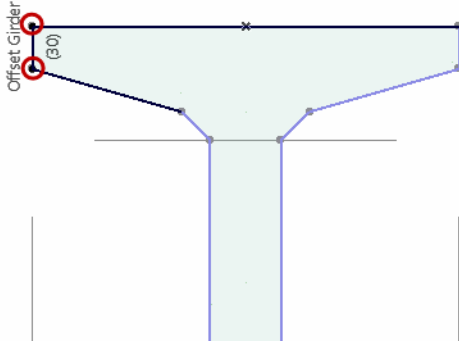
The two clicked edges do not have to be parallel.



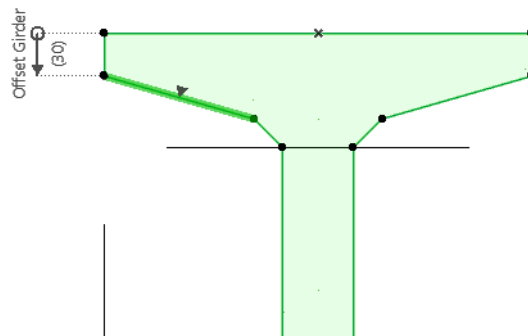
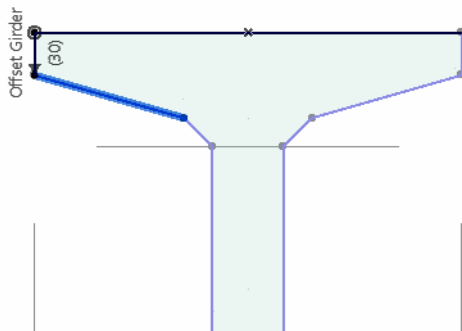
Node and Node with Neighboring Edge

Use this method to create a modifier whose anchoring Dimension is not necessarily perpendicular to the offset edge.

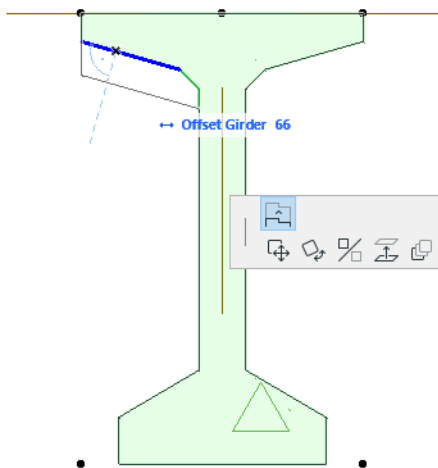
Click two nodes representing the Dimension.



Then click the edge to be offset. The edge need not be perpendicular to the Dimension, but it must be adjacent to one of the clicked nodes.



The result: The edge is offset perpendicularly, but is measured from the Dimension you specified.



Related Topics

[Edit Offset Modifier](#)

[Offset Modifiers: Tips and Tricks](#)

Edit Offset Modifier

Topics in this section

[Reassign Dimension anchor](#)

[Add/Remove Edges to One Offset Dimension](#)

[Add Dimensions to One Offset Modifier](#)

[Two Connected Modifiers](#)

[Remove Edges or Entire Dimension](#)

[Delete Modifier](#)

[Rename Modifier](#)

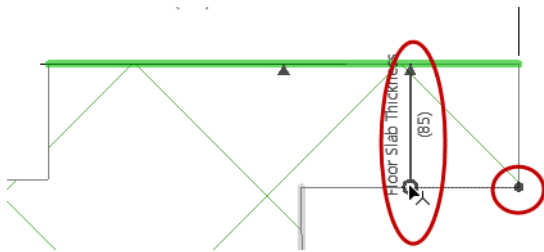
Related Topic

[Create New Offset Modifier - Example](#)

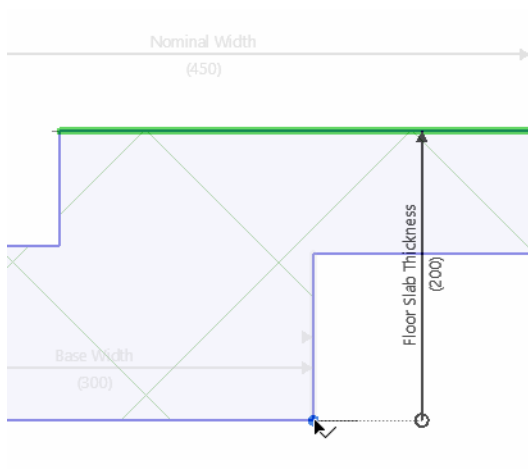
[Offset Modifiers: Tips and Tricks](#)

Reassign Dimension anchor

1. Activate the modifier in Profile Editor (click the Offset Dimension in Profile Editor, or select it in the Modifiers list of Profile Manager)
2. Check the feedback to see the edge/node anchor.



3. Click the Dimension end you want to redefine.
4. Move to another node or edge and click to reassign the Dimension anchor.

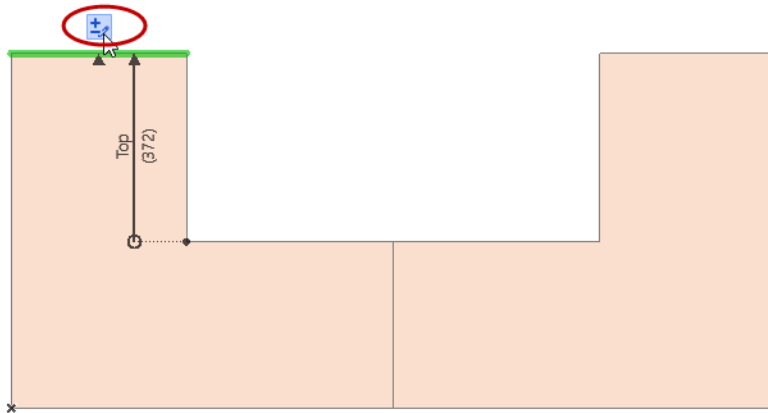


Add/Remove Edges to One Offset Dimension

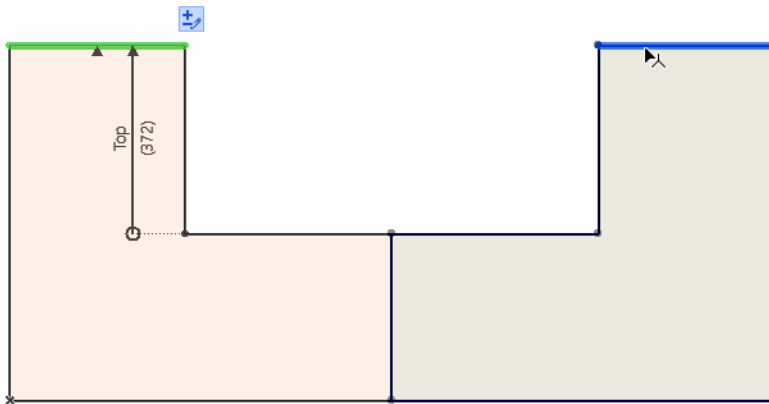
You can link multiple edges to a single Dimension. All edges will move together, at the same time.

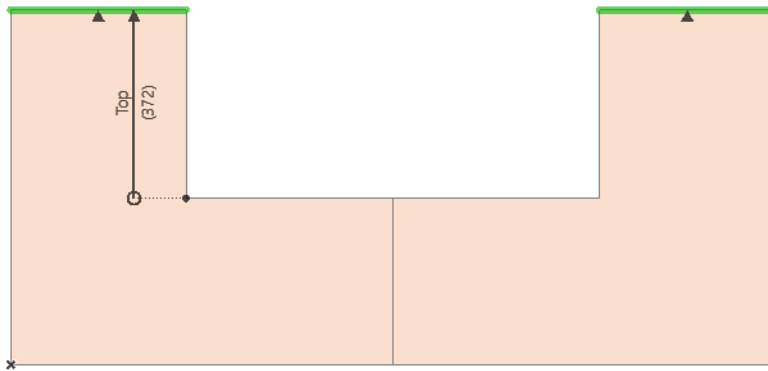
In this example, the existing modifier is called “Top”.

1. In Profile Editor, hover over the Dimension or an assigned edge of the **Top** modifier to see the grip.
2. Click the appearing grip: **Add/Remove Edge**.



3. As you move the cursor, the eligible edges (initially shown in black) are highlighted in blue. Click to choose an edge.
4. A second click defines its direction.



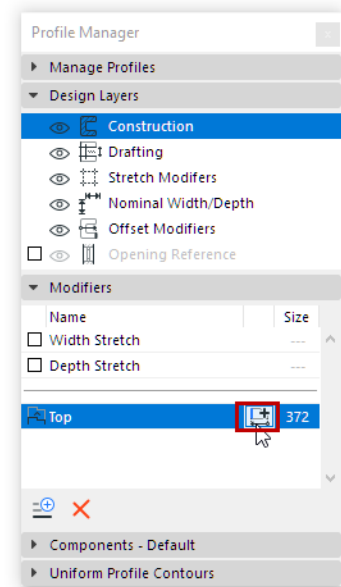
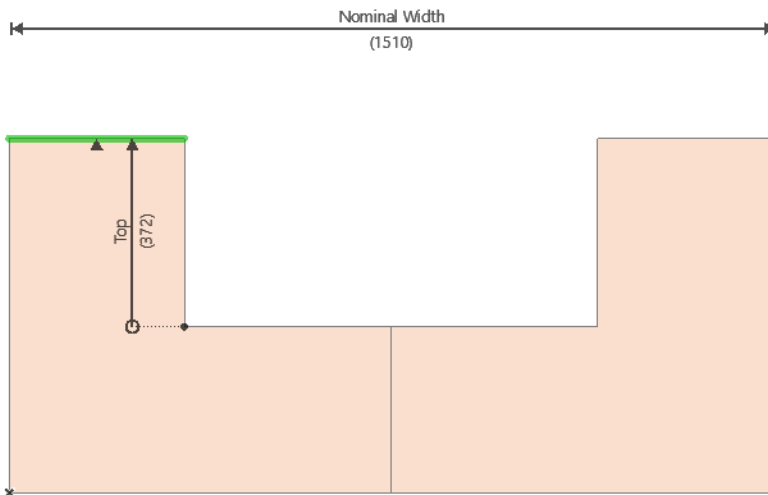


5. The second edge is added to the existing Dimension.

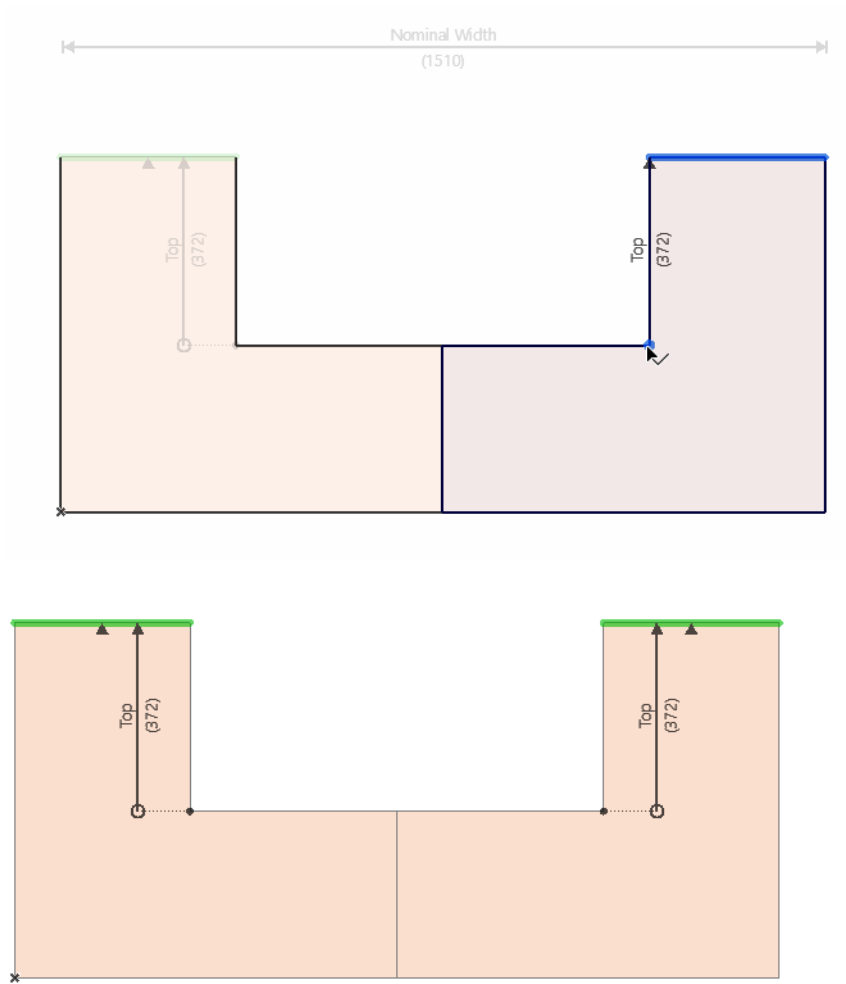
Add Dimensions to One Offset Modifier

You can add multiple dimensions to the same Offset Modifier. Every subsequent dimension will have the same length. This allows different edges in the same modifier to have dedicated anchors - in the case where those anchors are moved by other modifiers.

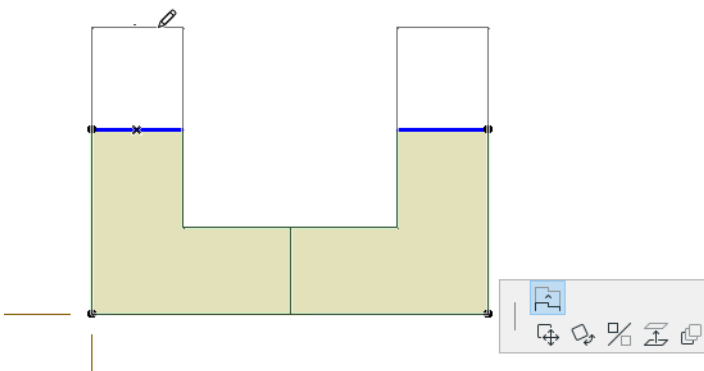
1. In Profile Manager, select the existing modifier (Top).
2. Click the **Add Dimension** icon.



- As you move the cursor, the eligible edges and anchor nodes are highlighted in blue. Click to choose an edge, then anchor the dimension in the usual way. Its dimension length is the same as the existing one defined in the modifier (set by the first dimension).

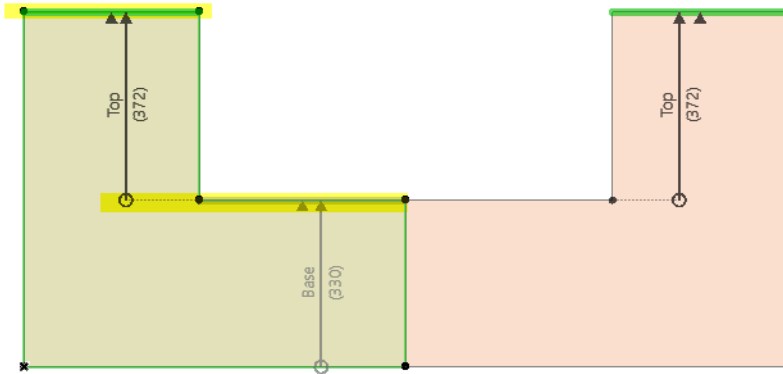


- The resulting single modifier controls two, separately anchored edges.

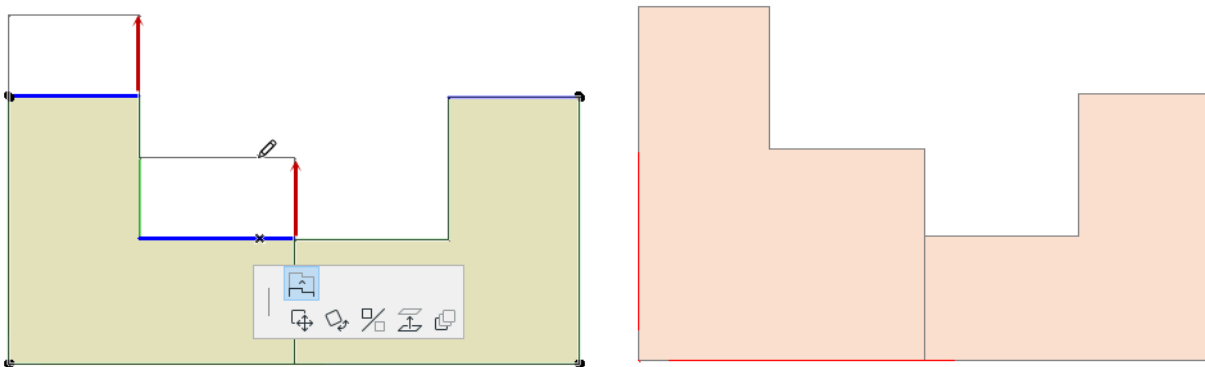


Two Connected Modifiers

In this example, the “Top” Offset Dimension on the left side is anchored to an edge which itself is another modifier (Base). Note that the right-side “Top” dimension has its own anchor.



If an anchor moves, so will the dimension which it anchors. In this case, if the Base is offset, the left-side Top dimension (anchored to the Base) also moves.

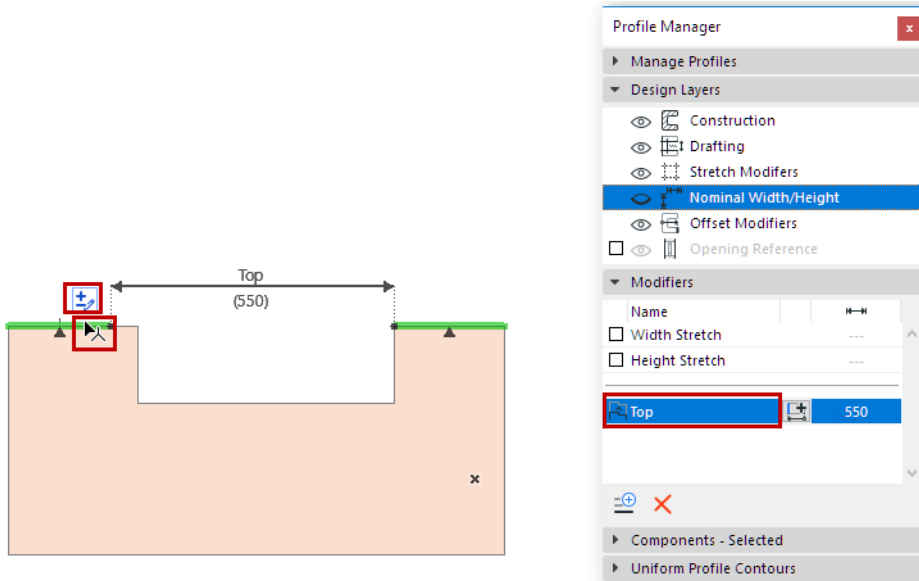


The right-side Top edge, however, remains in place: its anchor was not moved.

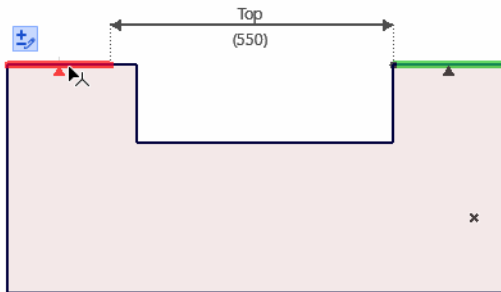
Remove Edges or Entire Dimension

1. In Profile Manager, activate the modifier you want to edit; or click a grey Offset Dimension to activate and select it
2. In Profile Editor, hover over the edge or dimension.

3. Click the appearing grip: **Add/Remove Edge**.

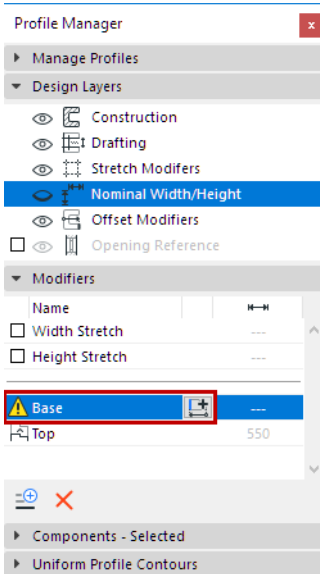


4. Hover over the edge so that it turns red: click to delete.



The edge is deleted.

If you hover over the last edge or dimension, the entire dimension and edge will be highlighted in red. When all dimensions in a modifier are deleted, that modifier gets a warning notice in Profile Manager.

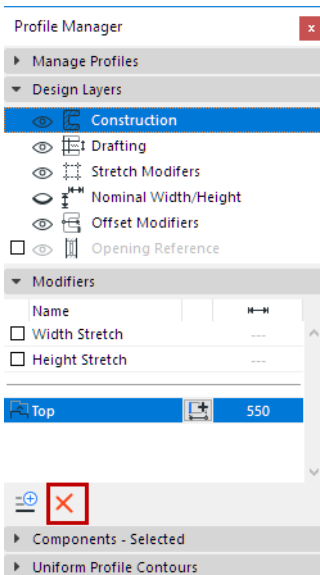


If needed, click **Add Offset Dimension** to create a new Dimension and assigned edge for that modifier.

Delete Modifier

1. In Profile Manager, select the modifier to be deleted.
2. Click Delete.

All Offset Dimensions and assigned edges in that modifier will be deleted.



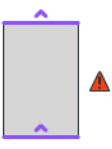
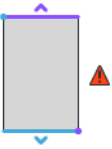
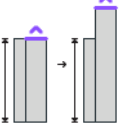
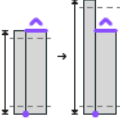
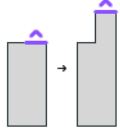
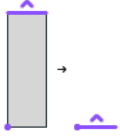


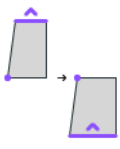
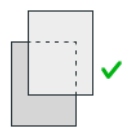
Rename Modifier

In the Modifier list, double-click its name to edit in place.

Offset Modifiers: Tips and Tricks

When creating or editing Offset Modifiers, it's worth keeping the following situations in mind.

	<p>Curved edges of Fills cannot be used in Offset Modifiers. Only straight segments are supported.</p>
	<p>Non-horizontal and non-vertical Fill edges that cross Stretch Modifier lines cannot be used in Offset Modifiers. This is because Stretch Modifiers break these edges to allow stretching/compression to occur between Stretch lines.</p>
	<p>In an Offset Modifier, two parallel Offset edges that face the same direction can prevent the Offset from working. A warning will appear when Storing the Profile. The Profile is saved, but this Modifier can't be used until the issue is resolved. To resolve it, try flipping the direction of one of the edges.</p>
	<p>Two Offset Modifiers connected to each other will cause an over-constraint, preventing either Offset from moving. A warning will appear when Storing the Profile. The Profile is saved, but this Modifier can't be used until the issue is resolved. To resolve it, try moving one Offset Modifier anchor to another point.</p>
	<p>Offset Modifiers do not affect or modify the Nominal Size of a profiled element. The Total Size is calculated from the entire bounding box of a profiled element, including Offset Modifiers.</p>
	<p>Modifying a profiled element's size (such as height, width or thickness) affects both Nominal Size and Stretch Modifiers. Offset Modifier values won't be affected, to maintain their constraint and distance to anchor point.</p>
	<p>An Offset Modifier that is adjacent to another edge will create a new edge when offset. This newly created edge will take its properties from the Fill as defined in the Profile Editor.</p>
	<p>Offset Modifiers can remove geometry, such as if you set a thickness to zero. This is supported and valid. However, if the entire element will disappear as a result, the action will be invalid, and the value will be reverted – to avoid a profiled element with no geometry.</p>

	<p>Offset Modifiers can cause geometry to flip, such as if you set a negative value. This is supported and valid.</p>
	<p>Overlapping Fills in the Profile Editor are supported. The geometry will be regularized at the element level based on the Building Material Intersection Priority of each Fill.</p>

Using the attribute dialogs

Applies to:

- Layers
- Surfaces
- Composites
- Building Materials
- Fills
- Lines

[Open attribute settings](#)

[Flat list or folder view](#)

[Resize panels in folder view](#)

[Use folders to organize attributes](#)

[Sort attributes](#)

[Rename attribute](#)

[New attribute](#)

[Duplicate attribute](#)

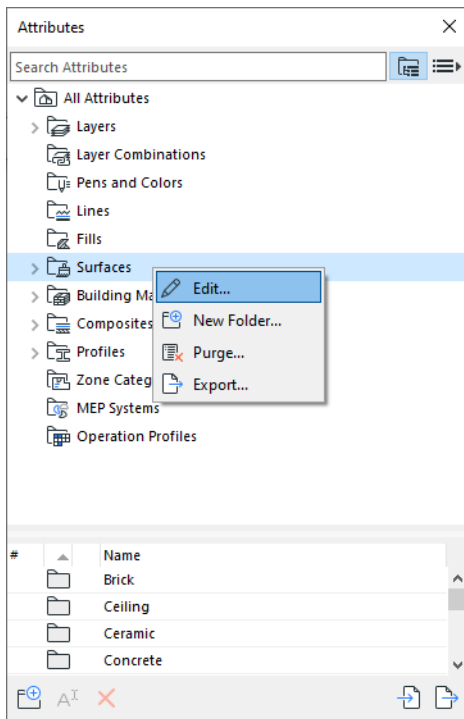
[New based on Selected \(Layers\)](#)

[New folder](#)

Open attribute settings

Use one of the following:

- **Options > Element Attributes**
- command from the Attributes toolbar (Windows > Toolbars > Attributes)
- From Attribute palette:
 - Double-click any listed attribute
 - Use the **Edit** context-menu command from a selected attribute or the attribute-type folder

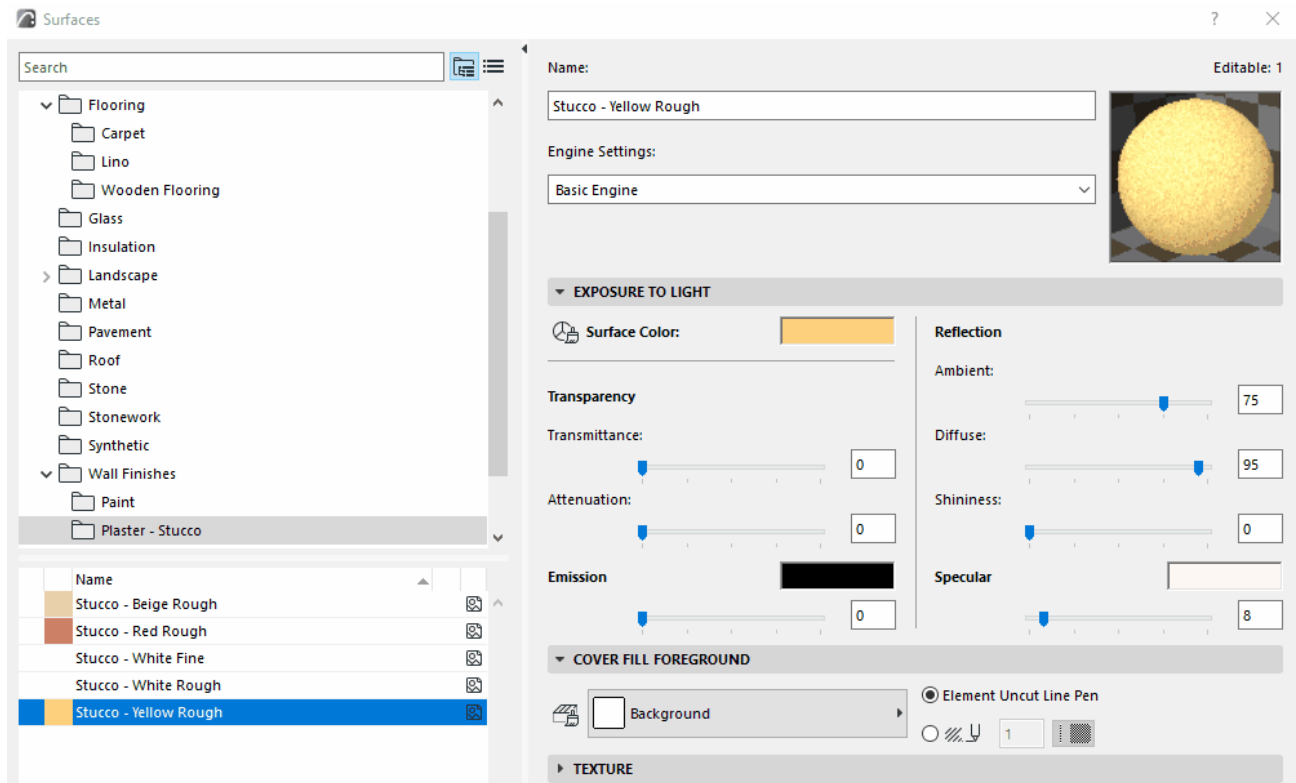


Flat list or folder view

The list of attributes can be displayed in folder or flat-list view.

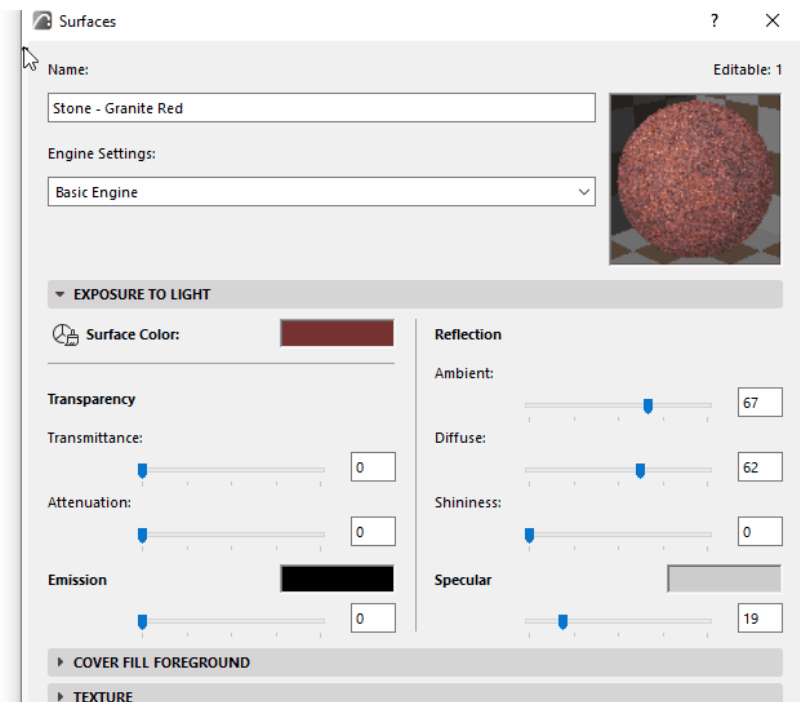
Use the icons to switch between flat list and folder view.

In folder view, click on any folder to see its contents (attributes or sub-folders) in the list below.



Open/close left panel

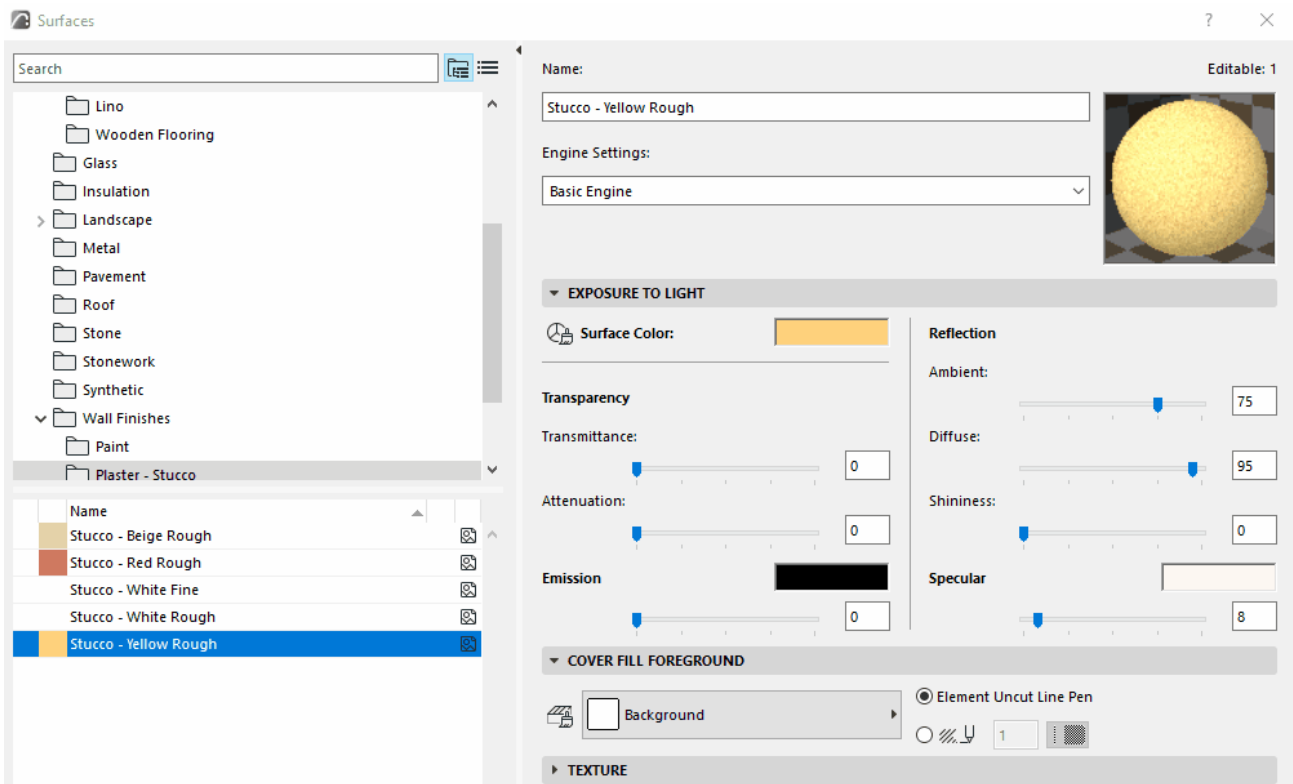
- Double-click anywhere on the splitter bar panel
- Click the black arrow at the top of the splitter bar



Resize panels in folder view

- Drag the splitter to the top to make the top panel disappear

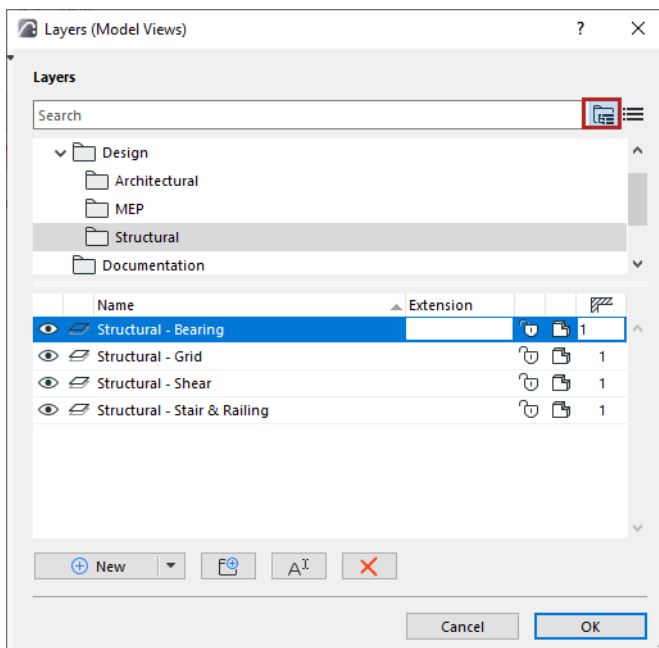
- Double-click the splitter to open/close panels



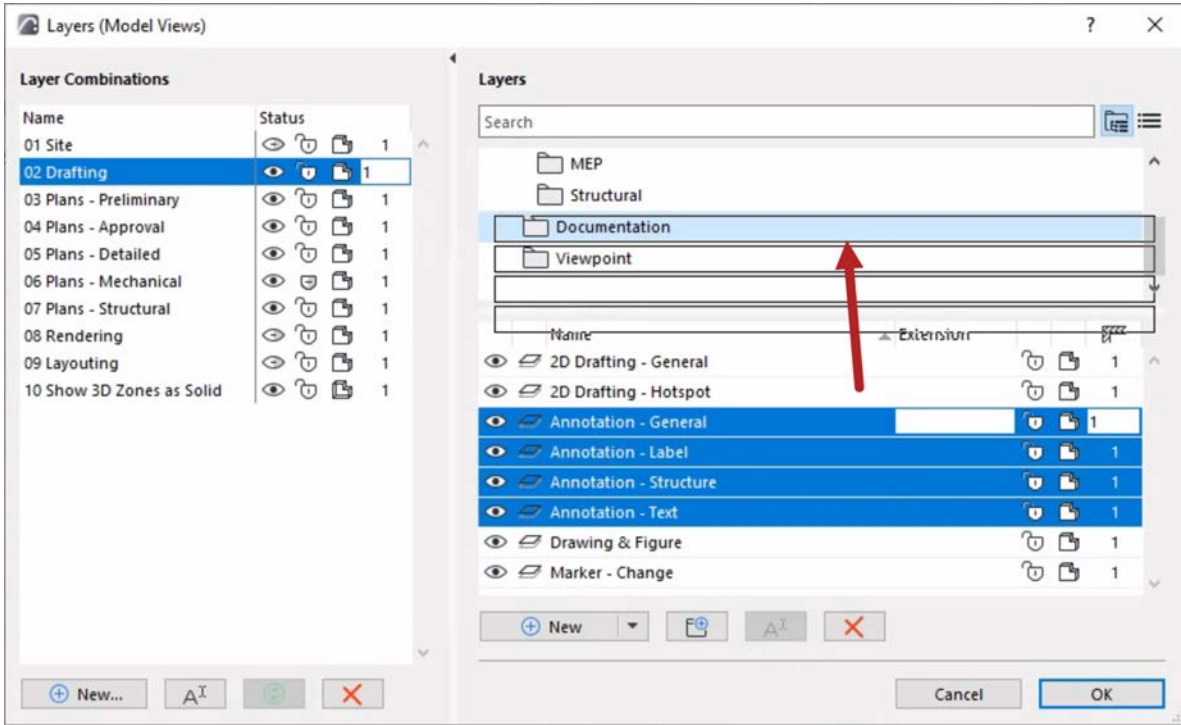
Use folders to organize attributes

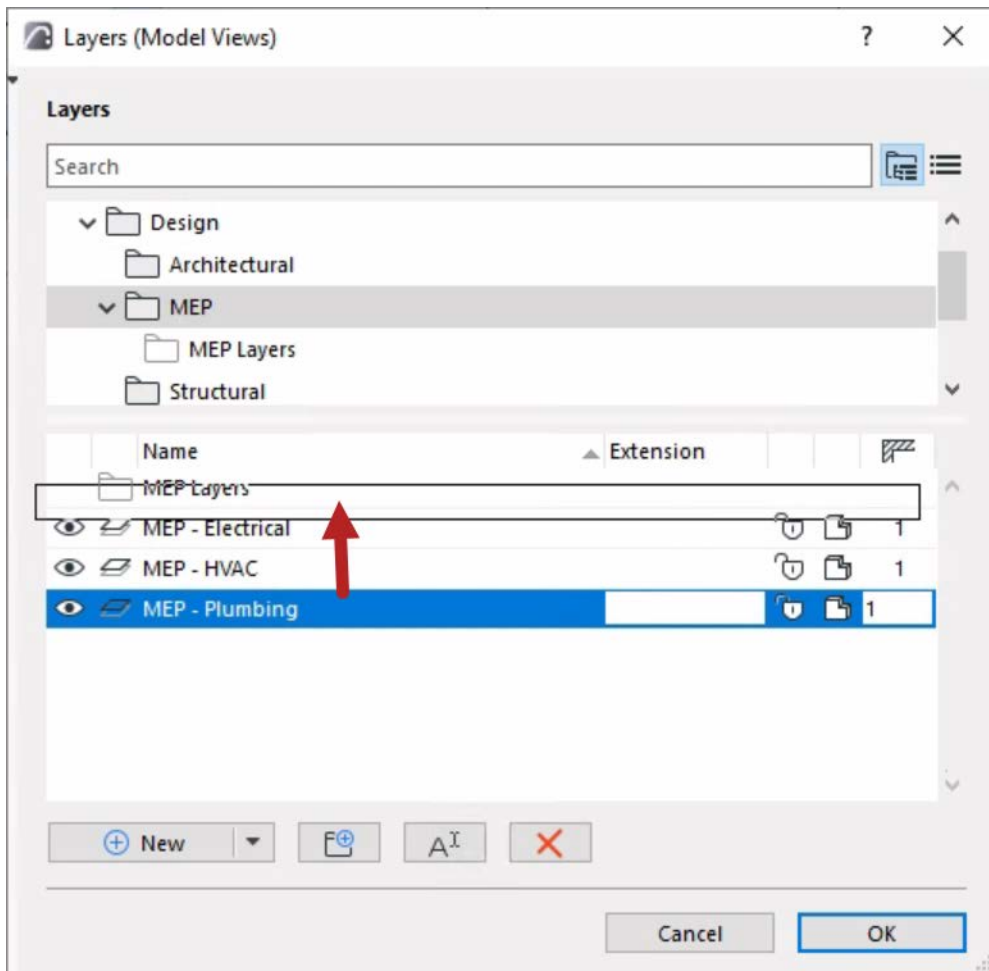
Folders are containers to help you keep track of your attributes. Use them to organize your attributes in any hierarchy.

In folder view, click on any folder to see its contents in the list below.



To reorganize the hierarchy and folder contents, just drag and drop attributes and/or folders from the bottom list into other folders, either in the top or bottom list.

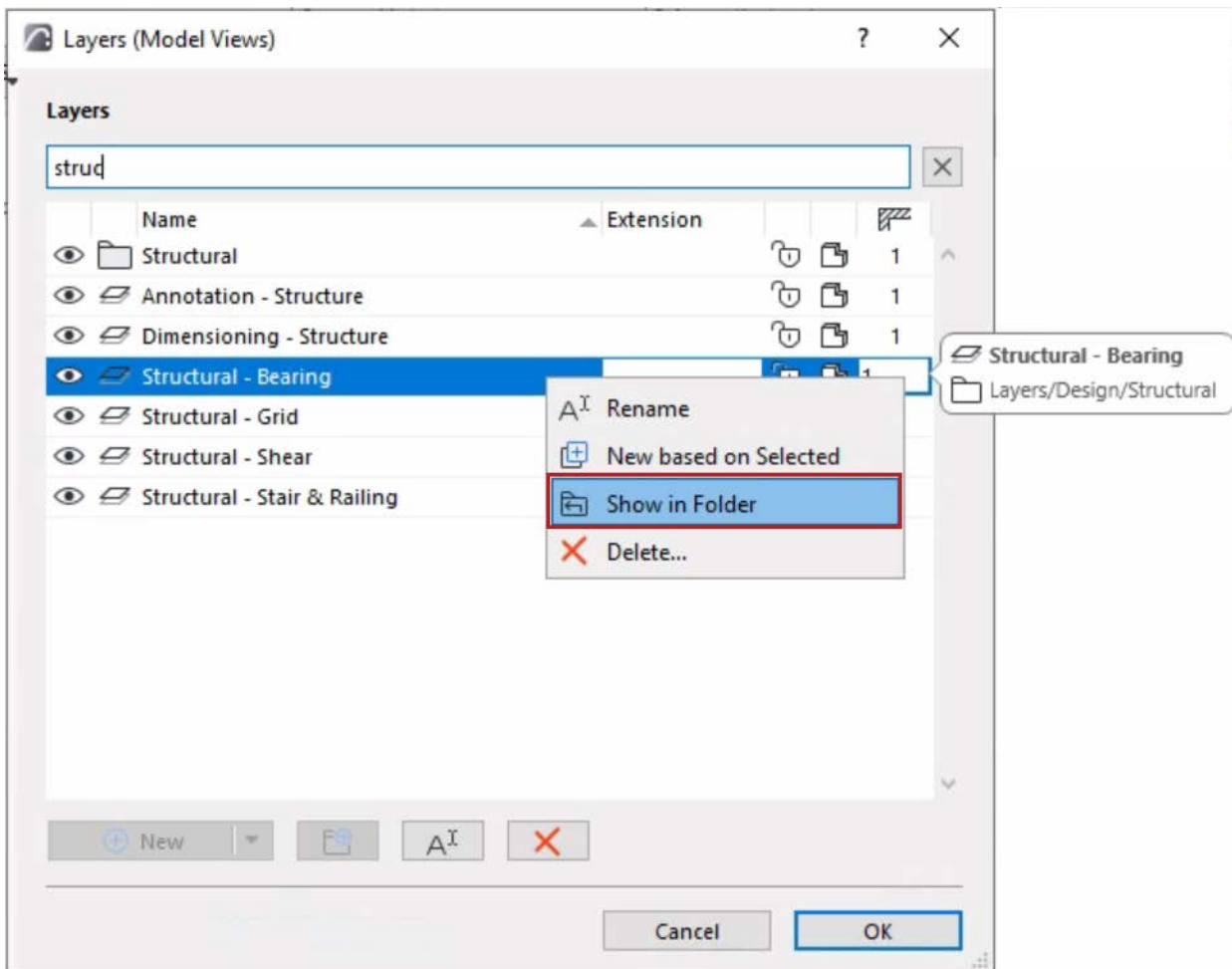




Locate item in folder structure

To see where an attribute is located in the folder hierarchy, do one of the following:

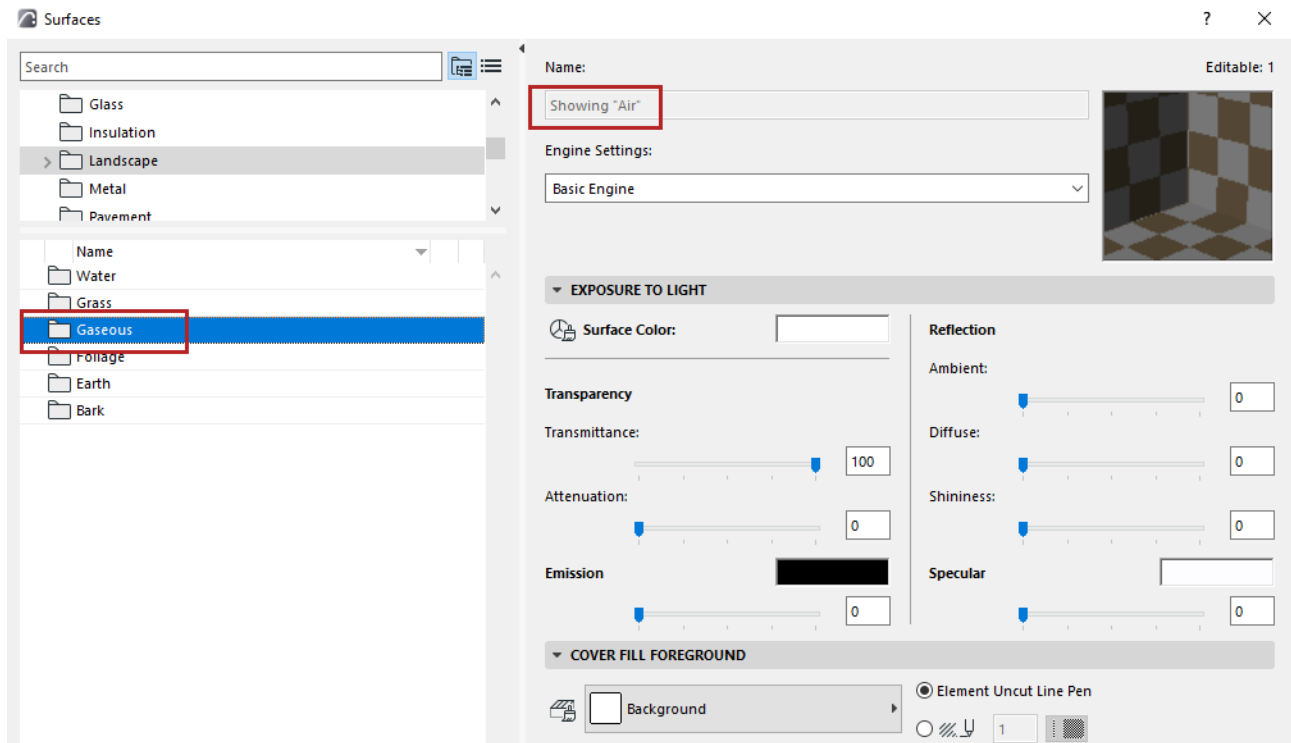
- Select the item, then use the “Show in Folder” context menu command
- Hover to see the info pop-up



Edit all items in sub-folder

In folder view, select a sub-folder to edit all its contents at once.

Hint: In folder view, select a sub-folder to edit all its contents at once.

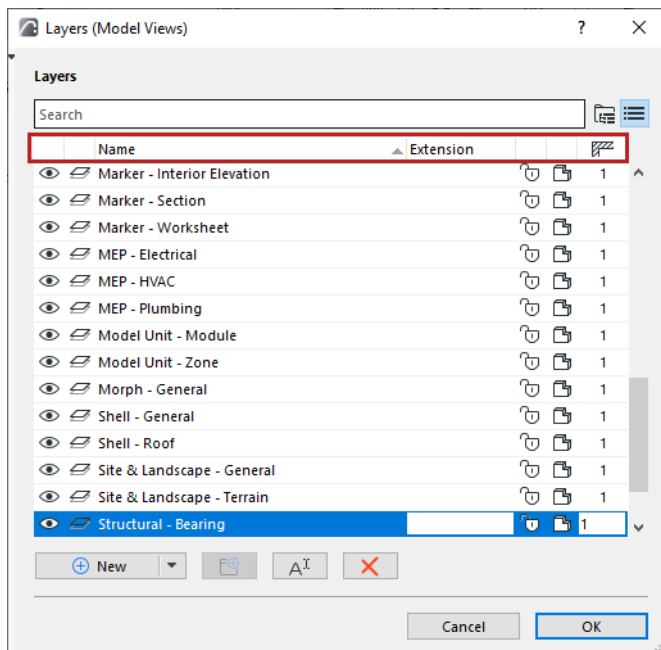


Sort attributes

In any list:

Items are sorted alphabetically by default.

Sort the list by clicking on any column header.



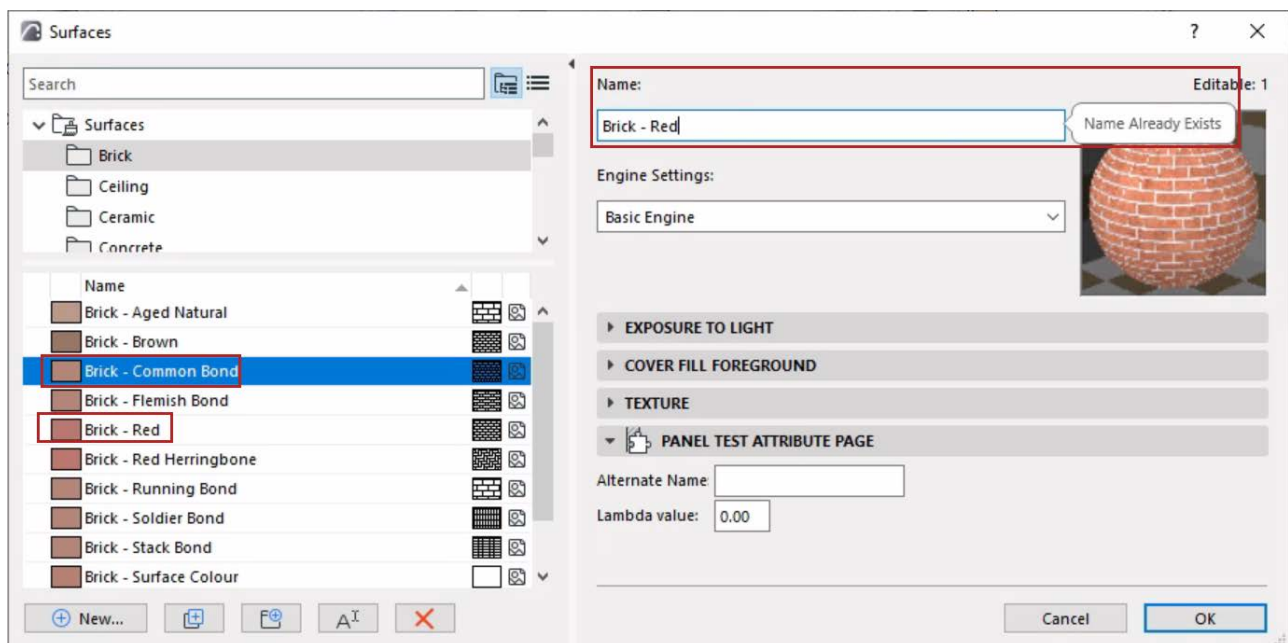
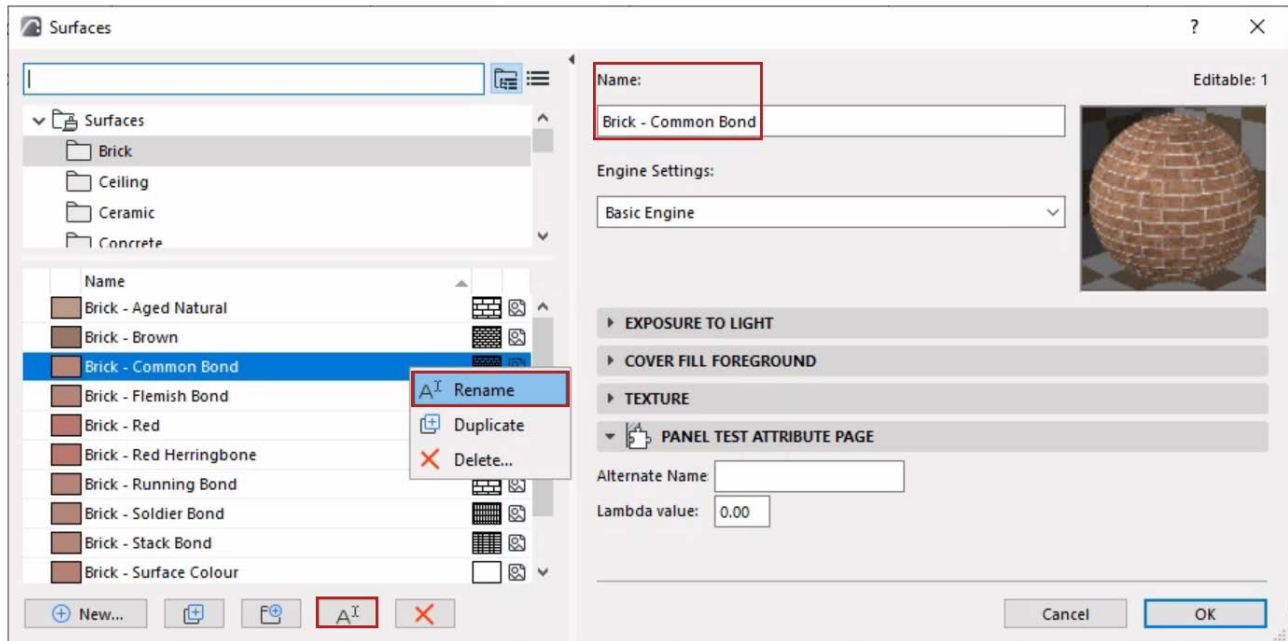
Rename attribute

Do one of the following:

- Click **Rename** from the attribute dialog, the Attributes palette, or the item's context menu

- Edit the Name field directly

Note: A bubble informs you if you try to use a name that already exists

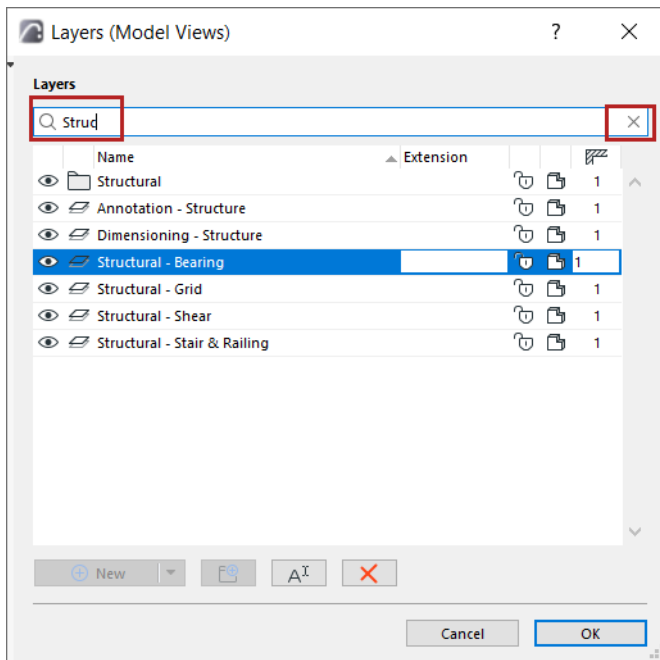


- Select an attribute or folder, then click its name to rewrite it

Search for attributes

Use the search field to locate attributes or folders by name.

Click X to exit the search.



New attribute

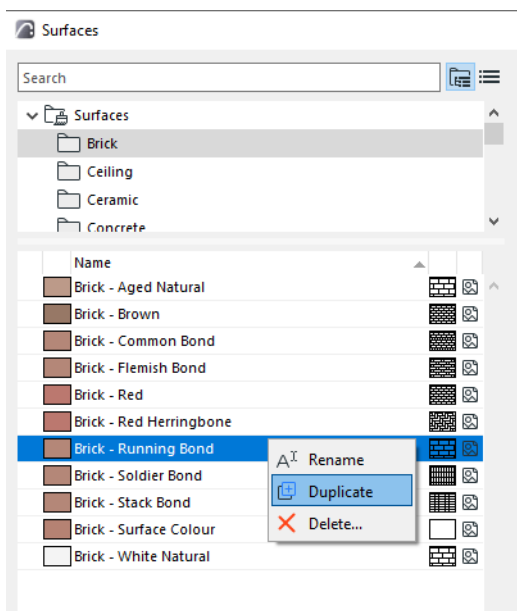
Do one of the following:

- Click the “New...” button
- Right-click in empty space in the bottom list to access the New command

Duplicate attribute

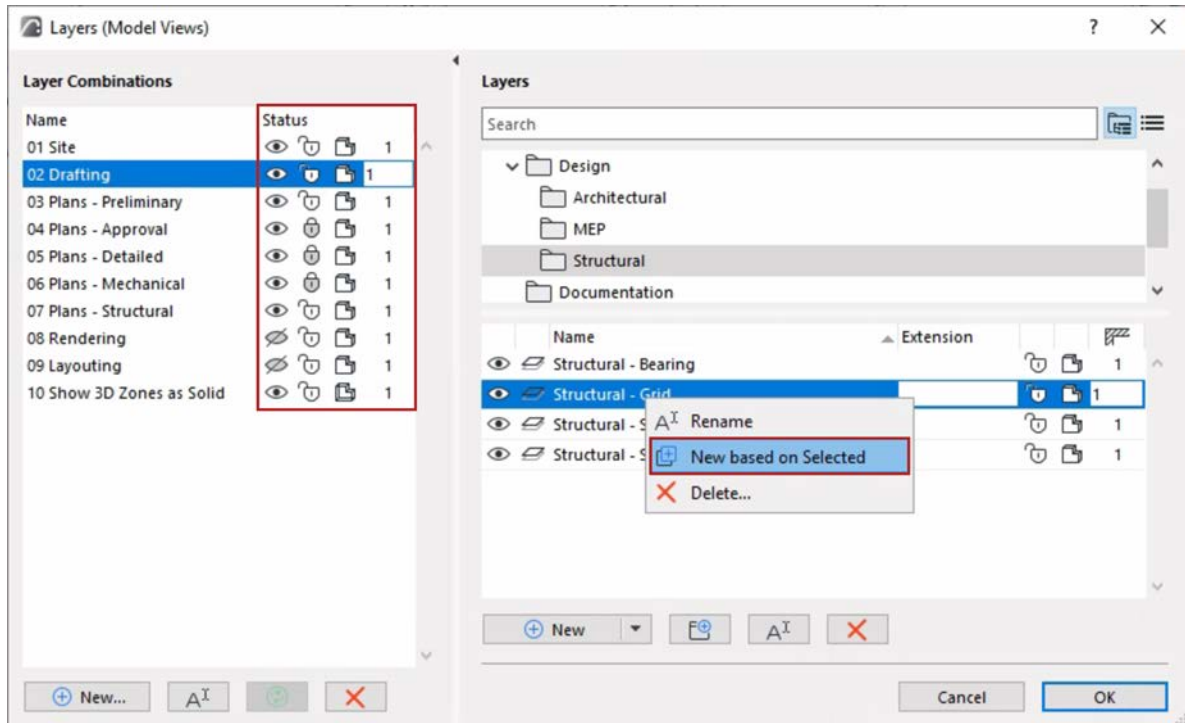
Do one of the following:

- Select any listed attribute, then click the Duplicate button
- Select any listed attribute, then context menu command
- Click the “New...” button, and use the Duplicate option from the appearing dialog



New based on Selected (Layers)

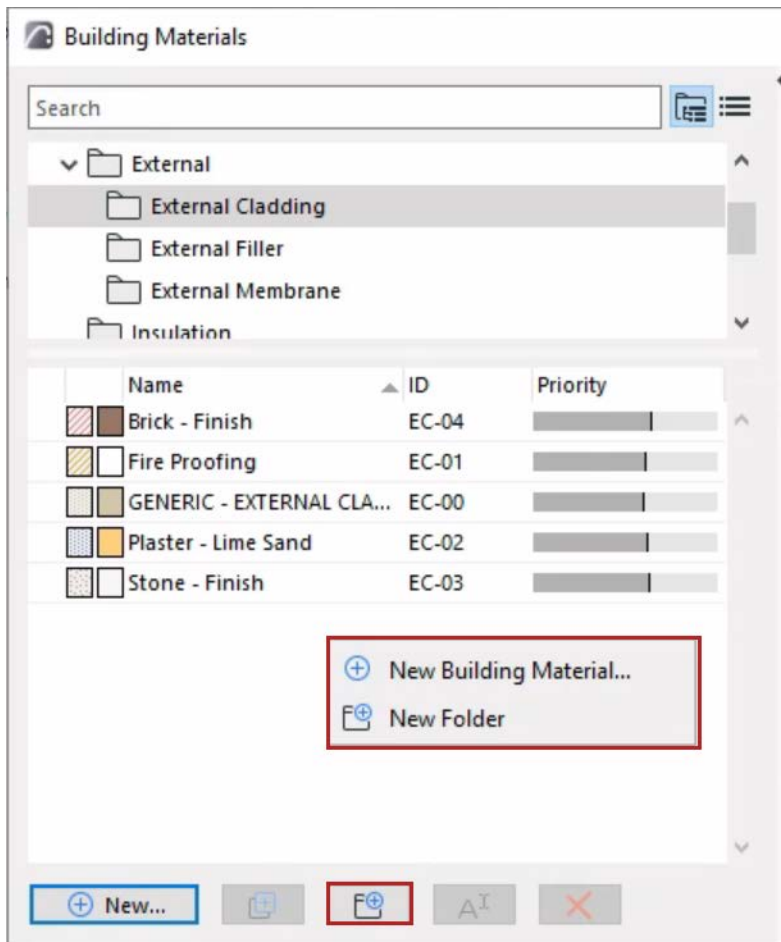
The new Layer will inherit the statuses (e.g. show/hide, lock/unlock) of the currently selected layer in all Layer combinations.



New folder

Do one of the following:

- Click "New Folder" button
- Select any folder, then context menu command
- Right-click in empty space in the bottom list to access commands



Attributes palette

The Attributes palette gives you direct, easy access to any attribute and its settings.

Use Attributes palette to

- List, manage and edit all the attributes of your current project
- Import attributes from another project or XML file
- Save any set of attributes as an XML file
- Purge unused attributes from the project

Note: In addition to the Attributes palette, the Attribute Manager dialog box (familiar from earlier Archicad versions) is available. In more complex import/export workflows, you might prefer its two-panel interface to compare two lists of attributes. [See *Attribute Manager*](#).

Open Attributes Palette

- **Options > Element Attributes > Attributes**
- Attributes toolbar



- From Profile Manager (click the Attributes shortcut)

Navigate in Attributes palette

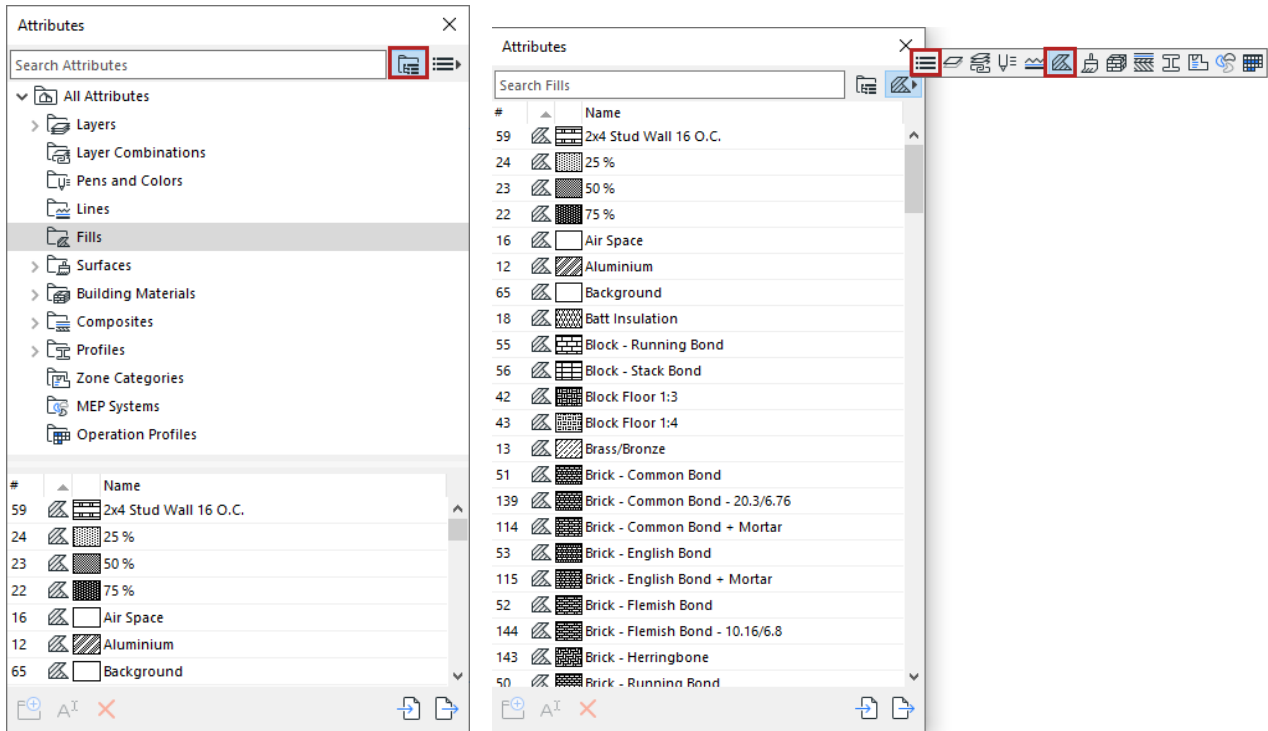
Project attributes can be displayed in folder or flat-list view.

In folder view of Attributes, the top section displays each attribute type in a separate folder.

This set of top-level folders is fixed, but you can create any number of sub-folders.

Click any folder to see its contents (any sub-folders and attributes) in the list below. Empty folders are grey.

Click the list view button to list all attributes in a single list, or to filter by type (click the attribute's icon to list only that type).



Attributes: Folder view; List view with option to filter by type

Related Topics

[Search and sort attributes \(Attributes palette\)](#)

[Manage attributes and folders \(Attributes palette\)](#)

[Edit attribute settings \(Attributes palette\)](#)

[Import attributes](#)

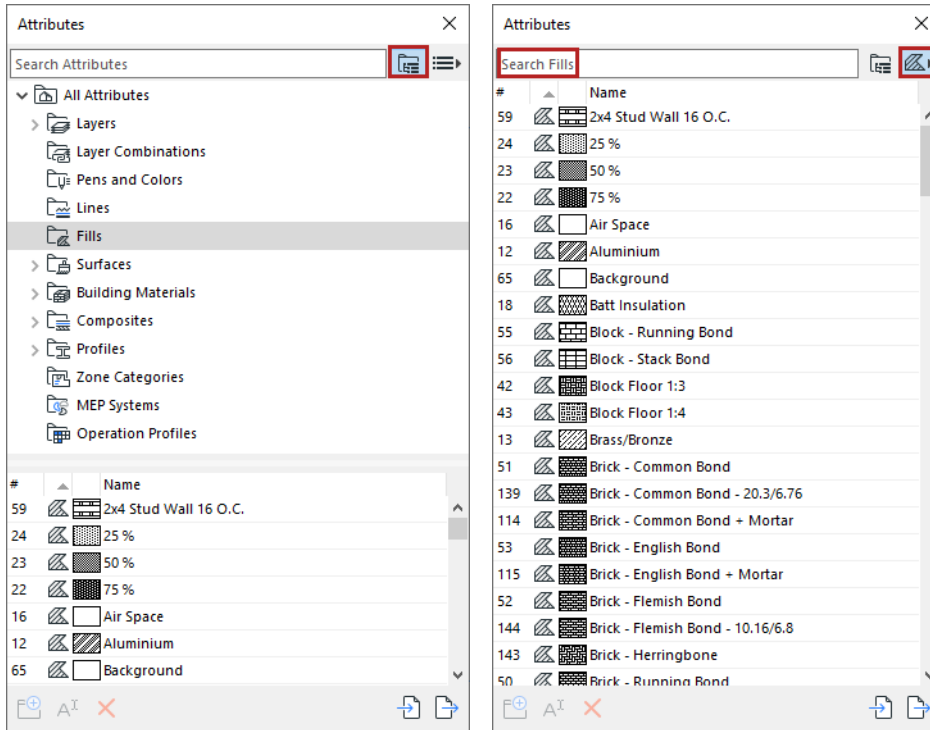
[Export attributes](#)

[Purge attributes](#)

[Reservation in Attributes palette; Reserve All Attributes \(Teamwork\)](#)

[Delete attributes](#)

Search and sort attributes (Attributes palette)



Search

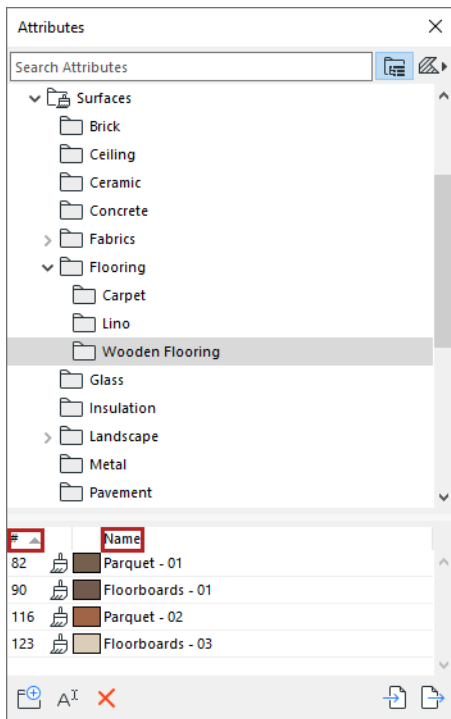
Use the search field at top to locate folders and attributes by name, inside the entire attribute collection.

In list view, if you have filtered the list by type, the search is limited to that attribute type.

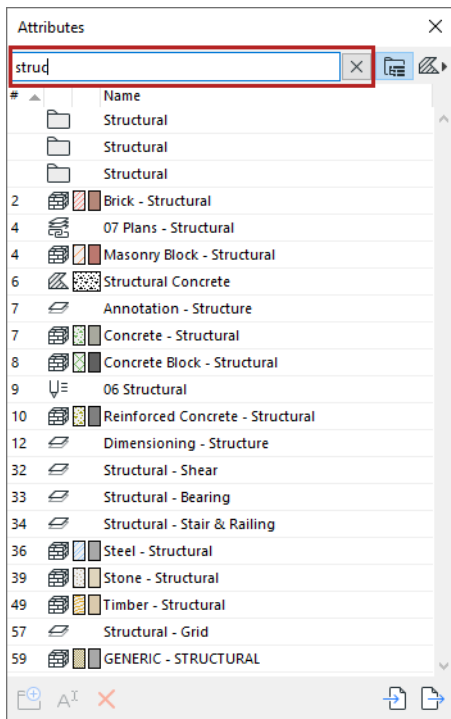
Click the X to exit the search.

Sort

- Sort the bottom list by name
- Sort listed attributes by index number
- Hover to see the item's name and location in the folder hierarchy



Sort by index number or name

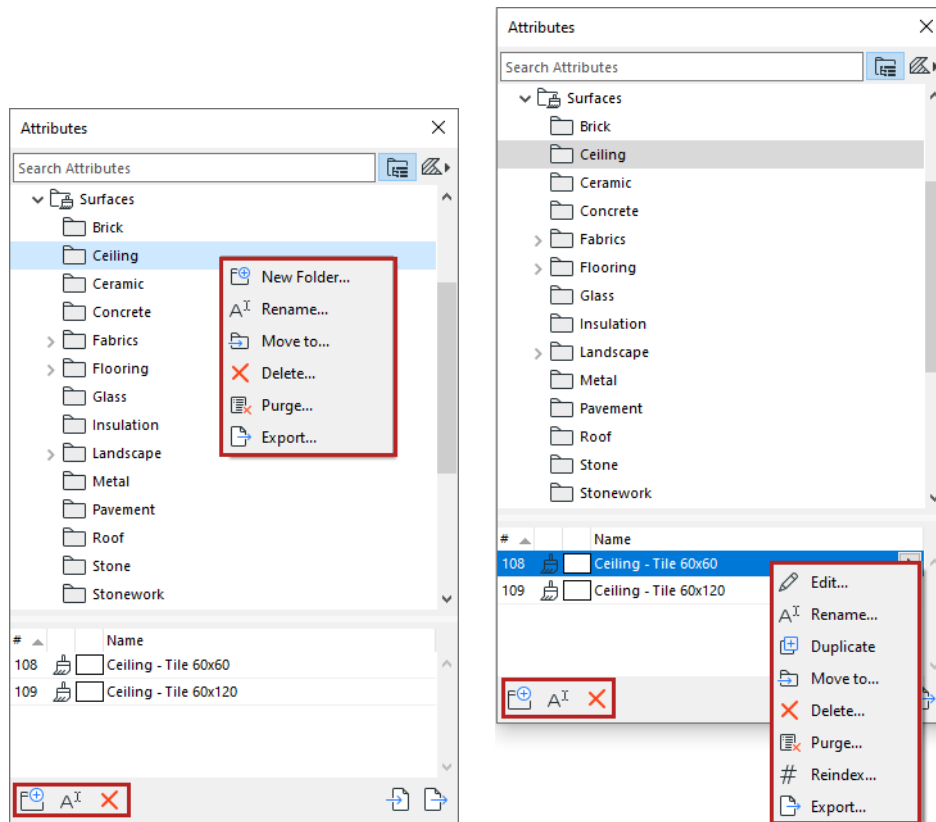


Search by name

Manage attributes and folders (Attributes palette)

For a selected attribute or folder:

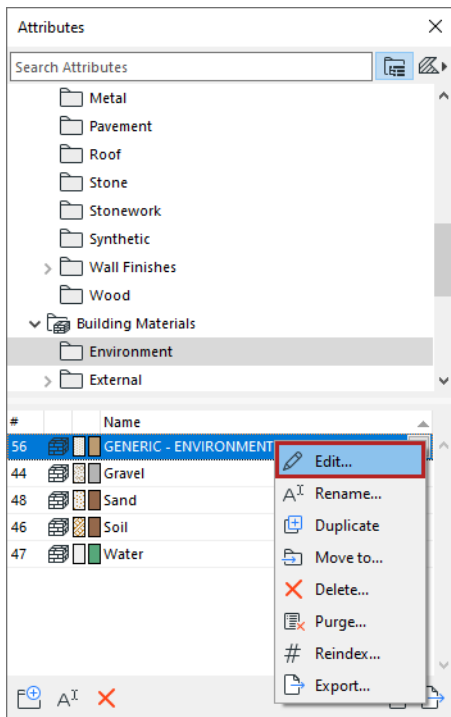
Use context menu or button commands.



Edit attribute

From Attributes:

Double-click any listed attribute, or use the **Edit** context-menu command to open its settings dialog.



Create or move folders

Within each main attribute folder, you can create any number of sub-folders at any level.

Drag and drop one folder at a time to any other level of the hierarchy.

Note: You can't have two child folders of the same name inside the same parent folder.

Merge two folders of the same name

Drag one of the folders into the parent of the other one. The contents of the two subfolders will be merged in a single folder.

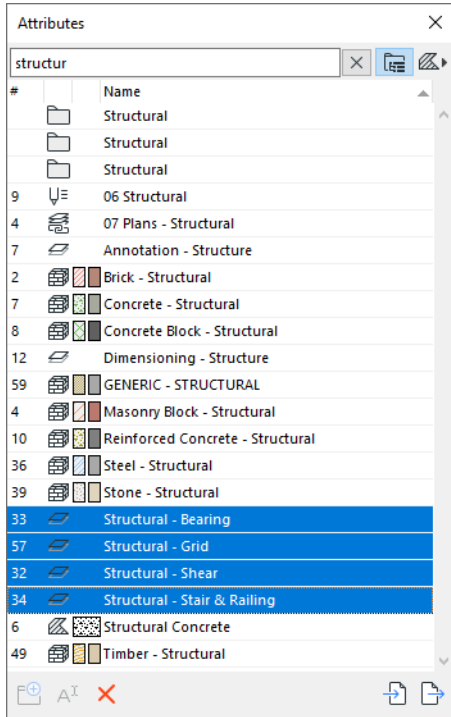
Move to

For selected folders or attributes:

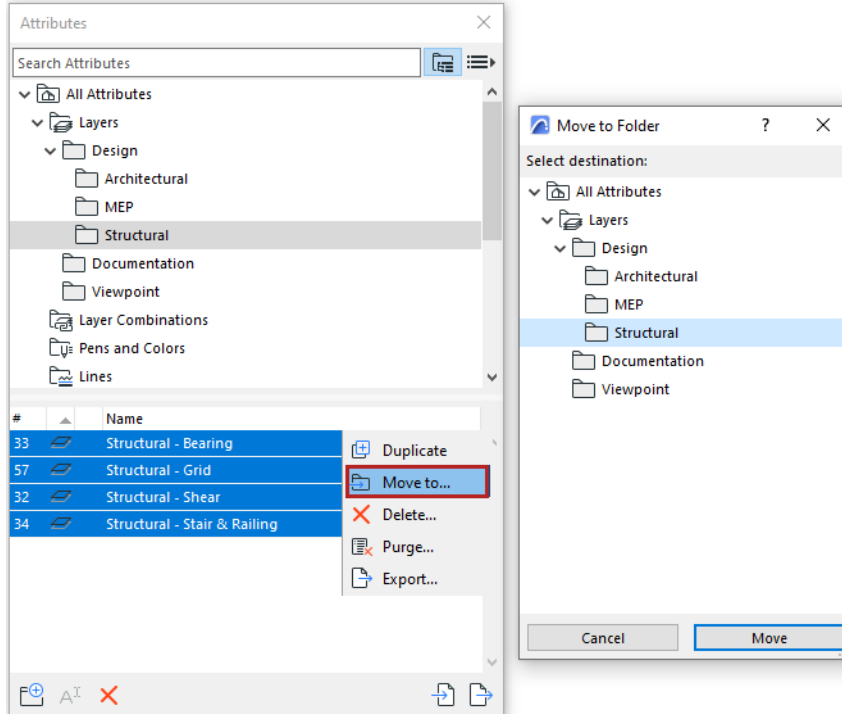
Click **Move to** from the context menu

For example

1. Use the Search bar to find and select particular attributes



2. Close the search. Your search selection is still active.



3. Click **Move to** from the context menu
4. Choose a folder to move the selected items.

Drag and drop attributes among folders

Select/multi-select, and drag and drop attributes from the bottom list into any folder.

Note: Attributes must be located under their own type. (You can't put a fill into a surface folder.)

Edit attribute settings (Attributes palette)

Double-click any attribute to open and edit in its settings dialog.

Or right-click a main attribute folder and use Edit to access settings.

Non-editable attributes

The following attributes are fixed and their settings cannot be edited:

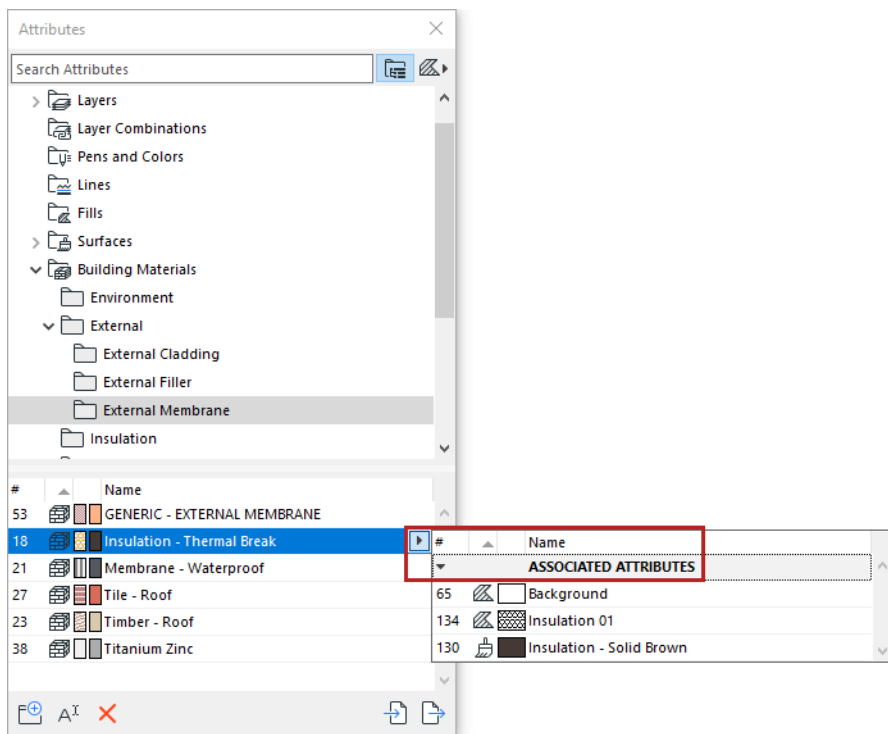
- Layers: Archicad Layer
- Lines: Solid Line
- Fills: Background, Foreground, Linear Gradient Fill, Radial Gradient Fill

Associated attributes

Fills, Surfaces, Building Materials, Composites, and Profiles have other attributes associated to them (e.g. a Building Material with its Fill and Surface definitions).

For a selected attribute: Click the pop-up arrow or use the right/left arrow keys to see its associated attributes.

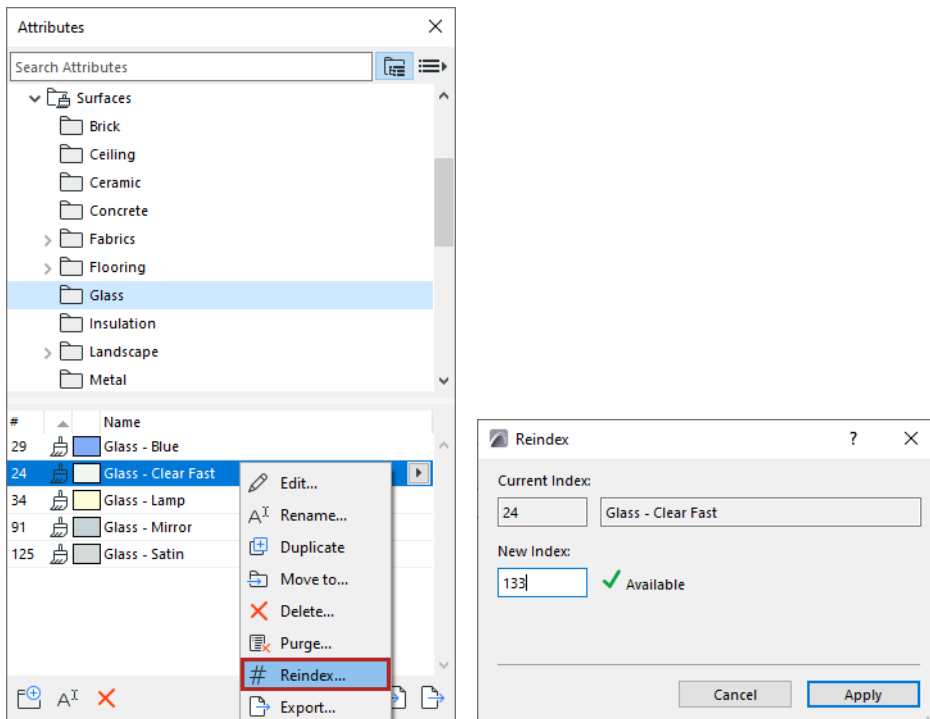
Double-click the associated attribute to open and edit its settings.



Attribute index number/Reindex

Elements refer to attributes by their **index number**.

- In the attribute list: click the number column to sort by index number
- To edit the number of a selected attribute, click **Reindex** from the context menu. The appearing dialog suggests an available index.
- You can enter any other new index, as long as it is not in use by another attribute of the same type. The dialog tells you whether the proposed new Name index is available or not.



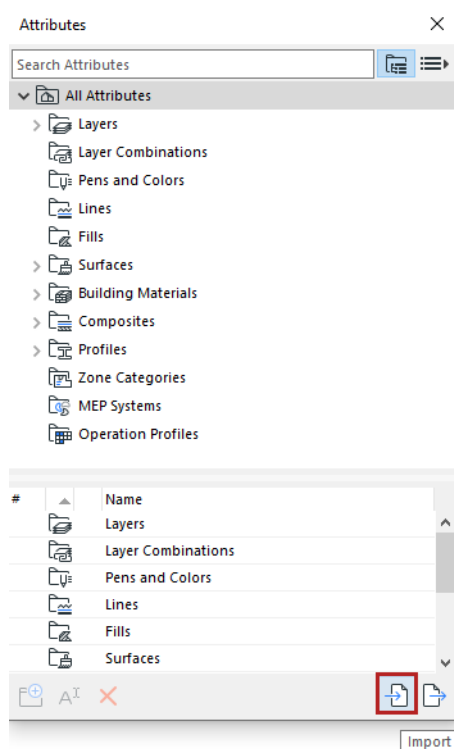
Reindex: Details

The Details button appears on the Reindex dialog in a Teamwork context. Click Details to see information on attribute reservation.

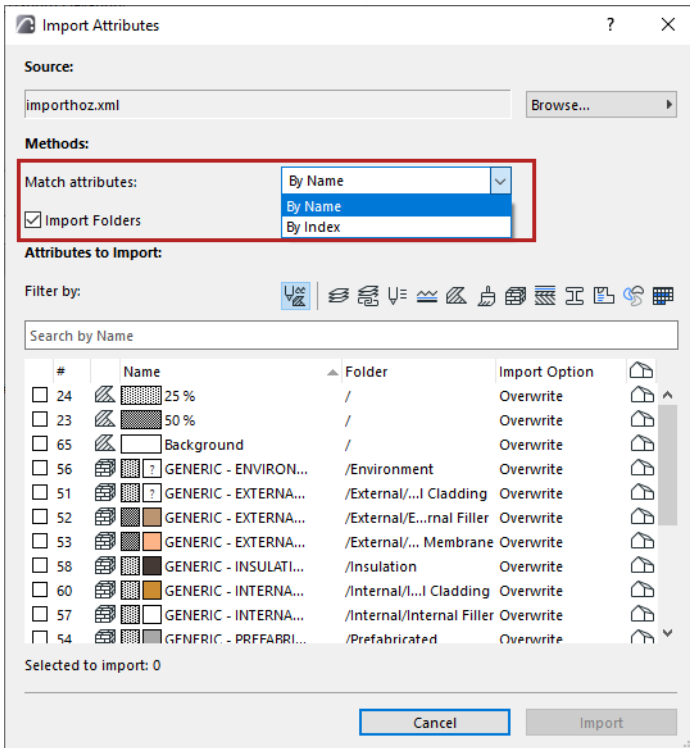
Import attributes

Note: In addition to the Attributes palette, the Attribute Manager dialog box (familiar from earlier Archicad versions) is available. In more complex import/export workflows, you might prefer its two-panel interface to compare two lists of attributes. [See *Attribute Manager*](#).

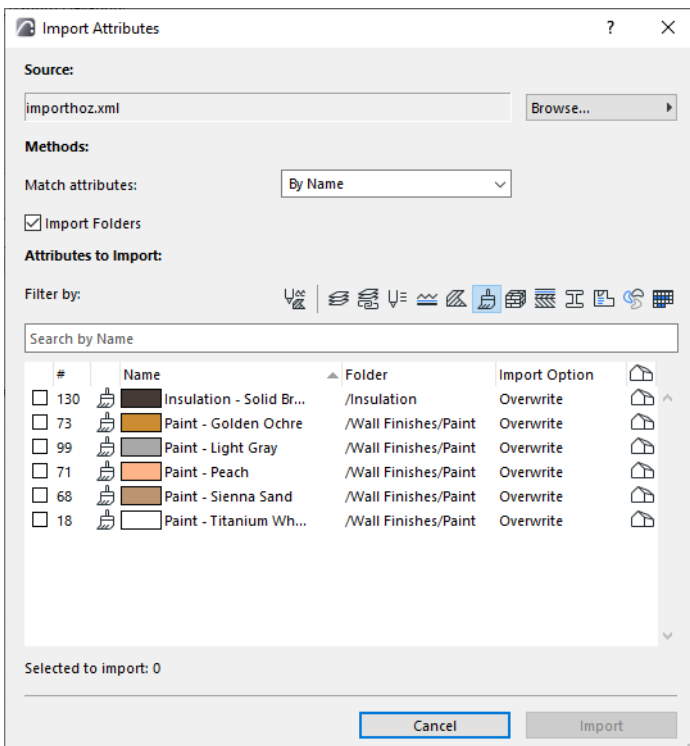
In Attributes:



1. Click the **Import** button.
2. The Import Attributes dialog opens. Click browse to choose a file containing the Archicad attributes you need (e.g. XML, .aat, or Archicad project).
3. Choose a method to match the incoming attributes with the existing ones in your current project.

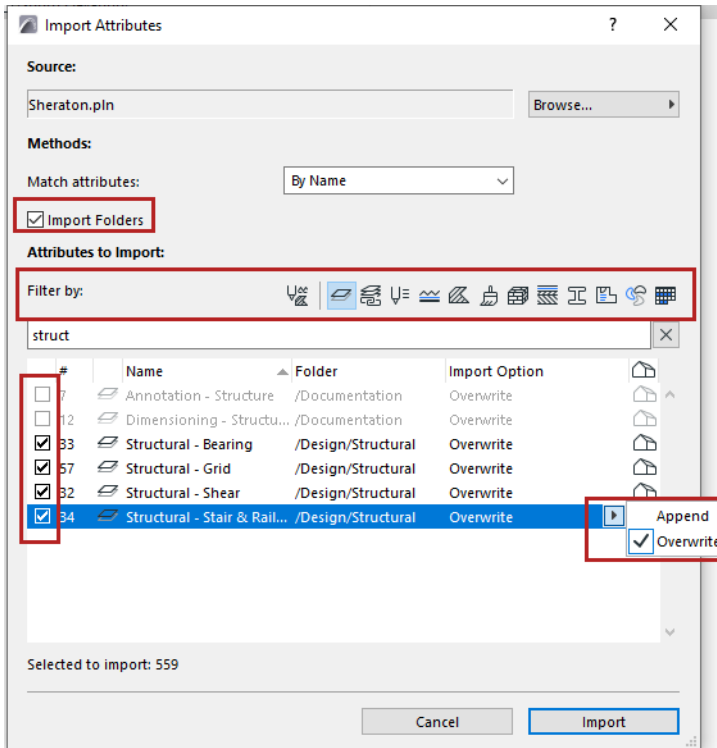


- Match by name: Overwrite existing attribute having the same name. If the name doesn't exist, append a new attribute.
 - Match by index: Overwrite existing attribute having the same index number. If the index doesn't exist, append a new attribute.
4. Filter the list of attributes by type, as needed.



5. For each attribute you want to import, check its box.

- Sort (by column headers) and multi-select as needed.
- The **Selected to Import** number is updated accordingly. (This is the total number of attributes of all types currently selected for import.)



Note: By selecting an attribute, all associated attributes will automatically be selected as well. You can deselect any of them by opening the popup.

6. Click **Import**, then confirm.

Associated Attributes

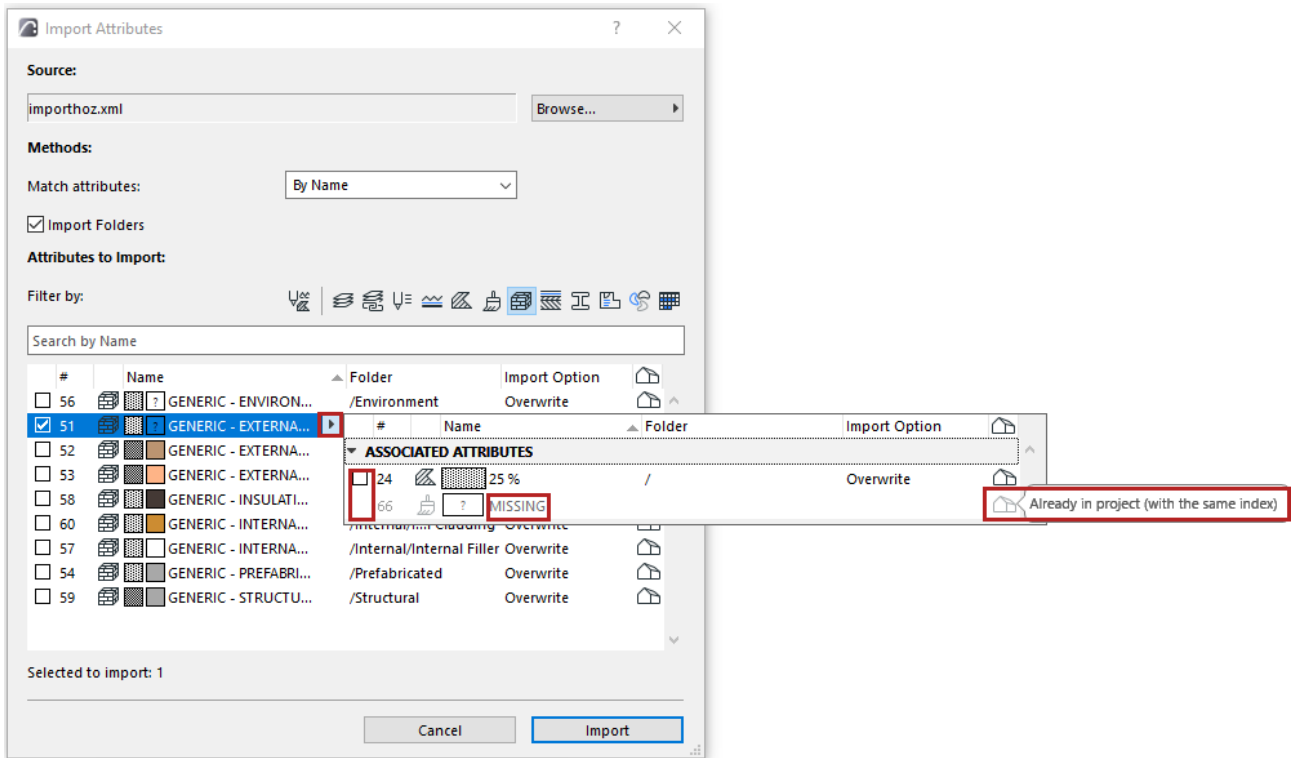
For a complex attribute, associated attributes are shown in a popup.

Access with the arrow button, or the right arrow key.

Check to import the ones you need.

The house icon signals whether the attribute is present in the project.

Double-click an attribute to jump to it.



See additional options below.

Import folders

Note: Only relevant for incoming files based on Archicad 26 and later. (Earlier versions did not have attribute folders.)

Checked by default. If you uncheck it, then the import process will ignore any folder hierarchy within the imported file, and the imported attributes will simply be added to the respective attribute's top-level folder (e.g. imported Surfaces are added to the Surfaces folder).

Append vs. Overwrite

If a listed attribute already exists in the project, a house icon appears at the end of the row.

You can either **Overwrite** the existing attribute, or **Append** the imported attribute: in this case, the appended attribute gets a name extension.

Note on Import of Building Materials

When you import a file via Attributes palette:

The incoming Building Material attributes may include Classifications and Properties that already have defined values.

But these values cannot be added to the host Archicad project unless the referenced Classifications and Properties *already exist* in the host project.

Therefore, before importing such Building Materials, review their incoming Classifications and Properties, and ensure that they already exist in your project (use Classification Manager and Property Manager).

If they do not exist, you can create the missing Classifications and Properties, and then import the Building Materials. All the referenced values will come into the project.

Otherwise, the assigned values are lost.

Export attributes

Use the Export function from Attributes palette to save selected attributes, or the contents of a selected folder, in two possible formats: XML or text.

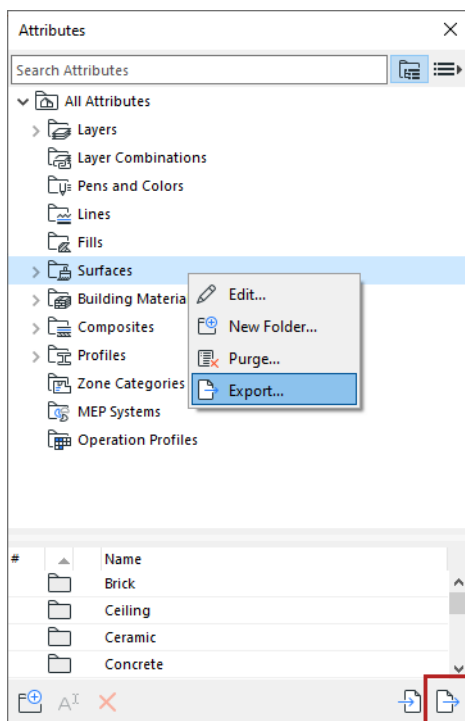
Note: In addition to the Attributes palette, the Attribute Manager dialog box (familiar from earlier Archicad versions) is available. In more complex import/export workflows, you might prefer its two-panel interface to compare two lists of attributes. See [Attribute Manager](#).

In Attributes:

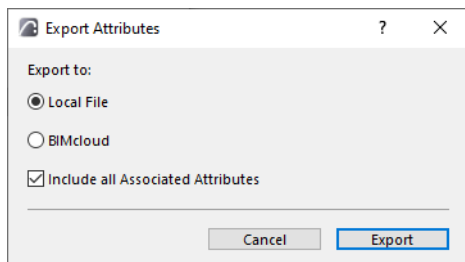
1. Select the attributes and/or folders to export.

Hint: Use search, sort, and multi-select as needed.

2. Click the **Export** button
 - from the bottom of Attributes
 - from context menu of a selected attribute or folder



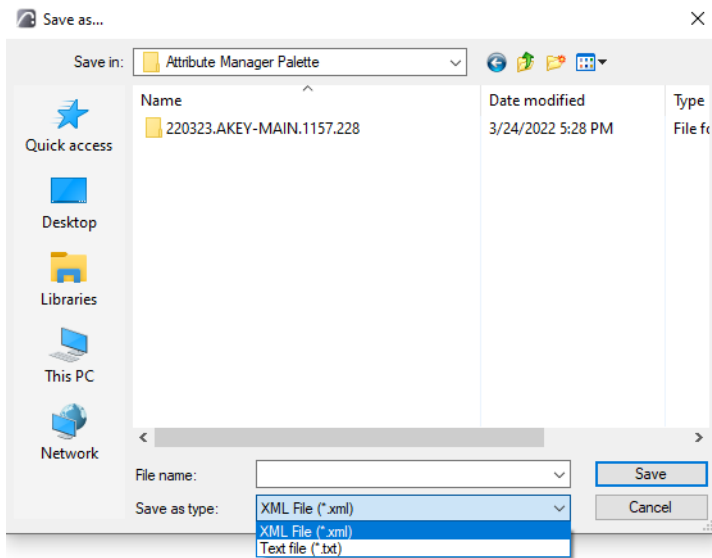
3. In the appearing dialog: Choose to export to either a local folder or to BIMcloud.
4. Check the box to **Include Associated Attributes**, if needed. (See [Associated attributes](#).)



5. Click **Export**.

Note: Empty folders are not exported.

6. Choose a file format:



- XML: Use this to transfer attributes among Archicad projects
- Text (.txt): This can be useful with the **Replace Attributes by File** function, which automates the process of deleting and replacing attributes. This function requires generation of a text file in a particular format, and running a command that calls this file.

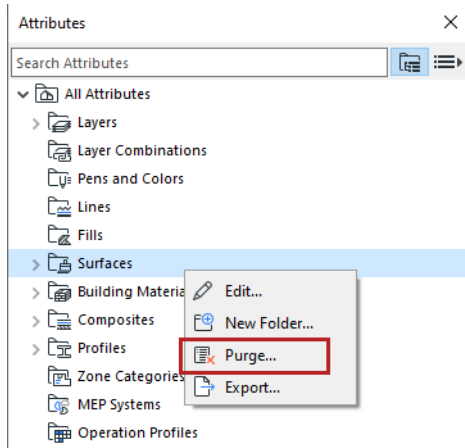
See a description of this process at [Attribute Migration article](#).

Purge attributes

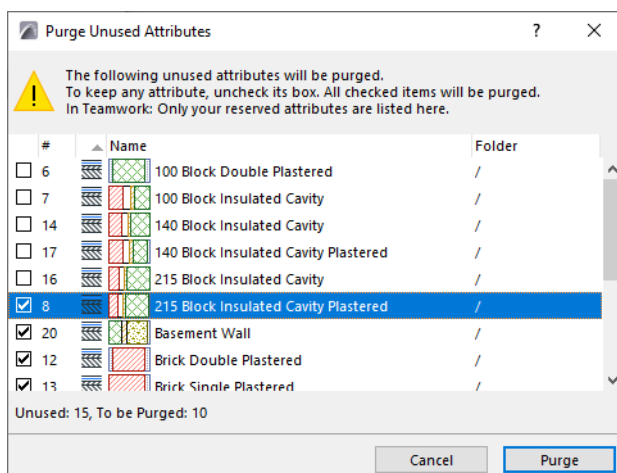
Use the Purge command to remove unused attributes from your project. You can only purge attributes that are not being used in your project. Therefore, unlike the Delete attributes command, “Purge” cannot result in missing attributes.

In Attributes:

1. Select attribute(s) or any folder.
2. From the context menu, choose **Purge**.



3. The next dialog lists the unused attribute(s) to be purged.
4. Edit the list:
 - All the listed attributes are checked by default.
 - For any attribute you don't want to purge, uncheck box at the left.
 - Sort (by column headers) and multi-select as needed.
 - The **To be purged** number is updated accordingly.



5. Click **Purge**.

See also [Delete attributes](#).

Delete attributes

When migrating a file, using file merge or working with hotlinks, you may need to get rid of the unneeded attributes.

Deleting an attribute folder means you delete all its contents.

Deleting attributes can result in missing attributes

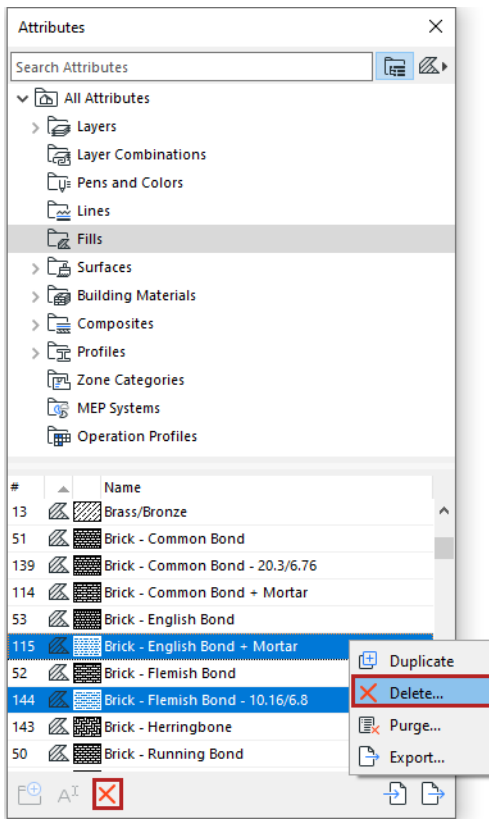
- in model elements which use those attributes
- in other attributes which are using the deleted item. (For example, if you delete a fill type that is also used as part of a composite.)

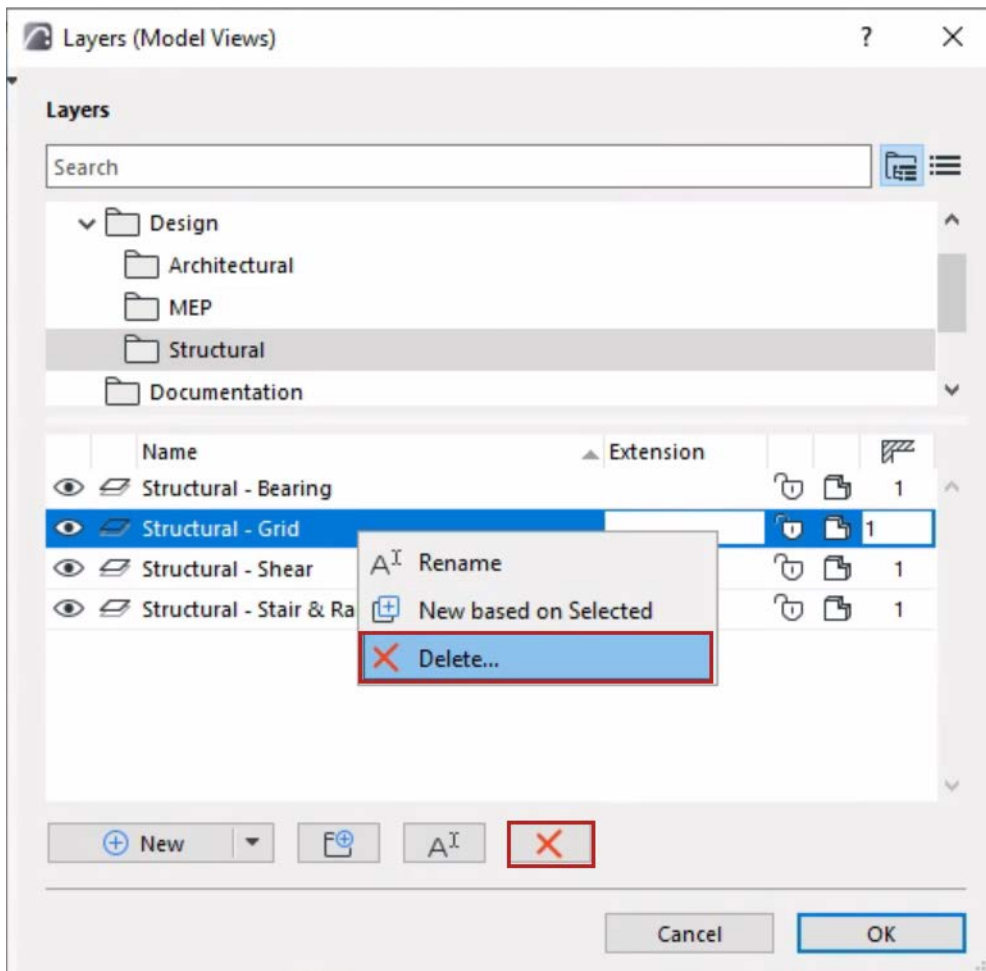
To avoid missing attributes, the Delete and Replace dialog pops up to help you choose replacement attributes.

Note: You can use the Purge command (from Attributes) to remove *unused* attributes from your project. Unlike the Delete attributes command, “Purge” cannot result in missing attributes. See [Purge attributes](#).

Delete and Replace

1. Find and select the attributes (or folder) you want to delete:
 - in the Attributes palette, or
 - in its respective dialog box (at **Options > Element Attributes**)
2. Click **Delete**
 - from the attribute dialog or Attributes palette
 - the item’s context menu
 - your keyboard Delete or backspace button

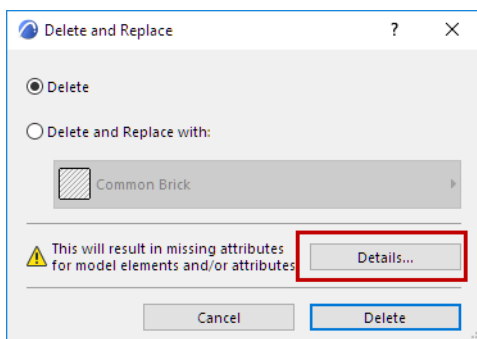




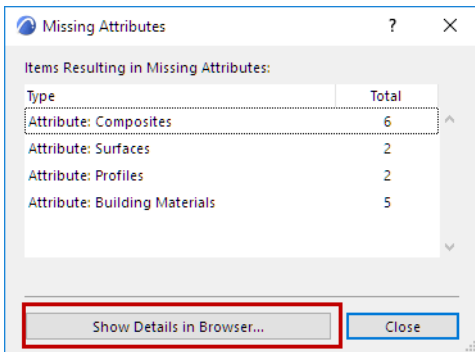
The **Delete...** button opens the **Delete and Replace** dialog for the following attributes:

- Layers (except the Archicad Layer) - see [Delete Layers and Move Elements](#).
- Lines (except the Solid Line)
- Fills (except Background, Foreground, Gradient Fills)
- Surfaces
- Building Materials
- Profiles
- Zone Categories

In the Delete and Replace dialog: choose either **Delete**, or **Delete and Replace**.



The warning icon is displayed if missing attributes will result. Click **Details** to show the number of missing attributes. For further detail on these items, click **Show Details in Browser**.

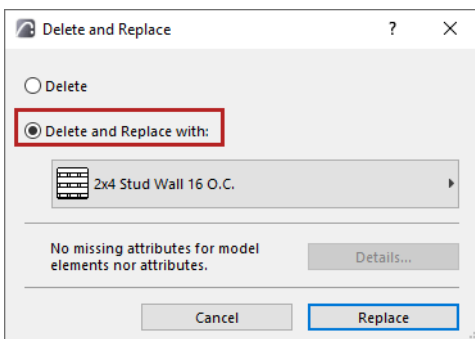


Note: Only elements and attributes that will be “Missing” are listed under Details. Elements that will have a missing attribute within its Composite or Profile are not listed. This is because the Composite/Profile will not be missing, just one of its components.

In Teamwork, **Details** will show the number of elements or attributes that will result in “Missing” attributes which are not reserved by you.

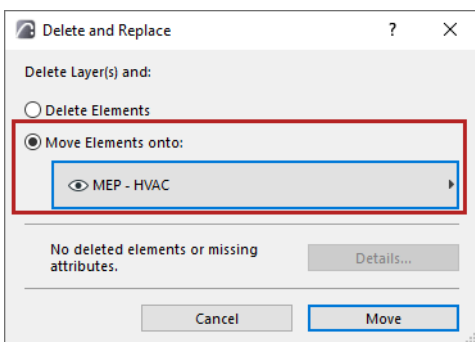
Delete and Replace

Choose this option to select a replacement attribute.



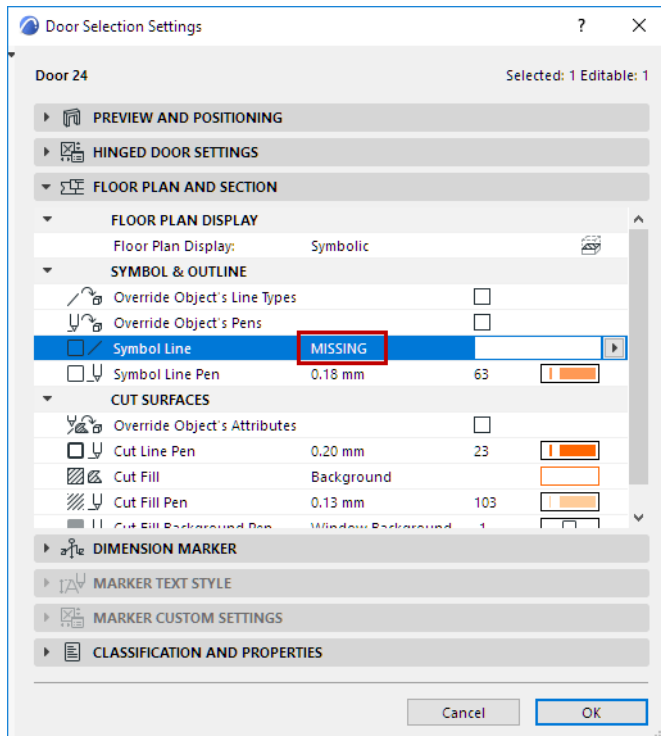
Delete Layers and Move Elements

If you delete a layer, you have the option to move its elements onto a different layer. Otherwise, when you delete a layer, you will also delete all the elements that the layer contains.



Missing attributes

Although attributes are saved with the project, it can still happen that your project contains elements which refer to missing attributes. You will notice this in the element's Settings dialog box, as in the case of the missing line type referenced by this door:



Missing attributes may occur:

- if you have copy-pasted elements from a different project using a different set of attributes
- if you have placed objects whose GDL script refers to attributes missing from your project
- if you have removed used attributes from your project via Attributes palette
- if you have removed used attributes via Delete and Replace Layer

To locate elements referring to missing attributes, use Find and Select.

See [Find and Select elements with missing attributes](#).

Find and Select elements with missing attributes

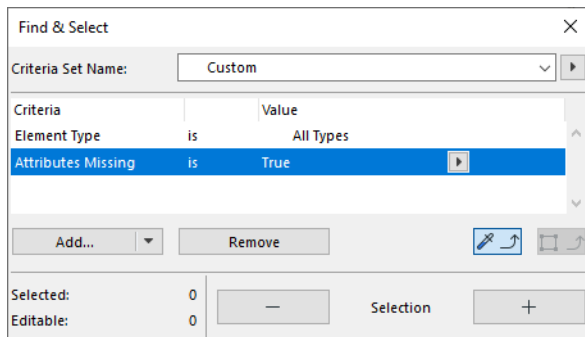
Although attributes are saved with the project, it can still happen that your project contains elements which refer to missing attributes.

[See also *Missing attributes*.](#)

To locate such elements easily, use Find and Select.

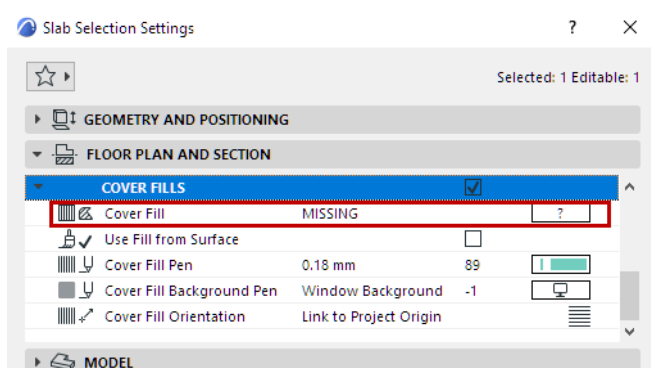
[See also *Find and Select Elements*.](#)

1. In the Criteria field of the Find and Select dialog box, choose a specific element type, or All Types.
2. Add “Attributes Missing - True” as the next criterion.

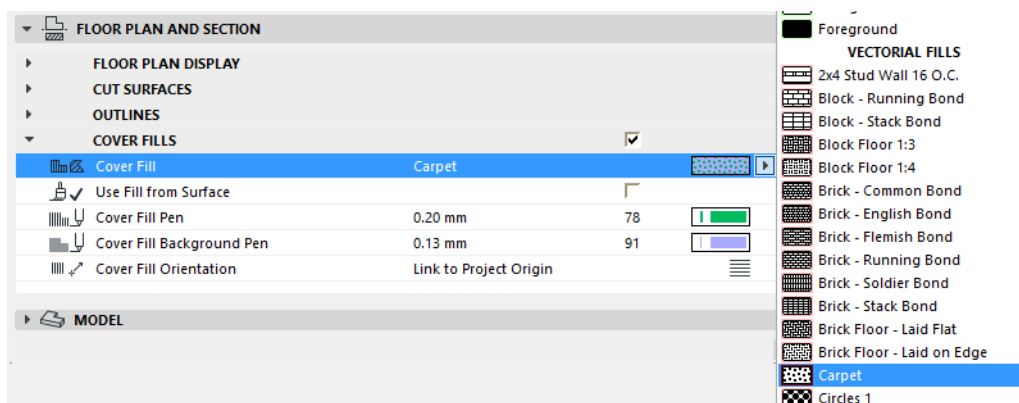


3. Click the Plus button to select all the elements that fit these criteria.

In this example, a slab has been identified and selected as using one or more missing attributes. In that Slab's Settings dialog box, it becomes clear that the missing attribute is a cover fill.



The easiest solution is to assign a different cover fill, from the fills that are available to the project, by clicking the Cover Fill attribute pop-up. Here we have chosen to use the Carpet fill.



Alternatively, if you know the location of the missing attribute you need (e.g. if it is part of another Archicad project), you can use Attributes palette to transfer the needed attribute from the other project to the current one.

See [Attributes palette](#).

Custom attributes of GDL objects

Some GDL object scripts include custom attributes. These attributes may be defined either as part of their individual object scripts or defined in the MASTER_GDL script.

When you load such objects into a project, their custom attributes are automatically merged into the project's attribute set as follows:

- If the Attribute definition is contained in the MASTER_GDL script, then once the library containing the MASTER_GDL script is loaded, the attributes are merged into the Archicad project attributes. Attributes with the same names are not replaced.
- If the Attribute definition is contained in the individual library part scripts, then
 - Fills and Lines are merged into the Archicad project.
 - Surface and Texture attributes are *not* merged into the Archicad project attributes.

Attribute Manager

Note: The **Attributes palette** (new as of Archicad 26) provides a streamlined interface for organizing and editing attributes – including folders and flat lists, and direct access to each attribute's settings. You can also use the Attributes palette for simple import/export of project attributes. [See Attributes.](#)

In addition, the **Attribute Manager** dialog box (familiar from earlier Archicad versions) is available. In more complex import/export workflows, you might prefer its two-panel interface to compare two lists of attributes. Its controls are described in this section.

Open the Attribute Manager from **Options > Element Attributes > Attribute Manager**.

Use this dialog to

- List and edit the attributes of your current project
- Import attributes from another project or XML file; edit and add them to current project attributes
- Save any set of attributes as an XML file

Topics in this section:

[View Attribute Lists by Type](#)

[Temporary List in Attribute Manager \(Right Side\)](#)

[Associated Attributes](#)

[Attribute Index Number](#)

[Checked Attributes](#)

[Search for Attributes](#)

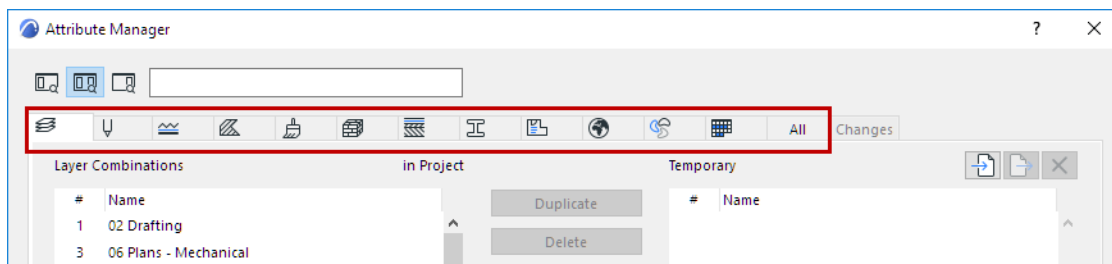
[Attribute Management Commands](#)

[Save Attribute List](#)

[Revert Changes in Attribute Manager](#)

View Attribute Lists by Type

Attribute Manager displays each attribute type on a separate tab page.

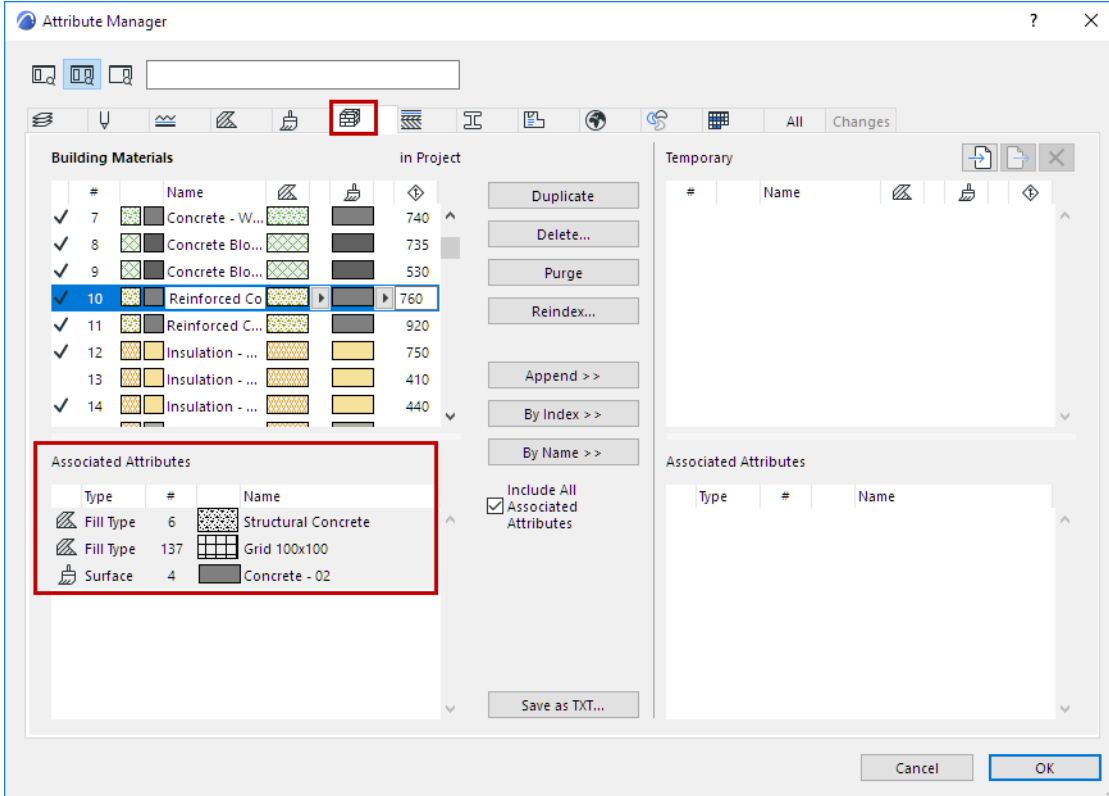


Click an icon at the top of the dialog box to switch to the corresponding attribute list.

- To sort an attribute list by a particular criterion, click the top of the desired column. For example, you can display pens by line width.
- The **All** tab combines all the project attributes into one list
- The **Changes** tab lists all pending changes to the current Project's attributes, as long as you have not yet closed Attribute Manager. ([See Revert Changes in Attribute Manager.](#))
- Clicking **OK** (in any tab of Attribute Manager) brings up a warning to confirm and apply the changes.

Associated Attributes

For each attribute type, the full hierarchy of associated Attributes is listed in Attribute Manager:



Associated attributes are read-only in Attribute Manager, but if you click on one, the relevant Attribute tab opens immediately.

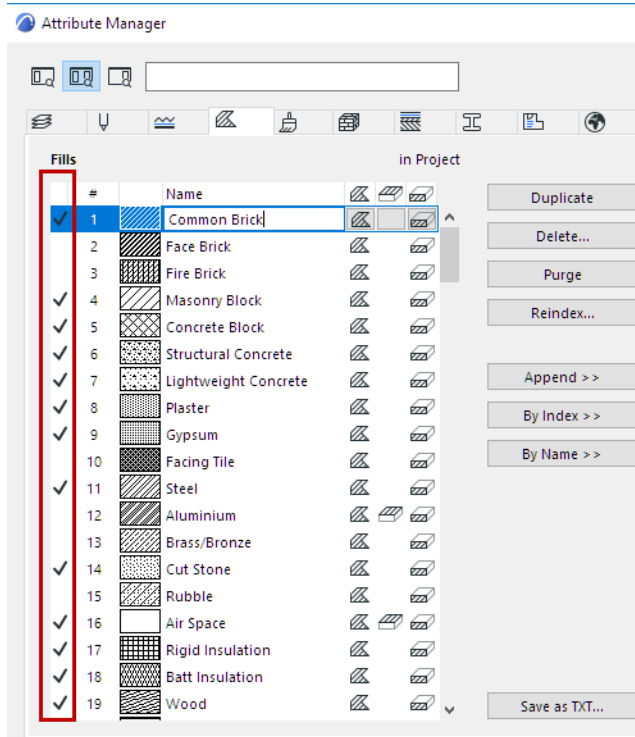
Attribute Index Number

Elements refer to attributes by their **index number**. To edit it, click **Reindex**.

Checked Attributes

A listed attribute has a checkmark if it is used in the current project:

- by placed elements
- by Favorite and Tool Default Settings
- in other Attributes (e.g. a Fill used in a Building Material)



Note, however, that an attribute shown as unused (without a checkmark) may in fact be referred by the GDL script of a placed object.

The “used with” checkmark is not available for the following Attributes: Layer Combinations, Pen Sets, Pens. [See also Purge \(Project Attributes only\).](#)

Temporary List in Attribute Manager (Right Side)

The right side of Attribute Manager is the “Temporary” list of attributes. Use this interface (on the right side) to transfer Attributes to and from the current project’s attribute set (on the left side).

You can edit and manage attributes in the Temporary list similarly to those of the Project list.

[See Append/Overwrite Attributes Between Lists.](#)

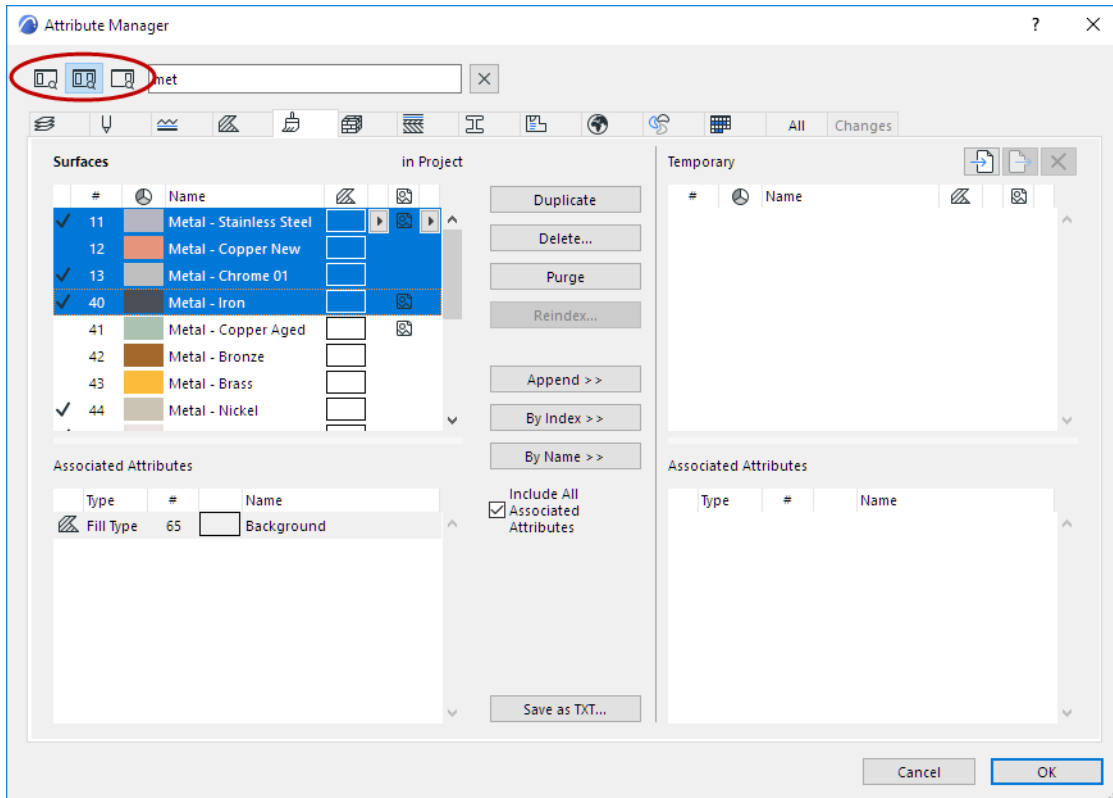
However, the Temporary list is cleared after you close Attribute Manager.

Search for Attributes

Enter any search term to locate Attributes on the current tab page, on either or both sides of Attribute Manager. This does not search among the Associated Attributes.

Note: To search for specific layers, make sure you first click in the “Layers” list to make it active. Use the icons at the left of the Search field to filter: Search Both Sides, or Right/Left Side Only.

Click the X button to clear the search.



Search for Attributes: Both Sides

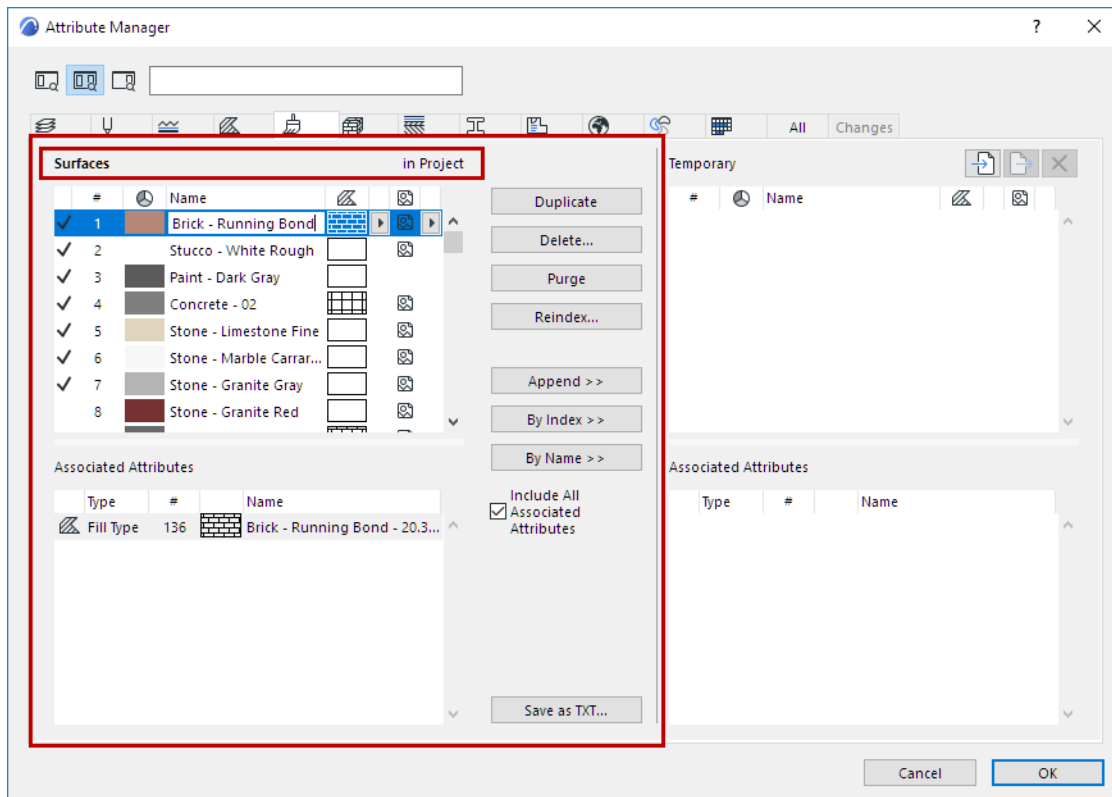
Attribute Management Commands

Use the command buttons in the middle of the dialog box to manage attributes from the lists on both sides of Attribute Manager.

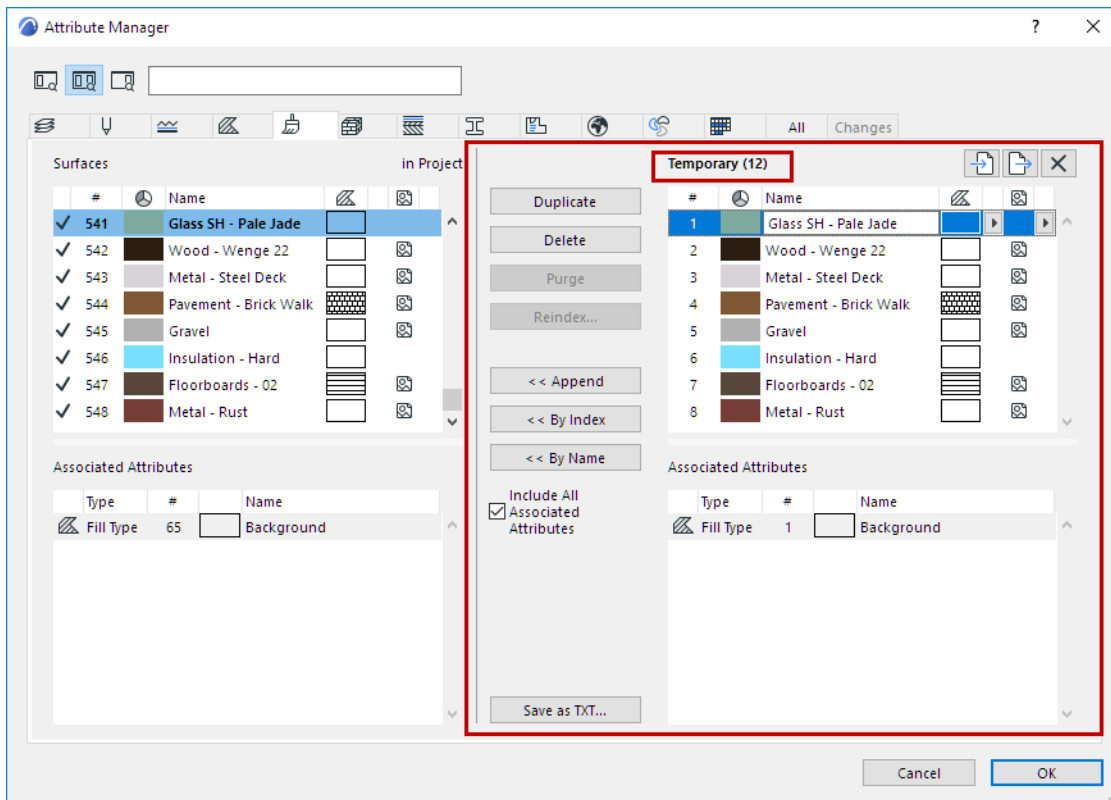
Attribute Manager commands apply to either the Project Attribute list (on the left) or the Temporary List (on the right) depending on which list is active.

Any changes you make in Attribute Manager which affect the current project's attribute set are listed on the Changes tab and can be reverted, until you close Attribute Manager.

[See Revert Changes in Attribute Manager.](#)



Commands to Manage Project Attribute List (left side)



Commands to Manage Temporary Attribute List (right side)

Append/Overwrite Attributes Between Lists

Use the following three commands to add Attributes from the Project List to the Temporary List and vice versa.

Append

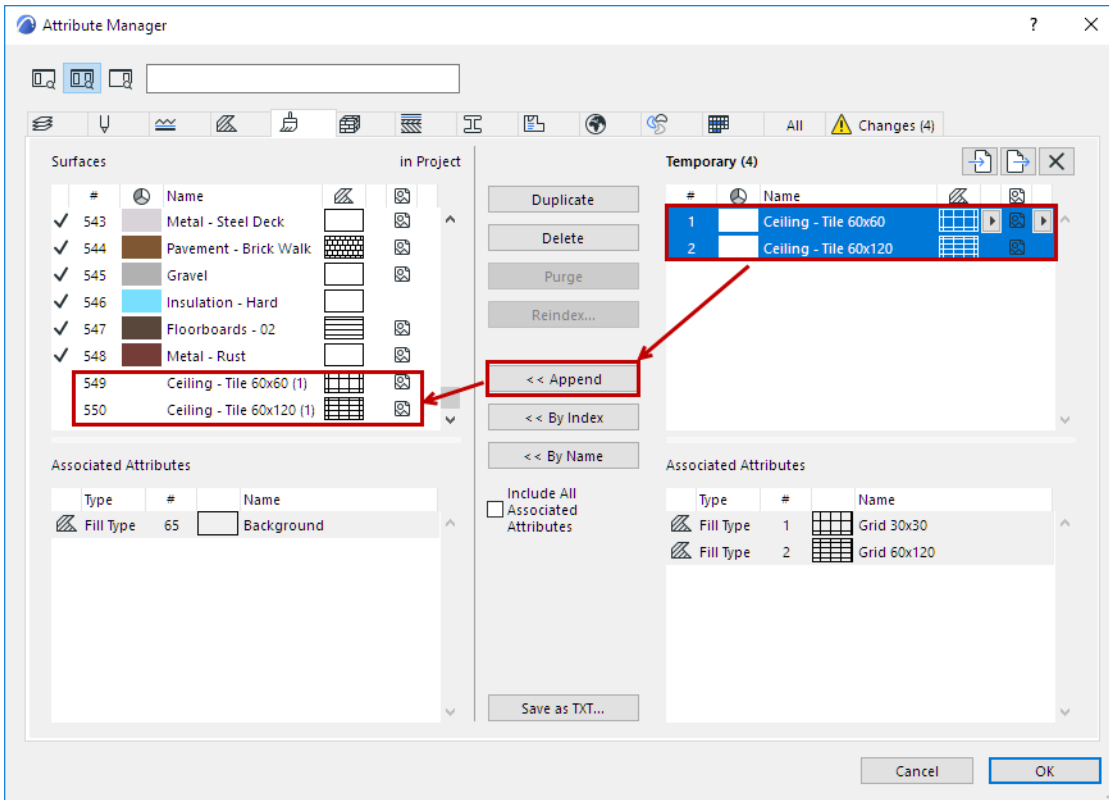
Adds the attribute from one list to the bottom of the other list, with a new index number. If an attribute of the same name already exists, a (1) will be appended to its name.

By Index

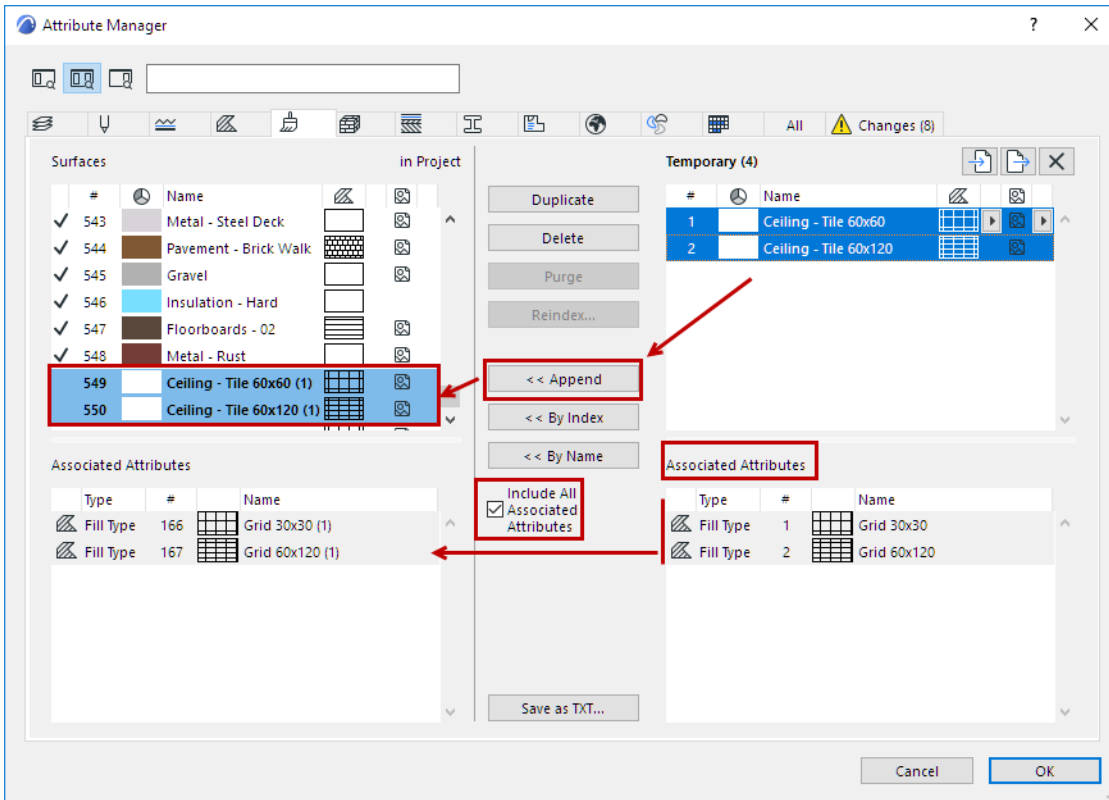
Overwrites the attribute in the other list that has the same index number. If the index doesn't exist, a new item will be created with the same index number.

By Name

Overwrites the attribute in the other list that has the same name. If the name doesn't exist, the attribute will be appended.



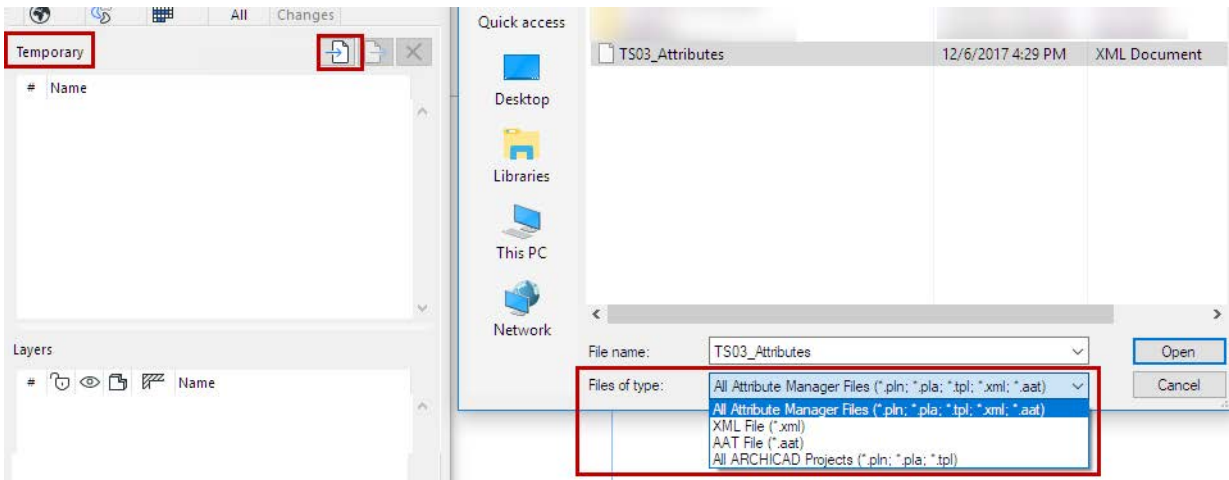
- **Include All Associated Attributes:** For each of the three transfer commands (Append/By Index/By Name), you can check this to also add its associated attributes.
- **Include All Associated Layers:** For each of the three transfer commands on the Layer Combinations page: check this to include the Layers.



Import Attributes to Temporary List

From the Temporary List side of Attribute Manager:

1. Click the **Import** button.
2. Browse for a file containing the Archicad attributes you need (e.g. XML, .aat, or Archicad project).



3. Click **Open**.

The attributes of the imported file now appear, by type, on the right side (Temporary list) of the corresponding pages of Attribute Manager.

Note: Remember, the **Import** command does not add any attributes to your current project.

Note on Import of Building Materials

When you import a file via Attribute Manager:

The incoming Building Material attributes may include Classifications and Properties that already have defined values.

But these values cannot be added to the host Archicad project unless the referenced Classifications and Properties *already exist* in the host project.

Therefore, before importing such Building Materials, review their incoming Classifications and Properties, and ensure that they already exist in your project (use Classification Manager and Property Manager).

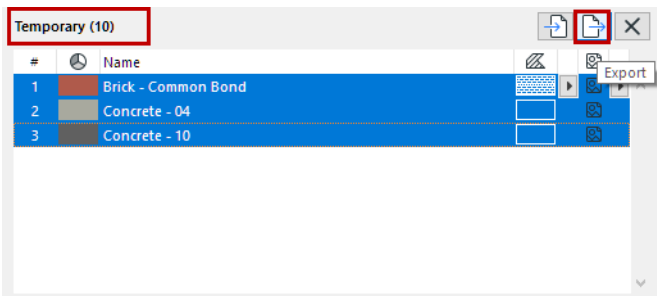
If they do not exist, you can create the missing Classifications and Properties, and then import the Building Materials. All the referenced values will come into the project.

Otherwise, the assigned values are lost.

Export Attributes to XML File

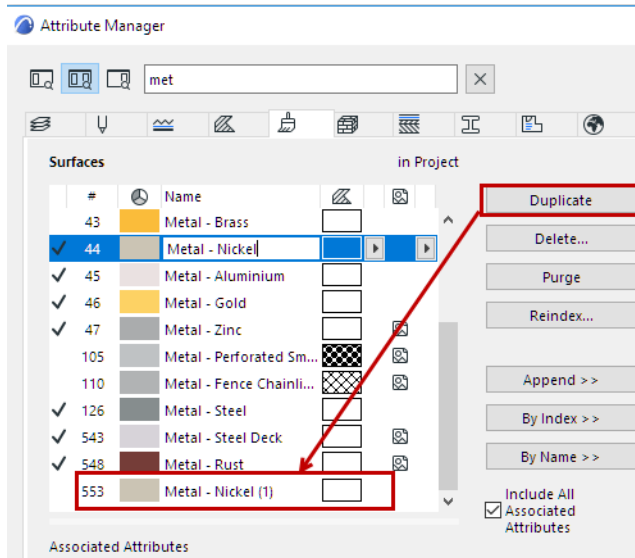
From the Temporary List side of Attribute Manager:

1. Select one or more attributes to export.
2. Click the **Export** button.



Duplicate

The Duplicate command adds the selected attribute - with a unique Index number - to the bottom of the current attribute list, where you can edit it further.



Delete/Delete and Replace

For items in the Temporary list (right side): Deletes attribute(s) from the Temporary list.

For Project attributes (left side): Deletes attribute(s) from the project.

The **Delete...** button opens the **Delete and Replace** dialog for the following project attributes:

- Layers
- Lines
- Fills
- Surfaces
- Building Materials
- Profiles
- Zone Categories

Here, you can choose a replacement attribute in one step.

[See *Delete attributes*.](#)

Notes:

- Deleting a Pen Set means that all Pens that it contains become Undefined, in all Pen Sets.
- Deleting a Pen from the Temporary list will make it undefined.

Purge (Project Attributes only)

Regardless of selection, this command removes all attributes in the list that are not currently used in your project - that is, all attributes that are listed *without* a checkmark.

The **Purge** command is grey

- if all the attributes of the current type are in use
- for attribute types that don't have checkmark feedback in Attribute Manager: Layer Combinations, Pen Sets, Pens.

Reindex

Elements refer to attributes by their **index number**. To edit it, click **Reindex**.

Save Attribute List

Click **Save as TXT** to save the currently open page of attributes as a text file.

Note: The text file can be useful for using the **Replace Attributes by File** function, which automates the process of deleting and replacing attributes. This function requires generation of a text file in a particular format, and running a command that calls this file.

[See a description of this process at *Attribute Migration article*.](#)

Revert Changes in Attribute Manager

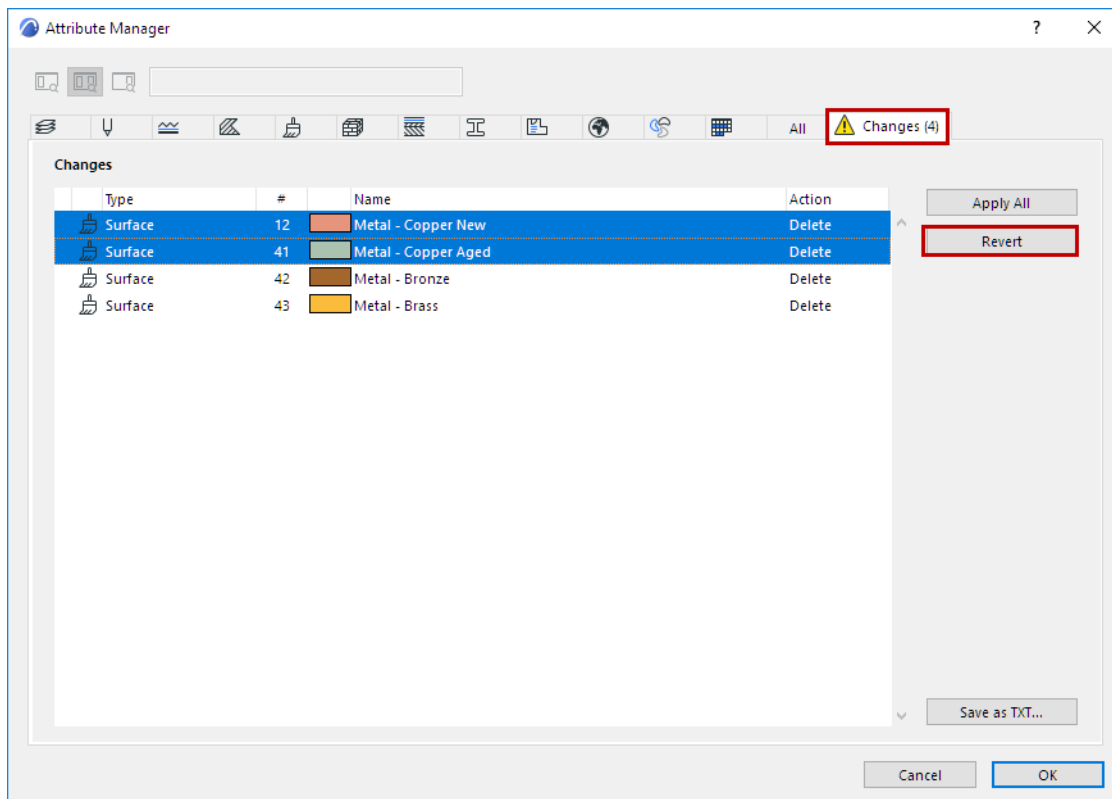
While you work in Attribute Manager, you can open the Changes tab to see all pending changes to the current Project's attributes.

This tab is only active if there are changes pending.

Clicking OK (in the Changes tab or any other tab) brings up a warning to confirm and apply the changes.

Revert

If needed, select any or all changes, then click **Revert** to undo, as long as you have not yet closed Attribute Manager.



Apply All

This will finalize your pending changes, without closing Attribute Manager. A warning appears before changes are applied.

Interaction

This section of the Archicad Reference Guide includes information on how to navigate around the Archicad project, and how to select and edit project elements. It also describes the many special Archicad aids that make input and editing more intuitive.

[Navigation in Archicad](#)

[Origins](#)

[Selection](#)

[On-Screen Input Aids](#)

[Editing Elements](#)

[Virtual Trace: Using References to Edit and Compare Model Views and Drawings](#)

[Consolidate Lines and Fills in Drawing Windows](#)

[Check for Linear Overlaps - Experimental Feature](#)

Navigation in Archicad

Archicad gives you several ways to find and open the project view or window you need.

Navigator

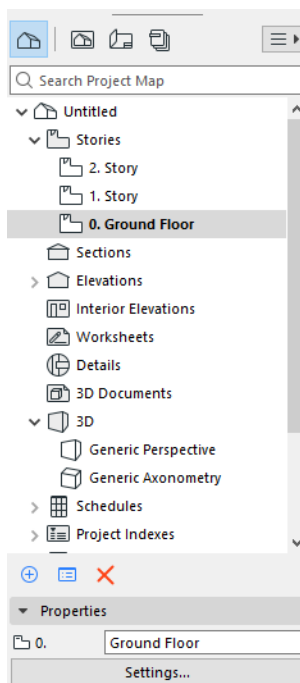
The Navigator palette contains all of your project's viewpoints, views, Layouts and Publisher sets, listed in four maps.

Find and double-click any item to open it.

Use the search field to locate items by name.

Note: When you open a new view or new layout, it can either replace the opened view/layout, or open in a separate new tab. Change this preference in **Options > Work Environment > More Options**.

Exception: You can have only one Floor Plan and one 3D window open at a time.



[See Navigator.](#)

Tab Bar

Click any tab, or click on its preview, to activate it.

[See Tab Bar.](#)

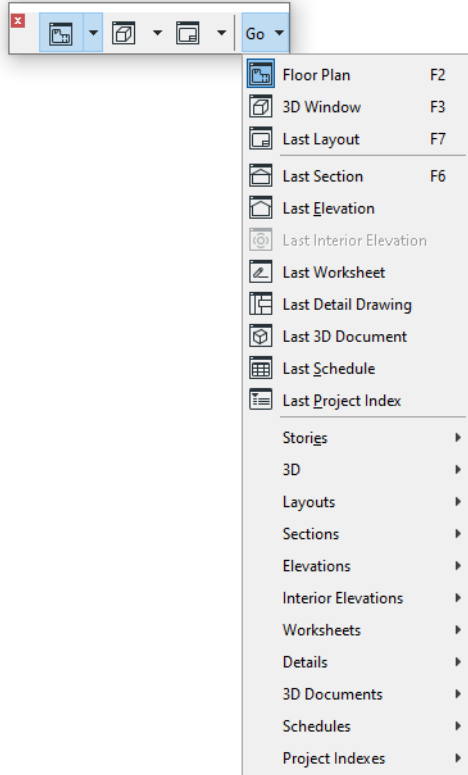


Window menu

Use Window > Navigate or other commands

Mini-Navigator toolbar

Open from Windows > Toolbars



Pop-up Navigator

A pared-down version of the Navigator palette, available in pop-up form at the right edge of the Tab bar. You can use this instead of the Navigator palette to save screen space.

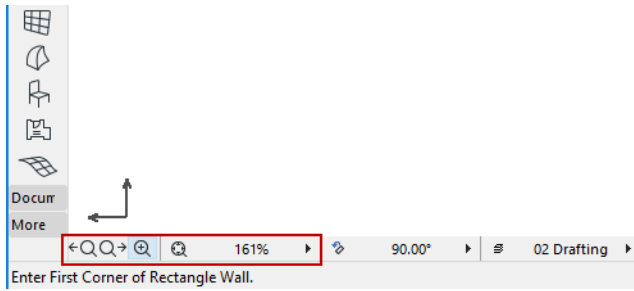
See [Pop-up Navigator](#).

Keyboard shortcuts

Shortcut	Window
F2	last Floor Plan
F3	last 3D
Shift+F3	3D (Perspective)
Ctrl+F3	3D (Axonometry)
F6	last Section
F7	last Layout

Pan and zoom

- Use commands at **View > Zoom** menu
- Use shortcut icons next to the bottom scrollbar



Related Topics

[Navigator](#)

[Organizer](#)

[Pop-up Navigator](#)

[Tab Bar](#)

[Zoom](#)

[Fit in Window](#)

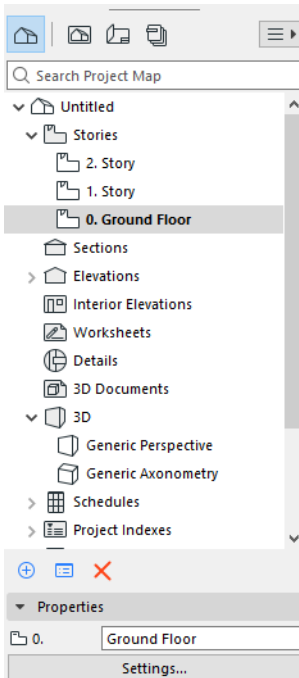
[Set Orientation](#)

[Navigator Preview \(2D\)](#)

[Navigation in the 3D Window](#)

Navigator

The Navigator palette is a tree structure containing the entire logical structure of your project. It is shown by default at the edge of your screen.



The Navigator contains all of your project's viewpoints, views, Layouts and Publisher sets, listed in four maps. Find and double-click any item in any map to open it.

Use the search field to locate items by name.

Once opened, the item is listed in bold.

Note: When you open a new view or new layout, it can either replace the opened view/out, or open in a separate new tab. Change this preference in **Options > Work Environment > More Options**.

Exception: You can have only one Floor Plan and one 3D window open at a time.

Maps of the Navigator

Project Map

Tree structure of project viewpoints, organized in folders. Viewpoint icons (in Navigator and tab bar) are black.

View Map

List of the project's saved views. A view is a stored version of a viewpoint; each view is defined by its View Settings, which you configure for a particular purpose. View icons (in Navigator and tab bar) are blue.

Layout Book

All the project's Layouts with their Drawings, optionally organized into subsets, and formatted based on Master Layouts.

Publisher

Publisher map items are views and Layouts collected in Publisher Sets. Set output methods and format here.

Organizer

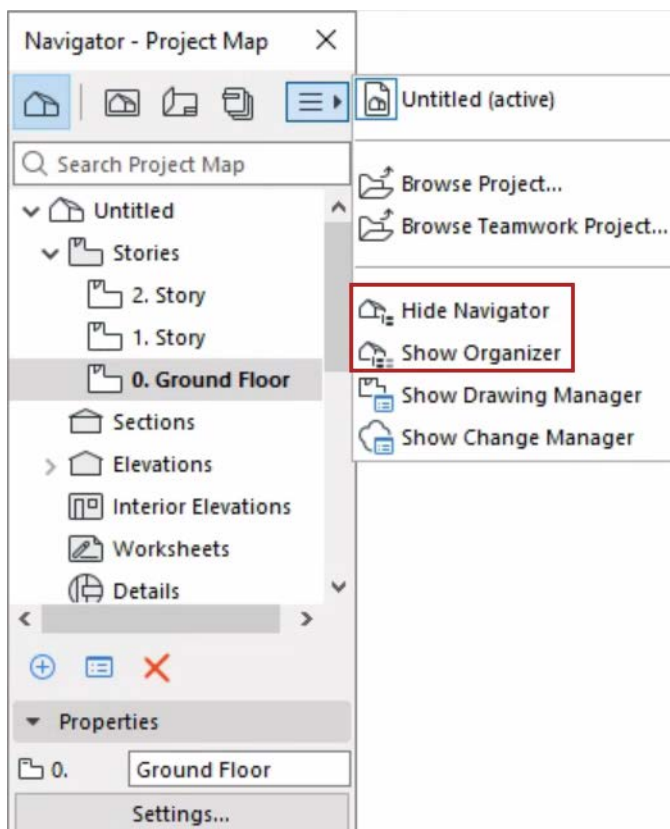
The Organizer is a special mode of the Navigator. The Organizer, with two tree structures side by side, makes it easier to move items from one map to the other.

Important: Operations involving Navigator/Organizer items (such as drag and drop between Navigator maps, deleting items from the Project Map or View Map, or adding items to the Publisher set) are not added to the undo queue, and are *not* undoable.

Show/Hide Navigator or Organizer

Do one of the following:

- **Window > Palettes > Navigator**
- **Window > Palettes > Organizer**
- Context menu commands from the palettes



Project Name in Navigator

The project gets a name once you save the project file.

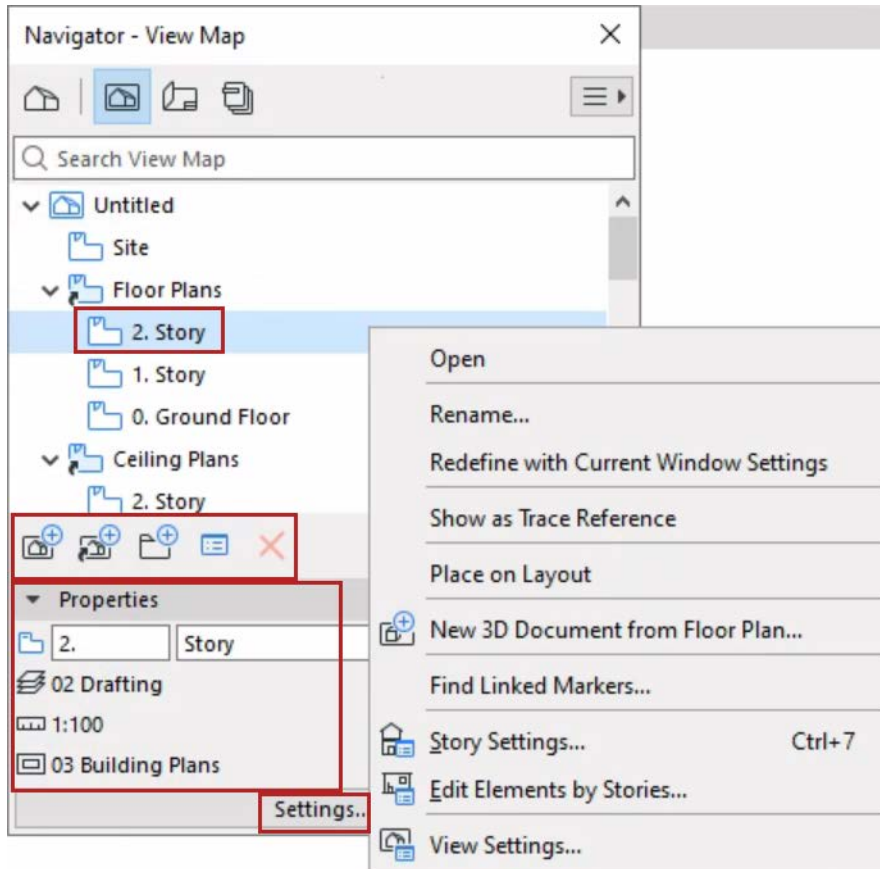
If you enter a different Project Name in **File > Info > Project Info**, the Navigator will display that name instead.

Commands in Navigator

For any selected Navigator item:

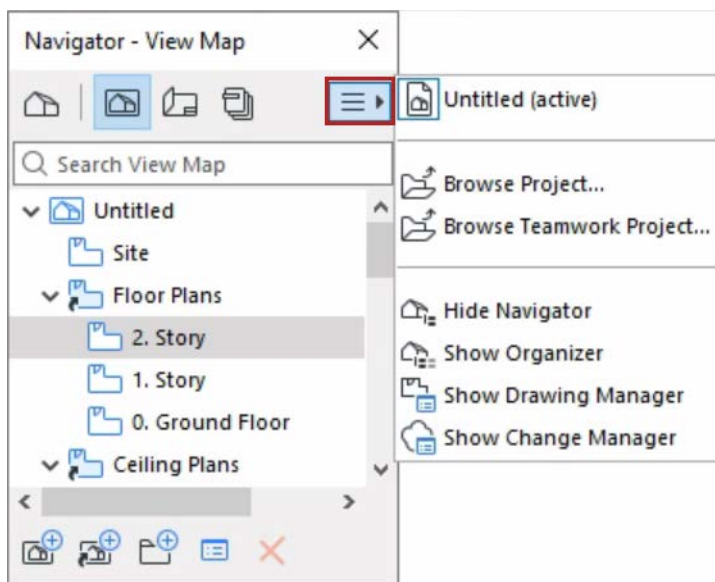
- Use context menu commands
- Click **Settings button**
- Use the command **buttons** at the bottom of the map

- View/edit a selected item's **Properties**



Show/Hide Related Dialogs

Use the Chooser pop-up to show or hide the Navigator palette and related dialogs, or browse another project:



See also [Place view from external Archicad project \(Navigator Project Chooser\)](#)

Related Topics

[Navigator Project Map](#)

Navigator View Map

Navigator - Layout Book

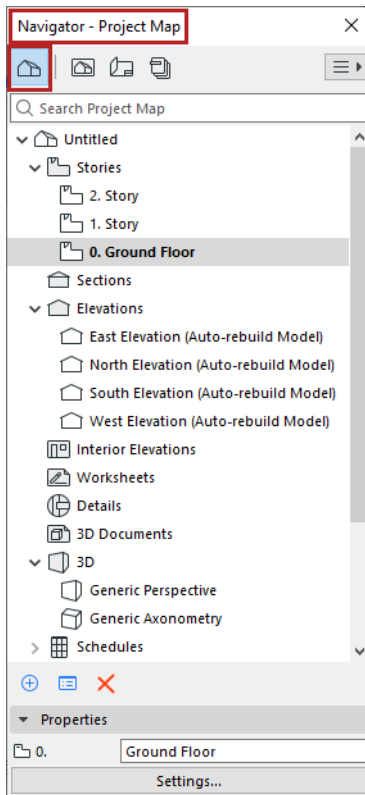
Pop-up Navigator

Publisher

Organizer

Navigator Project Map

Tree structure of project viewpoints, organized in folders. Viewpoint icons (in Navigator and tab bar) are black. Use the search field to locate items by name.



What is a viewpoint?

A viewpoint represents a window onto a particular part of your project.

Viewpoints are listed in the Navigator Project Map, organized into folders (one folder named for each type of viewpoint).

Note: Once you save a viewpoint's settings, a view is created (and added to the View Map) and can then be placed onto a Layout.

Folders of the Project Map

These folders are a fixed list of item types; you cannot add or delete folders here.

Stories

All the Stories defined for the project.

Note: In the Project Map, the Story ID is not editable. A Story ID is identical to its Story Number, which can be set in Story Settings.

Sections, Elevations, Interior Elevations, Worksheets, Details and 3D Documents

Folders containing corresponding viewpoints.

See [Archicad Model Views](#) for information about each of these viewpoint types.

3D

Contains 3D projections and any camera paths.

- Two default projections are provided: Generic Perspective and Generic Axonometry.
- When you place cameras in the project, their names appear in the 3D folder, grouped under the path.

[See *Camera Tool Settings*.](#)

Schedules

Contains the interactive schedules (Elements, Components, Surfaces) created as part of the project documentation. Use schedules as a basis for ordering construction elements from manufacturers and for checking the consistency of elements based on a set of criteria.

[See *Interactive Schedule*.](#)

Project Indexes

For example, View list indexes, Layout sheet, Drawing list.

[See *Project Indexes*.](#)

Lists

Predefined list schemes for Elements, Components and Zones

Info

Project Notes and Report (also available from the Window menu)

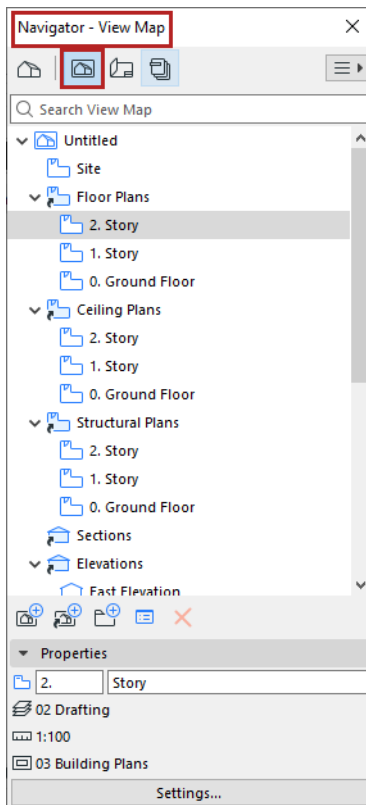
[For more information, see *Project Notes and Session Report Window*.](#)

Navigator View Map

List of the project's saved views.

View icons (in Navigator and tab bar) are blue.

Use the search field to locate items by name.



What is a view?

A view is a stored version of a viewpoint. Each view is defined by its View Settings, which you configure for a particular purpose.

View Settings

View Settings are the following:

- Layer Combinations
- Scale
- Partial Structure Display
- Pen Set
- Model View Options Combination
- Graphic Override
- Renovation Filter
- Design Option Combination
- Structural Analytical Model settings
- Floor Plan Cut Plane
- Dimensions

- Zoom and Rotation
- Image Settings (for 3D Window views): these include 3D projection settings (including zooming), filtered elements in 3D, 3D Style, 3D cutting planes, 3D Cutaway and PhotoRendering Settings.
- Existing selection (3D only)

See [View Settings Dialog Box](#).

Set up a View

To set up a new view, open a project window (Floor Plan, Section/Elevation/IE, 3D Document, Detail, Worksheet, 3D, Interactive Schedule, or List) and adjust any or all of its settings as needed.

It is convenient to use the Quick Options controls of the bottom scrollbar.

See [Quick Options Bar](#).

Even if the project content changes, the settings of your saved views will remain unchanged unless you expressly modify them.

Note: You can create a dynamic connection between project viewpoints and views if you clone a folder.

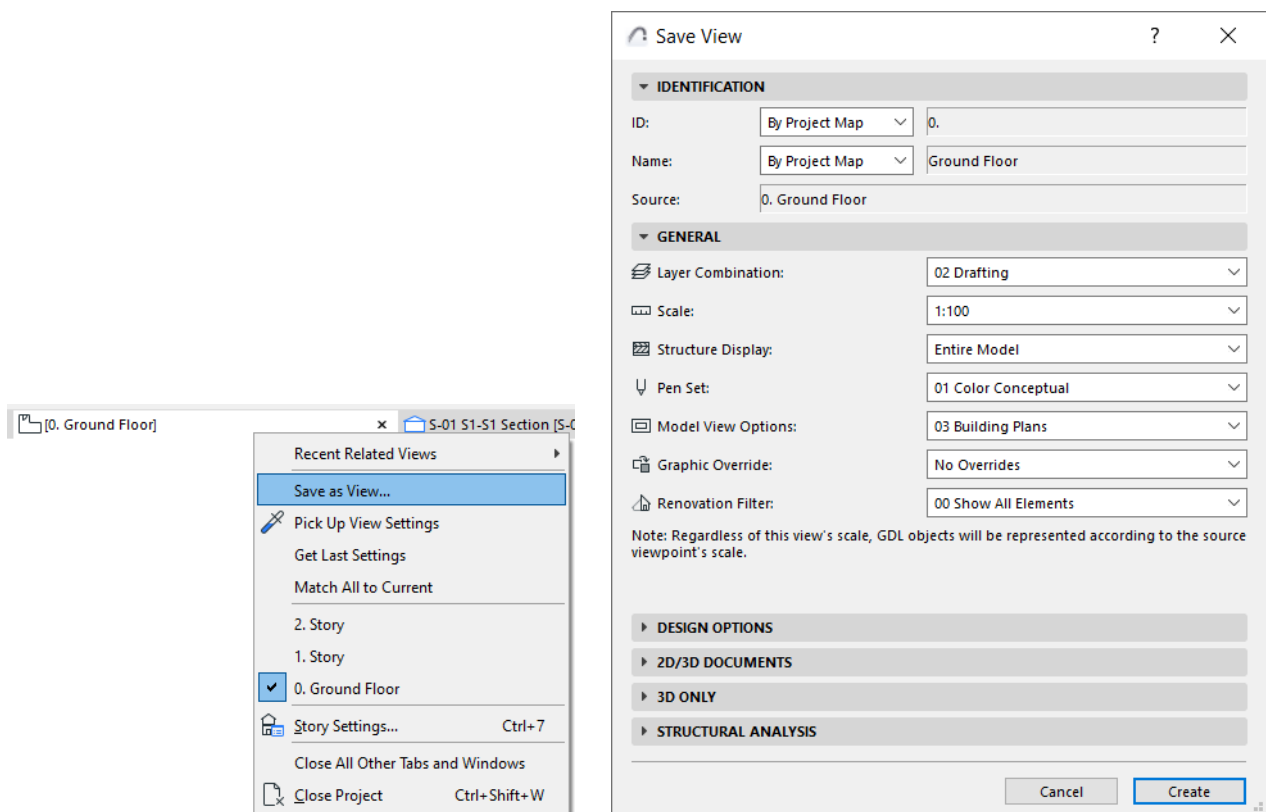
For more information, see [Clone a Folder in the View Map](#).

Save View

To save a view, do one of the following:

- Click **Save as View** from the tab's context menu
- Click **Save Current View** from the viewpoint's context menu in the Pop-up Navigator

The View Settings Dialog Box appears, allowing you to modify the settings before saving the view. Click **Create** to save the view.



Notice that the tab bar icon (blue) now indicates that the tab is a saved view.



- Use the **Save View and Place on Layout** command

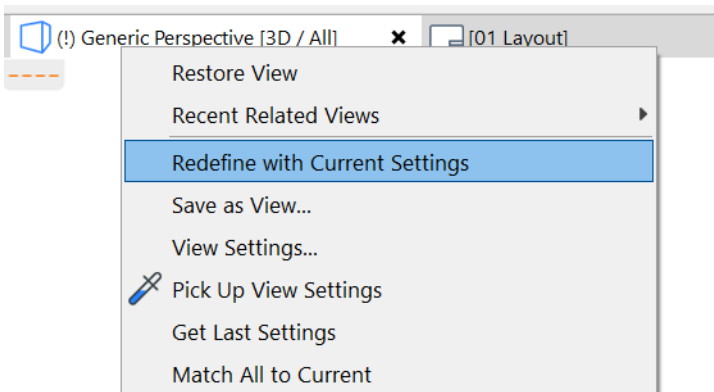
See [Save View and Place on Layout \(command\)](#).

Redefine View with Current Settings

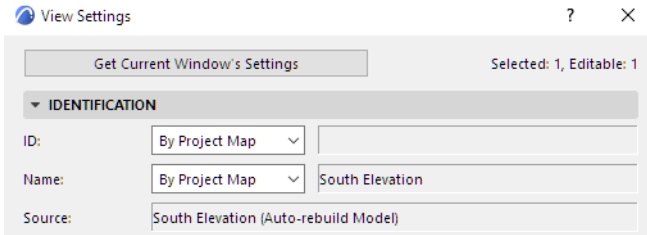
Use this if the settings of the currently opened view have changed compared to the saved settings, and you want to overwrite the saved settings.

Do one of the following:

- Use the Redefine command from the view tab's context menu



- In the View Settings Dialog Box, click **Get Current Window's Settings**.

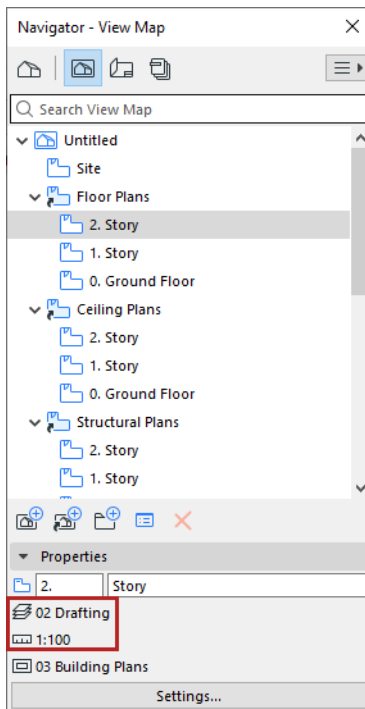


See also [Redefine Image Settings \(3D View\)](#).

Restore View in Navigator

If the current view that you are editing no longer matches its saved View Settings, a yellow “warning” icon will appear at the bottom of the Navigator, next to the changed settings, alerting you that the currently active view's settings no longer correspond to the original view settings.

For example, the image below indicates that the current window or tab's scale and layer settings are different than in the saved view.



To restore the view's saved settings, double-click the view in the Navigator.

Related Topics

[Restore View Settings from View Tab](#)

[Transfer Settings Among Tabs](#)

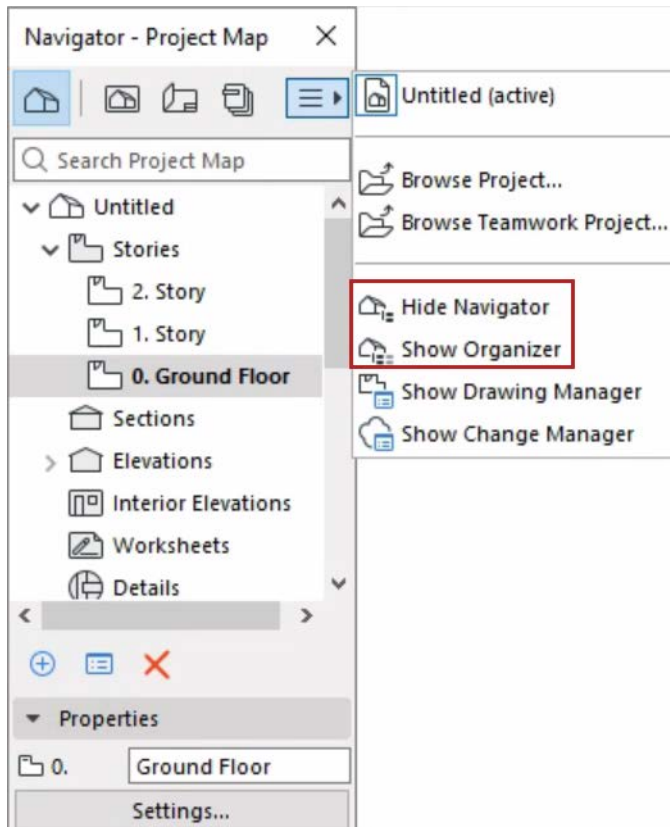
Organizer

The **Organizer** is related to and opened from the Navigator. It houses essentially the same controls, but has a double-tree structure to make it easier to move and copy views and layouts from one map to the other.

Display the Organizer

Show/Hide the Organizer using one of the following:

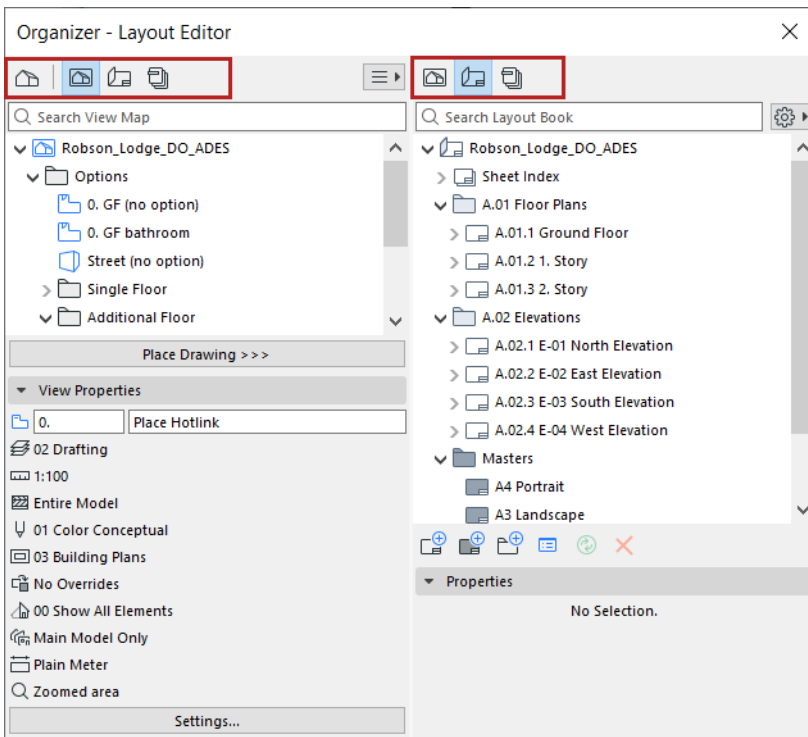
- **Window > Palettes > Organizer**
- Context menu commands from the Navigator or Organizer



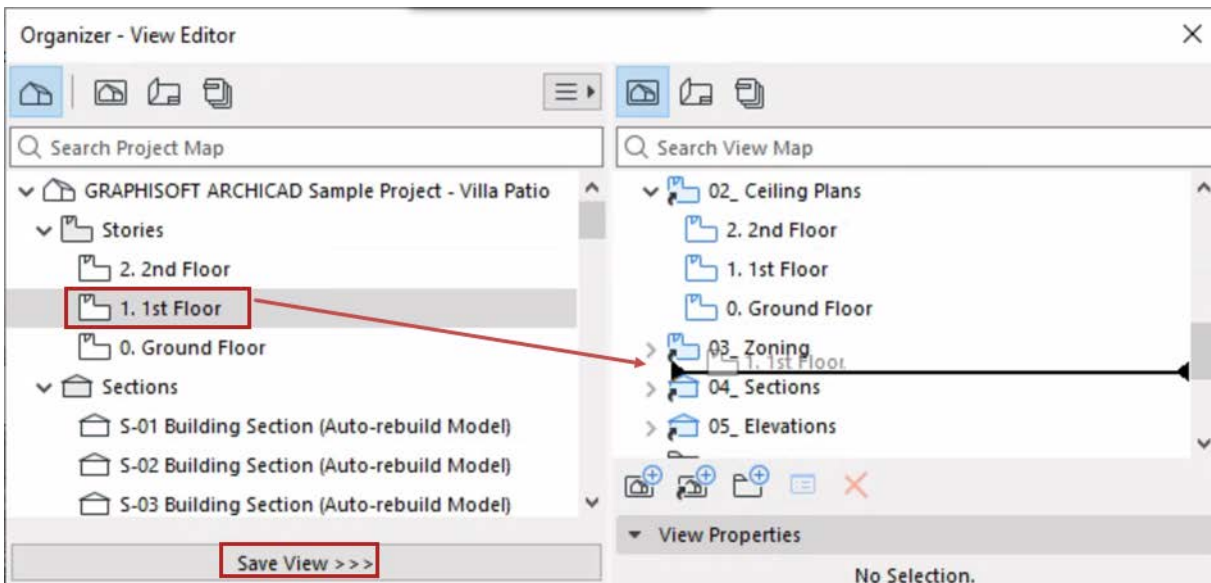
Organizer Modes

Use various modes of the Organizer to add or copy an item (view or layout) from the left side to the right-side tree structure (View Map, Layout Book or Publishing Set.)

Click the desired map icons on both sides of the Organizer to bring up one of the three modes of the Organizer:

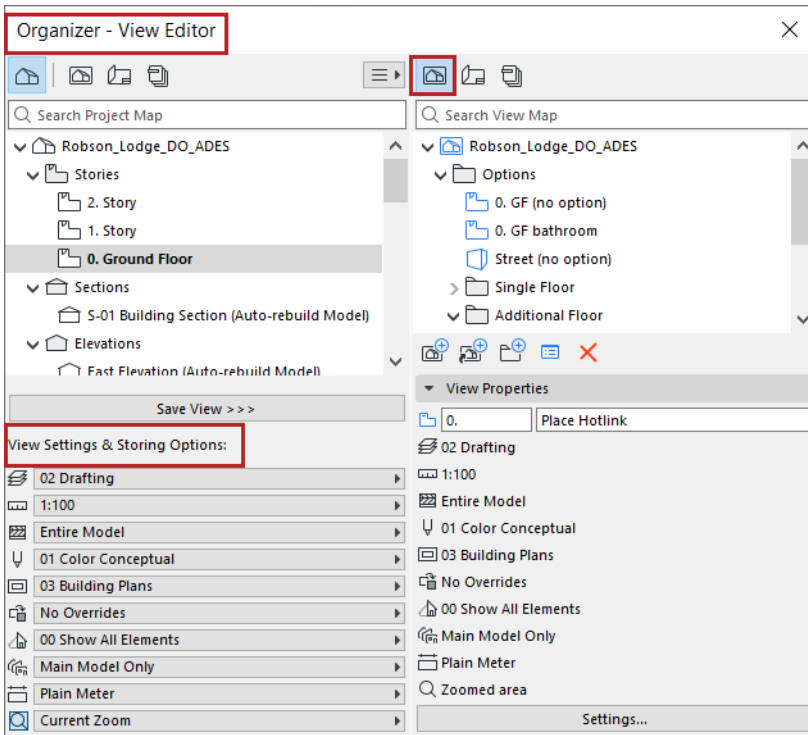


- **View Editor** mode (Project Map on the left, View Map on the right): Save views by selecting a viewpoint from the Project Map and clicking **Save view**, or drag and drop it into the View Map.

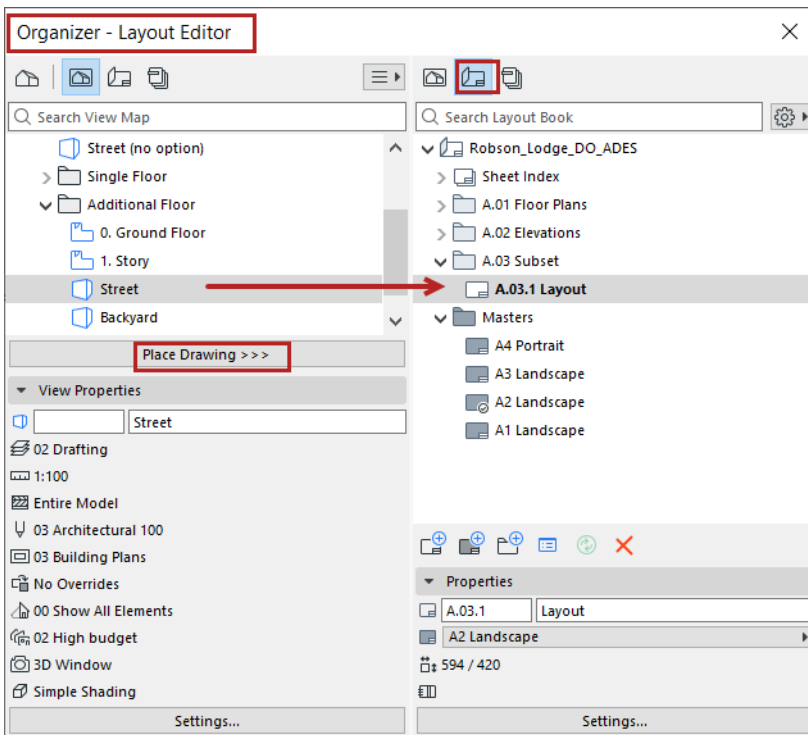


Organizer's View Editor Mode: Drag and Drop, or click Save View

In View Editor mode, you can access and edit the **View Settings and Storing Options** below the Project Map.

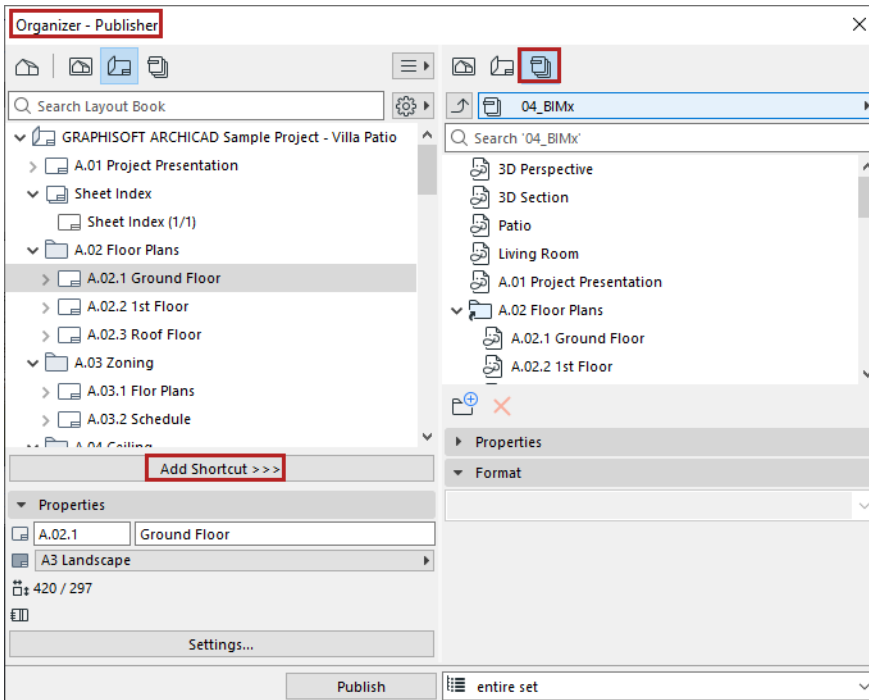


- **Layout Editor** mode (Layout Book on the right): Place Drawings onto Layouts by selecting a view from the Project or View Map (on the left), then clicking **Place Drawing**, or drag and drop it onto a Layout (on the right).



- **Publisher** (Publisher on the right): To add a view or layout (on the left) to a Publisher set (on the right), select a view or layout and click **Add Shortcut**.

If you select a Subset in the Layout Map, the **Add Shortcut** button will create a folder in the Publisher Set which corresponds to that Subset: if you change or add items in the original subset, the Publisher Set folder will reflect these changes automatically. However, if you *drag and drop* a Subset into the Publishing Set, a folder is created, but this folder is not linked to any subsequent modifications to the Subset.



See [Publisher](#) for more information.

Notes on the Organizer Modes

- The Project Map is available only on the left side.
- Certain combinations are not possible, and Archicad will always adjust to one of the three possible Organizer modes. (For example, if you choose the Publisher on the right, then the left side will switch from Project Map to View Map, because you cannot place items directly from the Project Map to the Publisher.)
- If you have the same map shown on both sides of the Organizer Palette, the **Copy Item** will duplicate the selected item on both sides (since both trees represent a single identical map).

Clone a Folder in the View Map

Cloning a folder means that you recreate a Project Map folder inside the View Map.

Consequently, for each new viewpoint you create in the Project Map, a counterpart new view will be created in its cloned folder in the View Map.

For example, if you have cloned the Project Map's Stories folder, then the cloned folder in View Map will always show all Stories of the Project, even new ones added after the creation of the clone folder.

A clone folder (unlike a regular View Map folder) has View settings which are applied to every view within that cloned folder.

Your Archicad template may include cloned folders by default.

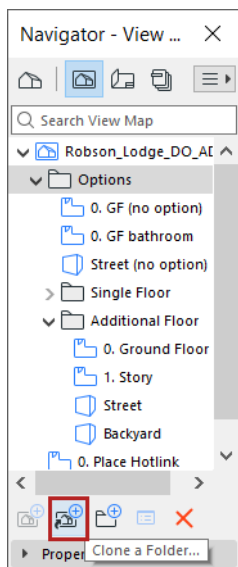
The **Clone a Folder** command is available in the Navigator Palette or Organizer's View Map (not in the Pop-up Navigator).

How to Clone a Folder (Drag and Drop)

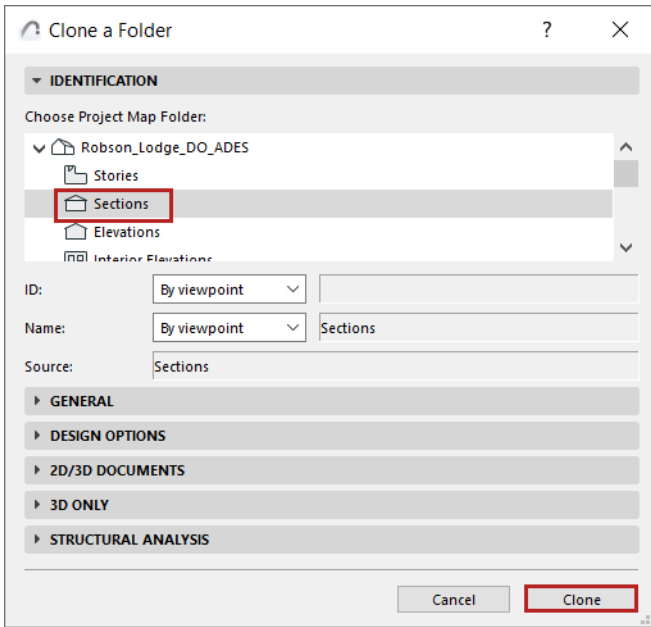
1. Open the Organizer.
2. Open View Editor mode.
3. Select a folder in the Project map.
4. Press Ctrl+Shift (Windows) or Opt+Cmd (Mac), then drag the folder from the Project Map to the View Map.

How to Clone a Folder (using Commands)

1. Click the Clone Folder at the bottom of the View Map (in the Navigator or Organizer). This opens the Clone a Folder dialog box.



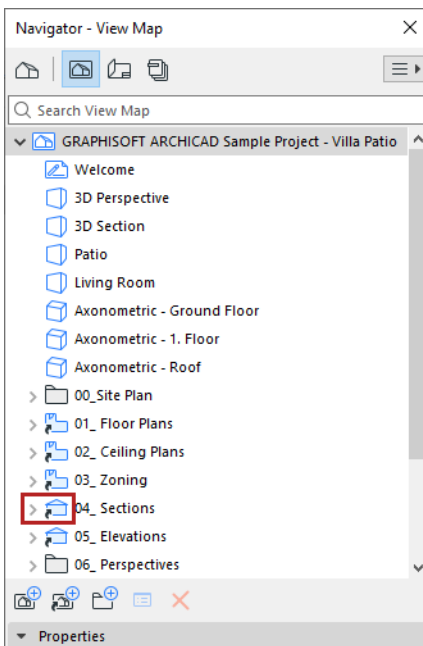
2. From the appearing **Clone a Folder** dialog: choose the folder whose clone you want to create in the View Map.



3. The rest of the controls are identical to those of View Settings. The settings you define here will be initially applied to every view in the cloned folder.

4. Click **Clone**.

The resulting cloned folder in the View Map is identified by the “clone” folder icon.



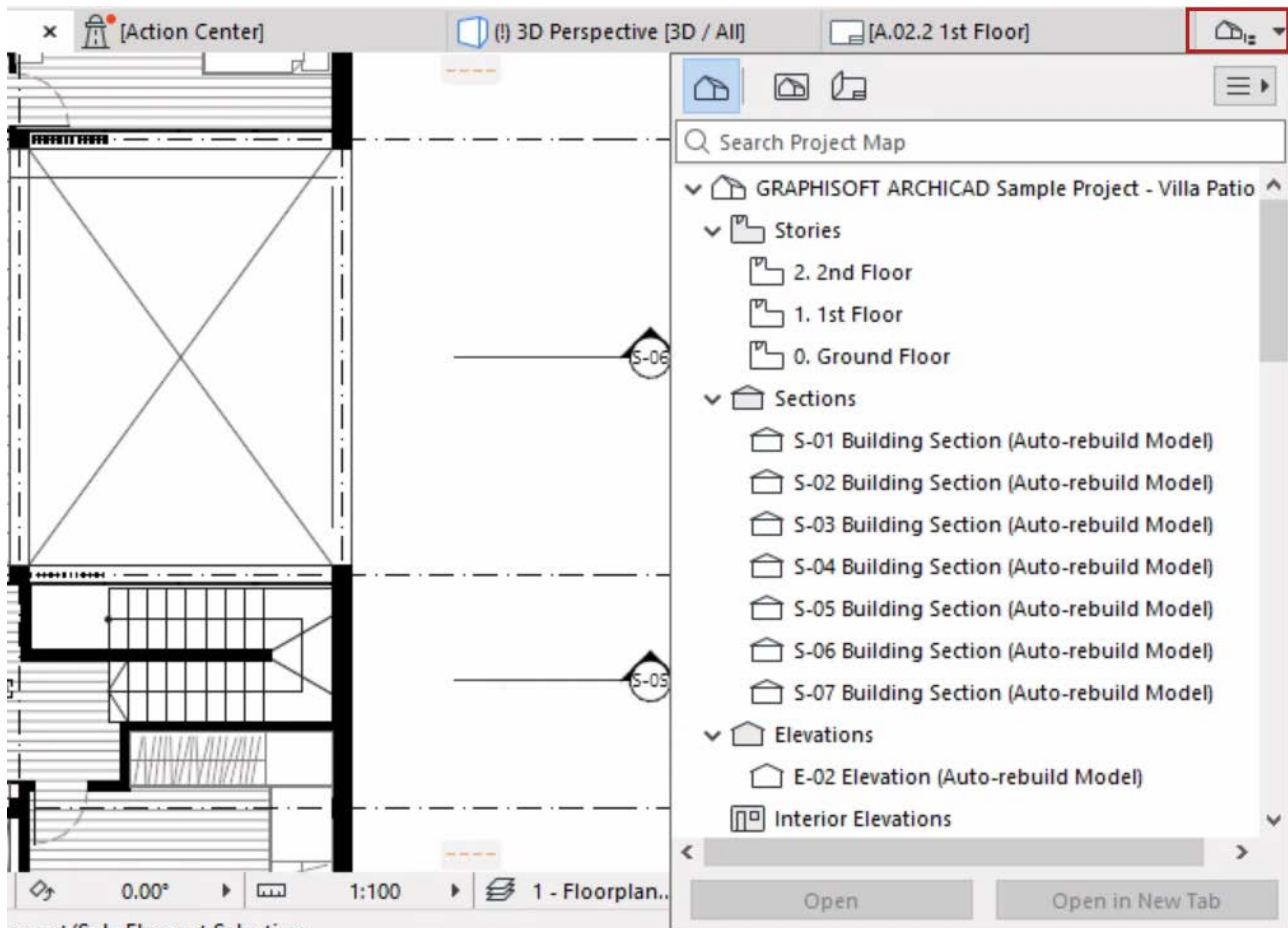
Pop-up Navigator

The pop-up Navigator is pared-down version of the Navigator Palette, available in pop-up form at the right edge of the Tab bar. You can use this instead of the Palette to save screen space.

The pop-up Navigator contains the Project Map, View Map and Layout Book (but not the Publisher).

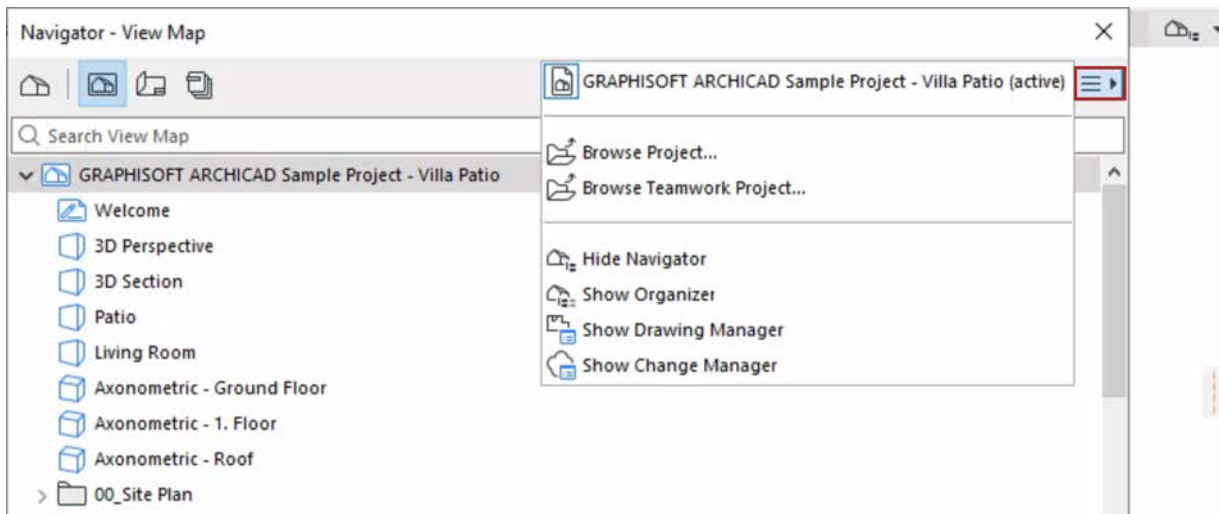
Access Pop-up Navigator

- click the icon at the right end of the Tab Bar
- or click **Ctrl/Cmd + Shift + N**



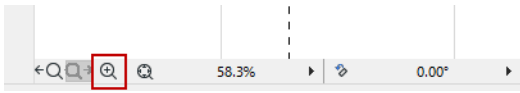
Show/Hide Related Dialogs

Use the Chooser pop-up to show or hide the Navigator palette and related dialogs:



Zoom

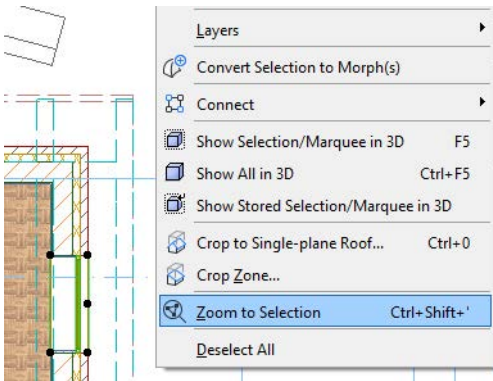
Set the window's Zoom using the controls at the bottom of the window.



When you save a view, its View Settings include the current Zoom value.

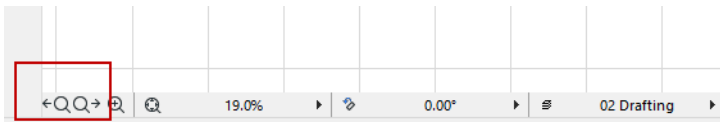
Zoom to Selection

Right-click anywhere on the screen to access this command and zoom in to the last-selected item:



Previous/Next Zoom

Click these shortcuts to move back and forth among the window's recently used Zoom values. Up to 20 Zooms are stored.



Zoom with Mouse Scroll Button

If you have a mouse with a scrolling button, use it for zooming in on (or out of) the area where the cursor is located.

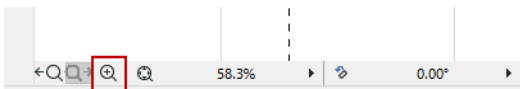
Note: In Work Environment, you can set a different preference for using the scroll-wheel (with or without the Alt button) for Zoom and Pan. [See *Input Constraints and Guides Dialog Box*](#).

Zoom with Keyboard

You can use numeric keypad shortcuts to achieve the scrolling zoom effect: press the '+' key for zooming in and the '-' key for zooming out.

Zoom Scrollbar Button

1. Click the **Zoom** button on the bottom scrollbar.



2. Draw a rectangle around the detail you want to enlarge.

Note: To double the current zoom level, double-click the Zoom button.

See also [Zoom and Rotate](#).

Zoom with Menu Commands

- Choose **View > Zoom > Zoom In** or **Zoom Out**.

Note: If these commands are not available in the View menu, you can use **Options > Work Environment > Menus** to add them.

For more information, see [Customizing Menus](#).

Home Zoom

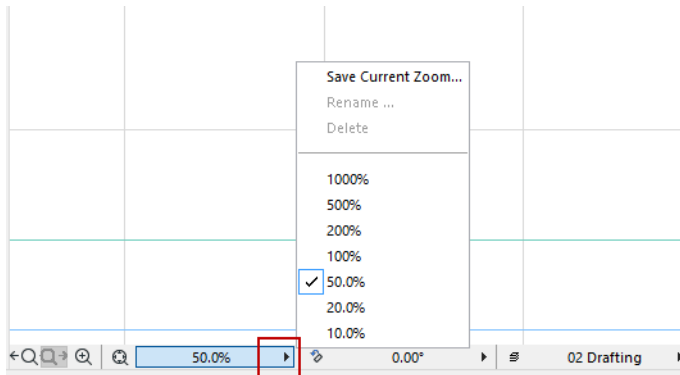
If you need to return frequently to a part of your project, you can define it as the home zoom by choosing the **View > Zoom > Set Home Zoom** command.

Return to this view with **View > Zoom > Home Zoom**.

The home zoom is saved with the project.

Predefined Zooms

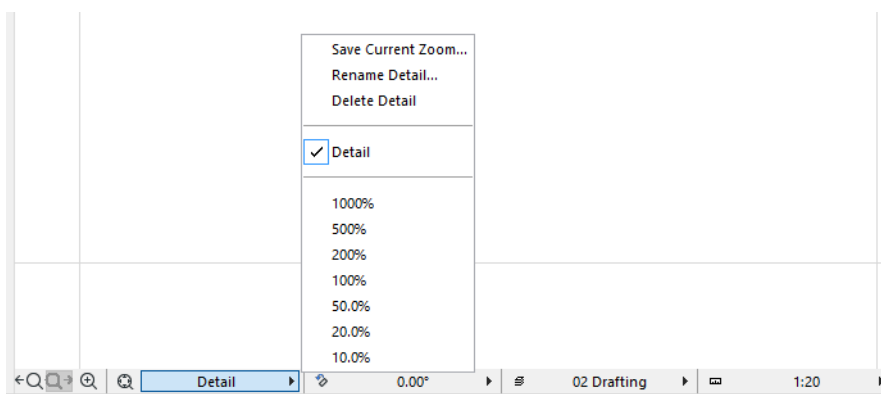
Click the Zoom pop-up to choose one of the predefined zoom values.



Use the pop-up commands to save, rename and delete predefined zooms.

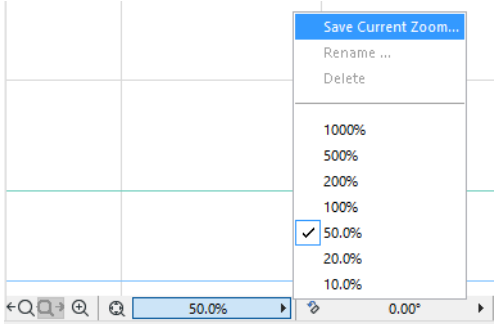
See [Save Zooms](#), below.

Home Zooms are not included in the list of saved zooms.

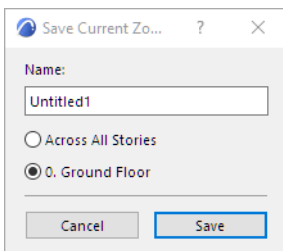


Save Zooms

Click the Zoom pop-up, then choose **Save Current Zoom**.

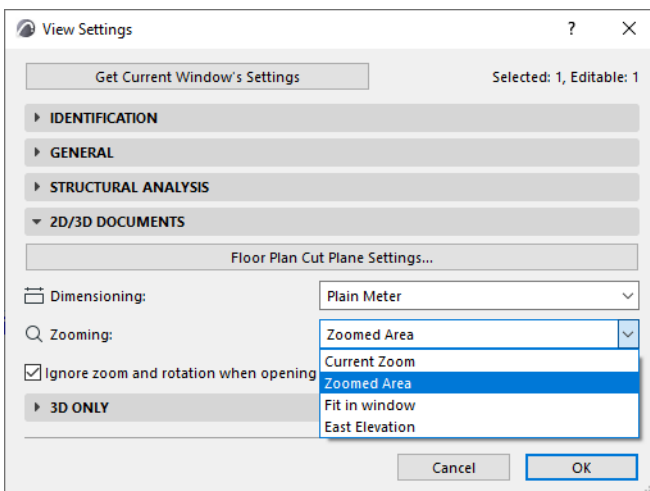


In the appearing dialog box, give the saved zoom a name.



By default, this zoom will be available only for the current story. To have this zoom available in all stories, click **Across All Stories**.

Custom-saved zooms you name here can be applied to any view, from the Zooming pop-up list in the 2D/3D Documents Panel of View Settings.



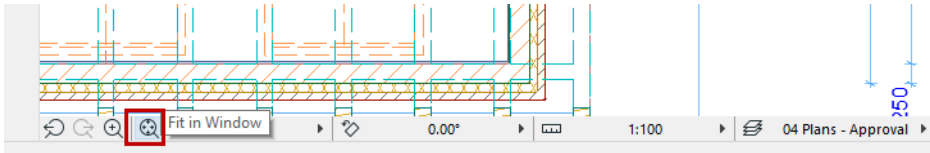
Related Topics

[Fit in Window](#)

[Set Orientation](#)

Fit in Window

Choose the **View > Fit in Window** command or click the corresponding option from the Quick Options bar at the bottom of the active window.



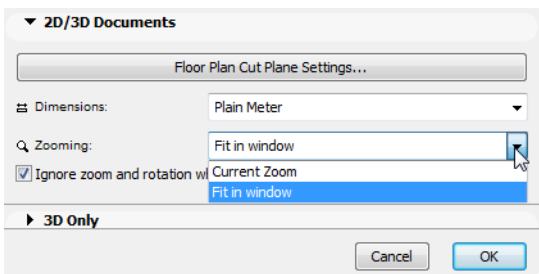
Another shortcut: Double-click the middle mouse button to Fit in Window.

If the current view has been rotated, the rotation remains intact.

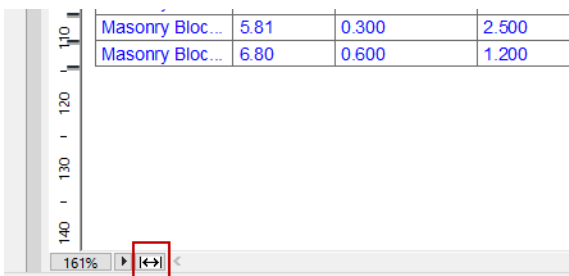
[See Set Orientation.](#)

If you wish, you can save a window's Fit in Window setting as part of any view's View Settings: choose Fit in Window from the Zooming pop-up in the 2D/3D Documents Panel of View Settings.

[See View Settings 2D/3D Documents Panel.](#)



Listing windows have an additional shortcut in their bottom scrollbar allowing you to **Fit the Width** of the generated list to the current window size.



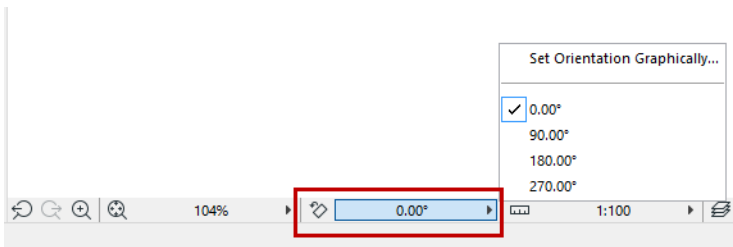
Related Topics

[Zoom](#)

[Set Orientation](#)

Set Orientation

Use this function to rotate the entire view on your screen to a preferred position.



See [Reset Orientation](#), below.

You can reorient the Floor Plan, Worksheet, Detail and 3D Document windows. The orientation value is saved as part of the “Zoom” value of every saved view, and on the published output created from that view.

Orienting the view does not rotate the project as such; the project coordinates remain unchanged. It just turns the window contents so that they are displayed at a more convenient angle on your monitor.

Note: There is a checkbox in View Settings: **Ignore zoom and rotation when opening this view**. If this is checked, then the displayed view will not use the saved zoom value, but rather the window’s last-used zoom value. Consequently, the rotation will not be seen either. Uncheck the box if you wish to see the saved zoom value in the window, together with the rotation. (Views placed on layouts will always display their saved zoom/rotation.)

Topics in this Section:

[Reset Orientation](#)

[Set Orientation Graphically](#)

[Rotate to Nearest Orthogonal Position](#)

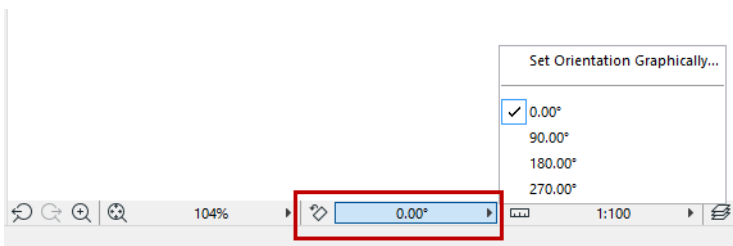
[Zoom and Rotate](#)

[Grids in Oriented View](#)

[Text in Oriented View](#)

Reset Orientation

1. Click the **Orientation** pop-up button at the bottom of your screen, or the **Set Orientation** shortcut button.

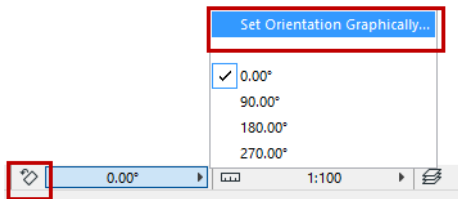


2. Choose from the list of predefined and recent orientation values.

Note: Archicad stores the last four of your custom orientation values in a Preferences file (the stored values are not project-specific, and not stored in your project file). If you set another (fifth) orientation, it will replace one of the stored ones.

Set Orientation Graphically

1. Click either the **Set Orientation** button, or **Set Orientation Graphically** pop-up command.



2. On screen, draw a rotation vector with two clicks.
3. Drag your mouse to rotate the screen contents to the desired angle, or enter a numerical rotation angle into the Tracker. Click a third time to complete.

The pair of arrows in the window's lower left corner also indicates the rotation.

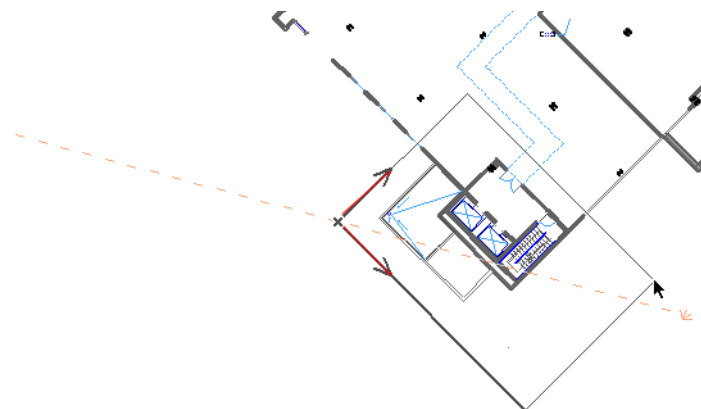
Rotate to Nearest Orthogonal Position

1. Click either the **Set Orientation** button, or **Set Orientation Graphically** pop-up command.
2. Move the cursor to an element edge.
3. **Ctrl+click** on the edge.
4. The window contents will rotate to the nearest orthogonal orientation.

Zoom and Rotate

To rotate the view and zoom in or out at the same time:

1. Click the Zoom icon from the bottom of your screen.
2. Press **Ctrl**.
3. Using three clicks, draw the zooming rectangle on screen at the desired angle.



When you complete the rectangle with the third click, the view will zoom in/out and be rotated simultaneously.

Grids in Oriented View

The on-screen Grid, if it was orthogonal to begin with, remains orthogonal after the rotation. If it was a rotated Grid, that Grid will rotate along with the view, its project coordinates remaining unchanged after the rotation.

Text in Oriented View

Text blocks

The way a Text Block behaves in a rotated view is set in Text Tool Settings in the Text Block Formatting Panel.

See [Text Block Formatting Panel](#).

Text Labels

See [Text Label Panel](#).

Dimension text

Dimension text is automatically adjusted to ensure that it will remain easily legible regardless of the view's orientation. (If you set a custom angle for dimension text, however, that angle will prevail.)

For dimensions whose text is set to "Horizontal" in Dimension Tool Settings, the text will remain horizontal with respect to the screen regardless of any rotation of the view.

See [Dimension Tool Settings](#).

Fill Area text

Will always be displayed horizontally on screen, regardless of the view's orientation.

See [Add Area Text to a Fill](#).

(If you set a custom angle for fill area text, however, that angle will prevail.)

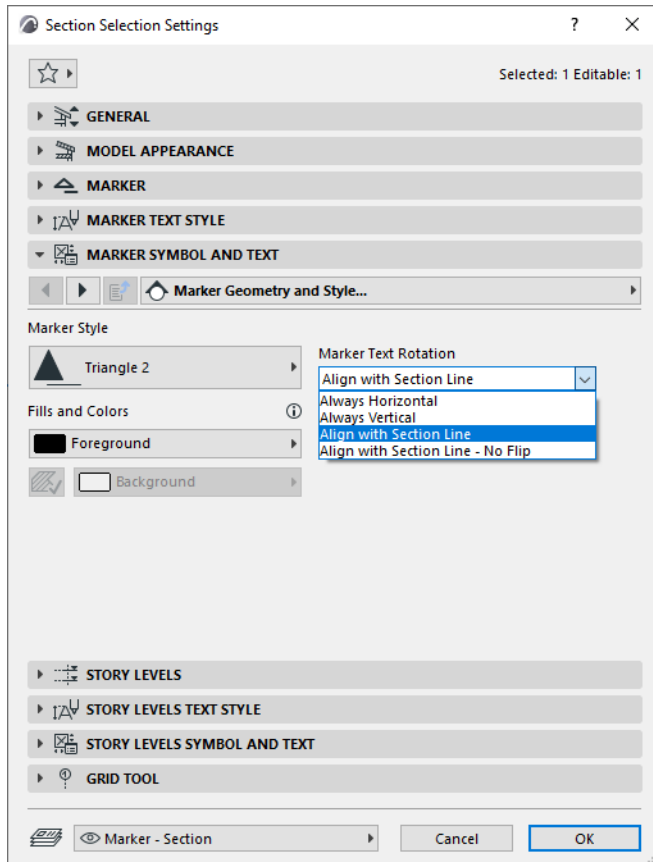
Text of GDL Objects in Rotated View

GDL Object-type elements may contain text that is affected by a rotated orientation. For example, options for **Text Orientation on Symbol** include:

- **Readable:** As the view is rotated, the object text rotates too, but is flipped as necessary to ensure text readability.
- **Align with Symbol:** Text rotates together with the symbol, with no other adjustments.
- **Always Horizontal:** Text remains fixed in a horizontal position on screen, regardless of the view's orientation.

Marker Text in Section/Elevation/Detail/Worksheet

In the Settings dialog box (Marker Panel), use the **Marker Text Rotation** parameter. This may include the following values, depending on the marker type:



- **Always Horizontal** and **Always Vertical** values will display text horizontally/vertically on screen, regardless of the view's orientation.
- **Align with Section Line** will keep the text lined up with the Section line, following any view rotation, and will flip the text if necessary to ensure readability.
- **Align with Section Line - No Flip** will keep the text lined up with the Section line, following any view rotation, without any "flip" adjustment for readability.

Text of Door/Window Markers

The text orientation parameter may include the following values, depending on the marker type:

- **Standard:** Text is always parallel to the wall, and is automatically readable (that is, flipped if necessary after rotation of the view).
- **Align with Marker Line:** Text remains fixed in line with the marker line and is automatically readable.
- **Horizontal/Vertical:** Width/Height/Sill/Fire Rating/Acoustic Rating texts are automatically readable.

Related Topics

[Zoom](#)

[Fit in Window](#)

Navigator Preview (2D)

The Navigator Preview Palette makes it easier to locate and zoom in on certain parts of your project, especially when working on larger projects.

For details on these controls, see [The 2D Navigator Preview Palette Controls](#).

To display the Navigator Preview Palette, use **Window > Palettes > Navigator Preview**.

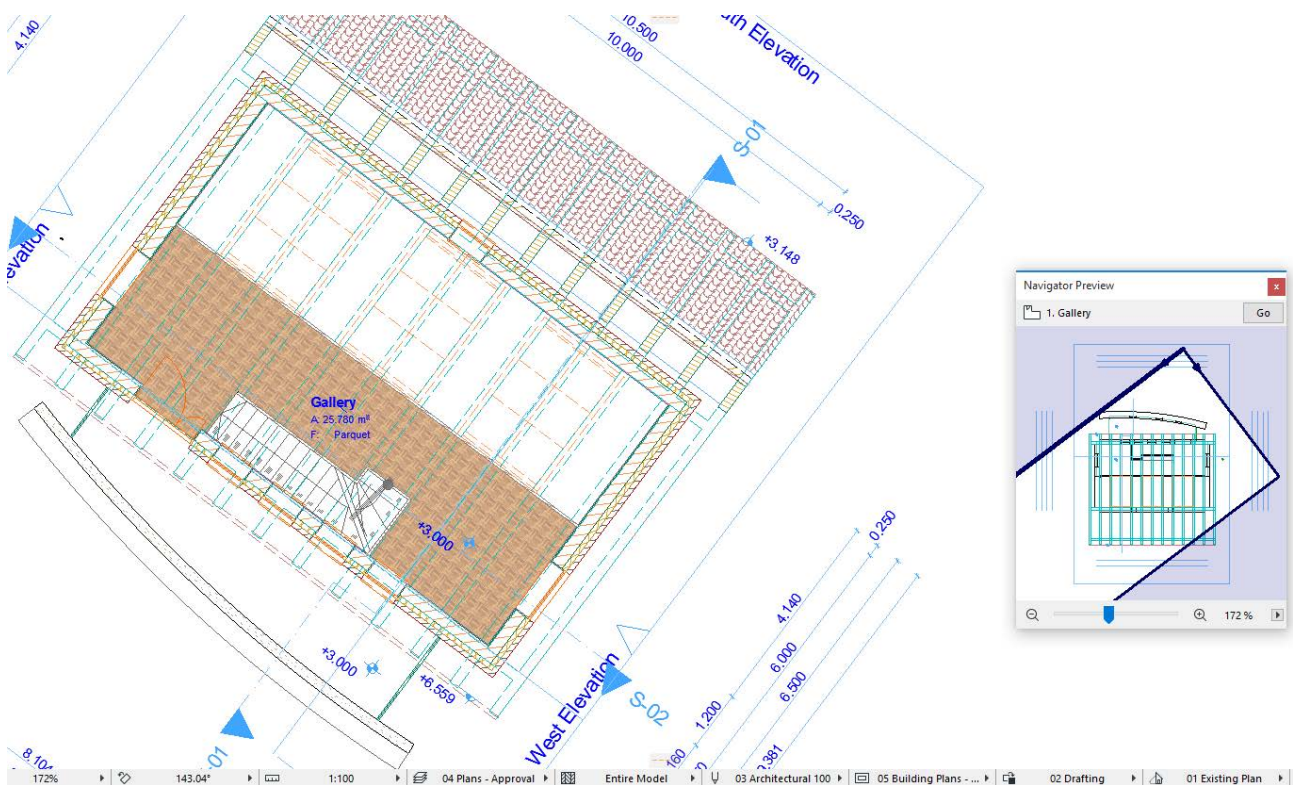


For 2D Windows, the Preview Palette displays a miniature copy of the entire contents of the current view. The frame inside the Preview Palette represents the active window at its current zoom. Use this frame to zoom and pan within the Preview Palette; this has the same effect as zooming and panning in the active window, while simultaneously giving you an overview of the entire window contents.

If the current zoom of the active window includes a rotated orientation, the preview shows the project in its original view, but the frame is rotated to match the angle of orientation.

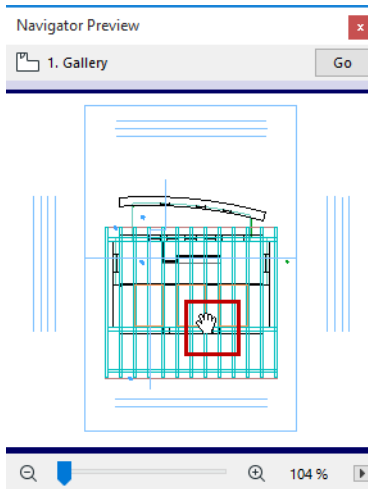
See [Set Orientation](#).

(The two heavier arrows of the frame indicate the bottom left corner of the screen.)

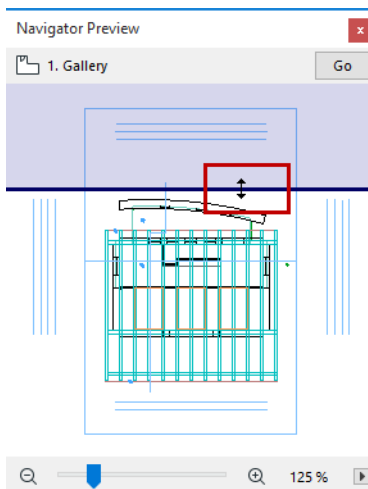


When previewing a 2D window in the Navigator Preview, you have the following zooming and panning options:

- To change the location of the frame, drag its enclosed area with the **Hand** Cursor (this lets you pan in the active window).

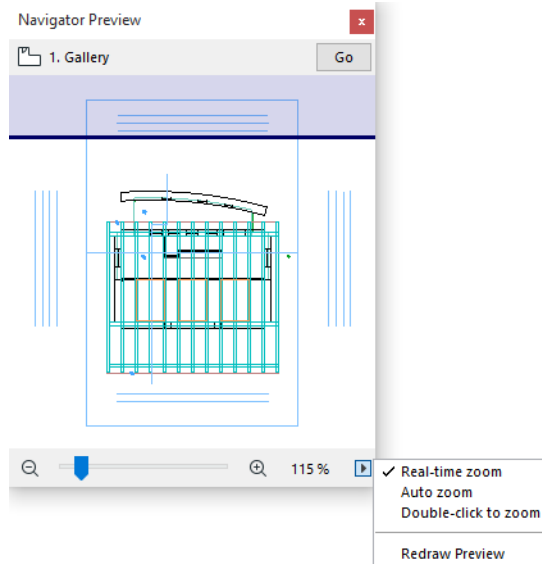


- To change the size of the frame, drag its sides or corners (this has the effect of zooming in and out within the active window).



- At the bottom of the palette, the plus and minus buttons and a sliding switch are additional zooming techniques: they allow you to zoom the current view in and out. Click the buttons or drag the slider to adjust the zoom.

- At the bottom right corner, a pop-up menu contains options for controlling the zooming techniques of the Navigator Preview. The default is **Real-time zoom**: as you move the cursor, the zoom is adjusted. **Auto zoom** will adjust the zoom after you finish moving the cursor. **Double-click to zoom** means you must double-click or press the **Go** button at top right to execute the chosen zoom.



- Use **Redraw Preview** if you have made changes in the current window that are not yet reflected in the Navigator Preview.

For information on saving this data, see [Save Navigator Preview Data with Project files](#).

Navigation in the 3D Window

Note: For general information on accessing the 3D window, and how to set up Perspective and Axonometric (Parallel) views, [see 3D Window](#).

The 3D window can be navigated using most of the basic 2D navigation methods. The keyboard, mouse scroll button, Fit in Window, Pan, Zoom, and command-based techniques all work the same way as in 2D.

See [Fit in Window](#), [Zoom](#).

The 3D window also contains unique navigation possibilities for exploring the model and orbiting around in it. These are described in the following sections.

Accessing 3D Navigation Commands

[Explore Model \(3D Navigation\)](#)

[Orbit \(3D Navigation\)](#)

[Navigator Preview \(3D\)](#)

[3D Connexion Enabler](#)

Accessing 3D Navigation Commands

The 3D Navigation commands are available only in the 3D Window.

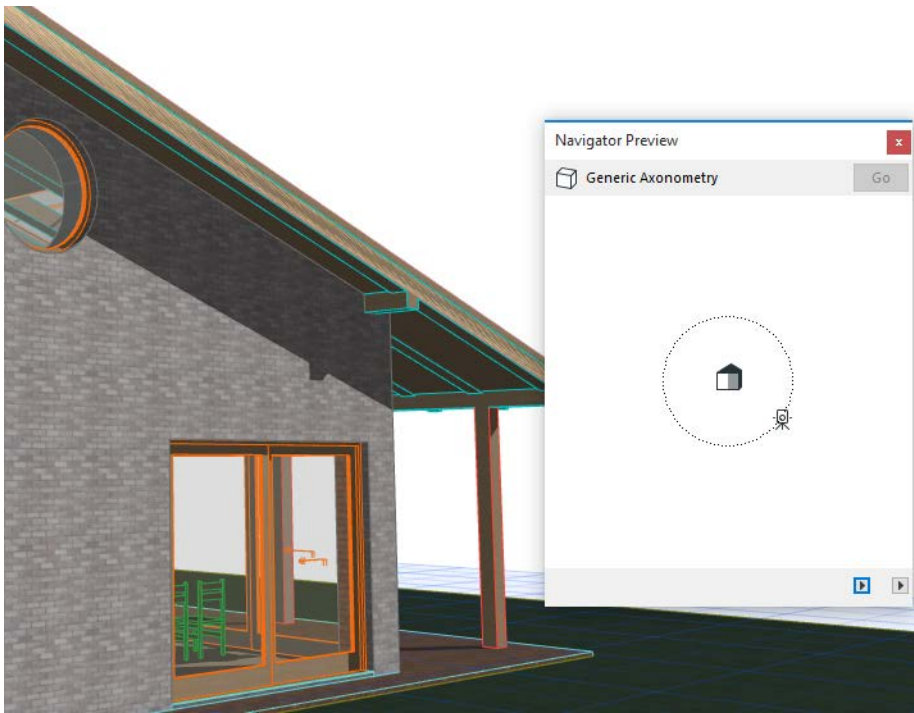
Use **Window > Toolbars > 3D Visualization**

or **Window > Toolbars > Simple 3D** to display these toolbars as needed.

If the 3D window is active, the Quick Options Bar contains shortcuts to the Orbit and Explore commands.



Note: The 3D Navigation controls from earlier versions of Archicad can be accessed by choosing **Window > Toolbars > Classic 3D Navigation** toolbar.



Related Topics:

[3D Navigation Extras](#)

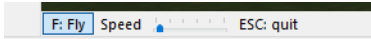
[Classic 3D Navigation Toolbar](#)

Explore Model (3D Navigation)

In Perspective views only, you can explore the model either by choosing the **View > Explore Model** or access the same command from a toolbar, or from the bottom scrollbar of the 3D window.



In Explore mode, use the mouse and the arrow keys of the keyboard for navigation (as when playing a video game on a PC.) When in this mode, you cannot edit the model or use any of the commands. The “Fly” option and speed slider also appear at the bottom of the screen as long as you are in Explore mode.

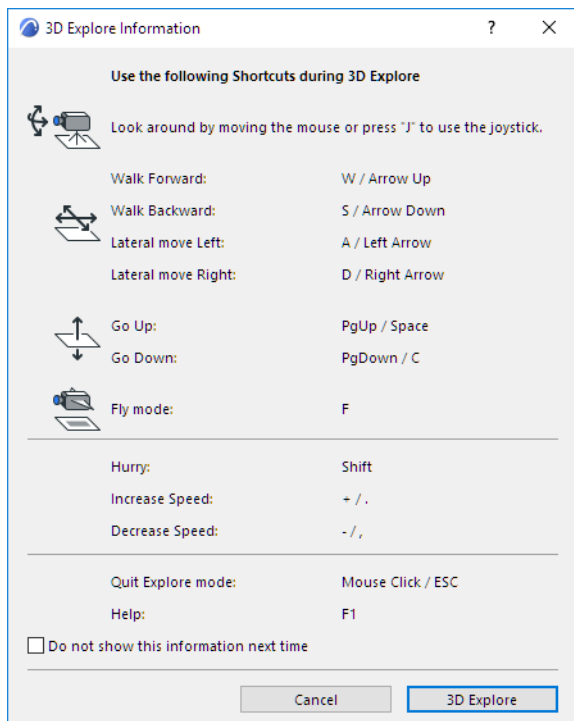


To **exit** Explore mode, click with the left mouse button or use the Esc key.

Note: Exploring the model is optimized for the Hardware acceleration engine.

When choosing the **Explore Model** command, an informational dialog box opens to give you a visual prompt on what you can expect from your various actions with the mouse and the keyboard.

Note: If you don't need to see this information every time you enter Explore mode, check **Do not show this information next time** at the bottom of the dialog box. To enable this or any other dialog box you have marked as “do not show,” use the **Enable all informational messages** command at **Options > Work Environment** (bottom of menu).



“Rotate” camera in Explore mode

Moving the mouse around will rotate the camera in the directions indicated by the arrows next to the camera on the picture above.



“Walk” forward and backward in Explore mode

Pressing the arrow keys, or the W-A-S-D keys, will walk you forward/backward and laterally in the directions indicated by the arrows in the picture above. Note that the Up arrow brings the model closer to you while the Down arrow moves it further away.



“Fly” around the model

In Explore mode, click “F” for Fly mode. Again, use the arrow keys or the W-A-S-D keys to move around, but this time the Fly mode combines the lifting effect with the rotating and lateral movements to simulate a walking movement on a horizontal plane or in space.



“Lift” the model up and down

Use the PageUp (shortcut: spacebar) and PageDown (shortcut: C) keys to simulate a lifting effect.



Speed Up/Slow Down in Explore Mode

Click the Plus and Minus keys on your numeric keypad to change the speed in which you explore the model. For a temporary speed increase, press Shift, then release it to return to your basic speed.

Orbit (3D Navigation)

For **Orbit** mode, choose **View > Orbit**, or access the same command from a toolbar or from the bottom scrollbar of the 3D window.



Once you enter Orbit mode, press the left button and drag the mouse to turn the model around its centerpoint (axonometry) or target point (perspective).

If you are in Orbit mode, you cannot edit the model. Click ESC to exit Orbit mode and return to editing mode.

Temporary “Orbit” Effects While in Editing Mode

While in editing mode (not in Orbit mode), you can temporarily simulate Orbit mode by pressing Shift together with the mouse scrolling button to orbit the model. Release the keys to stop orbiting.

This type of temporary Orbit requires the model to be redrawn and can result in slowed-down navigation.

Navigator Preview (3D)

To display the Navigator Preview Palette, click the shortcut button in the bottom scrollbar of any window; or use **Window > Palettes > Navigator Preview**.



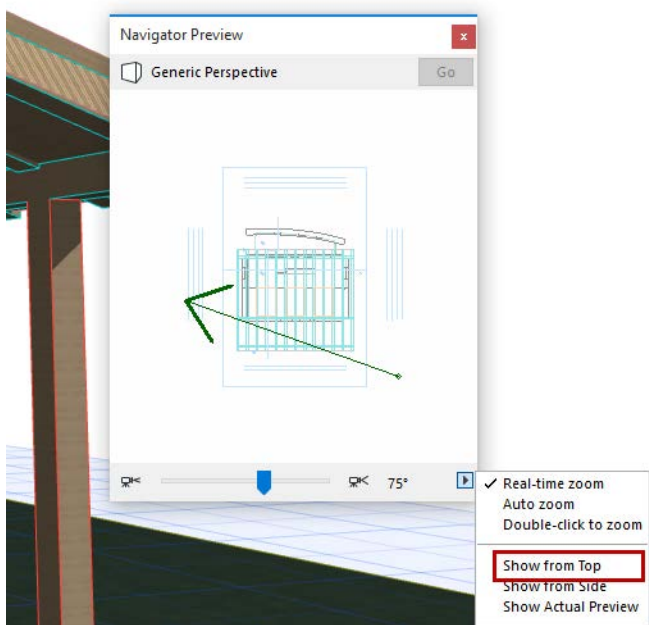
This palette allows you a quick way to modify your 3D Projections, whether perspective or parallel (axonometric).

[See 3D Projections.](#)

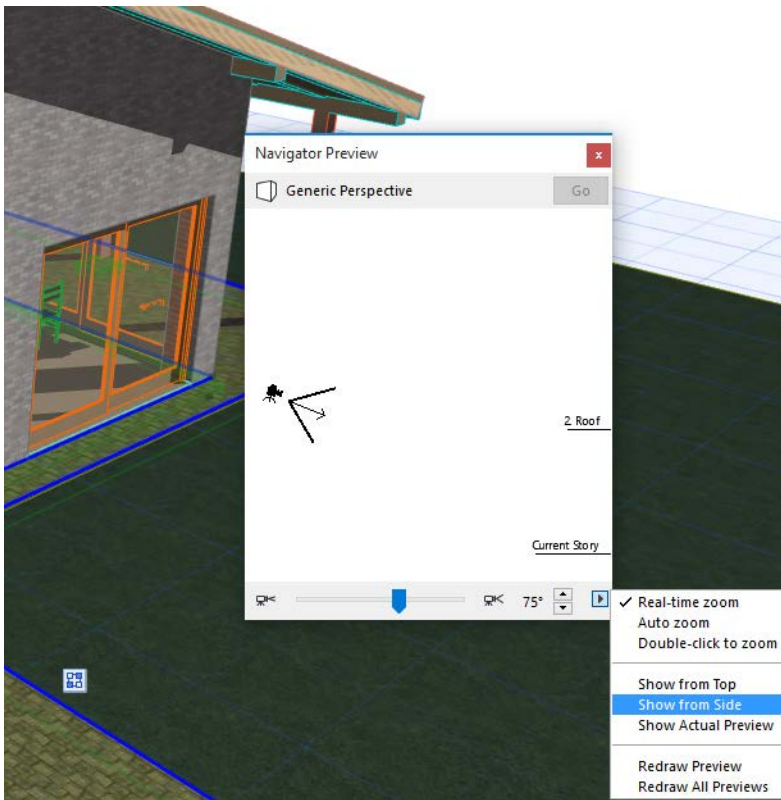
The options of the Navigator Preview depend on whether you are looking at a Perspective or Axonometric view.

For Perspective 3D views, edit the camera and target positions directly by clicking them in the Preview window and dragging them to a new location.

Note: Make sure that Show from Top is chosen from the pop-up menu in the Preview's lower right corner.

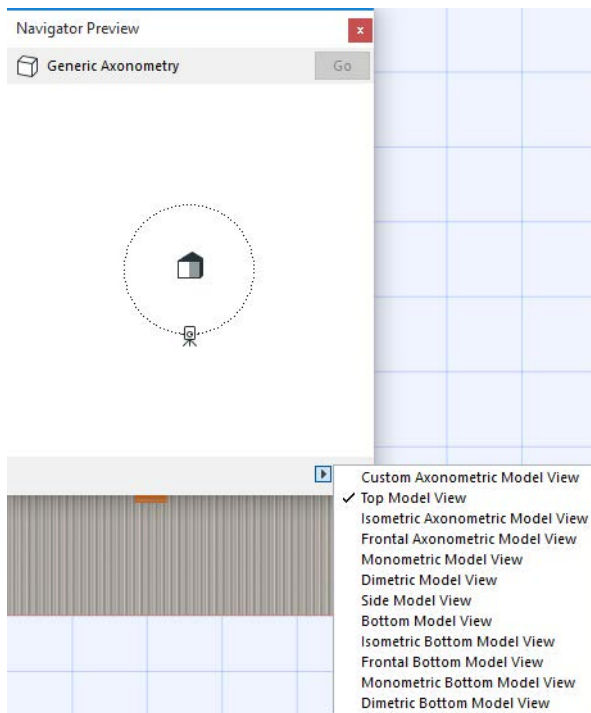


If you choose **Show from Side** view from the Preview options pop-up, you can easily re-set the camera height relative to the stories of the project: just click in the Navigator Preview and move the camera up or down.



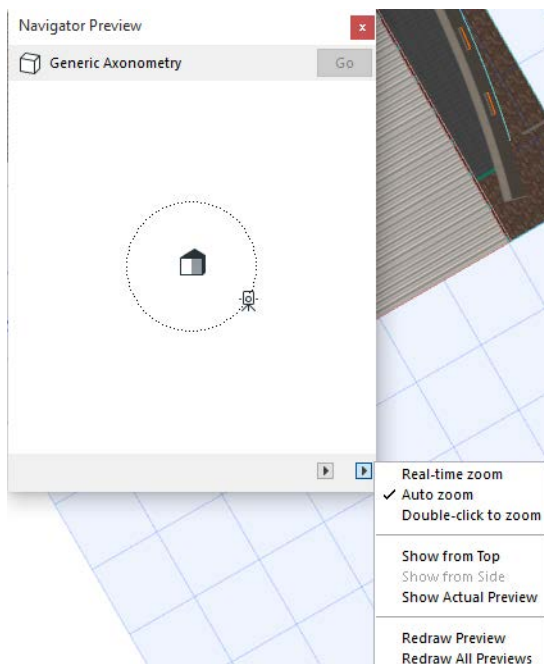
Or you can change the view cone, using the sliding switch or the small icons at its ends, in 5 degree increments.

For Axonometric (parallel) projections, the 3D Navigator Preview has two pop-up menus at the bottom right. From the left button, choose one of the 12 predefined projections to quickly switch to that view.



(These are the same predefined axonometries available from 3D Projection Settings).

To customize the projection, use the right-side pop-up button, then move the small camera icon inside the Preview window to set up a custom view.



For detailed information, see [The 3D Navigator Preview Palette Controls and 3D Projections](#).

For information on saving this data, see [Save Navigator Preview Data with Project files](#).

3D Connexion Enabler

The 3D connexion Enabler is an add-on functionality that allows you to use your 3Dconnexion device for navigating in Archicad's 3D window.

If you are using a 3Dconnexion device, use one of the two options at **View > 3D Navigation Extras > 3Dconnexion:**

- **Camera mode:** Move the device to navigate in the model by moving the camera position.
- **Object mode:** As you move the device, the model is moved accordingly. The camera viewpoint is fixed, while the model moves.

Origins

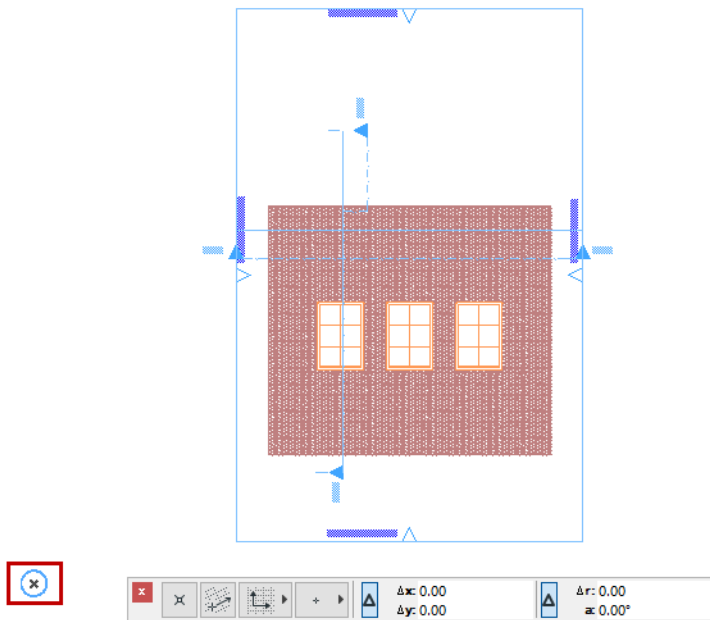
In Archicad, as in any coordinate system, all measurements are made in reference to an origin. The location of the origin is always interpreted as (0, 0).

Archicad defines three coordinate system origins:

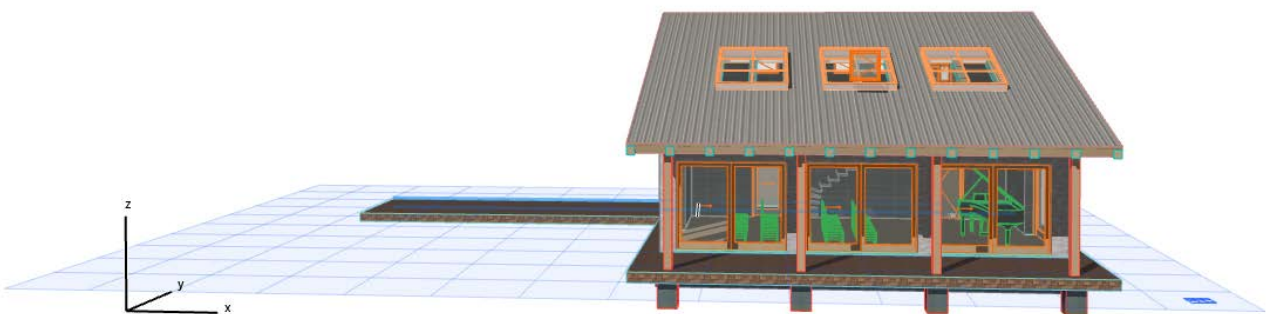
Project Origin

The **Project Origin** is a constant location which remains fixed for the life of your Project. The Project Origin of the coordinate system, marked with an X, is close to the lower left corner of the Floor Plan window, when using the default view of the default template. (The Coordinates Palette shows the coordinates as (0, 0).

In Floor Plan and other 2D views, the origin is always marked by a bold X.



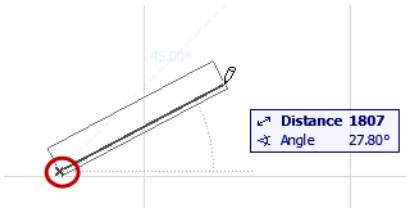
In 3D Views, the origins and their X, Y and Z axes are displayed with bold black lines.



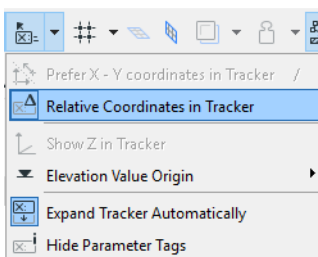
Edit Origin

The **Edit Origin** appears automatically when you click to begin input. It is a temporary origin that appears as an x during drafting and editing operations only.

In the Tracker, coordinates are measured from the **Edit Origin** by default: these are Relative Coordinates.



Note: Use the Tracker menu from the Standard Toolbar to toggle the **Relative Coordinates in Tracker** preference on or off.

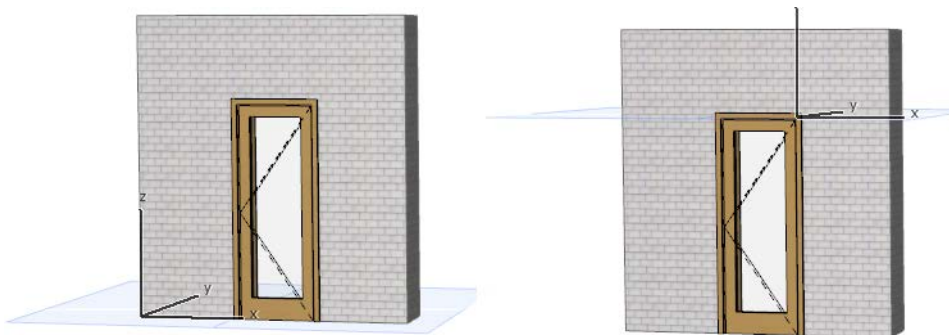


User Origin

By default, the **User Origin** is located at the Project Origin. However, the User Origin can be moved to any location, allowing you to “reset the zero point” to any location.

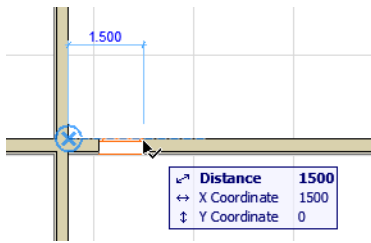
There are three ways to **reposition** the User Origin:

- At any element node: click **Alt + Shift** (Mac: Opt + Shift).
- At any location in a model window: Click the **Move User Origin** button from the Standard toolbar (or the Coordinates palette), then click at any point in the window. The origin is instantly relocated there.



See also [Editing Plane in 3D Window](#).

- Use a **Snap Reference** point as the User Origin:
When you snap to a Snap Guide during input, the User Origin is automatically relocated to the nearest Snap Reference point. A temporary blue X appears, indicating the User Origin for the current input.



Note: If you have manually relocated the User Origin, the Snap Reference is not used as a User Origin.

Return User Origin to Default Position

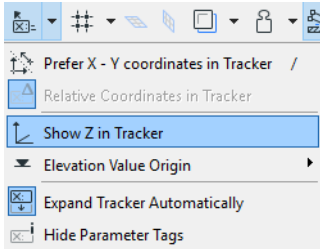
To return the User Origin to the Project Origin, double-click the User Origin button in the Standard Toolbar.



Elevation Values in the Tracker

During input, refer to the Tracker's Z field to check the elevation of the newly created element.

You must activate Show Z in Tracker from the Tracker's drop-down menu in the Standard toolbar. (This command is available during editing/input.).



If you are inputting a Wall, Column, Beam or Object using the Gravity function, the Z value will reflect the elevation of the underlying Slab/Shell/Roof/Mesh.

[See Gravity.](#)

If you switch the **Show Tracker** control to **Always** in **Options > Work Environment > Tracker and Coordinate Input**, then the Tracker's Z field will display elevation values even if input is not yet underway; just move the cursor to get feedback.

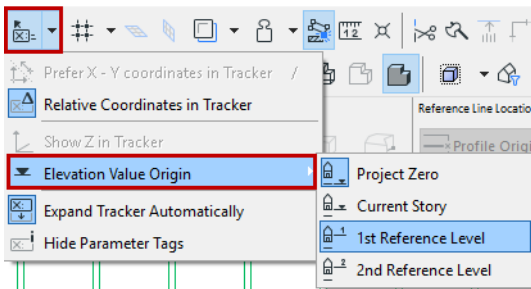
The Tracker can display elevation values in one of three different forms depending on your choice:

- from the Project Zero
- from the current Story (or in 3D, the User Origin)
- or one of the two optional Reference Levels defined in **Options > Project Preferences > Reference Levels**.

[See Reference Levels Preferences.](#)

To determine which reference level the Tracker's elevation (Z) feedback should use:

1. Open the drop-down menu of the Tracker icon from the Standard toolbar.
2. Go to the **Elevation Value Origin** sub-menu.



3. Choose the desired reference level.

Define Reference Levels at **Options > Project Preferences > Reference Levels**.

[See Reference Levels Preferences.](#)

Related Topics:

[Element Input Using Tracker](#)

Selection

Archicad makes it easy to select exactly the elements you need, even if they overlap.

[Selecting Elements](#)

[Special Selection Commands](#)

[Deselect Elements](#)

[Selection Highlights](#)

[Draw Selection Area](#)

[Selection Dots](#)

[Find and Select Elements](#)

[Define Element Criteria](#)

[Predefined Criteria Sets](#)

[Edit Selection Set](#)

[Store Selection Sets in Selections Palette](#)

[Marquee Area](#)

[The Intelligent Cursor](#)

Selecting Elements

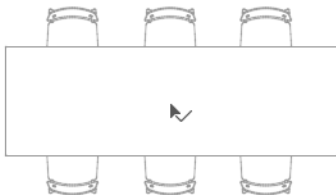
Point and click

With **Arrow tool**: Click the element

With any **other tool**: Shift + click

Or select elements by drawing a polygon around them.

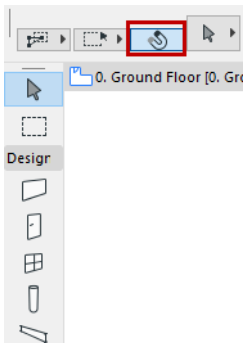
See [Draw Selection Area](#).



To select additional elements, press Shift, then select.

Click element surface to select

By default, you don't have to click a node to select the element - just hover over any part of the element, then click. This is because the Arrow Tool's "Quick Selection" method is on by default in the Info Box.



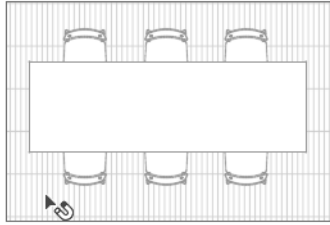
If you turned the Arrow Tool "Quick Selection" method off, you can still toggle it on the fly:

- Arrow tool: space + click
- Other tool: Shift + space + click

Element Info Tags

When you hover over elements, an info pop-up helps you identify them.

Note: If the Arrow tool is active, the element information pop-up appears on its own. With any other tool, press Shift to see the info tag.

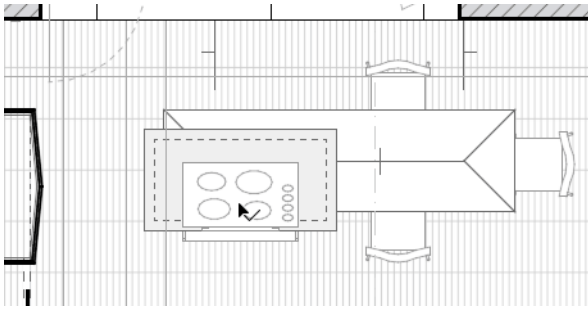


Select among overlapping elements

If you hover over several overlapping elements, the Info Tag displays the “Multiple Elements (TAB)” caption.

Press Tab to cycle the highlight among possible elements.

When the element you need is highlighted, click to select.



Configure Info tags

Enable, disable and set timing at **Options > Work Environment > Selection and Element Information**.

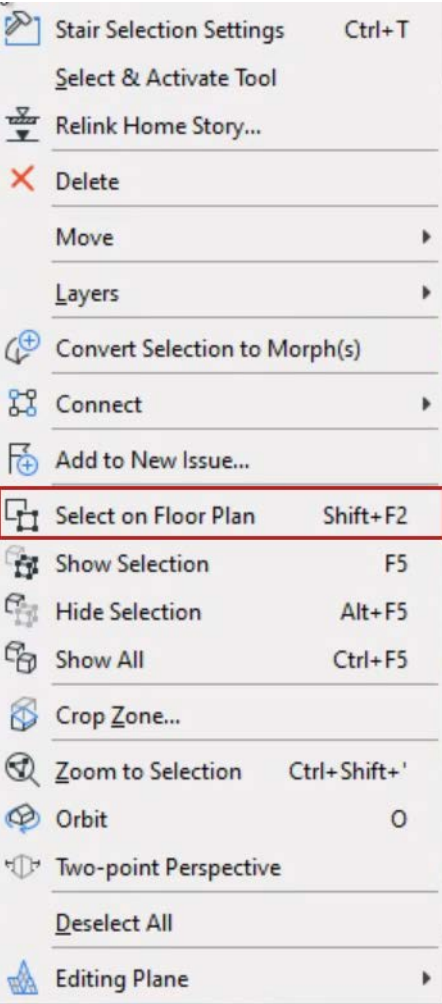
[See Selection and Element Information](#).

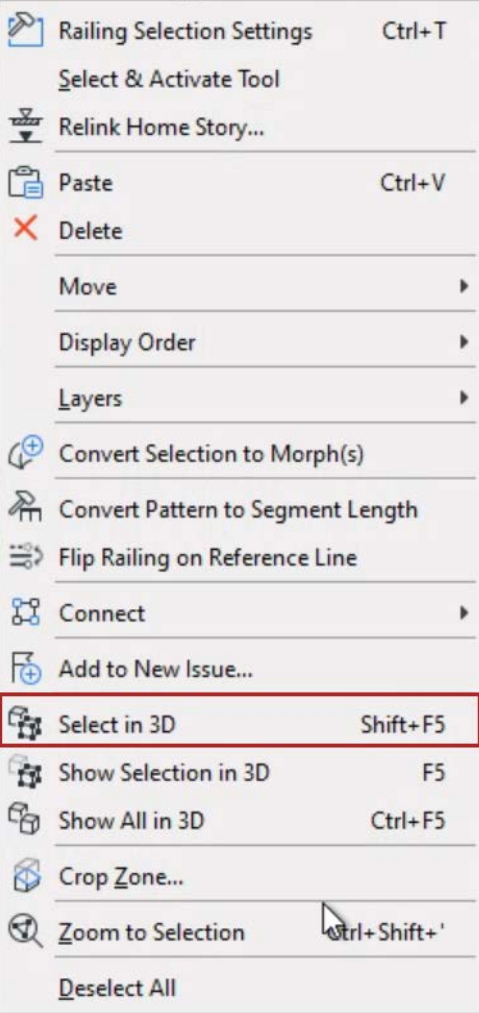
Related topics:

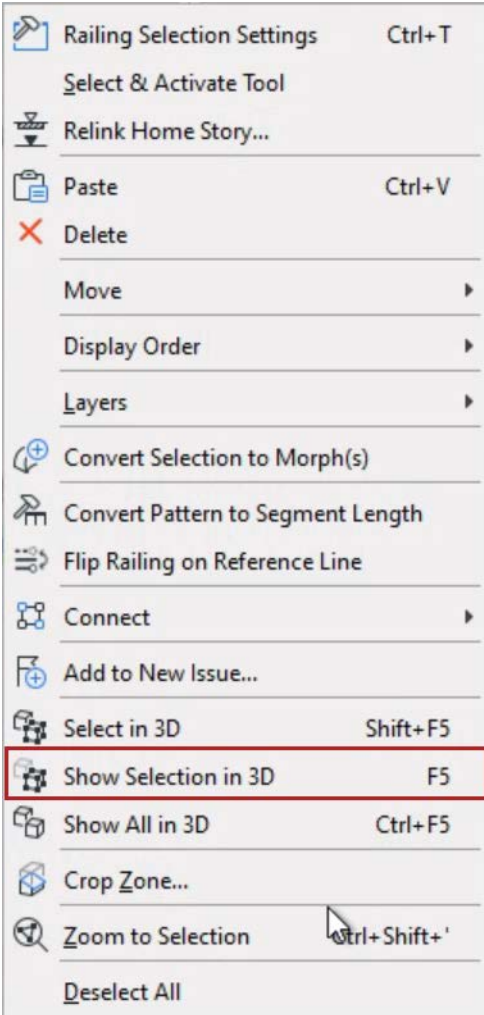
[Draw Selection Area](#)

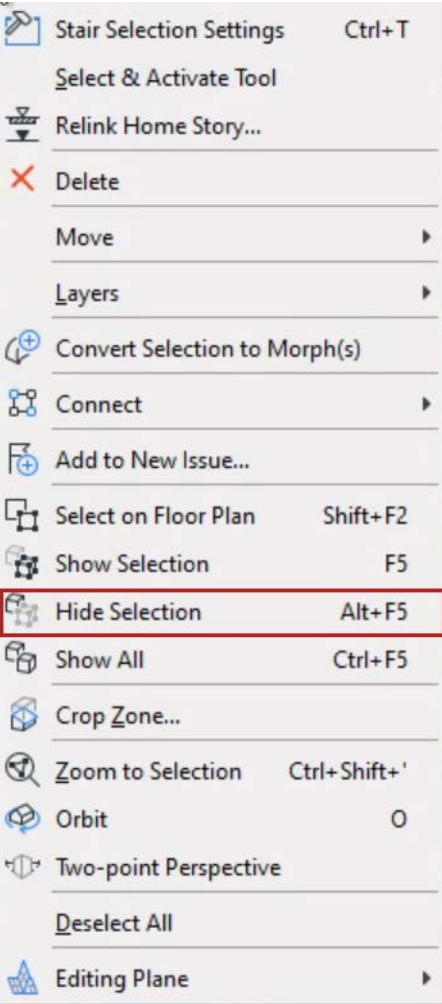
Special Selection Commands

Command	What it does	How to do this
Select All	Select all elements in window or in Marquee. (Selects elements on current story only.)	Activate Arrow or Marquee tool, then: Ctrl/Cmd+A or Edit > Select All
Select All (single element type)	Select all elements of a single type (e.g. all Walls). (Selects elements on current story only.)	Activate tool (e.g. Wall tool), then: Ctrl/Cmd+A or Edit > Select All (the command changes to the active element type, e.g. Select All Walls)
Find & Select	Select elements based on multiple criteria	Set up a filter to select elements defined by a wide variety of criteria. See <i>Find and Select Elements</i>.

Command	What it does	How to do this
Select on Floor Plan	Open floor plan, zoom to this selected element	<p>If element(s) are selected in 3D or Section:</p>  <p>Use this command from context menu</p>

Command	What it does	How to do this
Select in 3D	Open 3D window, zoom to this selected element	<p>If element(s) are selected in Floor Plan or Section:</p>  <p>Use this command from context menu</p>

Command	What it does	How to do this
Show Selection in 3D	Open 3D window, show this selected element only	<p>If element(s) are selected in Floor Plan or Section:</p>  <p>The screenshot shows a context menu with various options. The option 'Show Selection in 3D' is highlighted with a red border. Other options include 'Railing Selection Settings', 'Paste', 'Delete', 'Move', 'Display Order', 'Layers', 'Convert Selection to Morph(s)', 'Convert Pattern to Segment Length', 'Flip Railing on Reference Line', 'Connect', 'Add to New Issue...', 'Select in 3D', 'Show All in 3D', 'Crop Zone...', 'Zoom to Selection', and 'Deselect All'.</p> <p>Use this command from context menu</p>

Command	What it does	How to do this
<p>Hide Selection in 3D</p>	<p>Hide the current selection in 3D window</p>	<p>If element(s) are selected in 3D window:</p>  <p>Use this command from context menu</p>
<p>Zoom to selection</p>	<p>Zoom in on selected element(s)</p>	<p>In any window: Click empty space, then use this command from context menu or View > Zoom > Zoom to Selection</p>

Deselect Elements

To do this	Do this
Deselect all elements	<ul style="list-style-type: none">• Press Esc or <ul style="list-style-type: none">• Click in empty space
Deselect one element	Press Shift, then click element
Deselect several elements	<ol style="list-style-type: none">1. Activate Arrow tool2. Press Shift3. Draw a selection rectangle around elements

Selection Highlights

Highlights help you identify elements before and after selection.

Pre-selection highlight

Identifies an element before you actually select it.

Hover a selectable element, then **press Shift** to see its highlight.

Note: With the Arrow tool active, the highlight appears without Shift.

The pre-selection highlight is especially useful to

- Pick the right one among several overlapping elements
- Label or dimension elements
- Transfer parameters among elements ([see Parameter Transfer](#))
- Use Magic Wand

Set highlight colors

Configure highlight colors at **Options > Work Environment > Selection and Element Information**.

You can also turn off any or all highlights altogether.

[See Selection and Element Information.](#)

Draw Selection Area

Select elements by drawing a polygon around them.

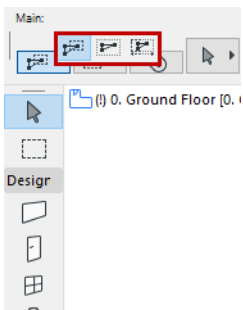
1. Activate the Arrow tool.
2. Click outside the element.
3. Draw a selection rectangle that includes at least one node of the element.

Note: This is the default method. See other methods and shapes below.

Selection Polygon Methods

To access these options in the Info Box, do one of the following:

- Activate the Arrow Tool
- With any other tool active, press Shift



Partial elements method (default)

Any element with **at least one node** inside the polygon is selected.

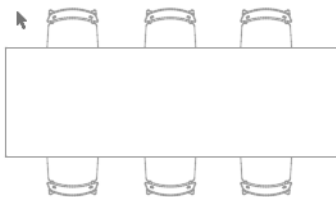
Entire elements method

Any element with **all nodes** inside the polygon is selected.

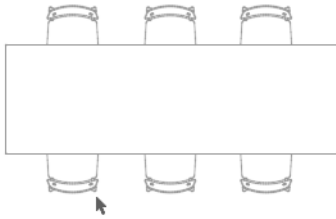
Direction-dependent method

Selection logic depends on the direction in which you draw the polygon.

- Draw from right to left: only elements entirely inside the polygon are selected.



- Draw from left to right: partial elements are selected.

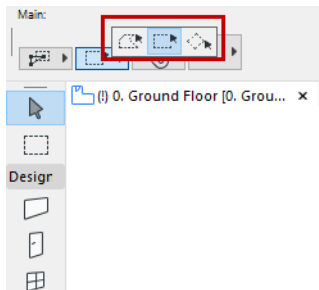


Selection Polygon Shapes

To access these options in the Info Box, do one of the following:

- Activate the Arrow Tool
- With any other tool active, press Shift

Selection area shape options are **Polygonal**, **Rectangular** and **Rotated Rectangular**.

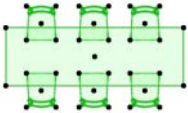
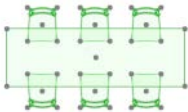
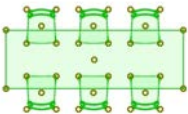






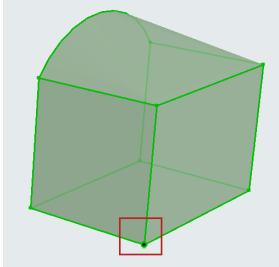
Related Topics:

[Selection Type \(to select Morph Sub-Elements\)](#)

[Watch the Video](#)

Selection Dots

Type	Example	Notes
Regular		
Locked		<p>An element is locked in any of these cases:</p> <ul style="list-style-type: none"> • It is on a locked Layer. • You have locked it by command (Edit > Locking > Lock) • It is part of a Hotlink Module or XREF. <p>Locked elements cannot be edited.</p>
Grouped		
Hotlink Module or XREF		<p>Elements are locked and cannot be edited.</p>
Editable hotspots (certain GDL Objects)		<p>Grab the hotspot to graphically edit the object (e.g. open or close the door).</p> <p>See <i>Graphical Editing Using Editable Hotspots</i>.</p>
Associative Railing		
Static Railing		

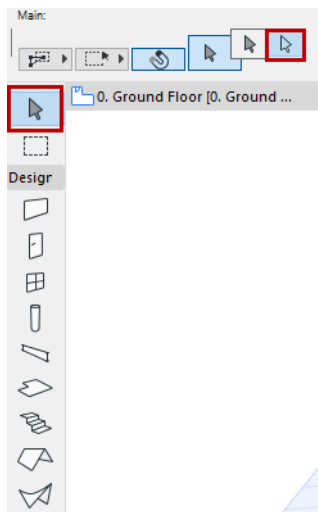
Type	Example	Notes
Morph		<p>A selected Morph has only a single (black) hotspot, which represents the point of its elevation value; but all other nodes/corners are also editable.</p>

Set Selection dot colors

Use **Options > Work Environment > Selection and Element Information.**

[See Selection and Element Information.](#)

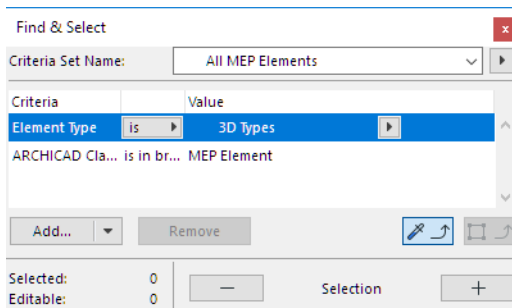
Selection Type (to select Morph Sub-Elements)



[See Morph Selection.](#)

Find and Select Elements

Use the **Find & Select Palette** to select and deselect elements based on defined criteria.



Open this palette from **Edit > Find & Select**;

or click the Find & Select shortcut from the Standard Toolbar:



The Find & Select palette can remain open as long as you wish on top of all windows.

Set up Criteria for Find & Select

To set up the Find & Select Criteria, you can use predefined Criteria Sets, or set up the criteria manually, as described in these sections:

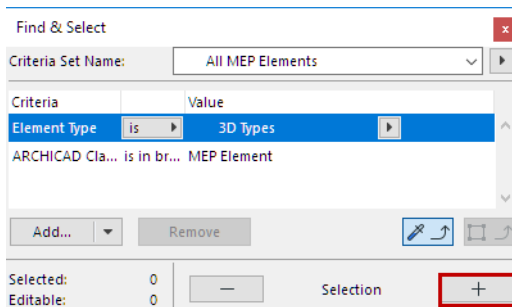
[Predefined Criteria Sets](#)

[Define Element Criteria](#)

[Define Criteria Based On Selected Elements](#)

Click Plus to Select

Once you set up the criteria, click the **Plus** button. All elements that fit your defined criteria are selected.



- Elements are selected on the visible layers of the currently active window.
- The **Selected** and **Editable** values at bottom left give you feedback on the currently selected elements in the project.

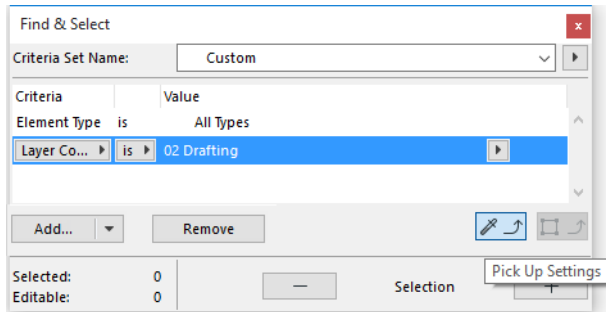
Deselect Elements

Click the **Minus** button to deselect all the elements that fit the current criteria in the Find & Select Palette.

Define Criteria Based On Selected Elements

Use this feature if you have added a number of criteria (e.g. Element Type, Layer, Surface) in the Find and Select dialog box and want to define their values based on an existing element. This is a quick way to define a **Find & Select** criteria set.

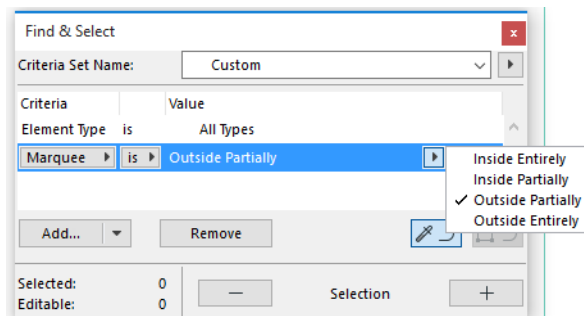
1. In the **Find & Select** dialog box, define the criteria you want to search by (e.g. Element Type, Layer).
2. Click the **Pick up Settings** button to activate it.



3. In the project, **Alt+click** an element whose settings you want to load.
4. The Element Type and Layer values of the clicked element are now loaded in the **Find & Select** dialog box.

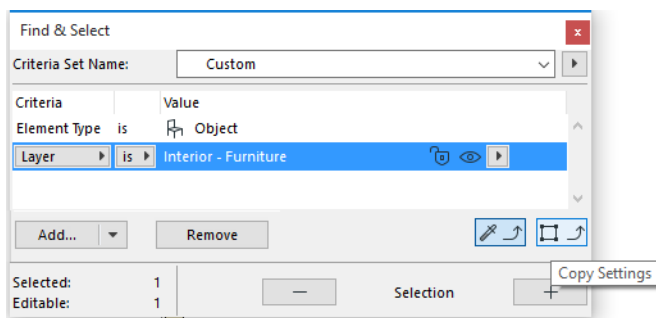
Use Marquee in Find & Select

If you have placed a **Marquee**, then the Find & Select includes the Marquee criterion automatically. You can then opt to search either inside or outside the Marquee.



Copy Settings

If any elements are selected, the second button - **Copy Settings** - becomes active. Click the button to load the settings of the last selected element into the **Find & Select** dialog box.



Related Topics:

[Find and Select Text Blocks/Labels](#)

[Find and Select Missing Library Parts](#)

[Find and Select elements with missing attributes](#)

[Define Element Criteria using IFC Data](#)

[Find and Select Element or Building Materials: By Property and Classification](#)

Define Element Criteria

In several Archicad functions, you specify a group of elements by criteria before carrying out the function:

- Find & Select
- Graphic Override Rules
- Collision Detection
- Reserve Elements by Criteria (Teamwork)
- Interactive Schedule Schemes

You can set up these criteria either manually, or by using predefined Criteria Sets. ([see: Predefined Criteria Sets](#))

This section describes manual criteria definition.

Criteria definition is largely identical in all four dialogs (Find & Select, Graphic Override Rules, Collision Detection, Reserve Elements by Criteria).

Topics in this section:

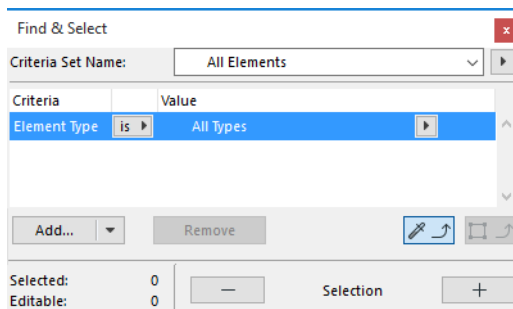
Define First Element Type

Define Additional Element Type

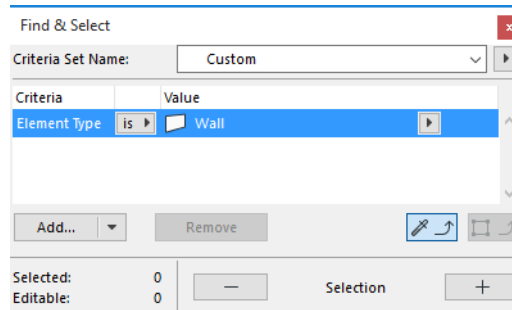
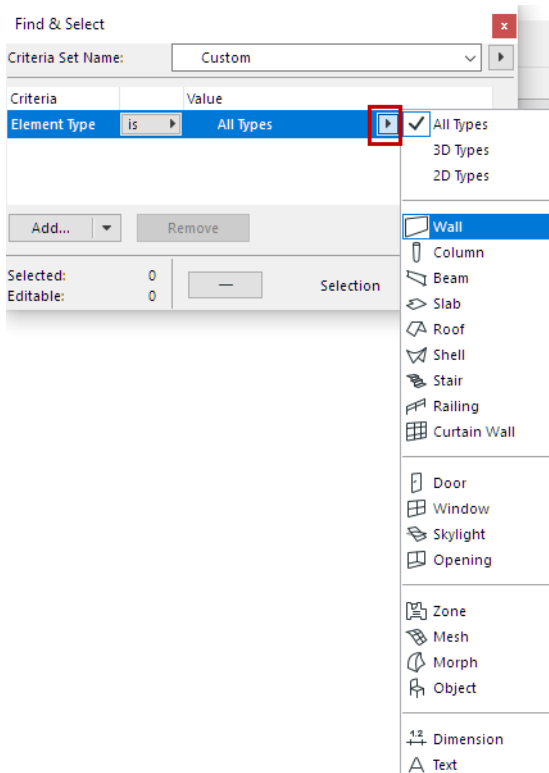
Define Additional Criteria: Parameters & Properties

Define First Element Type

1. In the **Criteria** column, the first line is always Element Type. Click to select it.



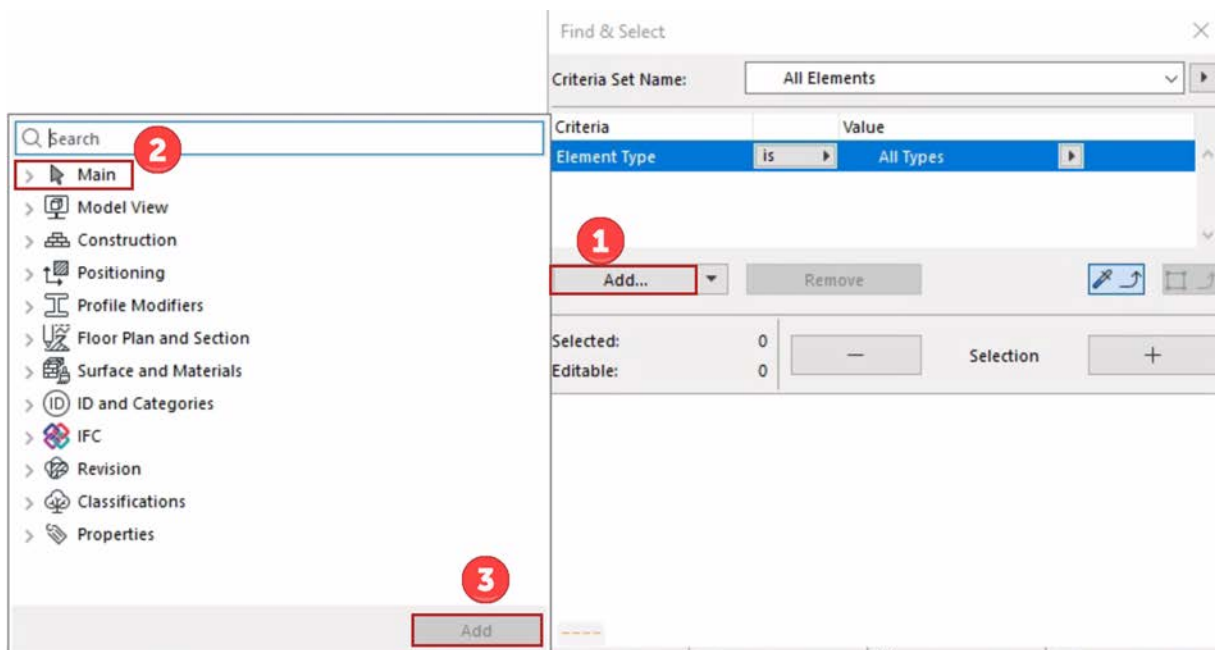
2. In the **Value** column, click the pop-up and choose an Element Type from the list (e.g. Wall).



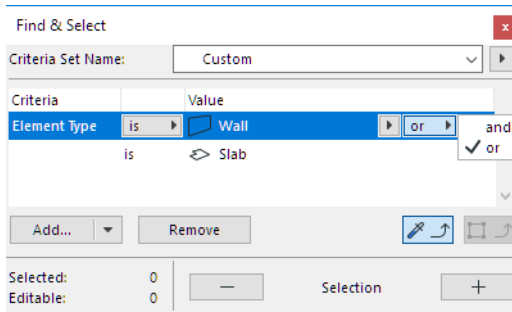
Define Additional Element Type

To add one or more additional Element Types:

1. Click the **Add** pop-up
2. Select “Element Type” from the list. (Use the search field at the top to help locate an item.)
3. Double-click, or click the list’s **Add** button to add it to your Criteria List.



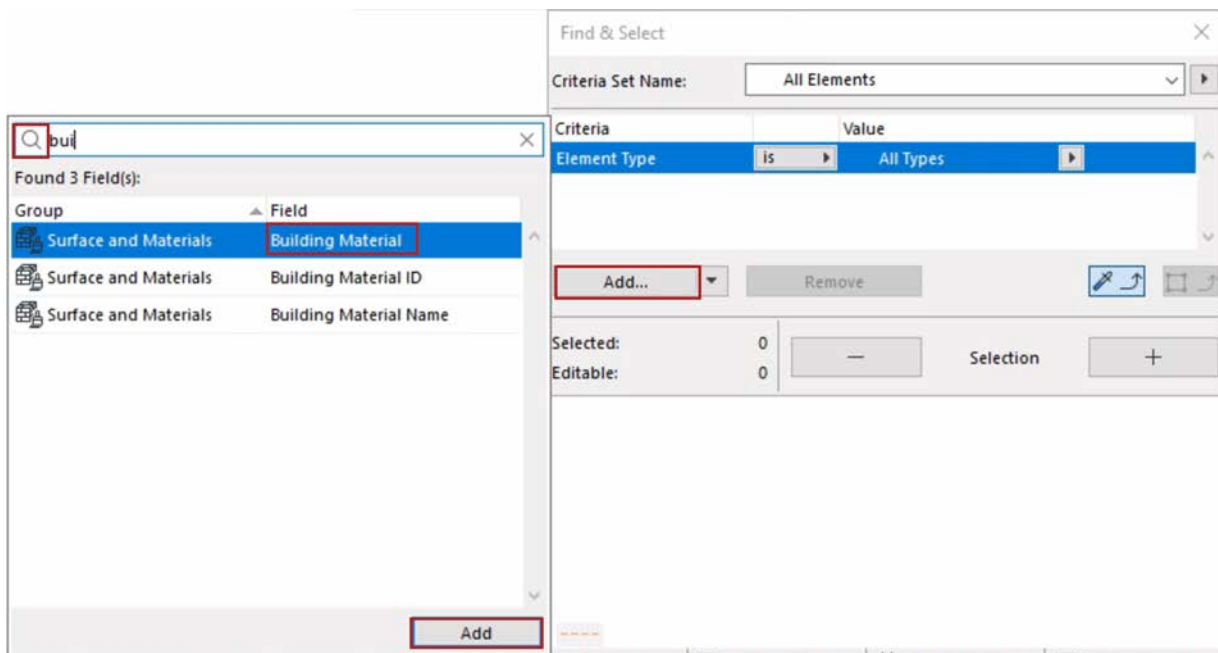
4. In the **Value** column, click the pop-up and again choose the Element Type from the list (e.g. Slab).
5. Use the **and/or** control to further define the Element Type criteria.



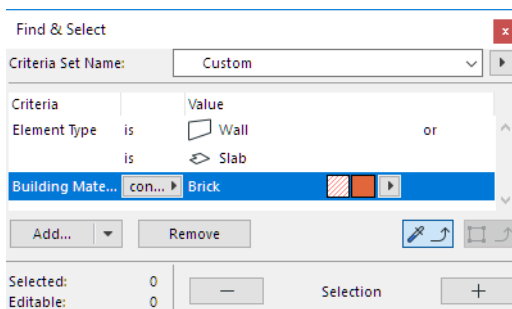
Define Additional Criteria: Parameters & Properties

To narrow the scope, add more criteria.

1. Click the **Add** button to bring up a list of Parameters & Properties.
2. Choose a criterion (use the search field at the top to find it more quickly), then click **Add** (or press Enter) to add it to the criteria list.

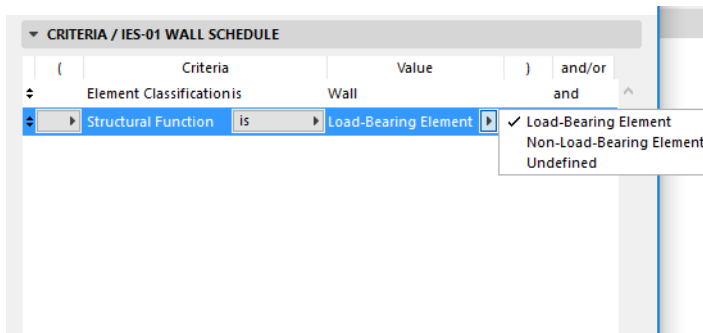


3. For each added criterion, define its Value from the pop-ups.



Available criteria depend on the Element Type(s) defined in the row(s) above.

For example, you can narrow the criteria based on the element's Structural Function Classification (e.g. Load-Bearing or Non-Load-Bearing.)



Note: Defining a “Layer” or “Layer Combination” as a criterion means that the program will find and select the elements which are *visible* according to the given Layer Combination.

Use IFC Properties or Library Part Parameters as Criteria

From the Add button's pop-up, access additional parameter/property types:

- **IFC Properties**

[See *Define Element Criteria using IFC Data*.](#)

- **Library Part Parameters** (Interactive Schedule Schemes only)

[See *Additional Criteria and Fields: Library Part Parameters*](#)

Related Topics

[Find and Select Elements](#)

[Graphic Override Rules](#)

[Collision Detection](#)

[Reserve Elements by Criteria \(Teamwork\)](#)

[Scheme Settings Dialog Box \(Interactive Schedule\)](#)

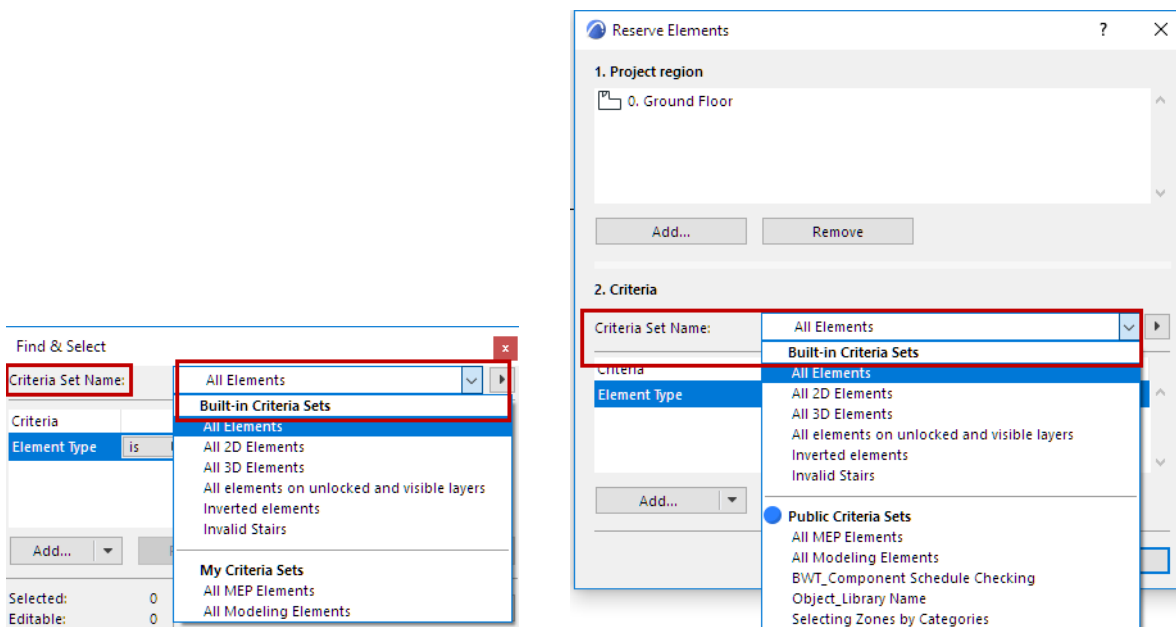
Predefined Criteria Sets

Several Archicad functions require that you specify elements by criteria. You can set up these criteria manually. But to streamline the search for elements, you can use built-in or customized Criteria Sets instead, in the following functions:

- Find & Select
- Collision Detection
- Reserve Elements by Criteria (Teamwork)

Built-In Criteria Sets

From the **Criteria Set Name** pop-up, optionally choose a Built-In Criteria Set:



- All Elements
- All 2D Elements
- All 3D Elements
- All elements on unlocked and visible layers
- Inverted elements

[See *Inverted Elements*.](#)

- Invalid stairs

[See *Toplinked Stairs and Changed Story Height*.](#)

Built-in Criteria Sets cannot be modified.

Note: The built-in **“All elements on unlocked and visible layers”** Criteria Set is designed to fit a workflow in which the project manager sets up views especially defined for Teamwork usage. You can create several views out of a single viewpoint, then vary the layer states in each view: set the layers of elements that logically belong together to be visible and editable (with the rest of the non-relevant elements locked), then name the view accordingly. For example, a view entitled “Ceiling Plan” might place all the paneling and mechanical structures onto visible and editable layers. This way, the designated user

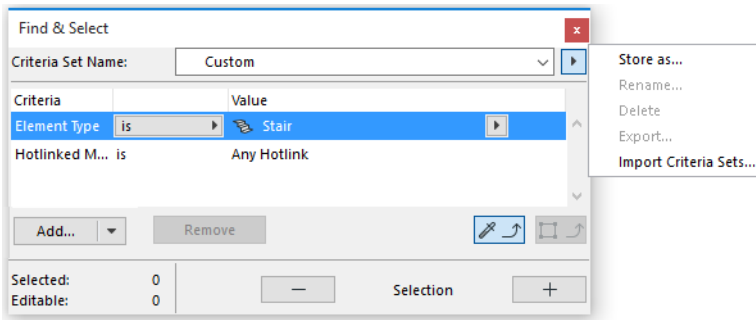
would reserve elements from the Ceiling Plan view by using the built-in “Elements on unlocked and visible layers” criteria set.

Public Criteria Sets (Teamwork) and **My Criteria Sets** are listed if you have already defined and saved custom criteria sets (see below).

Define and Store Your Own Criteria Sets

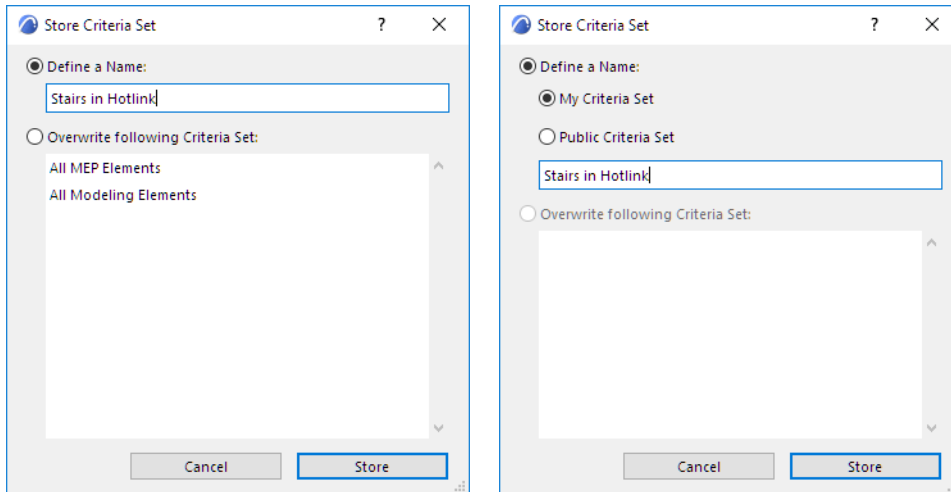
To store a Criteria Set:

1. Set up the criteria as desired. The Criteria Set name is now “Custom.”
2. Next to the Criteria Set Name field, click the pop-up button to access the relevant commands. Click **Store as**.

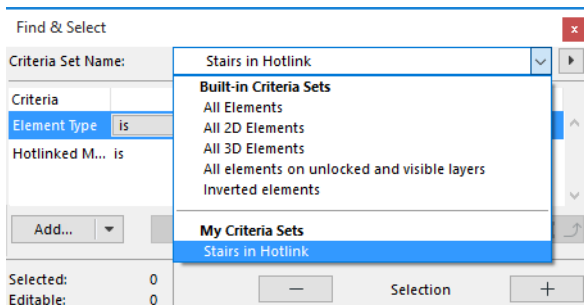


3. In the appearing dialog box, give a name to the Criteria Set and click **Store**.

In Teamwork: Choose either **Public Criteria Set** (to make it available to all project users) or **My Criteria Set** (if you do not wish to share it).



Your customized Criteria Sets are now stored in the Criteria Set pop-up, under “My Criteria Sets.”

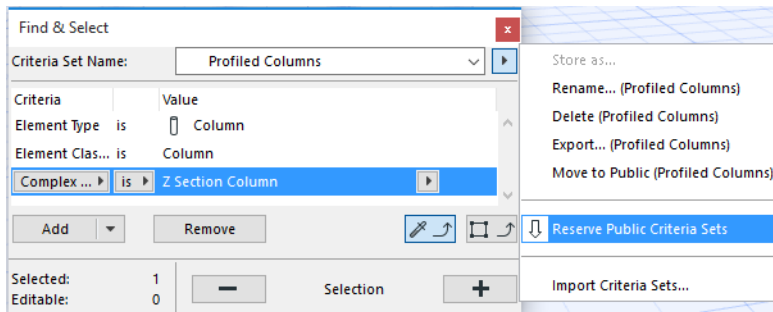


Use the commands from the pop-up to manage your Criteria Sets: Rename, Delete, Export or Import them (as.xml files).

Public Criteria Sets in Teamwork

In a Teamwork Project: These are criteria sets that you have created and stored a Public Criteria Set (see above). These criteria sets are part of the shared Teamwork project.

Public Criteria Sets can be reserved through the Find & Select dialog box: click **Reserve Public Criteria Sets** from the Criteria Name pop-up. Public Criteria Sets are shown as underlined if they have not yet been sent to the server.



This reservation means that you alone have the right to *modify* any of the stored Criteria Sets. However, any user - regardless of reservation status - can create, use and store new Criteria Sets.

Note: “My Criteria Sets” are not shared, and cannot be reserved.

Related Topics

[Find and Select Elements](#)

[Collision Detection](#)

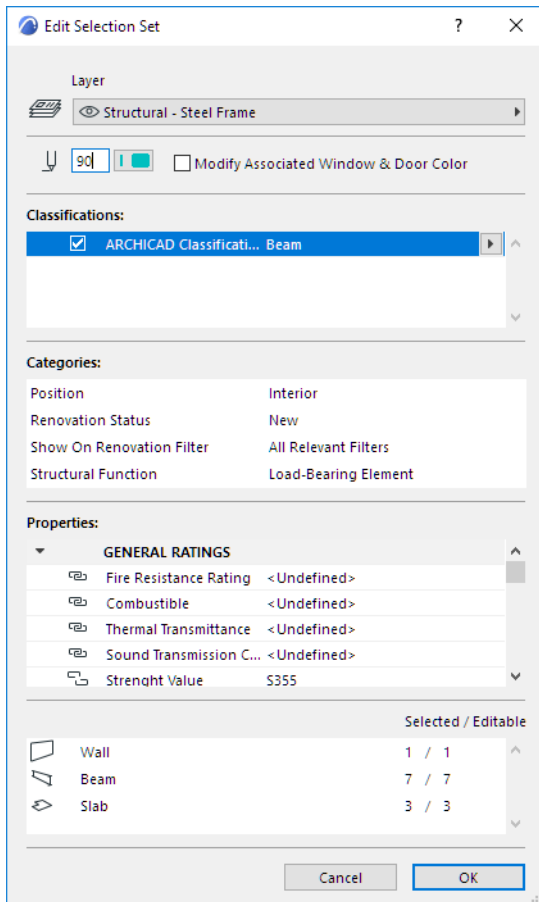
[Reserve Elements by Criteria \(Teamwork\)](#)

Edit Selection Set

Use this dialog box (**Edit > Element Settings > Edit Selection Set**) to edit specific attributes (layers, pencolors, classifications and properties) for a collection of dissimilar element types simultaneously, without affecting any other settings of these elements.

If you change any of these attributes in the **Edit Selection Set** dialog box, all the selected elements will take on the new values.

Note: The pen setting affects all cut pens of the selected elements (cut fill pens, cut line pen).

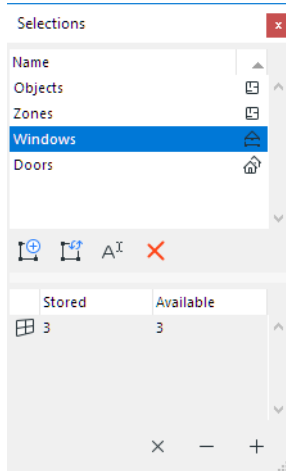


- **Modify Associated Window & Door Color:** Check this box to also modify the Pencolor/Penweight of Window and Doors contained in any selected Walls.
Note: If not checked, the openings will retain their individual Pencolors/Penweights.
- The pencolors of **GDL Object**-type elements can only be changed if the object is not set to be displayed with its own Symbol Pens (defined in the 2D symbol of the GDL Object). When editing the object, the standard Pen Color radio button will be preset to the edited Pen Color.
- All parts of composite elements, such as Walls, Dimension chains or Markers (made up of lines, Arrowheads and Text), are modified together.
- **Selected/Editable:** This area displays the number of elements of each type currently selected on the Floor Plan and in 3D.

Store Selection Sets in Selections Palette

Use the **Window > Palettes > Selections** palette to create saved, frequently accessed sets of selections that are stored with the Project.

The controls of the Selections Palette allow you to save any selection made on the Floor Plan, 3D window, Section/Elevation/IE window, or a Detail/Worksheet drawing. You can delete a selection or sort saved selections by name or by window of origin.



Use **Add Selection** to save this selection (you will be prompted to name it.)

Use **Redefine** to update any Selection Set based on current element selection.

Click the list headers to sort the items by name or the window they originate in.

The bottom part of the palette displays the contents of the selection set.

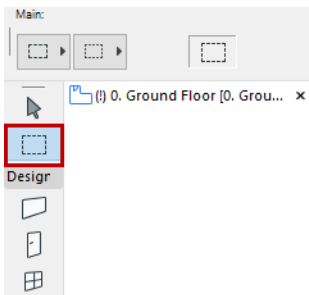
Use the X, - and + controls as follow:

- Click X function to select, in the project, *only* the elements in the highlighted selection set.
- Click - to deselect, in the project, the elements of the highlighted selection set.
- Click + to select all the elements of the highlighted selection set (in addition to any other current selection).

The operation is always performed on the currently active window.

Marquee Area

The **Marquee** tool is used:



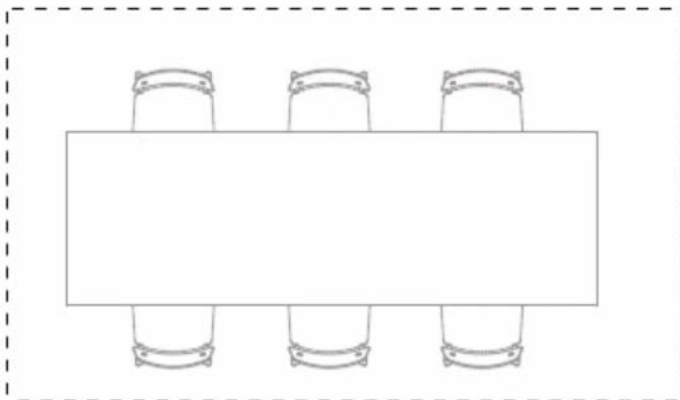
- Define a smaller area in which to select all of elements of a given type
- View only a certain part of the project in 3D
- To move or resize several elements simultaneously
- Save only part of the view in PDF or DWG format

The **Marquee** is available in all project windows.

Note: In PhotoRendering or picture windows, and the GDL Object's 3D view, the only operation available for a marquee selection is Copy.

Place Marquee

1. Select the Marquee tool.
2. Click in the project and define the Marquee area.



A Marquee area does not display selection dots on the included elements; the marquee area is shown by a dashed line.

For more options, see [Marquee Area Definition](#).

Remove Marquee

Do one of the following:

- Press **Esc**
- Use **Remove Marquee** from the context menu

- Begin drawing a new Marquee
- Double-click in the window outside the marquee area with the **Marquee** tool selected.

If you have explicitly selected further elements within the Marquee, pressing Esc will deselect these elements in reverse order, and finally remove the Marquee.

If you choose another tool, the Marquee area will *not* be removed.

Select All Elements in a Marquee

To select all elements inside a Marquee area with the Marquee Tool active, use the **Ctrl/Cmd+A** shortcut or the **Edit > Select All in Marquee** command.

View Marquee Area in 3D

Define the Marquee area, then use the **View > Elements in 3D View > Show Selection/Marquee in 3D** command; or select the same command from the context menu.

Any part of an element that falls inside the Marquee area will be included in the 3D view, even if there is no selection node or hotspot on this part of the element.

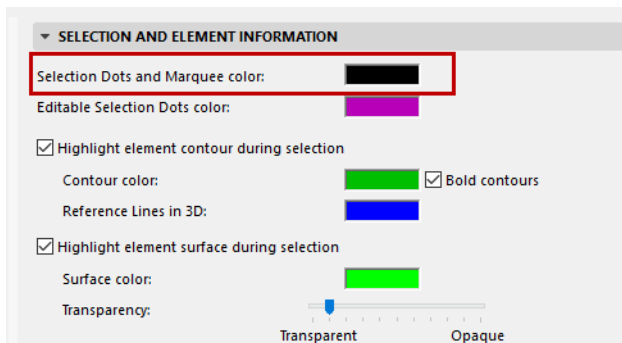
Note: If your plan includes both explicitly selected elements and a marquee selection, this command will disregard the marquee and show only the explicitly selected elements.

To further define how Marqureed elements are shown in 3D, use the “**Marquee Effects**” controls of the **View > Elements in 3D View > Filter and Cut Elements in 3D** dialog box.

To use the Marquee to define Photorendering output: see [Render Marquee Options](#).

Marquee Color

The color of the Marquee is the same as that of Selection dots: set this color at Options > Work Environment > Selection and Element Information > Selection Dots and Marquee color.



Topics in this Section:

[Marquee Area Definition](#)

[Save Marquee Area as PDF or DWG](#)

[Copy/Paste Marquee Area from Project Window](#)

[Stretching with the Marquee Tool](#)

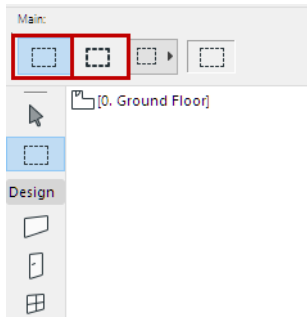
[Use Marquee in Find & Select](#)

[Other Editing Operations within the Marquee](#)

Marquee Area Definition

Single Story or All Stories

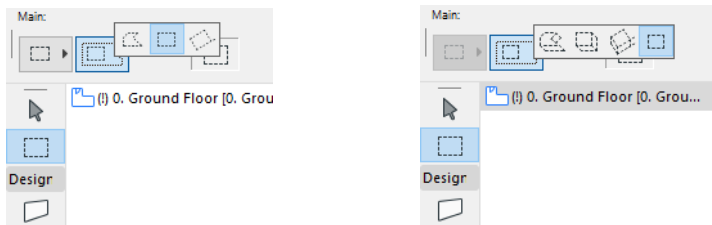
The **Marquee** tool has two definition methods available in the Info Box (Floor Plan only).



- **Single Story** method (thin contour): Marquee elements on the current Story only.
- **All Stories** method (thick contour): Marquee elements on all Stories.

Marquee Area Geometry

Choose one of the Geometry Methods from the Info Box.



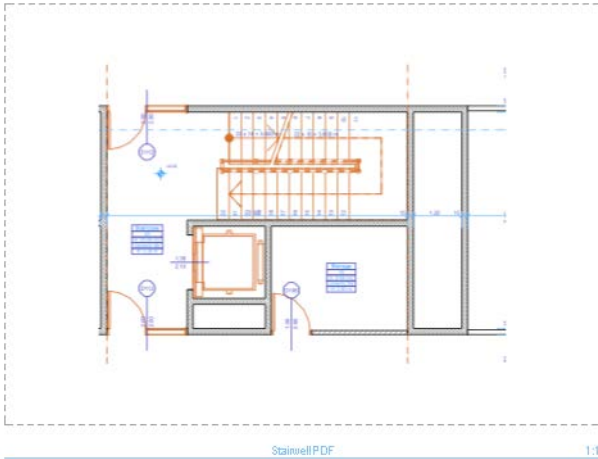
- **Polygonal:** Only straight segments are possible.
- **Rectangle** method: Click twice to define.
- Rotated Rectangle: Click three times to define.
- In the 3D window: Use the **Polygonal**, **Rectangle** or **Rotated Rectangle** methods to first define the base of the marquee area, then click again to define the height of the selected space. (In perspective view, the marquee base must be underneath the horizon.)
- The fourth option (3D window only) allows you to draw a **2D marquee** in order to copy elements from the 3D Window in the form of pictures.

For more information, see [Copy/Paste Marquee Area from Project Window](#).

Save Marquee Area as PDF or DWG

When saving a project view in PDF or DWG format, you may use the Marquee to delimit the part you wish to save.

After placing the Marquee, use the **Save As... PDF** or **Save As... DWG** command as usual. Only the Marquee area will be saved to the file, rather than the entire window contents.

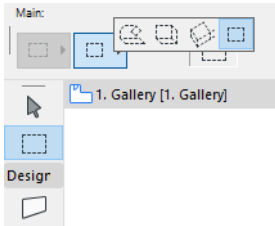


Save Document in PDF Format
Saving DWG/DXF Files

Copy/Paste Marquee Area from Project Window

You can use the Marquee tool to copy and paste an area of the 3D Window, the Floor Plan, a Section/Elevation/IE or 3D Document window, or a Model Picture Window.

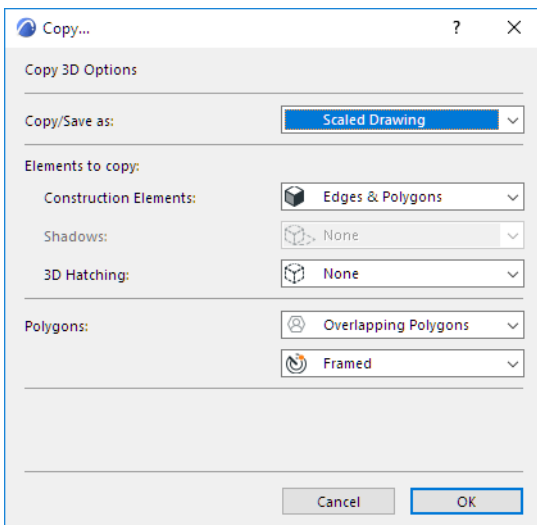
1. Generate the view you need.
2. Use the Marquee tool to select the area you wish to copy (if you are selecting from the 3D window, make sure that the 2D Marquee option is active).



3. Choose **Edit > Copy**.

If you are copying from the 3D window using the **Hardware acceleration 3D Engine**, all images will be pasted in pixel format.

If you are copying from the 3D window using the **Vectorial 3D Engine**, a dialog box appears during the Copy operation giving you options for how the image will be displayed on the screen.



For more information, see [Copy Dialog Box](#).

Note: If you are copy-pasting construction elements, they will be placed as 2D elements (points, lines and fills.)

You may find that the “exploded” 2D elements generated from the model often contain superfluous elements (extra line segments, overlapping or superfluous fills) that make such editing difficult. To make editing easier, use the **Linework** and **Fill Consolidation** functions on selected items in the window.

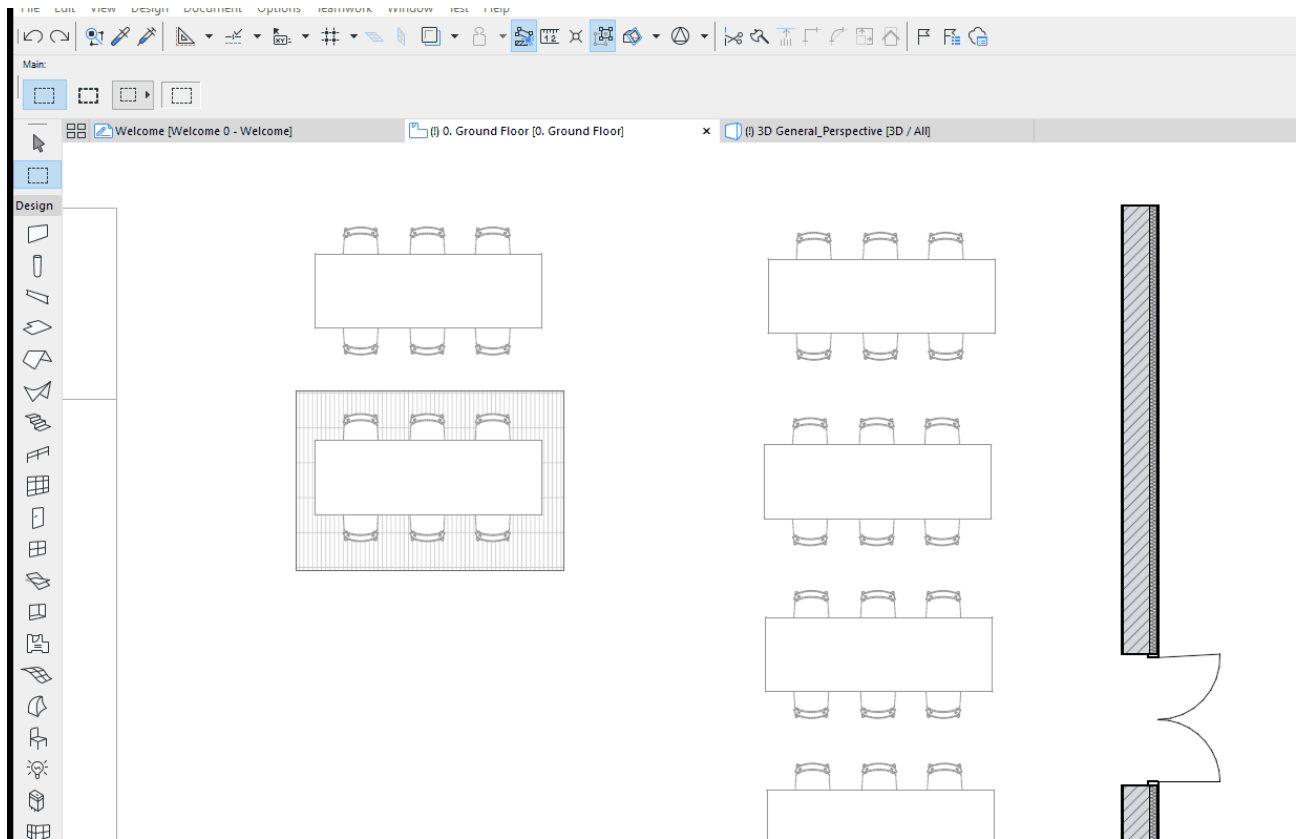
For more information, see [Consolidate Lines and Fills in Drawing Windows](#).

4. Move to the Floor Plan and choose **Edit > Paste**.

Note: You can paste into other 2D model windows, but the pasted elements will behave as 2D drawing elements only, not model elements.

5. Position the image on your Floor Plan using the Trident cursor.

6. Click outside the selection rectangle to complete the operation.
- Elevations are pasted to the Floor Plan as full-sized 2D elements.
 - Parts of Model Pictures are pasted as screenshots.



Other Editing Operations within the Marquee

If you wish to **Cut**, **Copy**, **Clear**, **Drag**, **Mirror**, **Rotate**, **Elevate** or **Duplicate** elements, they must have at least one node inside the Marquee area in order to be selected. The edit operations will act on the whole element.

Note: In Section/Elevation/IE, 3D Document, Detail and Worksheet windows, some editing commands are available only for 2D drawing elements.

See also [Stretching with the Marquee Tool](#).

On-Screen Input Aids

The following sections describe Archicad functions that assist in precise element input.

[The Intelligent Cursor](#)

[Selection Highlights](#)

[Tracker](#)

[Ruler](#)

[Measure Tool](#)

[Set up Construction Grid](#)

[Set up Snap Grid](#)

[Snap to Grid](#)

[Snap Guides](#)

[Guide Lines](#)

[Snap Points](#)

[Editing Plane in 3D Window](#)

[Magic Wand](#)

[Gravity](#)

[Mouse Constraints](#)

[Coordinate Constraints](#)

[Relative Construction Methods](#)
































[Snapping to Existing Elements](#)

[Cursor Snap Range](#)






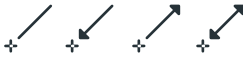




[Distance Guides - Experimental feature](#)

The Intelligent Cursor

The cursor changes shape as you work, depending on the tool you are using, and whether an editing/input operation is underway or not. Cursor variations help you identify particular nodes and edges, as an aid in editing.

	Empty Space	Reference Line	Other Edge	Other Edge - Perpendicular	Inter-section	Node of Reference Line	Other Node
Before Input (Arrow Tool)							
Before Input (Other Tools)							
During Input/Editing							
Magic Wand							
Trim Elements							
Pick Up Parameters							
Transfer Parameters							

Other Cursors

Cursor	Description
	Empty space: No selectable objects at this location
	Quick Selection option (for surface selection with Arrow tool)
	Drag/Rotate/Mirror Element Copy
	Drag/Rotate/Mirror Multiple Element Copies
	<p>Appears during element input/editing, when several options are available. E.g.:</p> <ul style="list-style-type: none"> • set direction of Roof pitch • when splitting element: choose which part of element should remain selected • set Limited depth of a Section • curved Walls placed with tangential method - choose preferred placement.
	Fill Handle
	Place Dimension/Zone Stamp/Fill Area, or Close Polygon
	Paste, then move contents of Marquee/Clipboard
	Choose Plane (Place Dimension in a 3D Document)
	Choose Line/Edge (Place Linear Dimension - Any Direction)

Tracker

Coordinate values are displayed in the **Tracker**, which follows your mouse movements and is available in any editable window type.

The Tracker gives you dynamic feedback on such values as edited lengths, angles, and hotspot values for GDL Object editing.

Also use the Tracker to input elements by defined values.

[See *Element Input Using Tracker*.](#)

Show Tracker

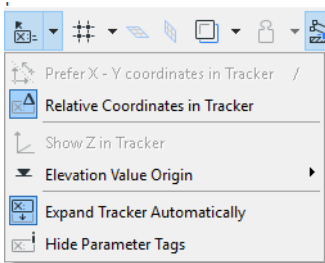
Do one of the following:

- Activate the Tracker toggle on the Standard toolbar



- Use the **N** shortcut. The Tracker will pop up and display the cursor position.
- Check **Show Tracker** in **Options > Work Environment > Tracker and Coordinate Input**. Here, you can also set Tracker colors, contents and behavior.

Some Tracker options are accessible from the pop-up arrow next to the Tracker toggle in the toolbar.



Element Input Using Tracker

When placing a new element, you can define its coordinates in the Tracker.

With the Tracker displayed, **do one of the following**:

- Start typing a value for the default parameter (the one shown in bold).
Note: This input method works if the “Enable direct input into Tracker fields” box is checked in **Options > Work Environment > Tracker and Coordinate Input**. This is the default setting.
- Press **Tab** as many times as needed to activate the required Tracker field, then type in the desired number.



- Click a field in the Tracker to activate it, then type in the desired number.

- Type a shortcut (**X**, **Y**, **Z** or **D/R** or **A**) to activate the corresponding Tracker field, immediately followed by the value. For example: **X3** or **Y-4**.

Notes:

- This input method works if the “Coordinate Shortcuts prefer Tracker” box is checked in **Options > Work Environment > Tracker and Coordinate Input**. This is the default setting.
- The **+** prefix is optional for positive values.
- You can type coordinate values in any order.
- If the Tracker is displayed in its compact form, typing the shortcut for a hidden parameter (or clicking Tab) will display it.

For another method of coordinate input, see [Coordinates Palette](#).

Finish input of Tracker value

- Checkmark to apply value
- Tab to apply value, then jump to next Tracker field
- Enter to apply value and exit Tracker
- Esc to cancel and exit Tracker

Finish element input

Press **Enter** or click the checkmark icon in the Tracker.

Display Snap Guides at Coordinate

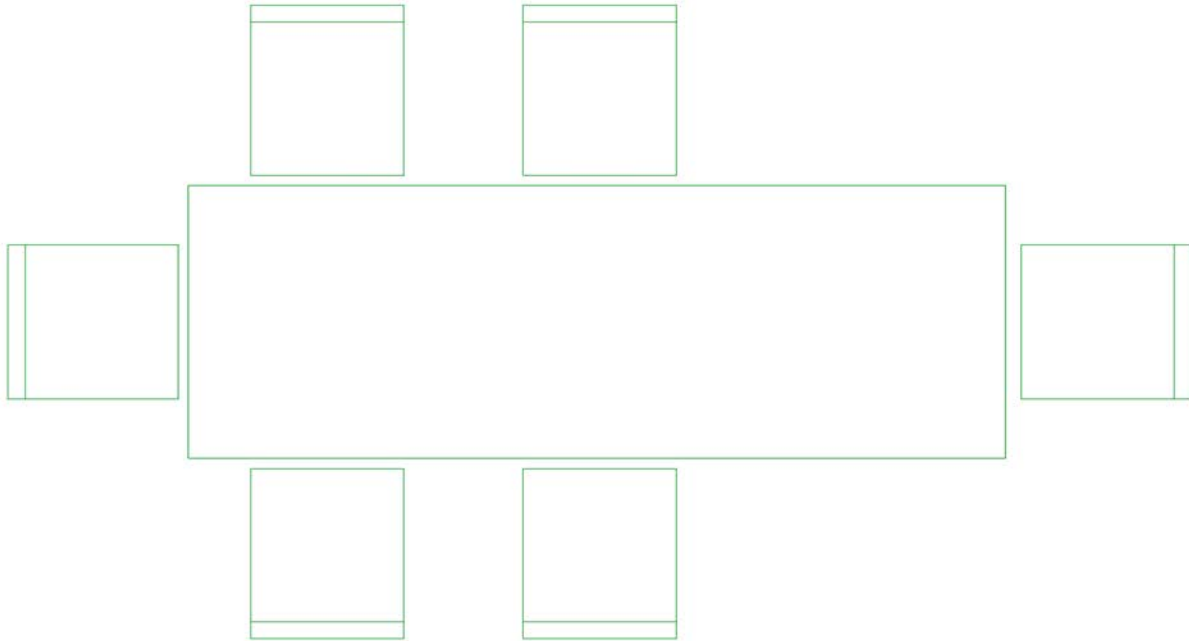
Click the crosshair icon in the Tracker. The edited element will then jump to that coordinate value and, if Snap Guides are active, the applicable Snap Guides will appear.

For more information, see [Pin or Place a Snap Guide Manually](#).

Use Math Expressions to Place Elements

Type the operator *after* the value. For example, typing $x3+$ will add 3 to the x value shown in the Tracker.

Operator	
20+	add 20
20-	subtract 20
3*	multiply by 3
3/	divide by 3

Example: Place a copy of a chair at twice the distance

User types 2 into Tracker's Distance field*

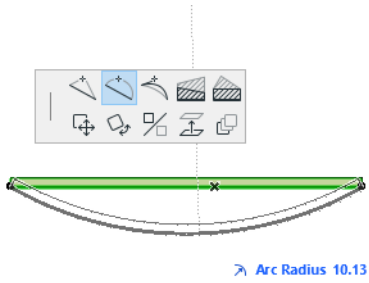
1. Select the first chair
2. Activate Move > Drag a Copy
3. Drag the copy to overlap the second chair. The Tracker shows this distance: 800.
4. In the Tracker, type 2* in the distance field
5. Press Enter. The chair copy is moved to a distance that is twice the distance between the first two chairs.

Notes for users of U.S. dimensions:

- When **3** is entered, Archicad assumes you mean three feet. Typing **3'6**, **3'6"**, or **3-6** will enter three feet, six inches.
- To divide or subtract in Tracker, press the slash or minus key twice, respectively (because pressing just once switches the input from feet to inches)

Parameters Shown in Tracker

The parameters appearing in the Tracker vary depending on the type of editing action that is underway. For example, if you select a straight segment and choose the Curve edge command from the pet palette, then the Tracker shows the "Arc Radius" parameter.



The Tracker always shows one of its parameters in **bold**; this is the default parameter.

Show or hide the names of the coordinates using the “Show/Hide Parameter Tags” from the Tracker pop-up menu.

Display XY Coordinates or Distance/Angle (Polar) Coordinates

During input, the Tracker displays either the XY or the DA (distance/angle) coordinate pair.

Switch from one to the other during input:

- Use the / (slash character) shortcut
- Click “Prefer XY Coordinates” or “Prefer Polar Coordinates” from the Tracker pop-up menu.



Expand Tracker Automatically

This option means that all the parameters will be shown as soon as you enter the Tracker (by entering a value or by pressing Tab).

Otherwise, you have to keep pressing tab to see additional parameters.



Coordinate Input Logic: Expert Options

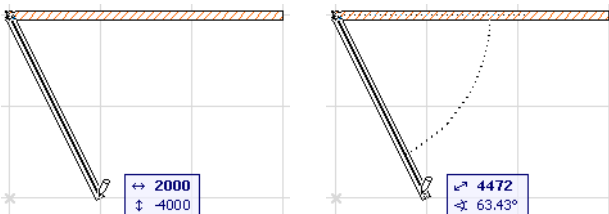
Special Tracker options are available to let you further customize the Tracker function, using the Coordinate Input Logic checkboxes in **Options > Work Environment > Tracker and Coordinate Input**.

[See Tracker and Coordinate Input.](#)

Origins of Coordinate Values in Tracker

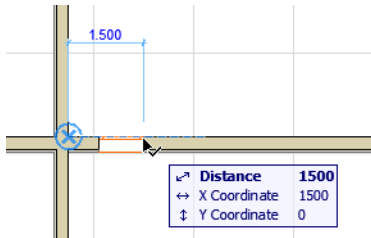
By default, the Tracker displays Relative coordinates.

- **Relative** Coordinates are measured from the **Edit Origin**.



Use the Tracker menu from the Standard Toolbar to toggle the **Relative Coordinates in Tracker** preference on or off.

When you snap to a Snap Guide during input, the User Origin is automatically relocated to the nearest Snap Reference point. A temporary blue X appears, indicating the User Origin for the current input.



- **Absolute** coordinates are measured from the **User Origin**. (If you haven't reset the User Origin, this is the same as the Project Origin). The Tracker displays Absolute coordinates if you have turned off the "Relative Coordinates in Tracker" toggle.

See [Origins](#).

A temporary dashed line is drawn between the origin and the cursor to indicate that you are using absolute coordinates.

Related Topics:

[Elevation Values in the Tracker](#)

Ruler

The Ruler is an editing aid that can be placed at the top and left side of the screen in every Archicad window, except the 3D Window and Schedules.

Use the Ruler as an aid in navigation: it lets you know where you are in the project, at the current zoom.

Show/Hide Ruler

Use the **View > Ruler** toggle to show or hide the Ruler.

You can also hide a ruler by right-clicking it and clicking the Ruler toggle.

As you move the cursor on screen, feedback on the ruler shows you the vertical and horizontal position of the cursor relative to the User Origin (or, if no User Origin has been created, relative to the Project Origin).

[See *Origins*.](#)

Every time you create a new User Origin, the measurements shown on the Ruler change accordingly. The measurement units are the Model Units you set in **Options > Project Preferences > Working Units**.

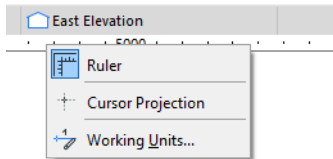
Place Orthogonal Guide Line Parallel to Ruler

At any time, click on the Ruler, then drag your cursor to anywhere in the window: a Guide Line parallel to the Ruler will be created.

[See also *Guide Lines*.](#)

Ruler Commands

The Ruler's context menu (right-click anywhere on the Ruler) gives you a quick way to access related commands:



- Click the **Ruler** toggle to hide the Ruler
- Click **Cursor Projection** toggle to include projection lines from the Ruler to the cursor position. (This will also turn on Cursor Projection in the 3D window.)
- Click **Options > Project Preferences > Working Units** to access this dialog box, where the Model Unit option determines the Ruler's measurement units.

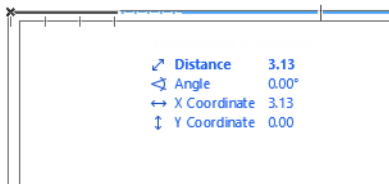
Measure Tool

The Measure tool, located in the Standard toolbar, allows you to measure distances and areas in the project.

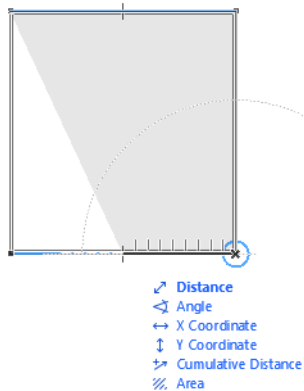


It is a temporary, on-screen function for quick feedback.

To use the Measure Tool, activate its icon in the Standard Toolbar, or use the M shortcut. Click on a starting point (the top left corner of the rectangular wall, in the image below) and start to trace the wall. The Tracker displays the horizontal, vertical and radial distance measured from the starting point, as well as the angle.



You can Measure any number of segments in sequence: each click represents a new starting-point from which the next distance will be measured. After two or more clicks, the Tracker also shows a cumulative distance and the area (shown by a temporary gray area on screen). The image below shows the Measure tool feedback after three clicks, as you approach the last corner of the rectangular wall.



You can also use the Measure tool while creating elements. When you start drawing a Wall, for example, you can activate the Measure tool on the fly, measure (or even edit) a value with the fields it displays and then hit Enter to transfer this value into the appropriate field of the Tracker.

Set up Construction Grid

Grids can be defined separately for the Floor Plan and any individual Section/Elevation/IE, 3D Document, Detail/Worksheet, or Layout window. Note that, in Floor Plan window, the same Grid settings will be used for all Stories.

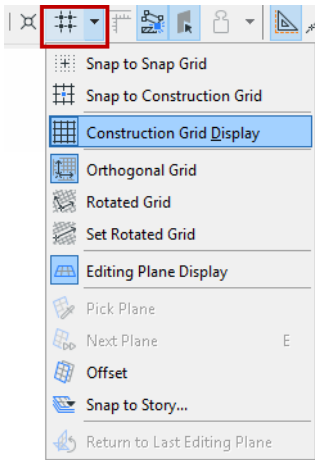
Note: You can also create, place and use custom local structural grids of different sizes and shapes using the Grid Tool.

See [The Grid Tool](#).

Show/Hide Construction Grid

Do one of the following:

- Toggle the **View > Construction Grid Display** command
- Use the Standard Toolbar's Grid pop-up menu

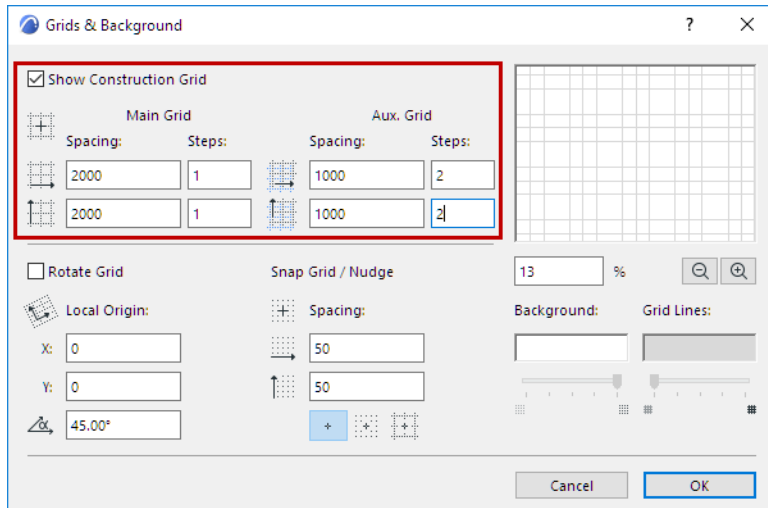


- Open **View > Grid & Editing Plane Options > Grids & Background**, and check **Show Construction Grid**.

Define Construction Grid and Auxiliary Grid

1. Open **View > Grid & Editing Plane Options > Grids & Background**.
2. Enter **Spacing** values to define the horizontal and vertical distances between grid lines.

Note: Spacing values also define the Nudge distance: use the Nudge shortcut to move elements by a predefined factor. (use Alt + Shift + an arrow key to move the element left, right, up or down). See [Nudging Elements](#).



Add Optional Auxiliary Grid

The Auxiliary Grid can be used if you want to create periodic or stepped grids. For example, to indicate the thickness of concrete walls or foundation footings.

To use it, set separate Spacing and Steps values for the Main Grid and the Auxiliary Grid.

Step values define the number of times to display the grid line before displaying the Auxiliary grid line, in both the horizontal and vertical directions. If the “Steps” value for any of these grid lines is 0, those lines will not be displayed.

In this example, you display one main grid line (spacing 2000), followed by two Auxiliary grid lines (spacing 1000); then the pattern repeats.

Note: All the values in this dialog box are based on the current measurement units defined in **Options > Project Preferences > Working Units**.

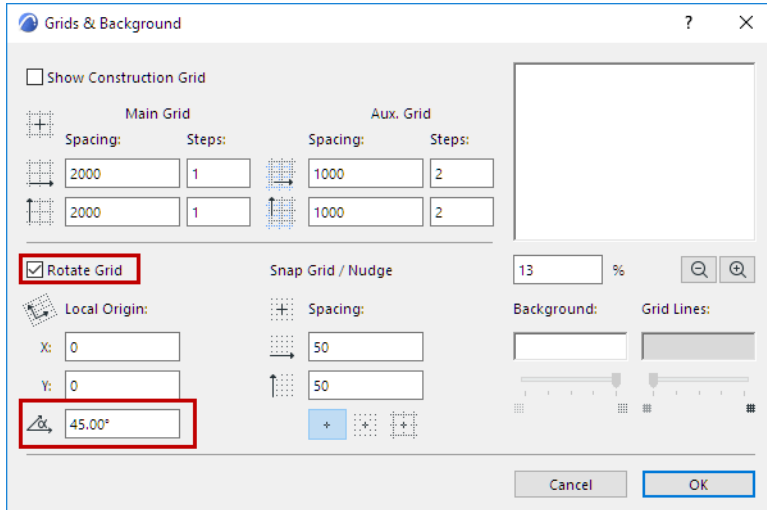
View the **preview** of the Construction grid (both the Main grid lines and Auxiliary grid lines). Use the percentage and **Zoom In/Zoom Out** controls (below the preview window) for zooming in the preview window.

Note: The grid is automatically hidden if it would be too dense on the screen, that is, if the smallest distance between two adjacent grid lines would be smaller than 3 pixels.

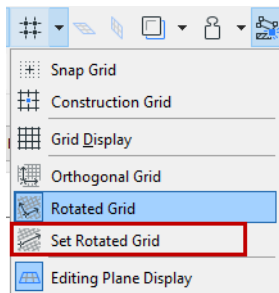
Rotate Construction Grid

1. Go to the **View > Grid and Editing Plane Options > Grids and Background** dialog box.
2. Check the **Rotate Grid** box.
3. Specify X and Y values to specify a **Local Origin** that is different from the Project Origin.

4. Define a Rotation Angle.



To define a rotated grid by hand: first, choose **Set Rotated Grid** from the Standard Toolbar, then draw a vector on screen or click on an existing straight segment: this vector will represent the rotated grid angle relative to the horizontal axis.



To display the rotated Construction Grid, make sure the Construction Grid is visible, then activate **Rotated Grid** button from one of the following locations:

- View > Grid & Editing Plane Options > Rotated Grid
- Use the Standard Toolbar's Grid pop-up menu
- Open **View > Grid & Editing Plane Options > Grids & Background**, and check **Rotate Grid**.

Related Topics:

[Set up Snap Grid](#)

[Snap to Grid](#)

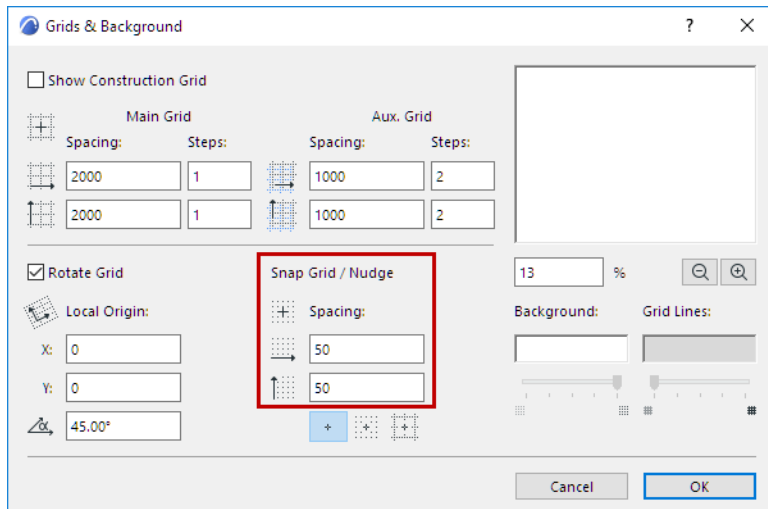
[Set Window Background and Grid Line Color](#)

Set up Snap Grid

The **Snap Grid** is an invisible grid that you can use optionally to make input easier, by having the cursor snap to a predefined distances.

To define a Snap Grid:

1. Open **View > Grid & Editing Plane Options > Grids & Background**.
2. Use the controls under Snap Grid/Nudge to define the intervals (horizontal and vertical directions) to which the cursor will snap when Snap Grid is activated.



You will usually enter the smallest unit of measurement relevant to your Project. You can set different horizontal and vertical Snap Grid increments, allowing you to reflect the unique geometry of your Project.

Note: The Snap increments you define here can also be used for the **Nudge** shortcut - to easily move any element by a small defined factor, using a keyboard shortcut. [See *Nudging Elements*](#).

Related Topics:

[Set up Construction Grid](#)

[Snap to Grid](#)

[Set Window Background and Grid Line Color](#)

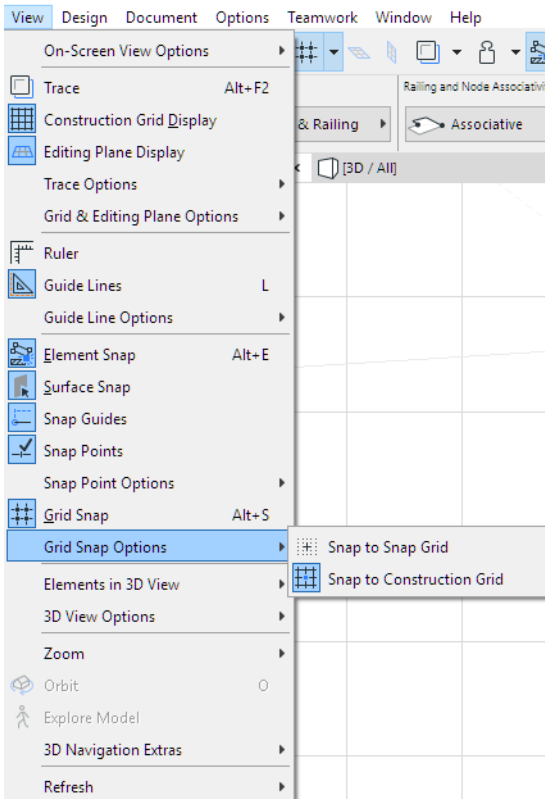
Snap to Grid

The **Grid Snap** function (Alt+S) assists in the precise graphic alignment of elements, even at low zoom levels. When grid snap is active, the cursor always snaps to the nodes of **either the Construction Grid or the Snap Grid**.

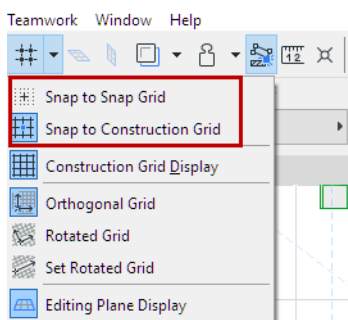
Activate Grid Snap

Do one of the following:

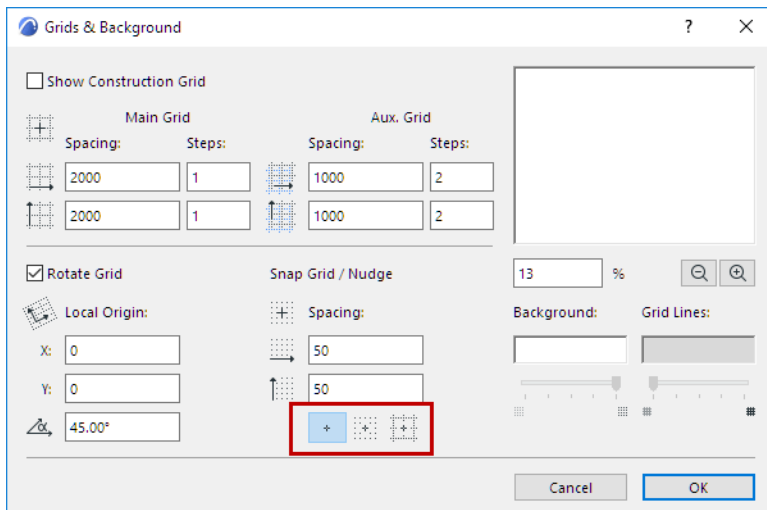
- From **View > Grid Snap Options**: Select **Snap to Snap Grid** or **Snap to Construction Grid**.



- Use **Snap to Snap Grid** or **Snap to Construction Grid** from the grid menu in the Standard Toolbar, or from the Coordinates Box.



- Select one of the three Snap buttons at **View > Grid & Editing Plane Options > Grids & Background**.



- No snap
- Snap to Snap Grid
- Snap to Construction Grid

Use the **Shift+S** shortcut to toggle among these three options.

- Use the **View > Grid Snap** menu command. The Snap function will use the last grid type used, either Construction or Snap grid. (Shortcut: **Alt+S**)

Deactivate Grid Snap

Do one of the following:

- Use the **Alt+S** shortcut key
- Use the **View > Grid Snap** menu command (deactivate the toggle).

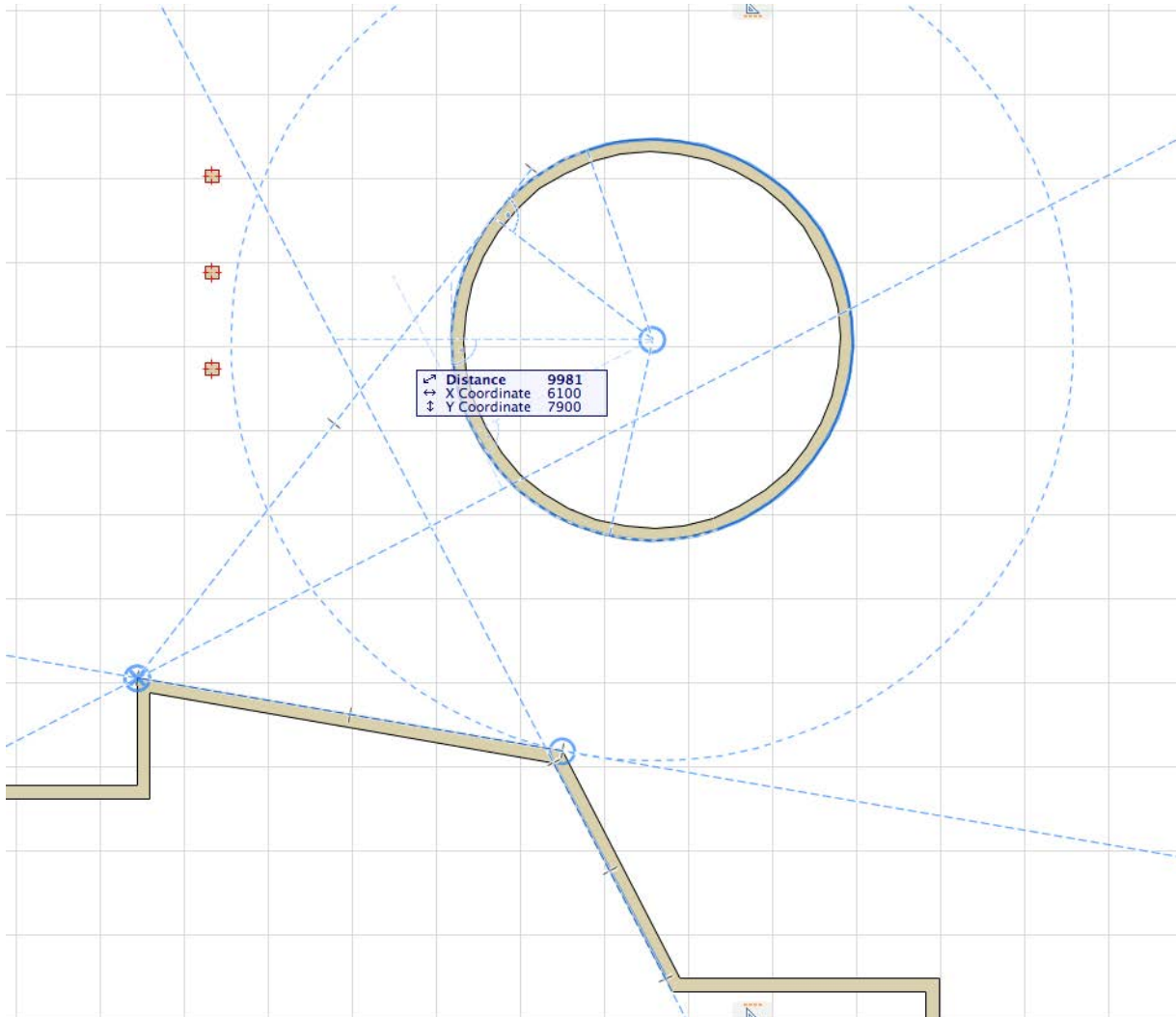
Related Topics:

[Set up Construction Grid](#)

[Set up Snap Grid](#)

Snap Guides

Snap Guides are temporary lines and arcs that appear along existing elements, to assist accurate element placement.



Unlike Guide Lines, Snap Guides are temporary and disappear automatically when the cursor is no longer in the vicinity. However, you can make a Snap Guide remain in place by pinning it.

You can also control which Snap Guides appear by marking particular points or edges as **Snap References**.

See the sections below on how to use Snap Guides:

Lock Input to Snap Guide

To constrain your input to the desired Snap Guide, do one of the following:

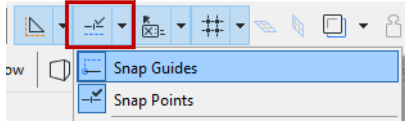
- Press **Shift** during input
- From the context menu, use **Lock to Guide Line/Snap Guide**

Turn Snap Guide Function On/Off

To activate/deactivate Snap Guides, do one of the following:

- Click the Snap Guides and Points toggle from the Standard Toolbar

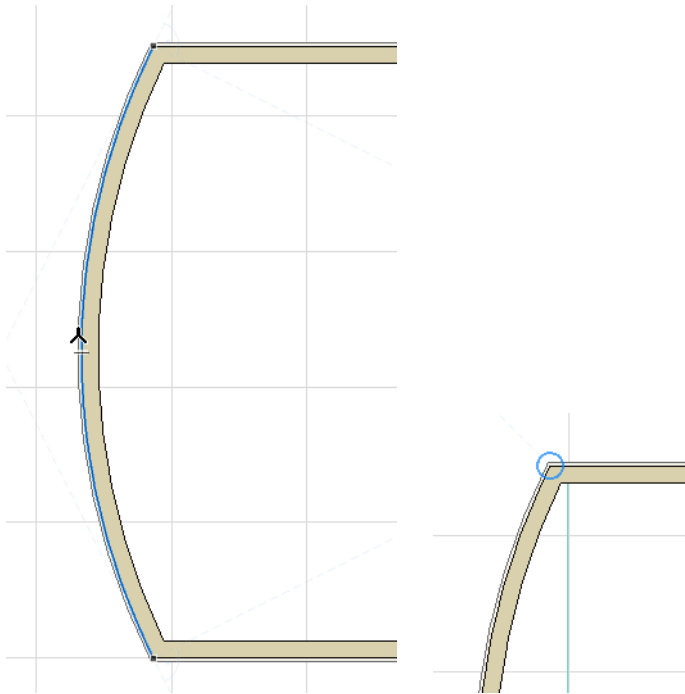
- Use the **Snap Guides** command in the toolbar dropdown
- Use **View > Snap Guides**



Activate Snap References

To make Snap Guides appear in more places that are relevant to your input, you can define any edges or points as **Snap References**.

Activate a Snap Reference (a dynamically appearing blue highlight) by hovering the cursor over element edges or points:



You can activate any number of Snap References simultaneously. Once you have defined Snap References, you will see additional Snap Guides appear which are relevant to those References as well as your current input vector.

[See Appearing Snap Guides.](#)

Again, to lock your input vector to any Snap Guide, press **Shift**.

Snap References stay in place, even if you move the cursor away, until you do one of the following:

- Complete input
- Click ESC
- Hover over the existing Snap Reference until it disappears, or click Q ([see Force Snap Reference \(Shortcut\), below](#)).

[Watch the Video](#)

[Watch the Video](#)

Force Snap Reference (Shortcut)

Click **Q** on any edge or point to “Force Snap Reference”, to highlight an edge or node as a Snap Reference instantly (regardless of the timer set in the Work Environment).

Note: The predefined Shortcut for this command is the **Q** key, but may vary depending on your localized version of Archicad.

Appearing Snap Guides

During input, Snap Guides will pop up automatically and temporarily.

Which Snap Guides will appear?

- Snap Guides which are relative to the location of your cursor (e.g. Horizontal/Vertical, Parallel to Grid, a fixed Incremental Angle), depending on the options set at **Options > Work Environment > Input Constraints and Guides**.
- Snap Guides relative to any **Snap References** you have marked, and to the relations among those Snap References.

Where applicable, icons on the Snap Guide indicate the relation:

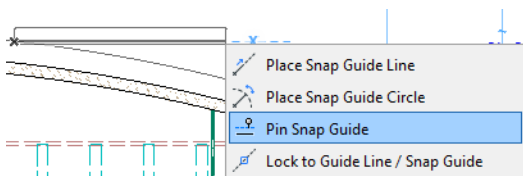
- Tangent
- Parallel
- Perpendicular
- Angle Bisector (if applicable)
- The line connecting two circles' centerpoints (if applicable)

Pin or Place a Snap Guide Manually

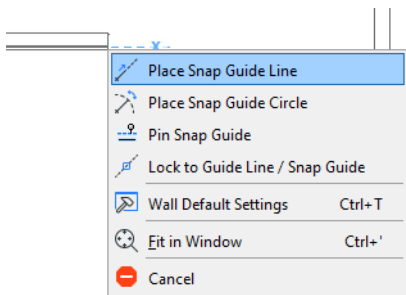
Unlike the automatically appearing Snap Guides, a pinned or explicitly placed Snap Guide is infinitely long and will remain visible, without fading out, until input is complete - even if you move the mouse away.

To pin a Snap Guide so that it does not disappear until input is complete, or to place one manually, do one of the following:

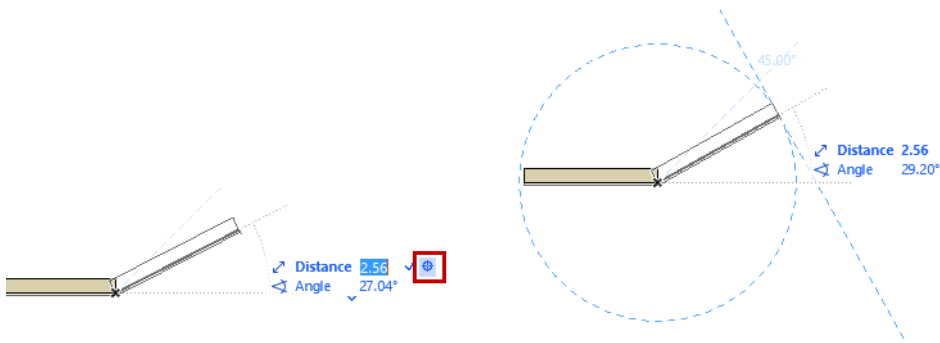
- From the context menu of an existing Snap Guide, use the **Pin Snap Guide** command:



- From the context menu during element input, use the **Place Snap Guide/Snap Circle** command:



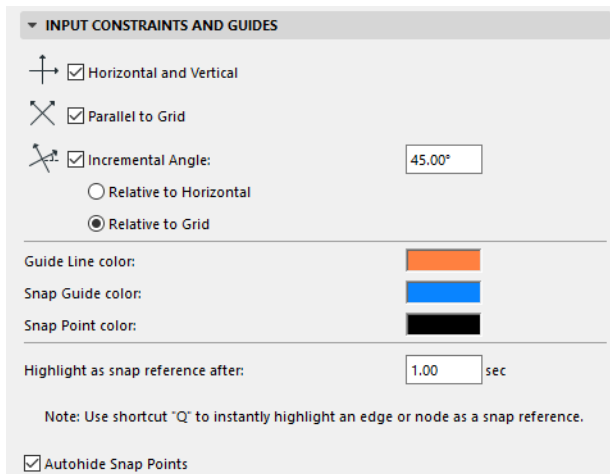
- During **Tracker** input, click the crosshair icon in the coordinate field. The edited element will then jump to that coordinate value and the applicable Snap Guides will appear.



Options for Snap Guides and Snap References

Options for Snap Guides/References are set at: **Options > Work Environment > Input Constraints and Guides.**

The color of the Snap Reference is the same as the Snap Guide color.



Guide Lines

Guide Lines (and Guide Circles) are drafting aids which help you find special points and projections, to ensure precise input in both 2D and 3D windows.

Unlike Snap Guides, Guide Lines are explicitly placed by you. They are always “permanent”, that is, they remain in place until you remove them.

Turn Guide Line Display on or off

Do one of the following:

- Use the toggle button in the Standard toolbar



- Use a shortcut key to toggle Guide Lines on and off

Note: The predefined Shortcut for this command is the **L** key, but may vary depending on your localized version of Archicad.

- Use the **View > Guide Lines** toggle

Lock Input to Guide Line

Once you have placed Guide Lines, use them to assist in precise element creation. To constrain your input to the desired Guide Line, do one of the following:

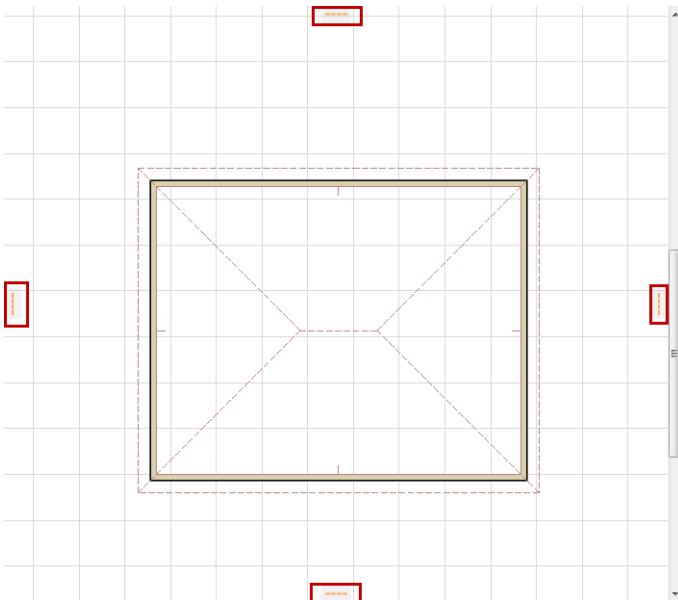
- Press **Shift** during input
- From the context menu, use **Lock to Guide Line/Snap Guide**

Place Guide Lines

1. Make sure Guide Lines are On.

See [Turn Guide Line Display on or off](#), above.

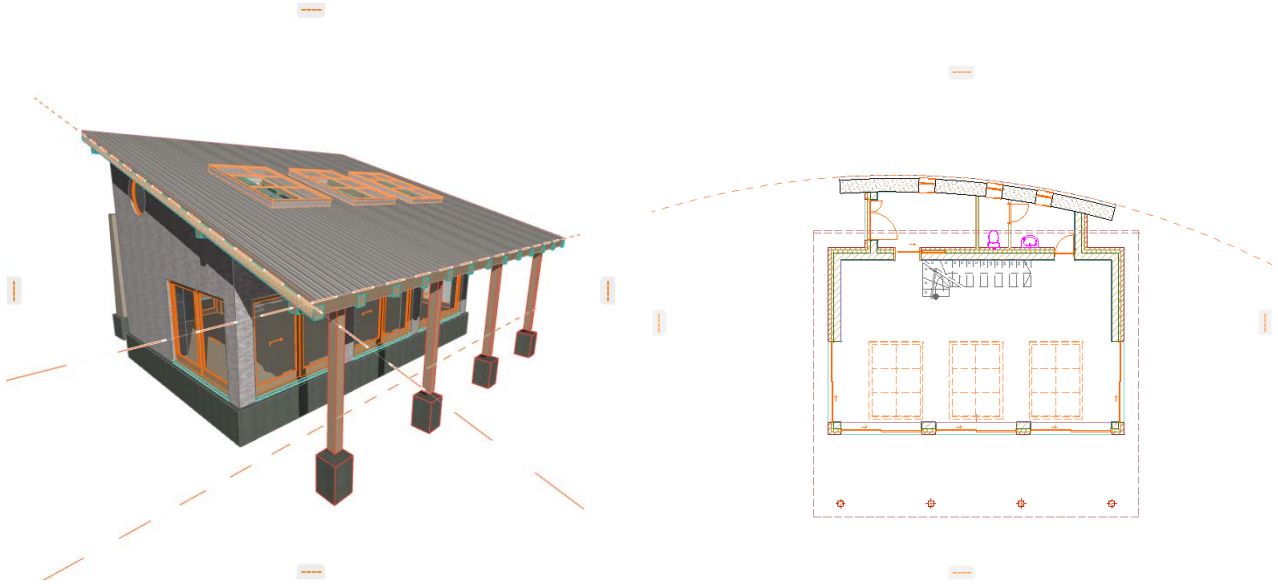
2. The window will display Guide Line tabs on all four sides of the screen.



3. Click any tab, then drag the appearing Guide Line to any location in the viewpoint.

4. Let go to place the Guide Line.

- If you place it along any existing **edge**, the Guide Line snaps to that edge.
- If you place it on a **curved element**, a corresponding Guide Circle appears.
- If you place it at a **point**, the Guide Line is placed at the corresponding main direction.
- You can also place it without snapping it to any existing element.



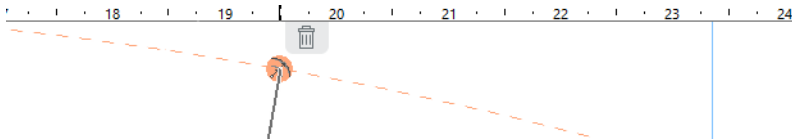
Other Ways to Create Guide Lines

- It is possible to draw a straight or curved Guide Line segment anywhere in the window, with two clicks. [See *Create Guide Line Segment*](#).
- If you are using the **Ruler**, click on the Ruler, then drag your cursor to anywhere in the window: a Guide Line parallel to the Ruler will be created. [See *Ruler*](#).

Cancel Guide Line Placement

During Guide Line placement, do one of the following:

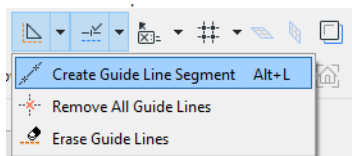
- Press **Esc**
- Drag the Guide Line back to any Guide Line tab, which is now shown as a garbage can.



Create Guide Line Segment

Use this command to create a Guide Line segment anywhere with two clicks.

1. Right-click any Guide Line tab, or click the Guide Lines pop-up menu from the Standard Toolbar.
2. Choose **Create Guide Line Segment**.

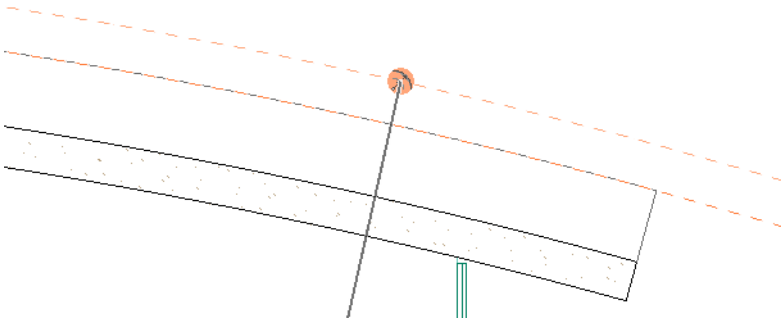


Note: This command is also available at **View > Guide Line Options**.

3. A pet palette appears. Choose a preferred line segment option, then draw the Guide Line or Guide Circle (arc).

Drag/Offset Guide Lines or Guide Circles

1. Hover over a Guide Line until its handle (the orange dot) appears.
2. Click-and-drag the handle to drag the Guide Line to a new location. (Or use the Tracker to define its new location numerically.)



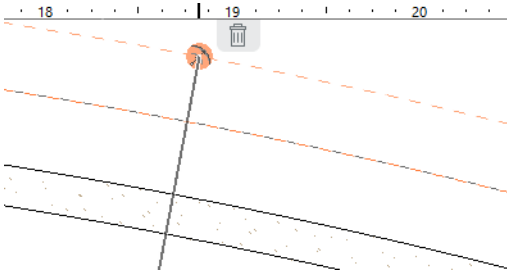
With a Guide Circle, this operation becomes a radial stretch (as also indicated by the icon).

If you drag the Guide Circle onto a linear edge, it is transformed into a straight Guide Line, and vice versa.

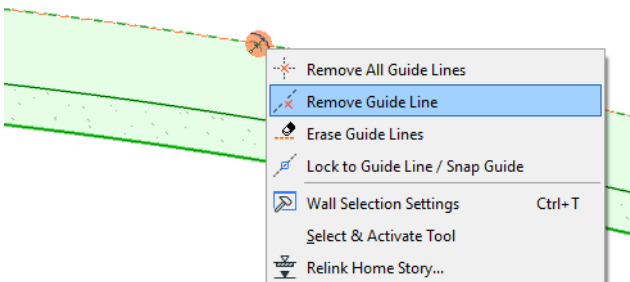
Remove Single Guide Line

Hover over a Guide Line until its handle (the orange dot) appears, then do one of the following:

- With the cursor over the Guide Line handle, press **Esc**.
- Click-and-drag the handle to drag the Guide Line to the nearest garbage can.



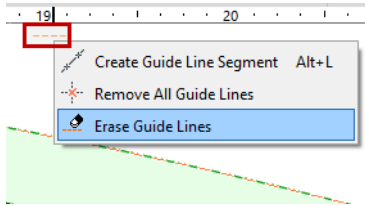
- Right-click, then choose **Remove Guide Line** from the context menu.



Erase Guide Lines

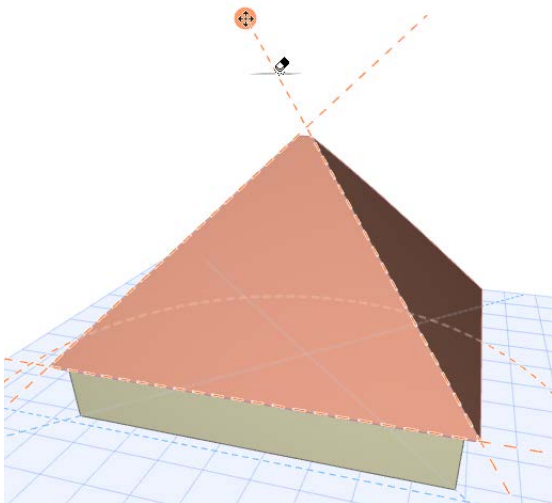
Use this command to remove several Guide Lines in a single operation:

1. Right-click any Guide Line tab in the window, or click the Guide Lines pop-up menu from the Standard Toolbar.
2. Choose **Erase Guide Lines**.

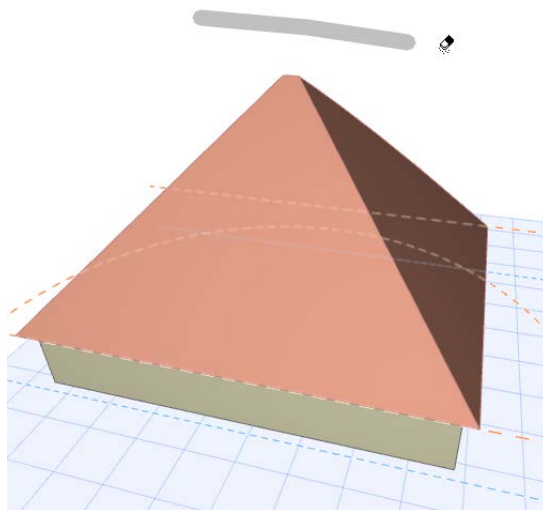
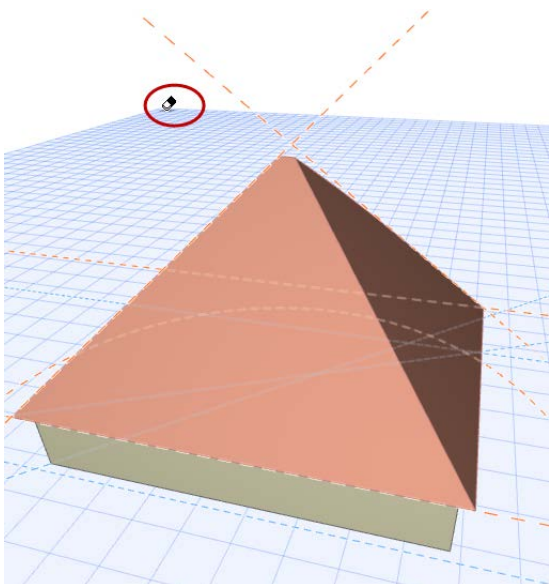


Note: This command is also available at **View > Guide Line Options**.

3. The Erase cursor appears.
4. Click anywhere on any Guide Line, one after the other, to erase them instantly.



Alternative, drag the Eraser cursor across the screen. Any Guide Line it touches is erased instantly.



5. When you are done, press **Esc** or double-click to exit Erase mode.

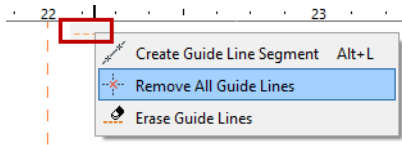
Remove All Guide Lines

This command removes all Guide Lines from ALL viewpoints in the project.

Do one of the following:

- Right-click any **Guide Line tab**, or click the Guide Lines pop-up menu from the Standard Toolbar. Then choose **Remove All Guide Lines**.

Note: This command is also available at **View > Guide Line Options**.



- Right-click on any single Guide Line to open the context menu. Then choose **Remove all Guide Lines**.

Guide Line Color

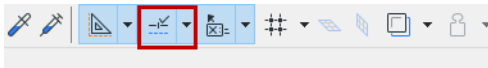
Set the Guide Line color at **Options > Work Environment > Input Constraints and Guides**.

Snap Points

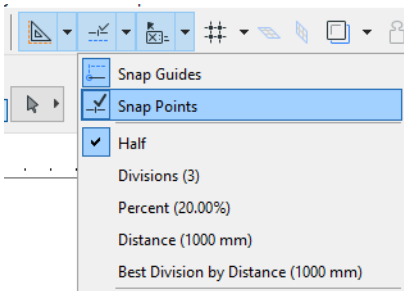
Snap Points are temporary hotspots that appear at particular intervals (defined by you) on elements, to help you place and draw new elements more accurately.

Activate Snap Points

1. Do one of the following:
 - Activate the **Snap Guides and Points** toggle on the Standard toolbar



- Turn on **Snap Points** from the Snap Guides and Points drop-down menu



- Click **View > Snap Points**

Using Snap Points

Hover your cursor over the element on which you want to see the Snap Points.

The Snap Points appear. Their number and position depend on the options set by you.

[See *Define Snap Point Intervals*, below.](#)

The Snap Points act as a temporary hotspot - you can snap to them with your cursor, and they also serve as Snap Reference points. ([See *Activate Snap References*](#).) You can start drawing new elements from these points or use them for editing other elements in relation to them.

Snap Points will disappear after about 5 seconds. However, if they are located along a Snap Reference segment, they will remain visible as long as that Reference does.

Note: To have the snap points remain visible until you either place an element or change views: disable the “Autohide Snap points” checkbox in **Options > Work Environment > Input Constraints and Guides**.

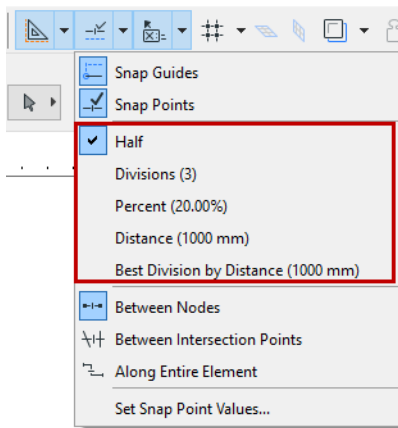
Another way to use snap points is to draw a temporary vector that displays the defined Snap Points.

[See *Snap Points on Temporary Vector*.](#)

Define Snap Point Intervals

Click the Snap Guides and Points drop-down menu from the Standard Toolbar.

The same commands are available from **View > Snap Point Options** menu, or from the Control Box.



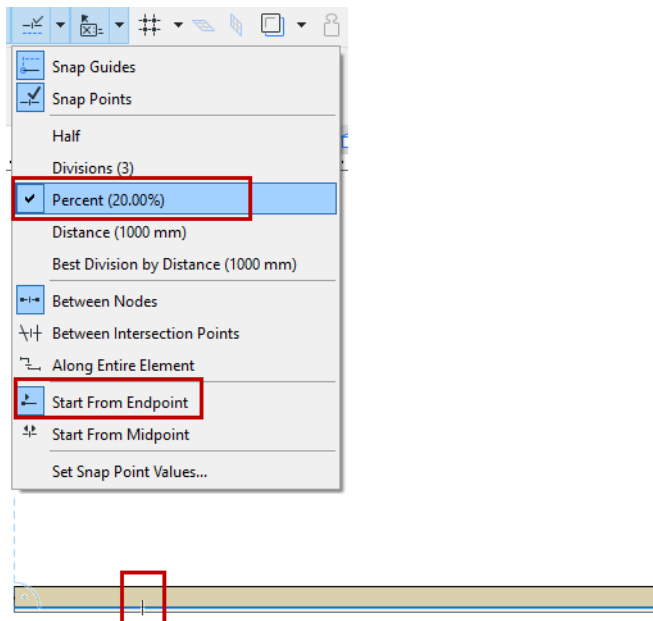
- **Half:** Divides an edge into two equal parts.
- **Divisions:** Divides an edge into the set number of segments. Range: 3-100.
- **Percent:** Divides an edge into two parts according to the set percentage.
- **Distance:** Divides an edge into segments that are the set distance apart.
- **Best Division by Distance:** Snap points will divide the edge into equal parts that are as close as possible in length to the distance defined here.

Note: Define your preferred divisions, percentage, etc. in the **Snap Point Values** Dialog Box.

See [Snap Point Values](#), below.

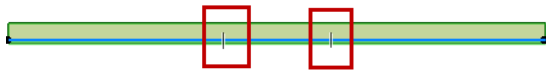
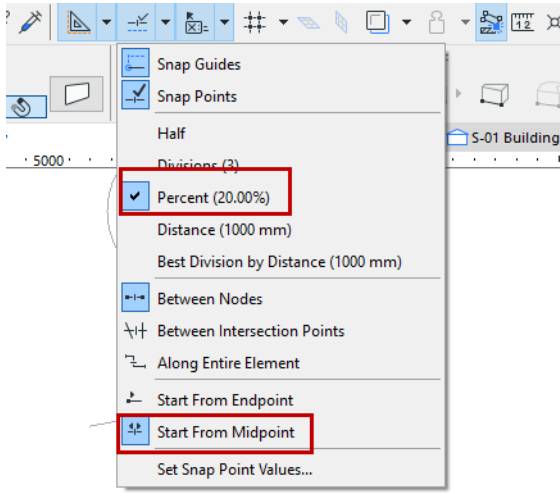
When using the Percent or Distance interval, you have two further options:

- **Start from Endpoint:** the Snap point is placed at the given percent/distance as measured from the element's endpoint that is closest to the cursor.



- **Start from Midpoint:** the Snap point is placed at the given percent/distance as measured from the element's midpoint.

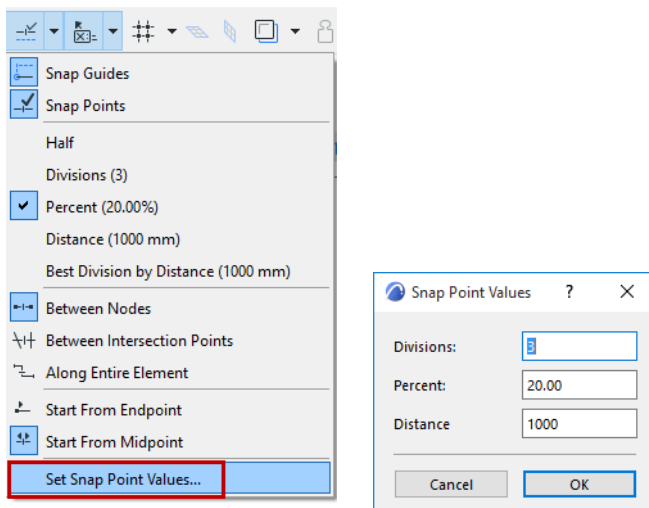
In either case, feedback shows starting point from which the element is marked.



Snap Point Values

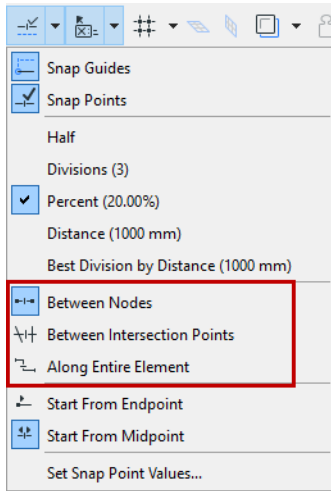
From the Snap Points menu in the Standard Toolbar (or **View > Snap Point Options**), click the **Set Snap Values** command.

In the appearing dialog box, define the values for your divisions, percentage, and/or distance.

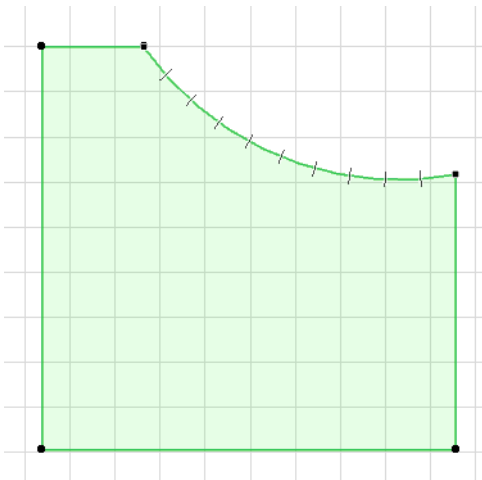


Snap Points Options

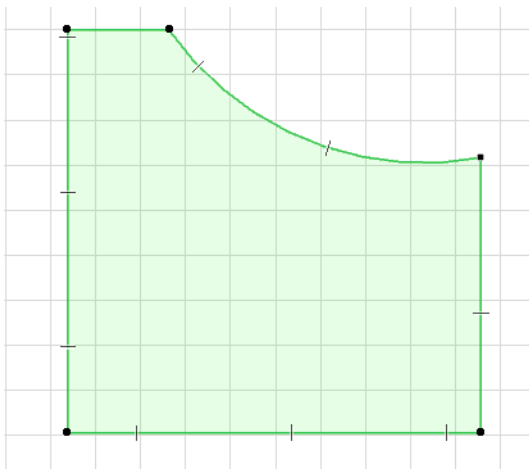
From the Snap Points menu in the Standard Toolbar (or the View menu), choose one of three options for setting your Snap Points:



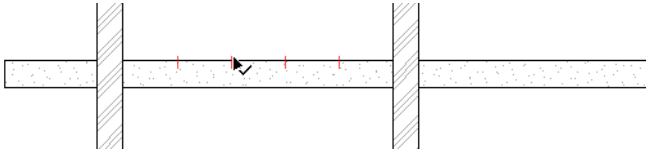
- **Between Nodes:** Snap points appear only on the chosen segment (between the neighboring nodes).



- **Along Entire Element:** Snap points appear at the defined intervals along the whole element (including polygon elements), regardless of any intersecting elements.



- **Between Intersection Points:** Snap points are placed on the chosen element only from one intersection point to the next.



Snap Point Color

To customize the color of your Snap Points, use the control in **Options > Work Environment > Input Constraints and Guides**.

Snap Points on Temporary Vector

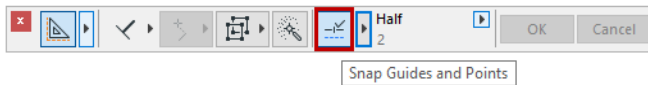
You can display Snap Points on a temporary vector used while drawing elements. The method can be activated with the Snap Point icon among the relative construction icons in the Control Box.



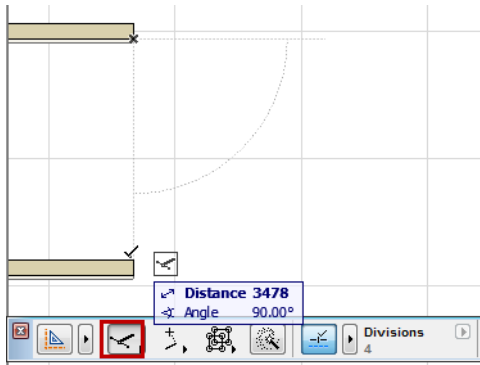
[See Control Box.](#)

In the following example, you will draw a Wall that starts at one-fourth of the gap between two endpoints of two other Walls.

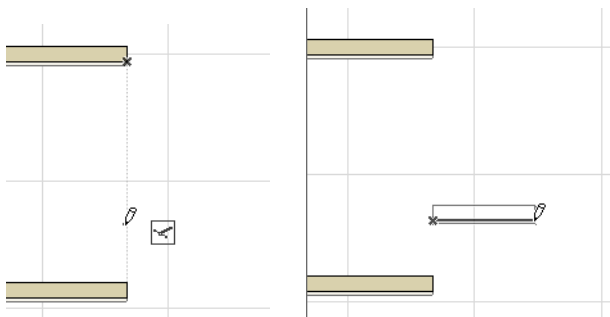
1. Start by choosing the Divisions option in Snap Point Settings, then type **4** as the number of Divisions.



2. Make sure that Grid Snap is inactive. (**View > Grid Snap** is off.)
3. Activate the Wall tool.
4. Choose the Special Snap Point icon in the Control Box, and click it to activate.
5. Draw reference vector from one Wall endpoint to another.



6. Snap Points are now available along the vector, at each of four divisions, as defined in the Control Box. Click at the desired location to start drawing the new Wall.



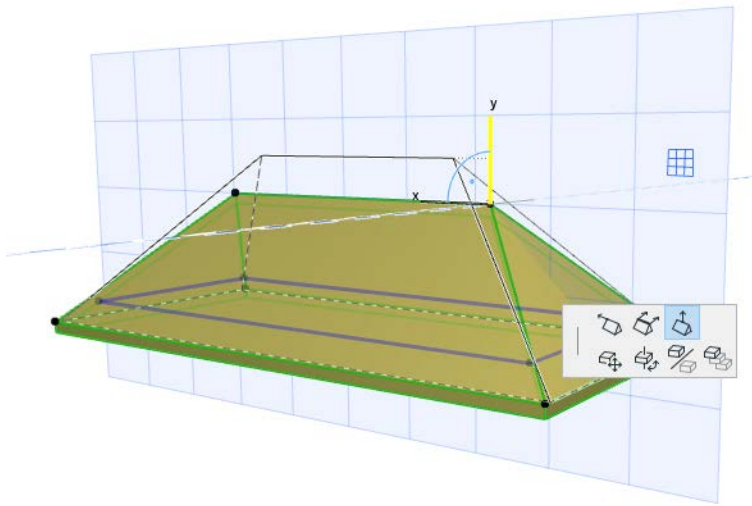
Editing Plane in 3D Window

When you open the 3D window, an editing plane automatically appears: new elements will be placed onto this editing plane.

If you begin element input by snapping to an existing element hotspot or surface, the editing plane will automatically move to that position.

You can reposition the editing plane manually at any time.

During editing, the editing plane may adjust itself automatically. For example, as you drag this roof ridge, the editing plane automatically becomes the vertical plane on which you drag the cursor.



Topics in this section:

[Editing Plane Display](#)

[Reposition Editing Plane](#)

Editing Plane Display

The Editing Plane is shown by default in the 3D window. To hide it, use the toggle at **View > Editing Plane Display**.

Note: Regardless of the state of this toggle, the Editing Plane will be temporarily displayed whenever necessary: for example, after you choose a plane for Ruled Shell input.

Display Editing Plane During Editing Operations Only

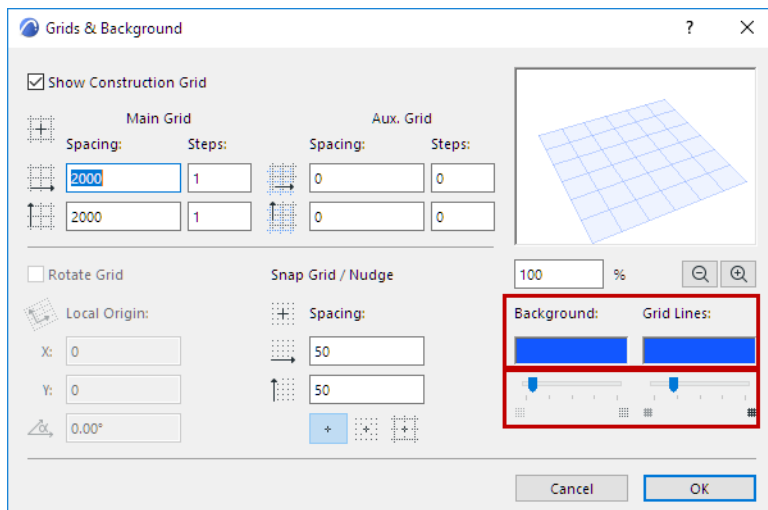
1. Go to **Options > Work Environment > On-Screen Options**.
2. Check the box to **Display Editing Plane during Editing Operations Only**.

If you check this box, the Editing Plane will appear only during the time you are actively editing in the 3D window (provided that the “Editing Plane Display” toggle is turned on.) When you are done with the operation, the Editing Plane disappears.

3D Editing Plane: Color and Grid

The 3D Editing Plane has a grid system that is the same as the one on the Floor Plan, and reflects the X-Y coordinate plane.

1. Activate the 3D window.
2. Go to **View > Grid and Editing Plane Options > Grids and Background** to set up your grid and color preferences.



Reposition Editing Plane

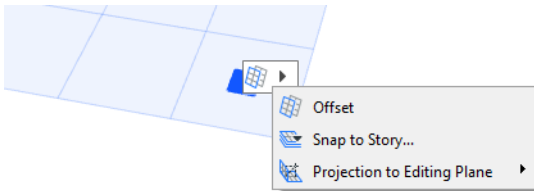
Use Editing Plane commands to change the default editing plane position, either before or after beginning element input. The available commands vary depending on your editing situation.

Note: The **Offset** and **Snap to Story** commands are available only before you begin input.

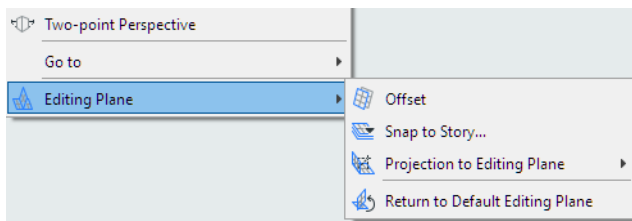
Where To Find Editing Plane Commands

To reposition the editing plane in the 3D window, either before or during editing, use the commands from one of the following places:

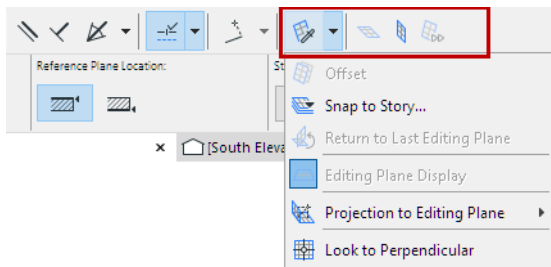
- Click the square-shaped editing plane “Grip”.



- Right-click to bring up a Context menu, then choose **Editing Plane**.



- Access the commands from the Drafting Aids Toolbar.



- Access the commands from **View > Grid and Editing Plane Options**.

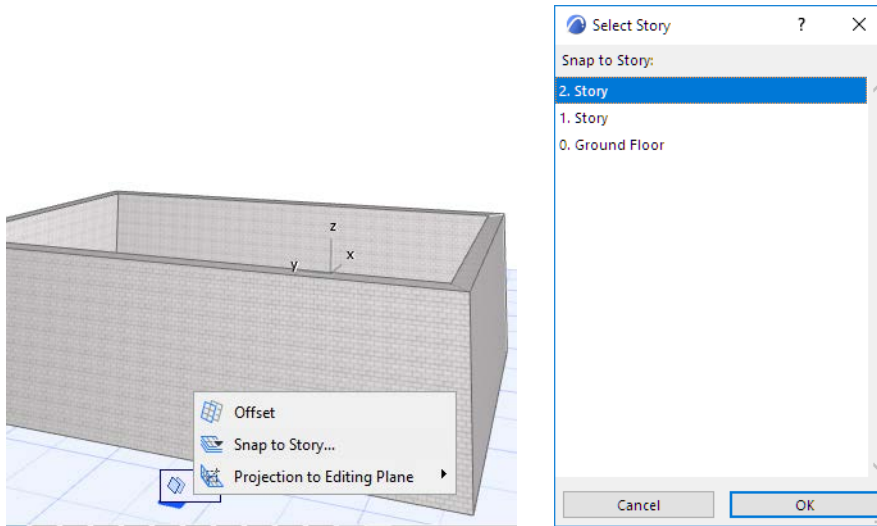
Snap Editing Plane to Story

Use this command prior to element input.

To snap the editing plane to a specific level corresponding to a project story:

- Bring up the Editing Plane commands.
- Choose **Snap to Story**.

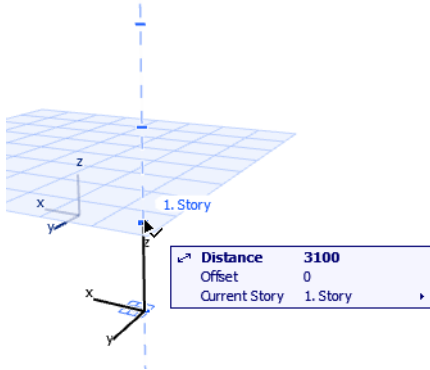
- From the appearing dialog box, choose the project story to which you want to move the editing plane.



Offset Editing Plane

Use this command prior to element input.

- Bring up the Editing Plane commands.
- Choose **Offset**.
- Use the cursor to drag the editing plane to the desired position, or define the offset distance numerically. Notice that a Snap Guide appears marking the story positions, to make it easier to place the editing plane.



Other Editing Plane Commands

Choose one of the options for your preferred editing plane orientation (the available options vary depending on the orientation of the current editing plane).

- **Next Plane:** Click this to switch to the next editing plane that makes sense in the editing context.
- **Pick Plane.**

[See Pick Plane.](#)

- **Horizontal:** Rotates the editing plane to a horizontal position.
- **Vertical:** Rotates the editing plane to a vertical position.
- **Offset.**

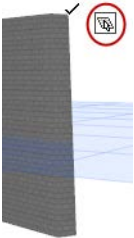
[See Offset Editing Plane.](#)

- **Look to Perpendicular:** Sets up the 3D window so that it is perpendicular to the Editing Plane.

- **Projection to Editing Plane.**
[See *Cursor Projection onto Editing Plane.*](#)
- **Lock to Plane.**
[See *Lock to Plane.*](#)

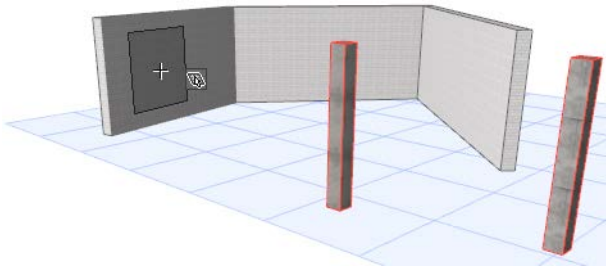
Pick Plane

Once you click this command, the gray rectangle “plane chooser” feedback appears.



Do one of the following to pick the desired plane for the editing plane:

- Move the cursor onto any plane of an existing element. A rectangle will help you see the plane.



- Click on any three points in space (such as three element hotspots) to define a plane.
- Click on an element edge plus another point (in either order) to define a plane.

Cursor Projection onto Editing Plane

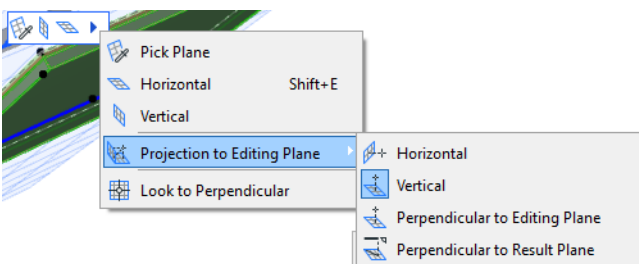
By default, as you perform graphical editing, your cursor is moving on the editing plane. If needed, you can redefine the direction of this cursor projection.

To change the cursor projection with respect to the editing plane:

1. Begin the editing operation.
2. From the context menu, select **Projection to Editing Plane**.

Alternatively, click on the **Editing Plane Grip** to bring up the same commands.

3. Choose one of the projection options:



- Vertical
- Horizontal

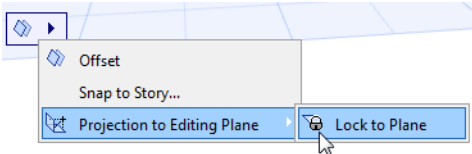
- Perpendicular to Editing Plane
- Perpendicular to Result Plane

Lock to Plane

Use this command lock input to the existing editing plane before you start creating the new element.

This is useful, for example, if you wish to create an element on the existing editing plane, by tracing the shape of an existing element (e.g. roof) on another plane.

1. Bring up the Editing Plane commands.
2. From the Projection to Editing Plane sub-menu, choose the **Lock to Plane** toggle.



The new element will be placed onto the editing plane, even if your cursor is on another plane.

Magic Wand

The Magic Wand (shortcut: Spacebar) saves you work by finding and tracing a linear or polygon shape from among existing elements, and then generating a new element based on that polygon.

- If you click on an edge or node, the Magic Wand traces the element and/or element chain.
- If you click in empty space, or on a surface, the Magic Wand traces the closed geometric area formed by the nearest elements.

Related Topics:

[How to Create an Element with the Magic Wand](#)

[Using the Magic Wand to Add/Subtract Polygon Shapes](#)

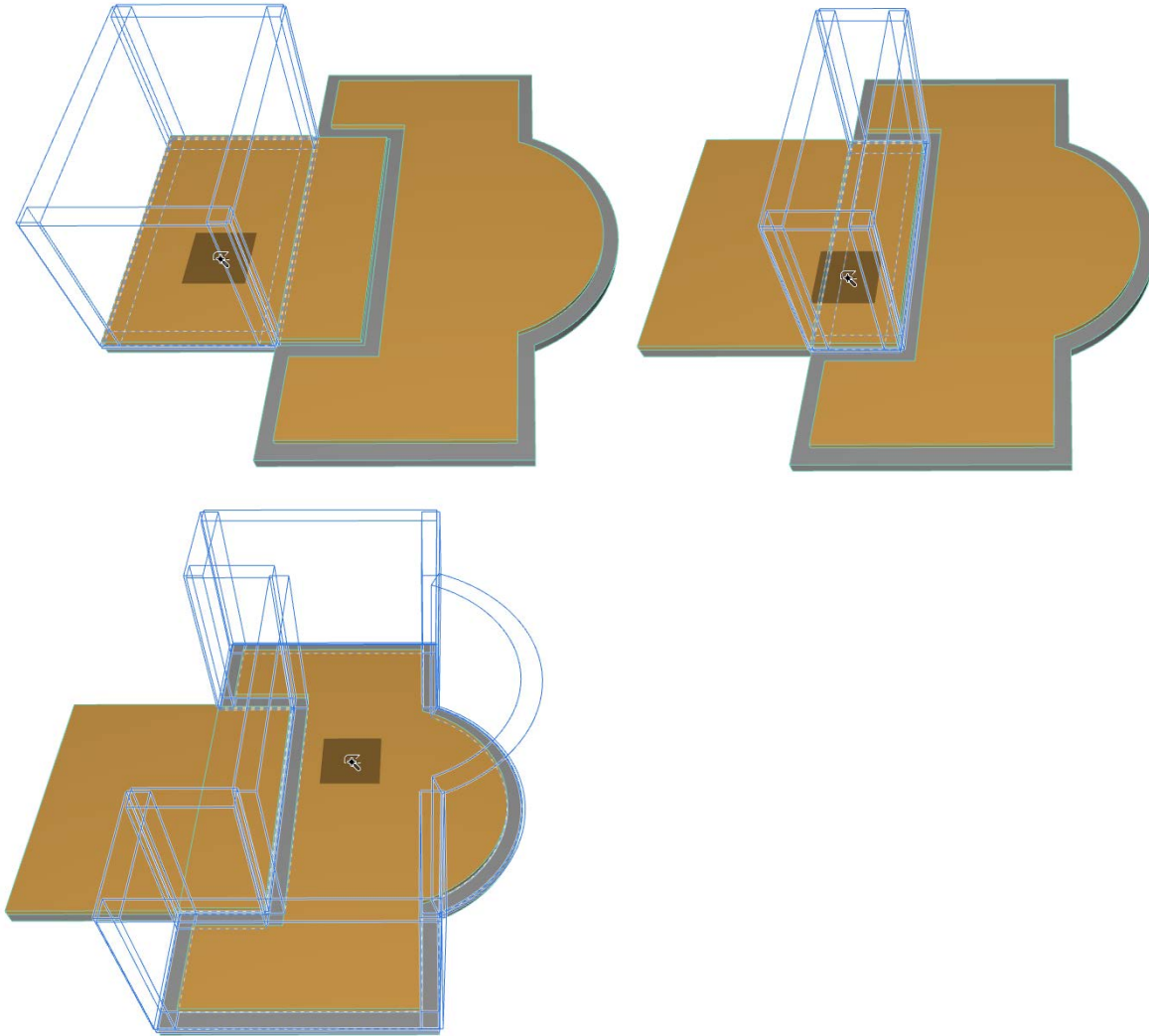
[Magic Wand Settings](#)

How to Create an Element with the Magic Wand

1. From the **Toolbox**, select the type of the new element you want to create.
2. Activate the Magic Wand by doing one of the following:
 - Press the Spacebar
 - Click the Magic Wand icon in the Control Box
 - Choose **Design > Outline Polygon with Magic Wand**
3. View the graphical feedback to see the potential result(s). (See below).
4. Click to automatically find and trace a polygon shape.

Graphical Feedback

While the spacebar is pressed, move the cursor around to view feedback on the possible resulting elements. Here, if you are using the Magic Wand to create Walls atop two intersecting slabs, the feedback shows you the possibilities:






Note: The Magic Wand preview highlight color can be set in the Work Environment settings.

[See Selection and Element Information.](#)

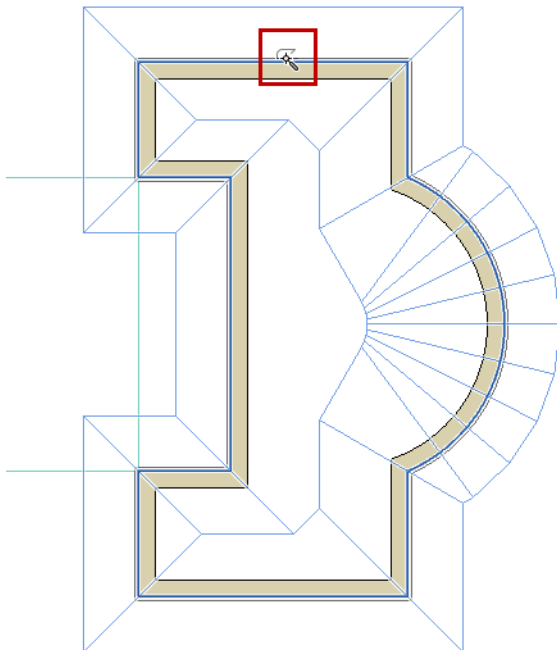
Magic Wand Cursor Icons

The Magic Wand has three different shapes for identifying:

nodes	
edges	
empty space and surfaces	

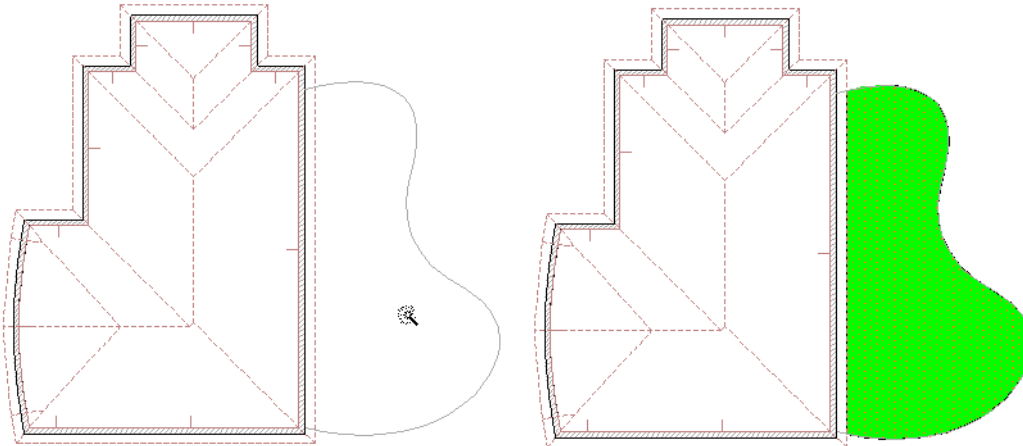
Magic Wand Examples

- Activate the Roof tool (Multi-plane method) and click with the Magic Wand on the edge of the Wall to instantly create a Roof whose pivot line traces the Wall polygon.



[Watch the Video](#)

- Activate the Fill tool, then click with the Magic Wand inside the empty area denoted by the curved line, to create a Fill of that shape.



Hints and Tips

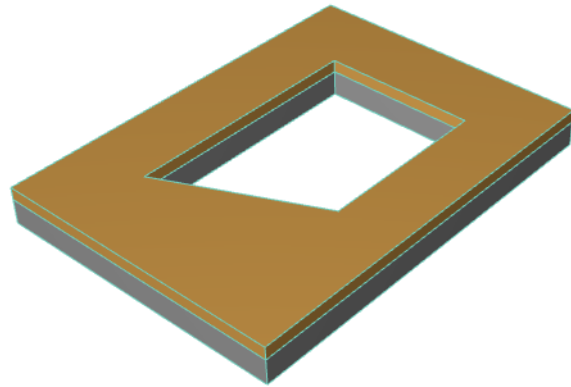
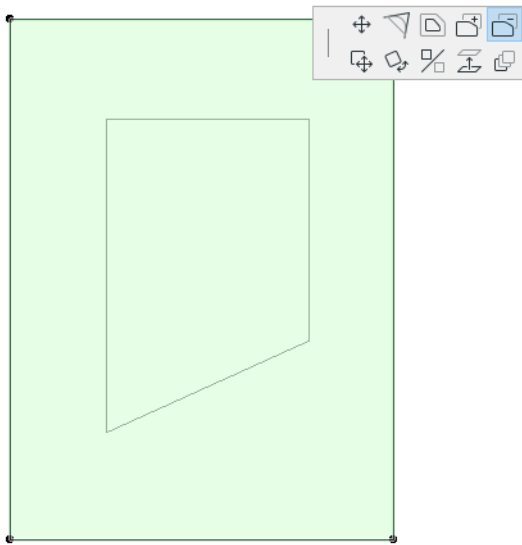
- New elements use the current default settings of the corresponding tool. You should always check that these are correct either before or after using the Magic Wand, especially if the elements are intended to have a specific relationship to each other, such as a roof resting on a Wall.
- You can refine the Magic Wand function by selecting one or more elements. In this case, the **Magic Wand** will only take the selected elements into account when searching for chained elements or a bounded area.
- When creating curved Walls and polygon-type elements based on Arcs, Circles and Splines, the approximation is defined in Magic Wand Settings.
[See Magic Wand Settings.](#)
- The new elements are not linked to the originals and can be manipulated independently.
- If Offset or Multi-Offset has been activated, you can complete the offset function after the Magic Wand has created the new polygon.
- If you are creating a single Roof, you must first draw a pivot line and define the pitch before creating the polygon with the Magic Wand.
- If you are creating a Fill with a custom origin, you will first generate the polygon with the Magic Wand, then draw the fill orientation vector.

Using the Magic Wand to Add/Subtract Polygon Shapes

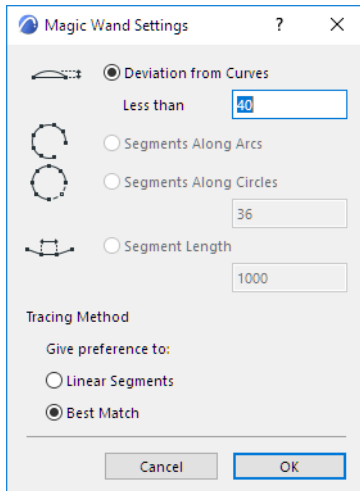
Use the Magic Wand to add or subtract the shape of an existing polygon (of the same or a different element type).

1. Select the polygon from which you wish to subtract another shape. In this example we want to subtract a polygon from a slab.
2. Choose the “Subtract from Polygon” icon in the pet palette.
3. Move the cursor onto the polygon that you want to subtract and activate the Magic Wand (press the spacebar).

4. Click to complete.



Magic Wand Settings



Use the **Design > Magic Wand Settings** command to open this dialog box.

When creating curved Walls and polygon-type elements based on Arcs, Circles and Splines, the approximation is based on the state of the **Magic Wand Settings** dialog box.

With the two radio buttons on the right, you can choose between two tracing methods.

- **Best Match:** Curved segments (when applicable) will follow as best as possible the natural form of Circles, Arcs and Splines.
- **Prefer Linear Segments:** Only linear segments are used.

The four radio buttons on the left allow you to choose from a set of options to approximate curves with linear segments.

- **Deviation from Curves:** Enter a value to define the maximum deviation of the polygon from the original curved element.
- **Segments Along Arcs:** Define the number of segments along an Arc.
- **Segments Along Circles:** Define the number of segments along a Circle. In this case, arcs will be transformed into a number of segments corresponding to the part of the circle they represent.
- **Segment Length:** Enter a value to define the segment length for transformations resulting in segments of uniform length.

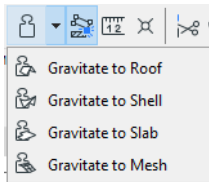
Note: If you choose the Best Match method and real curves cannot be created, Archicad will try to approximate curves in the hierarchical order represented in this dialog box.

Gravity

When placing a new Wall, Column, Beam or Object-type element (including a Level Dimension), you can use the **Gravity** function to automatically place the new element directly on top of a Slab, Roof, Shell or Mesh on the same story, thus taking on the elevation of the element it is placed on.

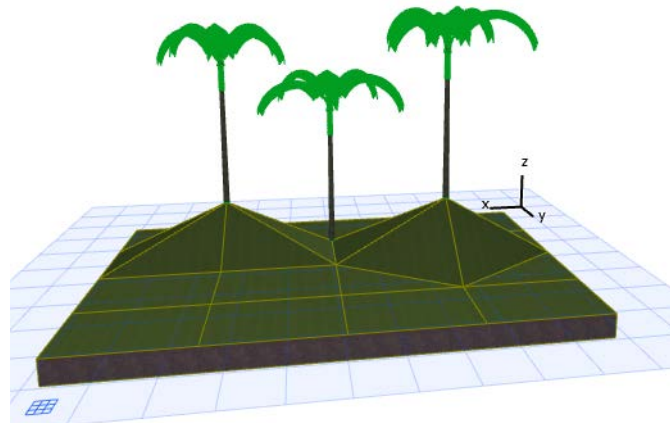
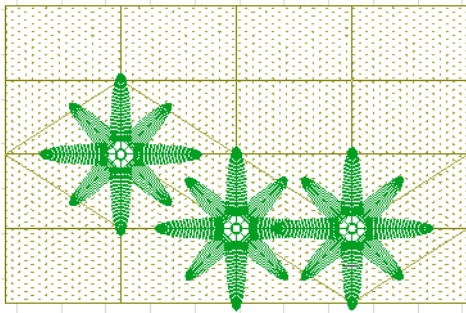
Gravity is useful if you are placing elements on the Floor Plan. (In the 3D window, the Surface Snap feature achieves this same effect. [See Surface Snap in 3D.](#))

To choose an option, use the Gravity controls in the Standard toolbar or the icons of the Coordinates palette.

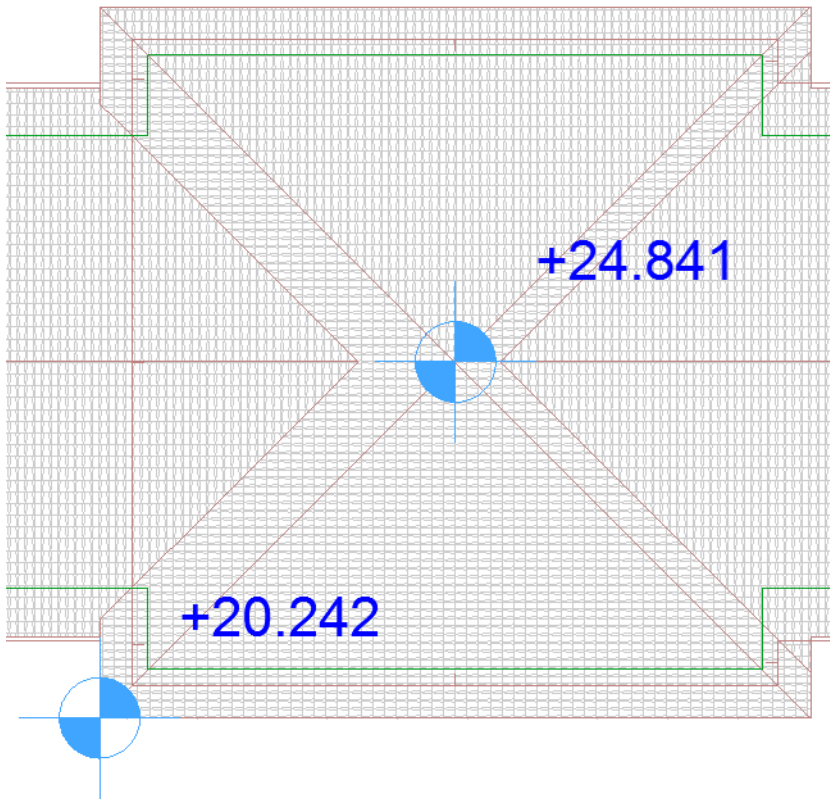


Note: Gravity is only used when placing new elements.

For example, to place trees on top of a Mesh, activate the Object tool, choose the tree, then select “Gravitate to Mesh” from the Gravity pop-up in the Standard toolbar. When you then place the trees, they will gravitate to the elevation of the Mesh at the point of placement.



Level Dimensions placed with **Gravity On**, above Slabs, Roofs, Shells or Meshes, are associated to them.



If the Gravity function is on (activate Gravity icon) while you are placing a new element, but there is no underlying Slab, Roof, Shell or Mesh, then the new element will be placed as usual, at the elevation defined in its Tool Settings.

Mouse Constraints

Archicad's Mouse Constraints are editing aids that can be used as an alternative to Snap Guides and Guide Lines.

[See *Snap Guides and Guide Lines*.](#)

The angle pairs they define can be used to lock the cursor at a particular drawing angle, by **pressing Shift during input**. The mechanism can only be engaged while drawing or editing an element, as indicated by the thick rubber band line shown in the Floor Plan or the ghosted element contour shown in the 3D Window. It temporarily locks the cursor's polar angle value in the Coordinates Palette using one of the angle pairs defined in the **Options > Work Environment > Input Constraints and Guides**.

[See *Input Constraints and Guides Dialog Box*.](#)

Using Mouse Constraints

Projection of Cursor with Mouse Constraints

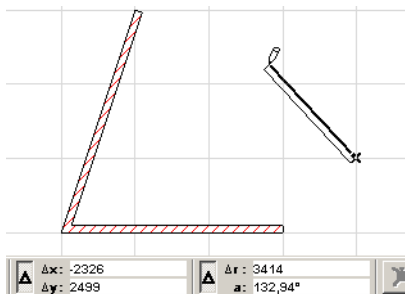
Using Mouse Constraints

Constraining the drawing direction helps you precisely align or connect new elements. To use the orthogonal direction sets for drawing a new element:

1. Make sure the Grid Snap Function is disabled.

[See *Snap to Grid*.](#)

2. Click to start drawing an element.



3. Move the cursor from this starting point and hold down the Shift key. Archicad will search for the closest enabled constraint direction.

Projection of Cursor with Mouse Constraints

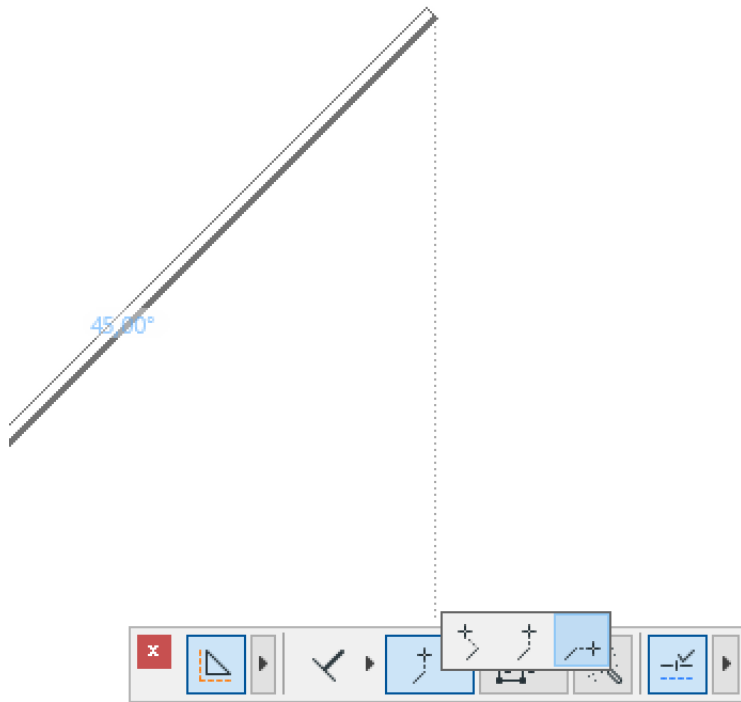
When the drawing direction is constrained by pressing **Shift**, a dotted line connects the unfinished element with the location of the cursor.

To switch among these options:

1. Display the Control Box. (**Window > Palettes > Control Box**)
2. Draw an input vector on screen.
3. Press Shift to enable a constraint.

The Cursor Snap Variants pop-up is now active in the Control Box. Move your cursor to this pop-up and choose the cursor snap variant you need.

[See *Control Box*.](#)



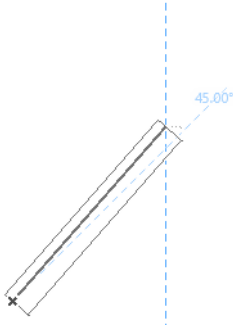
- Perpendicular to the constrained direction
- Horizontal from the constrained direction
- Vertical from the constrained direction

Coordinate Constraints

You can lock a cursor coordinate and thereby restrict the movement of the cursor by pressing **Alt** (Windows) or **Opt** (Mac) and either **X**, **Y**, **A**, or **R/D** on the keyboard.

If the Snap Guides and Snap Points function is active:

- **Alt/Opt + X** produces a vertical Snap Guide.



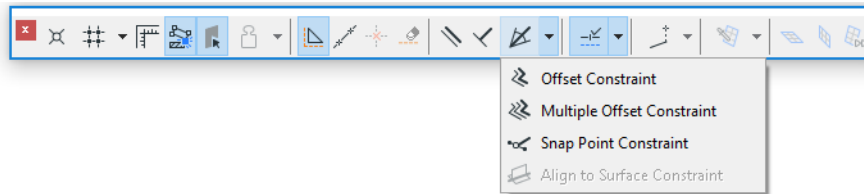
- **Alt/Opt + Y** produces a horizontal Snap Guide.
- **Alt/Opt + A** (having defined the Angle value numerically or drawing in a sample direction) produces a corresponding Snap Guide.
- **Alt/Opt + R** produces a Snap Guide Circle whose radius is determined numerically or graphically.

To unlock the constraint, just press the same key combination (e.g. **Alt/Opt + X**) again.

If the Snap Guides and Snap Points function is inactive, the locking function works the same way, but you will not see the Snap lines/points.

Relative Construction Methods

In addition to Guide Lines and Snap Guides, the relative construction methods also constrain the element construction direction to a constant angle or offset. They are accessible from the Drafting Aids toolbar:



They can also be accessed from the third button of the Control Box.



Note: The Control Box is not visible by default. It is available at **Window > Palettes**.

See [Control Box](#) for more information.

Each Relative Construction Method is described in the following sections:

Parallel and Perpendicular Constraints

Angle Bisector Constraint

Offset and Multiple Offset Constraints (Relative Construction Methods)

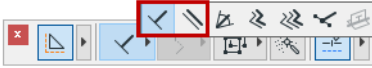
Aligning Elements to a Surface in 3D

Snap Points on Temporary Vector

Parallel and Perpendicular Constraints

Use these commands to draw new elements that are either parallel to or perpendicular to a reference line or existing element edge.

The Parallel and Perpendicular constraint commands are accessible from the Drafting Aids toolbar:



They are also accessible from the Control Box.

Note: The Control Box is not visible by default. It is available at **Window > Palettes**.

See [Control Box](#) for more information.

1. Make sure that the Grid Snap Function is inactive.

See [Deactivate Grid Snap](#).

2. Choose the Perpendicular or Parallel icon and click it to make it active.

3. Select a line or an element edge as a reference by clicking it (the Mercedes feedback will indicate an edge), or draw a temporary reference line using the cursor. The cursor icon will indicate which method (parallel or perpendicular) is active.

4. Draw the new element, which will be perpendicular or parallel to the reference line.

Angle Bisector Constraint

Using the Angle Bisector relative construction method, you can draw elements bisecting the angle formed by other elements or temporary reference vectors.

This command is accessible from the Drafting Aids toolbar:



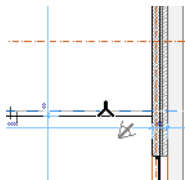
It is also accessible from the Control Box.

Note: The Control Box is not visible by default. It is available at **Window > Palettes**.

See [Control Box](#) for more information.

You can either start drawing the new element before activating the Angle Bisector method and defining the reference vectors, or the other way round. Before using Angle Bisector constraint, make sure that Grid Snap is disabled.

Click the Angle Bisector icon, then define the reference vectors by either clicking on two existing element edges or lines, or drawing two temporary lines. While you are clicking or drawing the reference lines, the cursor icon will show that you have activated the angle bisector constraint.



The mouse is constrained to the bisecting angle between the two reference lines.

Offset and Multiple Offset Constraints (Relative Construction Methods)

The **Offset Constraint** and **Multiple Offset Constraint** methods help you create elements offset from a reference line, using any tool that has a linear or polygonal input method.

1. Activate the tool with which you want to draw the new element.
2. Choose or activate the **Offset** command using one of the following methods:
 - the drop-down menu of the Drafting Aids toolbar



- the Control Box
 - [See Control Box.](#)
- 3. Draw a reference line (e.g. parallel to an existing element). The cursor displays the offset or multiple offset icon.
 - **Note:** You can also create this reference line using the Magic Wand.
 - [See Magic Wand.](#)
- 4. A rubber band outline appears; drag the cursor to the final location of the new element.
- 5. If you are using the regular **Offset** command: Click once to place the element.
- 6. If you are using **Multiple Offset**: drag and click as many times as the number of elements you wish to place; double-click, or click **Cancel** in the Control Box, to place the final element.

Aligning Elements to a Surface in 3D

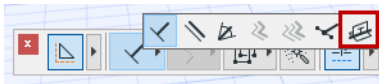
The **Align to Surface** relative construction method is only active when working in the 3D Window. It enables you to fit elements onto non-horizontal planes when working in 3D views.

Note: The 3D Editing Plane enables you to do the same functions graphically.

See [Editing Plane in 3D Window](#).

Using the Constraint when Creating New Elements

1. Open the 3D Window.
2. Choose the tool for the element you wish to add.
3. Adjust your User Origin if necessary.
4. Activate the **Align to Surface** Relative Construction method in the Control Box.



5. Click on the surface to which you want to align your new element.
6. Finish drawing the element in the usual way.

Using Align to Surface with Existing Elements

1. Activate the 3D Window.
2. Adjust the User Origin if necessary.

For more information, see [Origins](#).

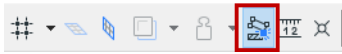
3. Select the element you wish to edit.
4. Choose the **Edit** command you wish to execute, e.g., stretch an edge of a slab to the intersection of a roof and the User Origin.
5. Click a reference point or edge to start editing.
6. Select the **Align to Surface** icon.
7. Finish editing the element.

Snapping to Existing Elements

Element Snap

Archicad helps you to move or reshape elements by snapping them to existing ones as part of a move operation.

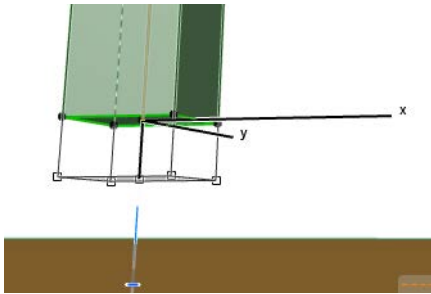
The Element Snap function can be enabled and disabled using the **View > Element Snap** toggle command. (This command is also available on the Standard Toolbar.)



Element Snap can be toggled on and off while you work using the E shortcut.

Note: Element Snap can only be used when Grid Snap is disabled.

If you select an element and then start moving it with Element Snap active, you will see that its special points change to small squares.



These special points include:

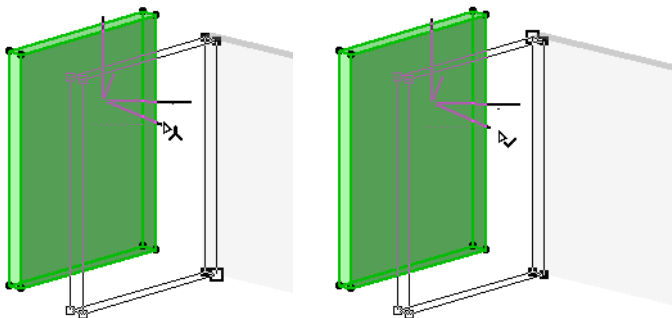
- The selection dot points of the element including anchor points
- The division points of the element (if Snap Points are enabled.)

[See Snap Points.](#)

- The tip of the cursor

When one of these selection dots encounters a node, anchor point or edge of another element where the cursor changes its shape to indicate a special relation, that selection dot becomes larger showing that the element is attracted to the sensitive point. You can then click to join the two elements without having to place the cursor at the actual joining point.

If more than one snapping point touches an eligible point on the other element, use the spacebar to cycle through possible points.

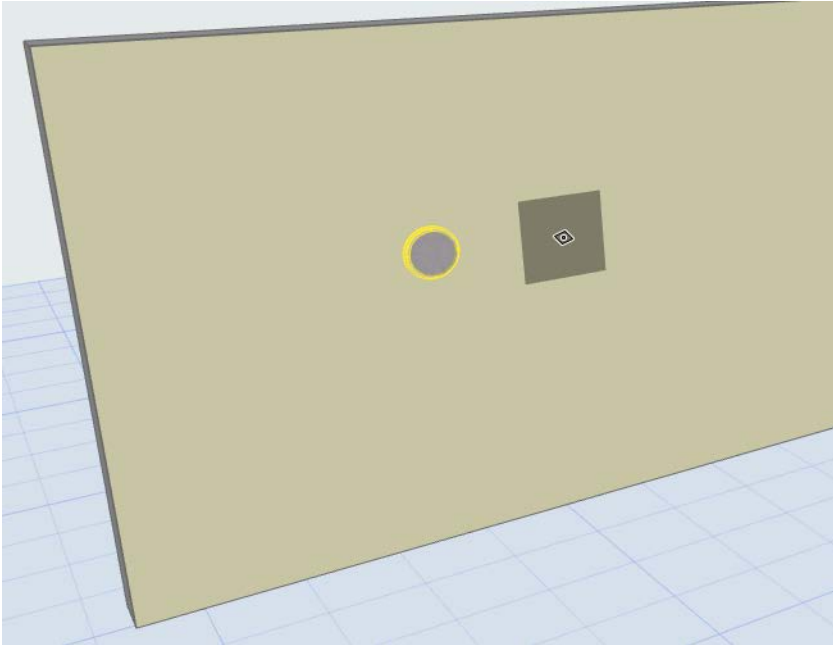


Surface Snap in 3D

Surface Snap is enabled by default, with the **View > Surface Snap** toggle.

Your cursor will snap to any surface (as well as edges and points) in the 3D window - for easy input among elements in space.

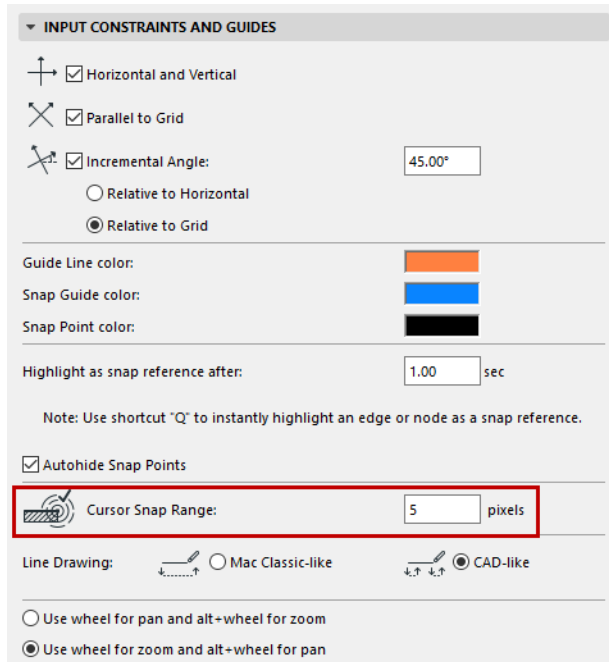
For example, with Surface Snap On, it is easy to place the Wall Lamp in the right place:



(You can achieve the Surface Snap effect in 2D windows, too, by using the Gravity feature. [See Gravity.](#))

Cursor Snap Range

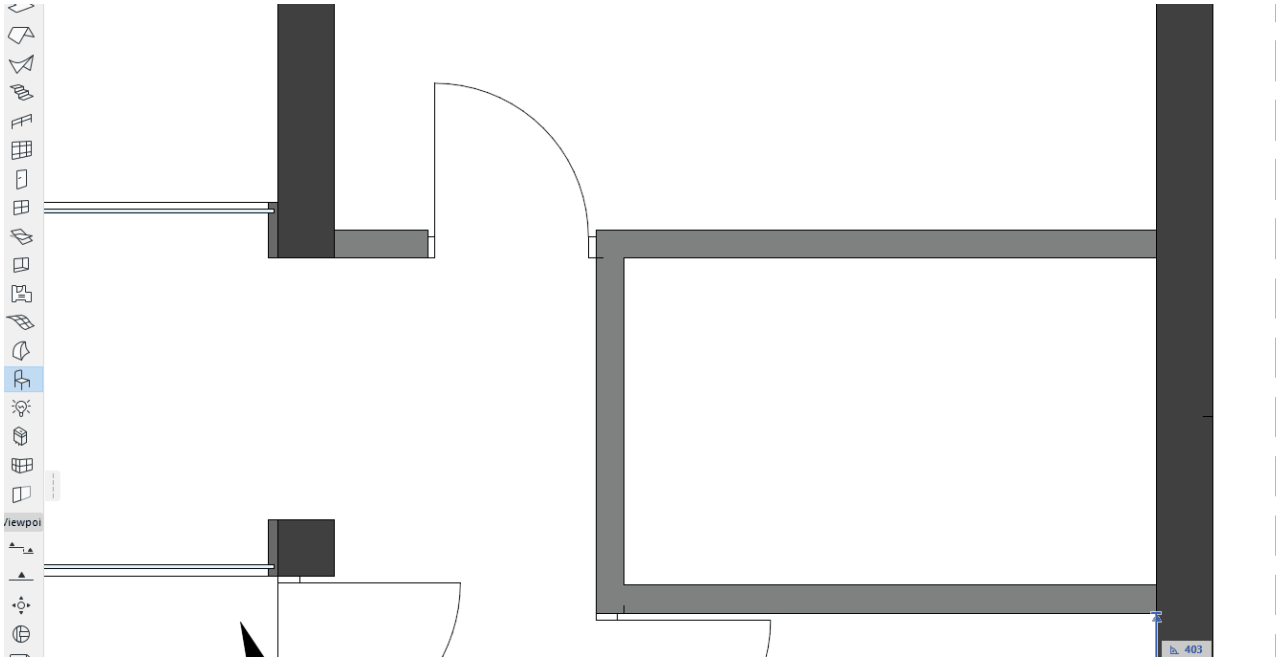
To help locate hotspots on the screen, the cursor gravitates toward edges and nodes. The attraction of the cursor by elements is defined in pixels in the bottom section of the **Options > Work Environment > Input Constraints & Guides** dialog box.



Cursor Snap Range can be set between 1 and 9 pixels. 3-4 pixels is a comfortable setting, but at small zoom levels you may hit hotspots that you did not wish to snap to. If you have good pointing skills, use 1 pixel and you will not be forced to zoom in.

Distance Guides - Experimental feature

Use Distance Guides to input and move elements accurately on the Floor Plan, with pop-up guides that show dynamic distances from surrounding “context” elements. You can use Distance Guides both during and after element placement.



Enable Distance Guides

This is an experimental feature. Make sure you enable it at **Options > Work Environment > More Options > Experimental Features**.

See [Experimental Features](#).

Turn Distance Guides On/Off

Once Distance Guides are enabled, you can toggle it on or off at one of the following:

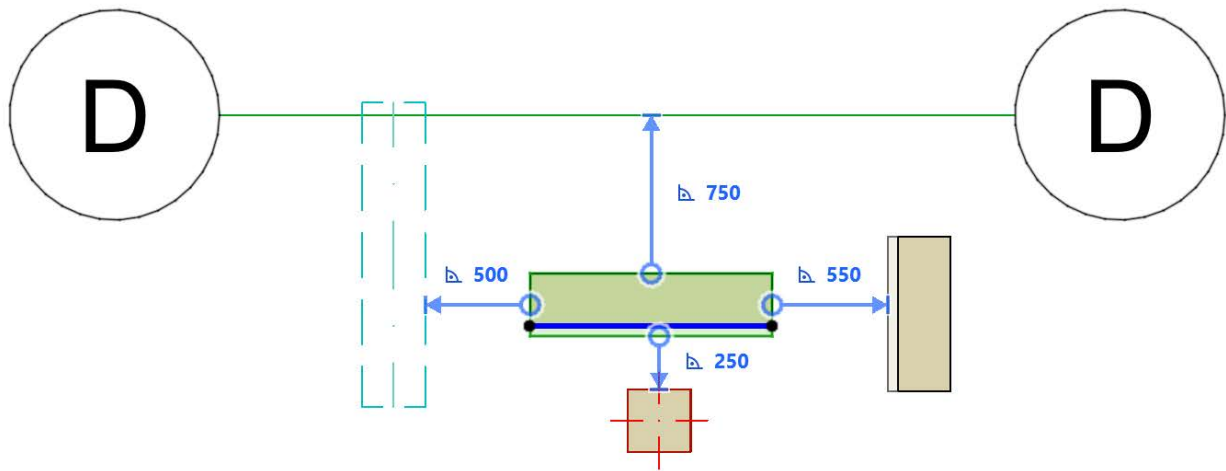
- Options > Work Environment > Tracker and Coordinate Input
- Add the toggle to a toolbar



Here you can also set whether the guides should measure from element edges, or from their reference lines.

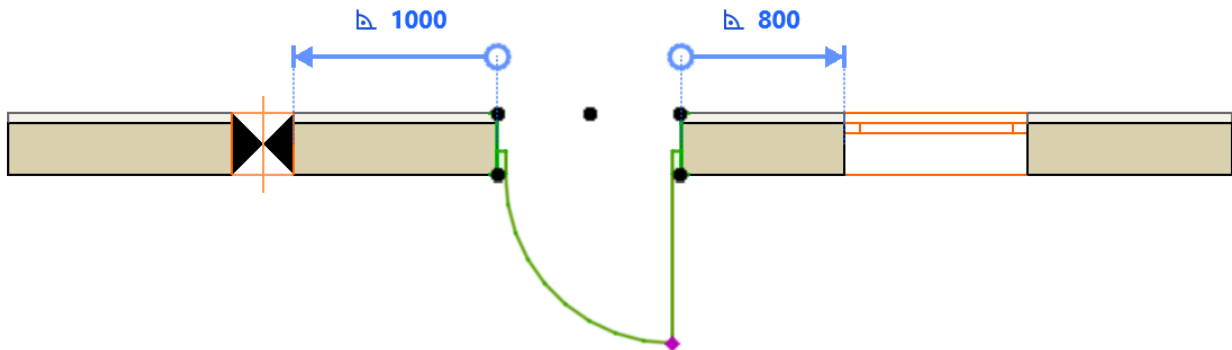
Which elements display Distance Guides?

The following “context” elements show Distance Guides when you select or input a standalone element:



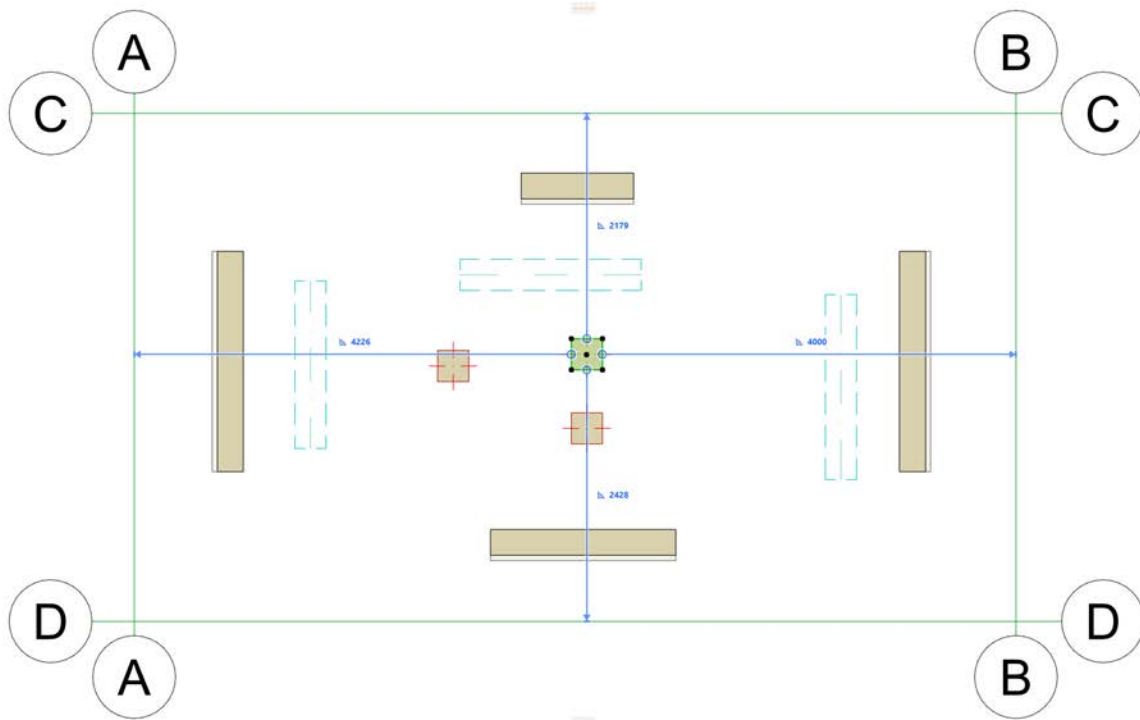
- Grid (without the marker part)
- Wall
- Beam
- Column

When you select or input an associative element (like a door in its host Wall, or an opening in a Beam), the Distance Guides are logical depending on the context.



Distance Guides - Priority

When several element types (visible in the current zoom) can all be used as context elements, Distance Guides “prefer” elements in the following order:



1. Grid lines
2. Walls
3. Beams
4. Columns

Use Distance Guides with selected elements

When you select an element, Distance Guides appear with distance dimensions from the surrounding “context” elements. (Curved elements are not supported.)

1. Select a single element of the following types on the Floor Plan:
 - Wall
 - Beam
 - Column
 - Window/Door
 - Opening
2. Edit the location of the selected element using the Tracker, taking advantage of feedback from Distance Guides.

[See *Element Input Using Tracker*.](#)

Use Distance Guides for element input

1. Activate one of the following element tools:
 - Wall
 - Beam
 - Column
 - Window/Door
 - Opening

2. Define cursor location by using the Tracker. You must use the keyboard to enter the Tracker and edit the values.
3. Distance Guides indicate the perpendicular distance from cursor position to the surrounding “context” elements. (Curved elements are not supported.)

Note: The cursor position you define is the start point of the new element’s Reference line/axis.

Wall Reference Line

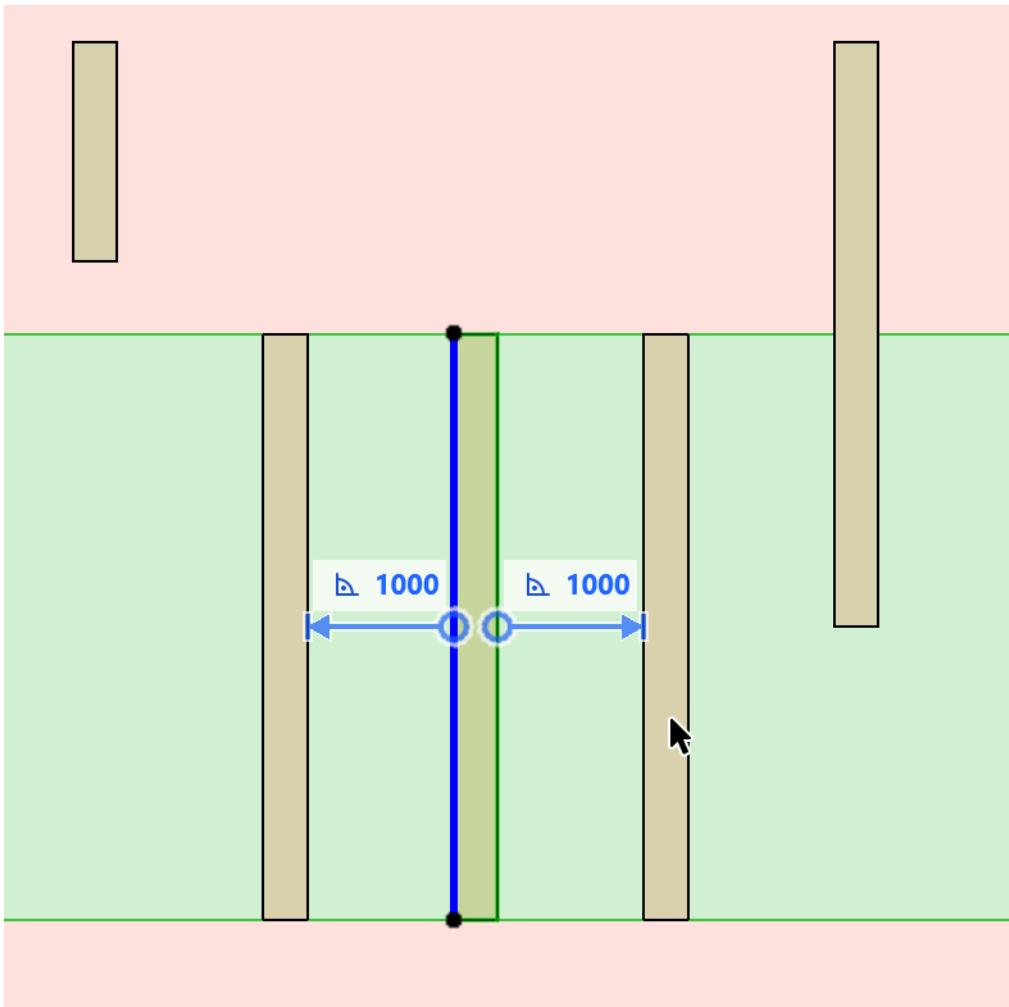
Beam Reference Axis

Column End Cuts and Reference Axis

Relink Distance Guide to another reference edge

For a selected element on the Floor Plan, you can relink the Distance Guide to another edge.

1. Point your cursor at the arrowhead at the end of the Distance Guide line
2. Hover to see the feedback
3. Click the arrowhead to edit it
4. Click on any valid edge (if you were to extend the body of the selected element, any element edge that it would intersect is valid)



Editing Elements

The following sections describe all aspects of Archicad element editing, including moving elements and modifying their size.

Basic Editing Operations

Edit Selection Set

Cancel Operations (ESC Key, Backspace)

Pet Palettes

Move Elements

Offset All Edges

Intersect Two Elements

Align Elements

Distribute Elements

Modify Element Sizes

Explode into Current View

Duplicate Elements

Drag & Drop

Parameter Transfer

Element Transfer Settings

Group Elements

Lock/Unlock Elements

Display Order

Consolidate Lines and Fills in Drawing Windows

Basic Editing Operations

Topics in this section:

[Cut](#)

[Copy](#)

[Paste](#)

[Copy/Paste Elements Between Stories](#)

[Copy/Paste Elements Between Projects](#)

[Delete](#)

[Undo/Redo](#)

Cut

The **Edit > Cut** command removes selected elements from a Project and places them on the Clipboard for future use via the **Paste** command. It can also be used for dialog box or Coordinates Palette numeric field contents.

Note: The **Cut** command is not available in the 3D Window.

The selection can be made with either the **Arrow** or the **Marquee** tool, or by Shift+clicking.

If you cut a construction element from a Model-type Section/Elevation/IE window or a 3D Document, the element is also cut from the Floor Plan.

In Text type windows, you can use the **Cut** command as in any word processor.

Copy

The **Edit > Copy** command puts the selected construction or text elements on the Clipboard, but the selected elements are *not* removed from the original document.

The selection can be made with either the **Arrow** or the **Marquee** tool, or by **Shift+clicking**.

You cannot create new construction elements in Section/Elevation/IE or 3D Document windows using Copy/Paste. (The only exception is if you use the “**Drag a copy**” command in a model-type Section window.)

Note: To Drag, Rotate, or Mirror a Copy, press Ctrl while executing the regular Drag, Rotate or Mirror command.

In the 3D window, the **Copy** command is only available with the Marquee tool.

Paste

Use **Edit > Paste** command to insert the contents of the Clipboard onto the current Project or a text Window.

When *pasting* cut elements into a Section/Elevation/IE, 3D Document or Detail/Worksheet window, the elements are pasted as drawing primitives (points, lines and fills).

You cannot create new construction elements in Section/Elevation/IE or 3D Document windows using either Cut/Paste or Copy/Paste.

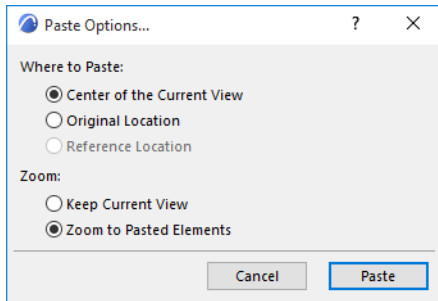
The **Paste** command is not available in the 3D window.

If you **Cut/Copy** and then **Paste** a selection between stories at the same zoom level with no Panning operations in between, the contents of the Clipboard are pasted into the same position they were originally cut or copied from.

If there are several possible options for locating the pasted Clipboard contents, the Paste Options dialog box appears. (The dialog box appears if the pasted elements cannot appear in their entirety in the Window at the

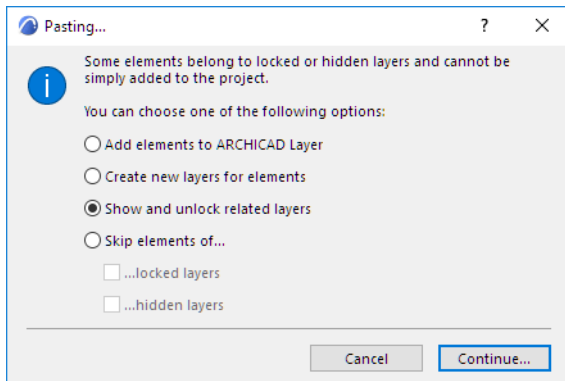
current zoom level, or if it is possible to locate the pasted elements relative to either the Active view or a Reference view.)

See also [Paste Options Dialog Box](#).



Copy/Paste Elements from Locked or Hidden Layer

You cannot paste elements that are on a locked or hidden layer. If you try it, the following dialog box appears:



Choose one of the options to manage this situation (either skip those elements, unlock/unhide the layers, or define another layer to place them on), then click **Continue** to paste.

Copy/Paste Elements Between Stories

The [Edit Elements by Stories Dialog Box](#) provides a quick way to copy elements from one story to another without having to redraw them for each story.

Copy/Paste Elements Between Projects

When pasting elements to another project, remember that element attributes are identified by name.

- If the pasted elements refer to attributes that do not exist in the target Project, the necessary attributes will be appended.
- If the elements refer to attributes that do exist in the target project but are different from those of the original project, they will take on the attributes of the target project.

The Clipboard can be used not only to hold Archicad elements for pasting, but also to import bitmap graphics and other types of information. You can paste multi-line texts from word-processing applications into Archicad.

Delete

The **Edit > Delete** command removes selected items from Project or text Windows. The deleted elements are not preserved on the Clipboard. They are easily retrieved by Undo.

The keyboard equivalent of Delete is hitting the **Backspace** or **Delete** key. Deleting construction elements in any model window (Floor Plan, 3D, model-type Section/Elevation/IE, or 3D Document) will also clear these elements from all the other windows.

Note: Deleting any additional (i.e. manually added) elements from a model-type Section/Elevation/IE window or a 3D Document, or any elements from a drawing-type Section/Elevation/IE window will leave all other windows unchanged.

Undo/Redo

The **Undo** and **Redo** commands allow you to undo (revert to a previous step) and redo a large number of construction operations. The name of the command to be undone is displayed in the menu along with the **Undo** or **Redo** command.

The number of steps that can be undone and redone (the “undo queue”) is set using in the **Options > Work Environment > Data Safety & Integrity** dialog box (Undo Limit).

Warning: The undo queue is cleared entirely whenever you do the following operations:

- Delete or add a story, or change story height
- Update a Hotlinked module
- Send/Receive commands in Teamwork

In text type windows, only the last editing step can be undone/redone. (Text-type windows include all Listing windows created with the Calculation function, GDL script windows, Project Notes, and Report windows.)

Important: Operations involving Navigator/Organizer items (such as drag and drop between Navigator maps, deleting items from a map, or adding items to the Publisher set) are not added to the undo queue, and are *not* undoable.

See also [Cancel Operations \(ESC Key, Backspace\)](#).

Cancel Operations (ESC Key, Backspace)

By successively pressing the **Esc key**, you will achieve the following actions:

1. Exit numeric input
2. Exit navigation (Orbit, Explore)
3. Remove fixing
4. Remove Snap Reference
5. Exit input
6. Deselect all
7. Remove Marquee
8. Change to Arrow tool

During input, pressing the **Backspace** key will cancel the operation.

Warning: Pressing Backspace with any items selected will delete those items!

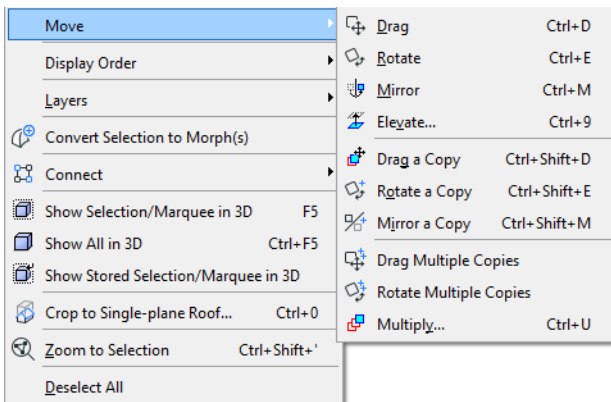
Move Elements

You can move elements individually or collectively. You can “**nudge**” them in increments; you can **drag, rotate** or **mirror** them along a horizontal plane and through vertical displacement, that is, by changing their **elevation** values.

Note: To Drag, Rotate, or Mirror a Copy, press Ctrl while executing the regular Drag, Rotate or Mirror command.

To move elements, you can:

- Select them and choose the corresponding menu command from the **Edit > Move** menu (with any tool active in the Toolbox) or the **Move** commands in the context menu of a selected element:



- With any tool active, click on a node and then choose the desired shortcut in the appearing pet palette.

Nudging Elements

Dragging Elements

Rotating Elements

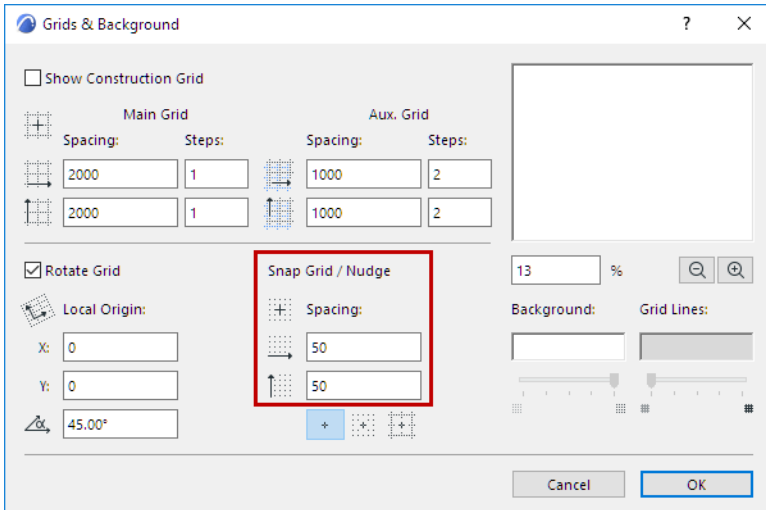
Mirroring Elements

Elevating Elements

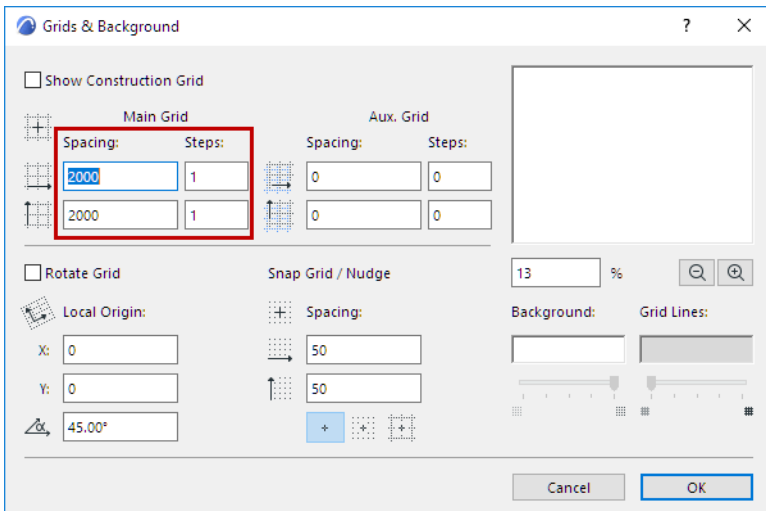
Nudging Elements

A quick way to move elements on the screen is to use “Nudge” feature to move the selected element(s) left, right, up or down (in 3D, along the horizontal plane). The Nudge distance is defined in **View > Grids & Editing Plane > Grids & Background**.

To “Nudge”, use the shortcut **Shift + an arrow key**. The value is defined under Snap Grid/Nudge.



To Nudge in a larger increment, use the shortcut **Shift + Alt/Opt + an arrow key**. The element will be nudged using the value defined for the Construction Grid spacing.



Dragging Elements

To drag one or more elements to a new location:

1. Select one or more elements, then do one of the following:
 - select the **Drag** icon from the pet palette



- select **Move > Drag** from the context menu
 - use the **Drag** command shortcut (**Ctrl+D**)
 - select **Edit > Move > Drag**
2. Move the cursor to drag all selected elements to a new location.
 3. Click to place.

In the 3D window, dragged elements are constrained to the current 3D Editing Plane. You can change this plane as needed.

[See Reposition Editing Plane.](#)

Drag a Copy

To Drag a Copy, select one or more elements, then do one of the following:

- press Ctrl while executing the regular Drag command
- select **Edit > Move > Drag a Copy**
- select **Move > Drag a Copy** from the context menu

Drag and click to place a copy. The original element(s) remain in place.

Drag Multiple Copies

To drag several copies of an element:

1. Select one or more elements, then do one of the following:
 - press Ctrl (Cmd) + Alt while executing the regular Drag command
 - select **Edit > Move > Drag Multiple Copies**
 - select **Move > Drag Multiple Copies** from the context menu
2. Drag, then click to place a copy.
3. Repeat as many times as needed.
4. Double-click to complete.

Rotating Elements

You can rotate selected elements in the Floor Plan or in the 3D Window.

In Section/Elevation/IE, 3D Document, and Detail/Worksheet window, you can rotate drawing elements only.

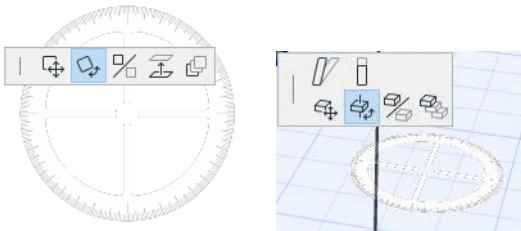
In the 3D Window, rotation is performed across a horizontal plane.

Note: Some elements - Curtain Walls, Shells, Morphs - can be free-rotated (that is, on a plane that is not horizontal).

See also [Rotating Beam and Columns](#)

1. Select one or more elements, then do one of the following:

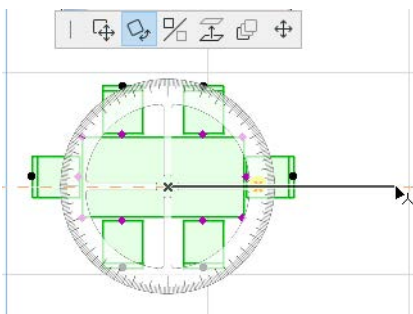
- select the **Rotate** icon from the pet palette



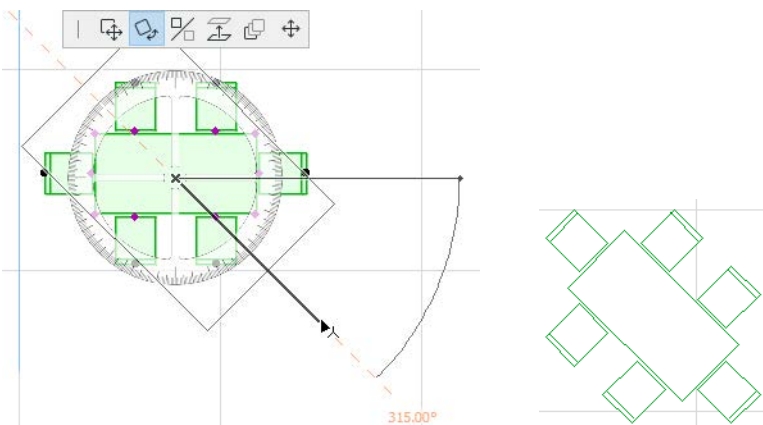
- select **Edit > Move > Rotate**
- select **Move > Rotate** from the context menu
- use the **Rotate** command shortcut **Ctrl (Cmd) + E**.

2. Click to define the center of rotation of the selected elements.

3. Click to define the starting point of the rotation arc and its radius.



- Move the cursor and click to complete the rotation.



Rotate a Copy

1. Select one or more elements, then do one of the following:
 - press Ctrl while executing the regular Rotate command
 - select **Edit > Move > Rotate a Copy**
 - select **Move > Rotate a Copy** from the context menu.
2. Click to define the center of rotation of the selected elements.
3. Click to define the starting point of the rotation arc and its radius.
4. Move the cursor and click to complete the rotation.

The original element(s) remain in place.

Rotate Multiple Copies

To rotate several copies of an element:

1. Select one or more elements, then do one of the following:
 - press Ctrl (Cmd) + Alt while executing the regular Rotate command
 - select **Edit > Move > Rotate Multiple Copies**
 - select **Move > Rotate Multiple Copies** from the context menu
2. Click to define the center of rotation of the selected elements.
3. Click to define the starting point of the rotation arc and its radius.
4. Move the cursor and click to complete the rotation. Repeat as many times as needed.
5. Double-click to complete.

The original element(s) remain in place.

Rotating Shell Profiles

Shell profiles can also be rotated graphically, using the Rotate Profile command from the pet palette of selected Shells.

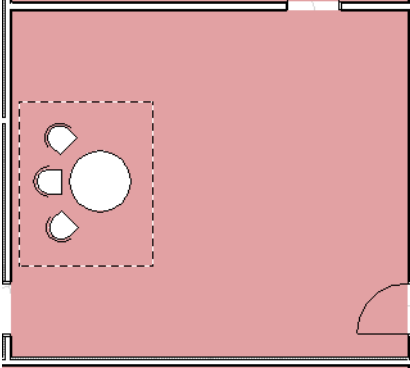
[See *Graphical Editing of Shells*.](#)

Mirroring Elements

Mirroring creates a mirror image of selected elements on the Floor Plan or in the 3D Window.

To Mirror an element:

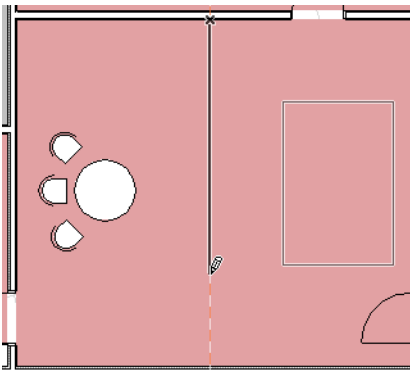
- Select the Element(s) you want to mirror. Here, we want to place the table and chairs in the facing direction, at the other side of the room.



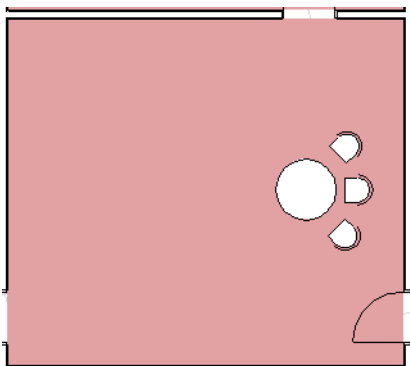
- Select the **Edit > Move > Mirror** command, or choose the **Mirror** command from the context menu or the pet palette.



- Define the mirroring axis with two clicks, as if you were drawing a Line.



The mirrored elements will be placed after the second click.



Notes:

- To Mirror a Copy, press Ctrl while executing the regular Mirror command.

- Any number of Doors and Windows can be mirrored at a time. The mirrored Door/Window will always be in line with the wall that contains the original. Select the Door/Window; click once. This click is the axis of the mirroring action.
- Text, Line arrowheads and Dimensions will not be mirrored, only their bounding box.
- Mirrored Text blocks, Labels, Dimensions and Fill polygon area labels are always readable from the bottom and the left side of the drawing.
- Zone stamps cannot be mirrored.

Elevating Elements

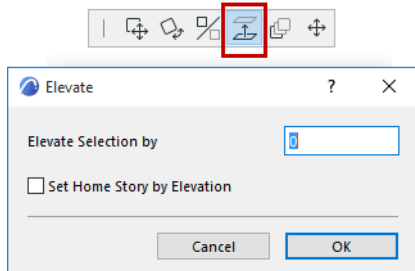
The **Elevate** command allows you to move selected elements vertically along the Z axis.

For information on elevating Mesh points: see [Edit Elevation of a Mesh Point](#).

In the Floor Plan and Section/Elevation/IE windows:

1. Select the element.
2. Choose the **Edit > Move > Elevate** command, or the Elevate command from the pet palette.

This brings up the Elevate dialog box.



3. Type the value by which you wish to raise or lower all selected elements.
4. To allow the element's Home Story to be automatically reset to reflect its new elevation, check **Set Home Story by Elevation**.
5. Click OK.

This command is well-suited to changing the elevations of large numbers of similar elements (i.e., walls, landscaping objects, floor tiles, and lighting fixtures). It is the best way to preserve the vertical relationships between objects while moving them.

Note: In the Section/Elevation/IE window, you can also elevate elements simply by clicking and dragging.

See also [Change Home Story by Elevation](#).

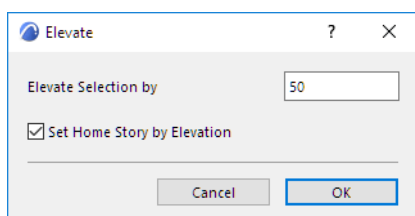
In the 3D Window (Using Drag):

1. Select the element.
2. Choose the **Drag** command from the pet palette.
3. Drag the element to its new position along the Z axis, or define the distance numerically. Notice that a Snap Guide appears, marking the story positions, to make it easier to place the element.

In the 3D Window (Using Elevate command):

1. Choose the **Edit > Move > Elevate** command.

This brings up the Elevate dialog box.



2. Type the value by which you wish to raise or lower all selected elements

3. To allow the element's Home Story to be automatically reset to reflect its new elevation, check **Set Home Story by Elevation**.
4. Click **OK**.

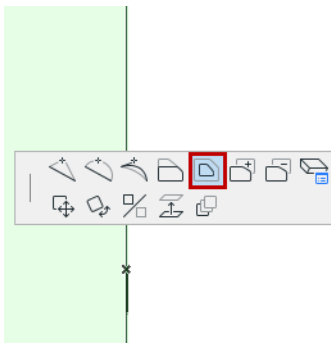
Offset All Edges

Use the **Edit > Reshape > Offset** command to offset all edges of an existing element. This function works on:

- all elements that are polygonal (these include splines as well as the more obvious polygon-type elements)
- multiple connecting elements of a single type, provided they are all selected: for example, two line segments that meet at a point

See also [Offset Edge - Fixed Area](#).

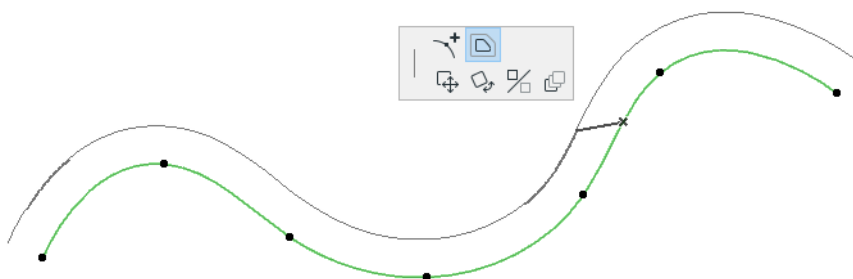
1. Select the element(s) you wish to offset.
2. Choose the **Offset all edges** command in one of the following ways:
 - the offset icon from the Pet Palette (click a node, an edge, or a reference line endpoint of the selected element);



- from the **Edit > Reshape** menu;
 - from the Edit Elements toolbar;
3. A ghost contour of the proportionally increased or reduced element follows the movements of the cursor. Click when you are satisfied with the new element size.

Use Offset to Create One or More Element Copies

1. As described above, select the original element and activate the **Offset** command.
2. Click the **Ctrl** key (**Mac: Opt**) and drag the cursor to the offset location.
To create multiple copies: Click the **Ctrl+Alt** keys (**Mac: Cmd+Opt**), then drag the cursor.
3. Click to place the new element(s).



Offset Constraint

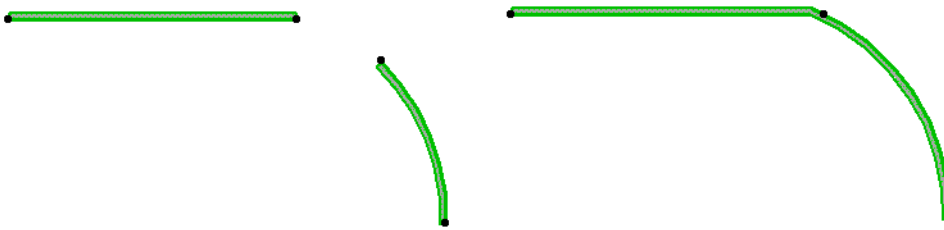
See [Offset and Multiple Offset Constraints \(Relative Construction Methods\)](#).

Intersect Two Elements

You can intersect two selected elements with each other by having them meet at their endpoints' nearest point.

This command works on Lines/Polylines, Arcs, Walls, and Beams.

1. Select the two elements.
2. Choose the **Edit > Reshape > Intersect** command.



Notes:

- This command can work only if the two elements are visible on the same story. They do not have to have the same Home Story, but they must be at least visible on the same story. (If the elements have different Home Stories, and their visibility is set to “**Home Story Only**,” this command will not work.)
- When intersecting Lines or Polylines: if only one ends intersects after the first Intersect command, repeat the command to intersect the other ends as well.

Related Topics:

[Intersect Single-plane Roofs](#)

[Intersect Morph Faces \(Generate Edges within a Morph\)](#)

Align Elements

Use this menu (**Edit > Align**) to align selected elements with each other, or to a custom-drawn target line, using a variety of criteria:

- Align elements **right** or **left**;
- Align elements to the **top** or **bottom**;
- **Center** elements vertically or horizontally;
- “Special”-align elements to **any point** on an existing element, or to a temporary line/arc drawn by you.

[Watch the Video](#)

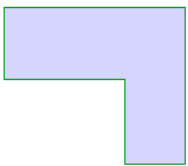
The **Align** commands are only active if you have selected at least two elements.

When using the **Align** commands, the expression “right” refers to the rightmost element on the x-axis in the window you are looking at (e.g. Floor Plan, Section, 3D). The expression “left” refers to the left-hand direction on the x-axis.

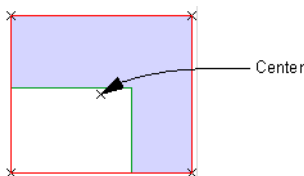
Similarly, “top” means the element with the highest y-coordinate in the current window, and “bottom” is the element with the lowest y-coordinate.

Archicad identifies an element’s right/left/top/bottom/center point according to an (invisible) bounding box around the element.

For example, suppose you want to align the following Corner Table object with other furniture:



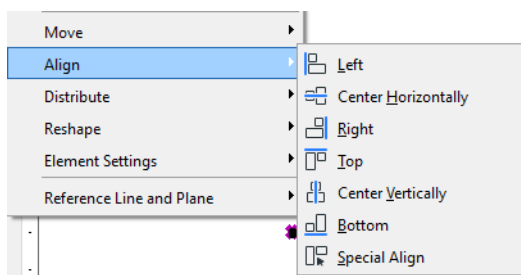
Archicad will use the centerpoint of the table’s bounding box (here, illustrated in red):



Note: If you want to use a different anchor point of the elements you are aligning, use the Special Align options (**Edit > Align > Special Align**).

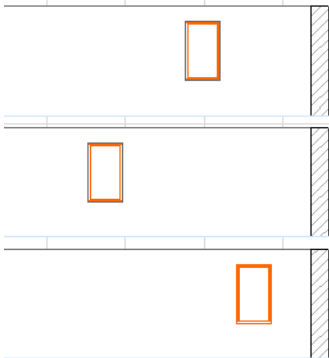
How to Align Elements

1. Select the elements you wish to align.
2. Go to **Edit > Align** and choose one of the following operations:

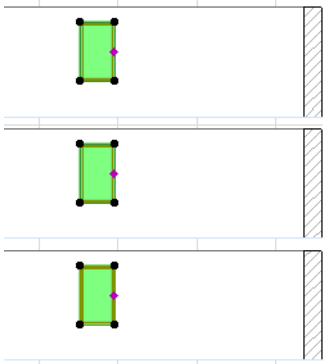


Left: If you choose **Edit > Align > Left**, all selected elements will be aligned to the leftmost bounding-box point of the element that is located farthest to the left.

In this example, we want to align the three windows shown in Section view.

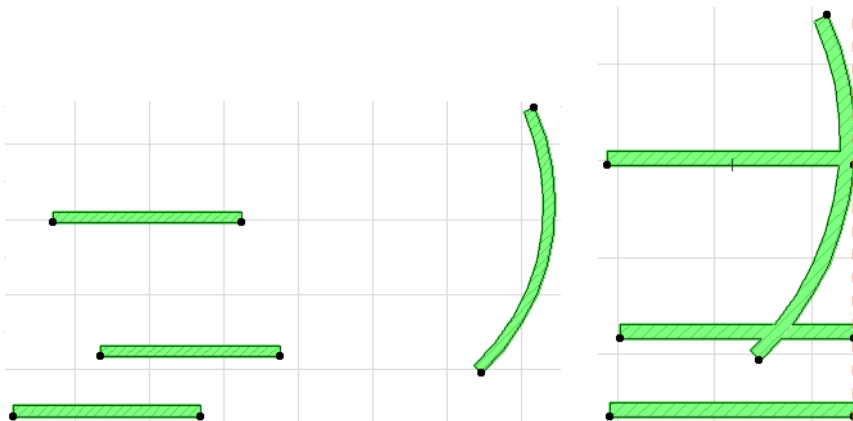


Select all three windows and use **Edit > Align Left**.



Right: If you choose **Edit > Align > Right**, all selected elements will be aligned to the rightmost bounding-box point of the element located farthest to the right.

- In the illustration below, the arc wall is the rightmost element; the rest of the walls (their rightmost points) are aligned to the rightmost point on the arc wall's bounding box.



Top/Bottom: If you choose **Edit > Align > Top or Bottom**, all selected elements will be aligned to the topmost (or bottommost) point on the element that is the top (or bottom) element of the selected group.

- In the example below, we want to move up the window on the left, so that it lines up with other two windows.



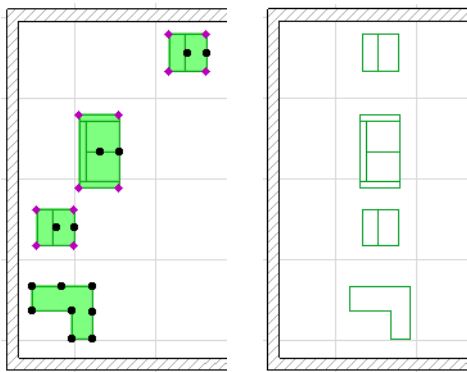
- To achieve this, we select all the windows, then use **Edit > Align > Top** to obtain the desired result.



Center Horizontally: If you choose **Edit > Align > Center Horizontally**, Archicad will determine the selected elements' top and bottom y-coordinates, and then align all the elements horizontally (using each element's centerpoint), along a horizontal line halfway between the top and bottom elements.

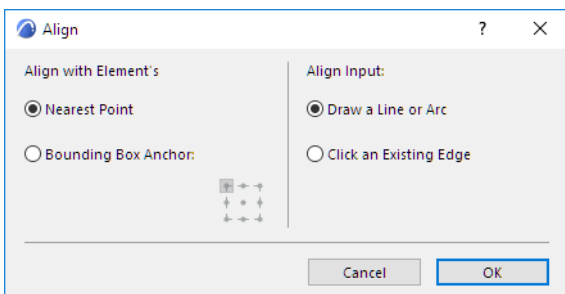
Center Vertically: If you choose **Edit > Align > Center Vertically**, Archicad will determine the selected elements' leftmost and rightmost x-coordinates, and then align all the elements vertically, along a vertical line halfway between them.

- For example, in the following Floor Plan, we will center the selected furniture vertically.

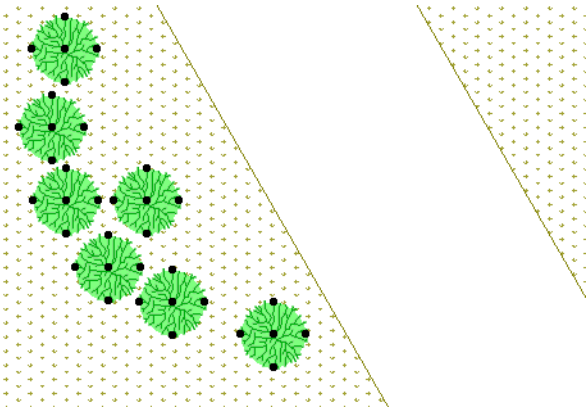


Special Align

The options in **Edit > Align > Special Align** give you great flexibility in lining up selected elements. You can choose **which point** to align, and you can choose **what** to align the elements to: a temporary line/arc drawn by you, or any existing line/edge.



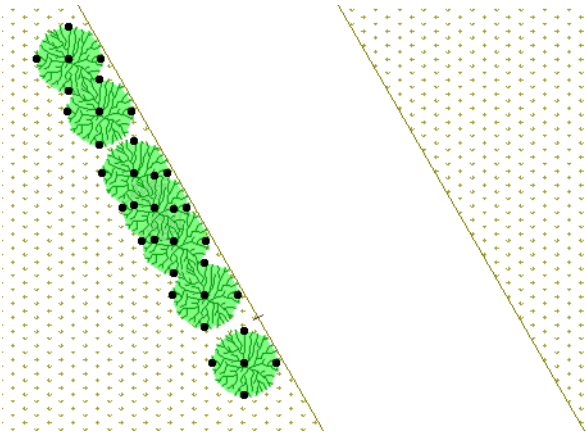
In this example, we wish to align the selected tree objects to the edge of the garden path.



Go to **Edit > Align > Special Align** and choose the **Nearest Point** option: each tree will line up along the path using each tree's bounding box point that falls closest to the path.

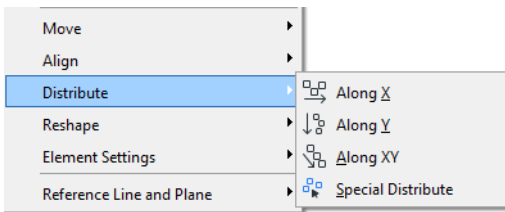
Choose the **Click an existing line or element edge** option.

Click OK to close the dialog box, then click the edge of the path (here, the edge of the mesh). The trees line up along the path.



Distribute Elements

Use this menu (**Edit > Distribute**) to evenly distribute selected elements using a variety of criteria:

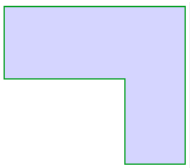


See also the *Distribute* option of the *Multiply* command, at [Multiply Elements](#).

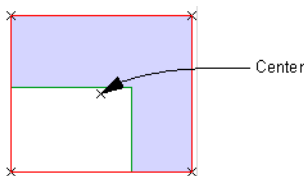
The **Distribute** commands are only active if you have selected at least two elements.

Archicad identifies an element's right/left/top/bottom/center point according to an (invisible) bounding box around the element.

For example, suppose you want to distribute elements that include the following Corner Table object:



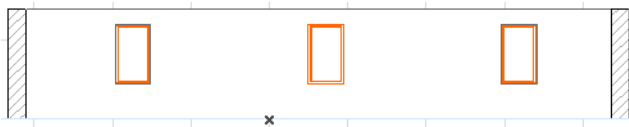
Archicad will use the centerpoint of the table's bounding box (here, illustrated in red):



Note: If you want to use a different anchor point of the elements you are distributing, use the Special Distribute options (**Edit > Distribute > Special Distribute**).

Along X: The two selected elements at either end will remain in place; the rest will be evenly distributed between them.

For example, suppose you have three windows distributed evenly along a wall.



- As a result of a design change, the wall is lengthened and you add two more windows.

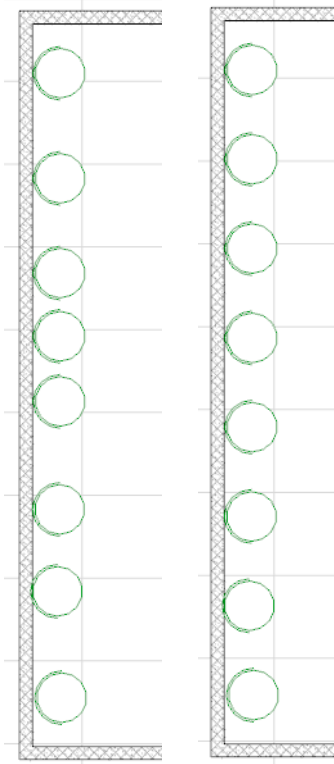


To ensure they are all distributed evenly, select all the windows and use **Edit > Distribute > Along X**.



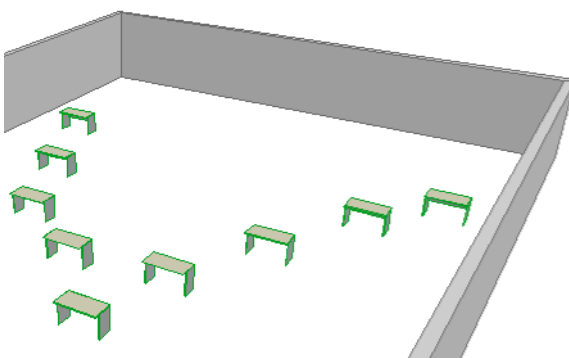
Along Y: The two selected elements at either end (vertically) will remain in place; the rest will be evenly distributed between them.

For example, suppose you want to distribute chairs evenly against the back wall of the room. Select them, then use **Edit > Distribute > Along Y**.



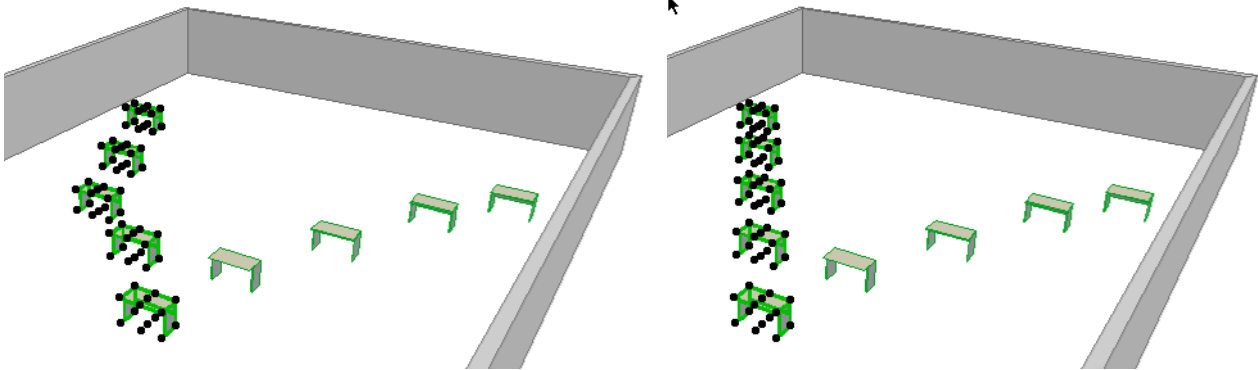
Along XY: The two selected elements at either end (at top left and bottom right) will remain in place, and the rest will be distributed evenly along an XY diagonal.

For example, you have nine desks in the classroom placed roughly in a V formation.



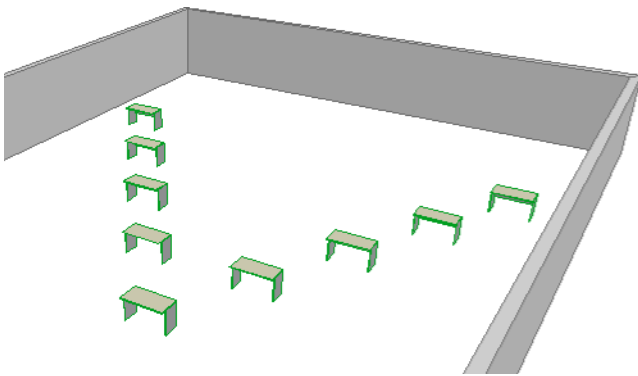
To distribute the desks precisely, select a group of 5 desks and use **Edit > Distribute > Along XY**.

First group:



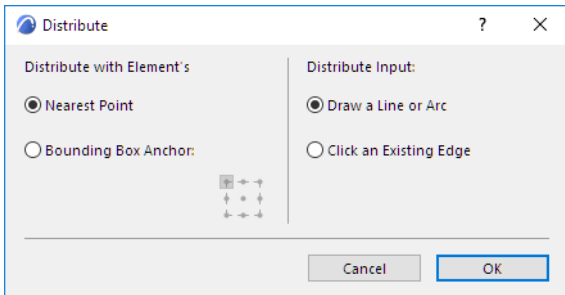
Repeat with a second group of five chairs.

Final result:

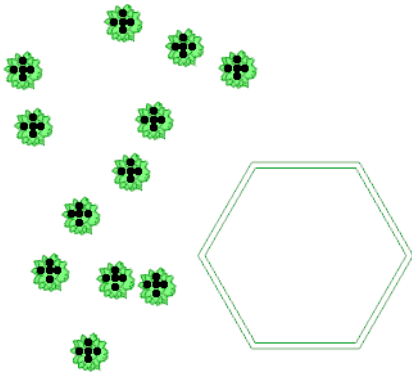


Special Distribute

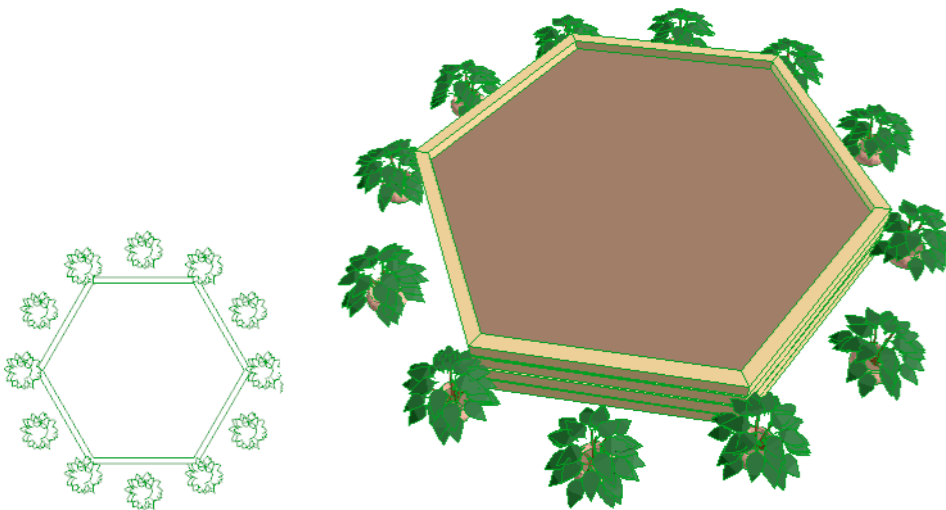
The options in **Edit > Distribute > Special Distribute** let you choose **which point** to distribute and **along what** to distribute the elements: a temporary line/arc drawn by you, or any existing line/edge.



Suppose you want to distribute potted plants in a circle around a hexagonal soil holder object:



Use **Edit > Distribute > Special Distribute**. Choose the “Draw a line or arc” option, then click OK to close the dialog box. Click to start drawing and choose the “Arc by centerpoint” pet palette option. You will create a temporary circle around the hexagonal object. As you complete the circle, the potted plants will be distributed around it.



Modify Element Sizes

Most of the operations can be performed in three ways:

- **Edit** menu command
- Keyboard Shortcuts
- Commands from Pet Palettes

In some cases, only the pet palette is available.

Some general considerations:

- Some GDL Object type elements are programmed to allow only specific sizes, or include smart editing hotspots to edit their shape graphically.
- **Stretch** is available for most types of selected elements in both the Floor Plan and the 3D Window. Only drawing elements can be stretched in the Section/Elevation/IE, 3D Document and Detail/Worksheet windows.
- The **Stretch** command cannot be used on polygonal elements, but you can stretch their edges or displace any of their nodes graphically.

Trim Elements Using Scissors

Reshaping Polygons and Chained Elements

Stretching with the Marquee Tool

Stretch Height

Resize (Enlarge or Reduce) Elements

Offset All Edges

Splitting Elements

Adjusting Elements

Adjust Elements to Slabs

Intersect Two Elements

Create a Fillet or Chamfer

Adding Element Nodes

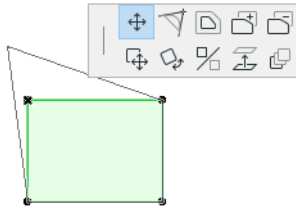
Move Nodes

Curve/Straighten Element Edge

Reshaping Polygons and Chained Elements

Move Polygon Node

To move a **node** of a selected polygon (including a polygonal wall), click the node and use the pet palette's **Move node** option.

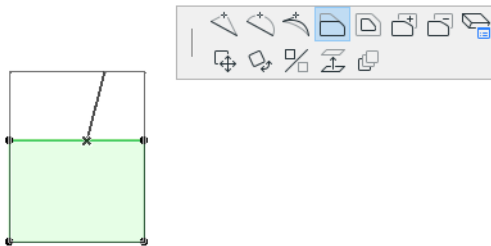


The two neighboring edges will follow the stretching movement and the shape of the polygon will change accordingly.

Offset Polygon Edge

To apply an **offset** on a whole side of a selected polygon, click an edge and use the pet palette's **Offset edge** option. This will stretch the neighboring edges.

This offset feature is available for chained elements as well as polygons. Selected elements will be treated temporarily as a polygon, and all polygon reshaping pet palette options will be available.



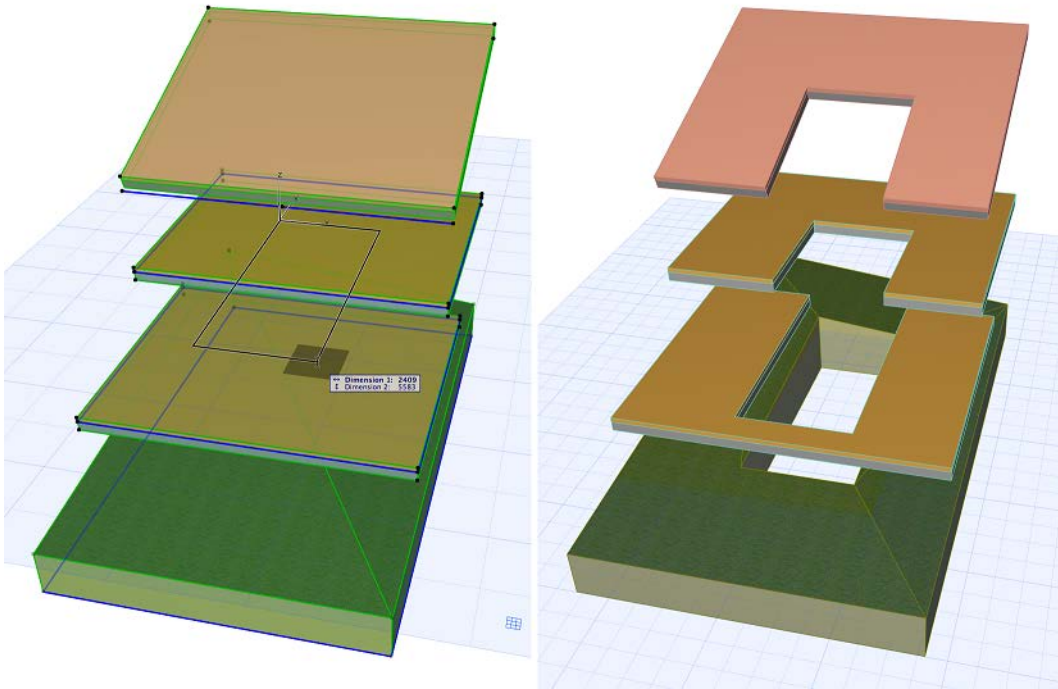
Offset Edge - Fixed Area

This option is available for Zone polygons only.

See [Offset Edge - Fixed Area](#).

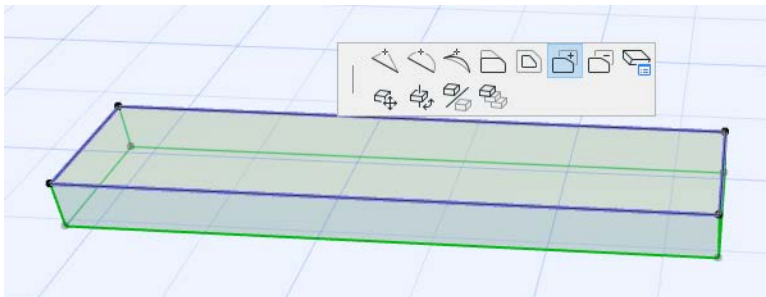
Adding and Subtracting Polygonal Shapes

You can use the pet palette to add to the shape of a polygonal element, or subtract new polygons from it. (This will work on multiple selected elements, too.)



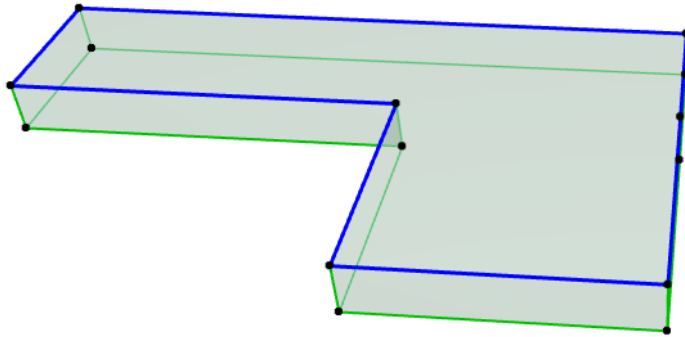
To add a new polygon to the existing one:

Select the polygon and click it either on an edge or a node. From the pet palette, choose the icon with the + sign (**Add to Polygon**).

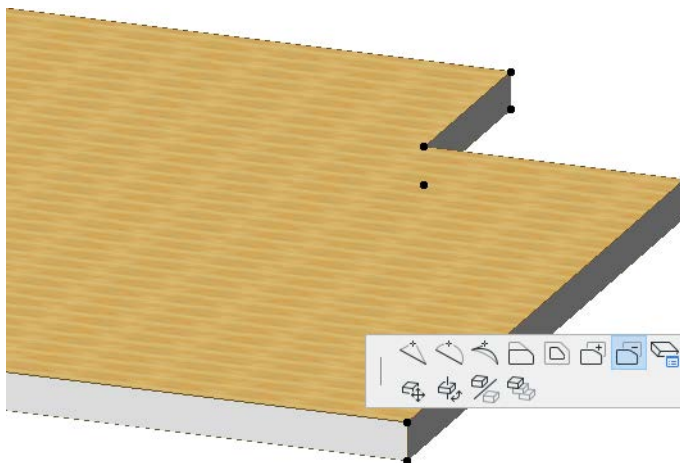


Draw the new polygon shape. It must intersect with the selected one or at least they should have a common edge.

Click to finish drawing the new shape, which will be automatically added to the original one.



The process is the same for subtracting a shape from a polygon, but choose the **Subtract from Polygon** icon.



You can also use the Magic Wand for adding/subtracting a polygon shape.

[See *Using the Magic Wand to Add/Subtract Polygon Shapes*.](#)

[See also *Graphical Editing of Roof Geometry* and *Create a Hole in a Roof*.](#)

Stretching with the Marquee Tool

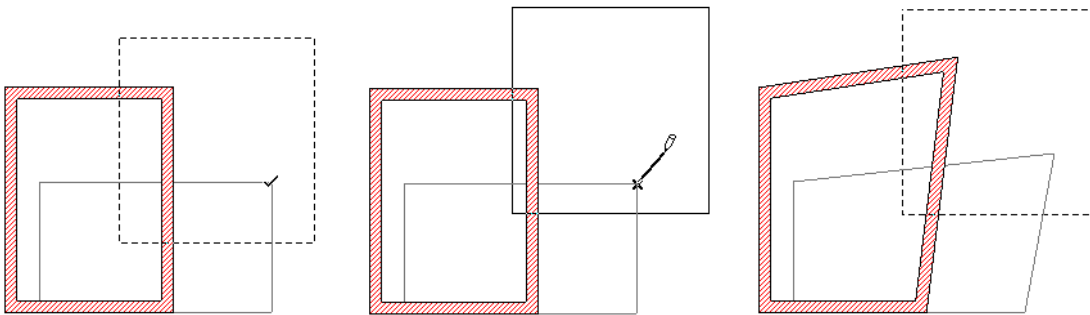
You can stretch multiple polygons or linear elements along a particular vector by using the Marquee tool.

To stretch elements inside a Marquee area:

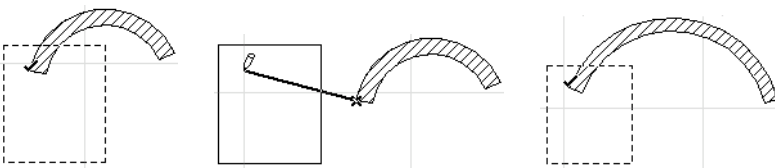
- Draw a Marquee so that the nodes or endpoints you want to stretch fall inside the Marquee. Any nodes you want to keep intact should be outside the Marquee.
- Define a stretch vector: with the Marquee tool active, click on a node or endpoint inside the Marquee area, then click on a second point.
- Alternatively, choose the **Edit > Reshape > Stretch** command, and then define the stretch vector by clicking any two different points.

The shape of all linear elements that have one of their endpoints inside the Marquee area, and polygonal elements that have nodes inside the marquee area, will be stretched along this vector.

Note: If you don't want to stretch an element whose node is inside the Marquee, lock the element (select it, then use **Edit > Locking > Lock**; or lock the element's layer.).



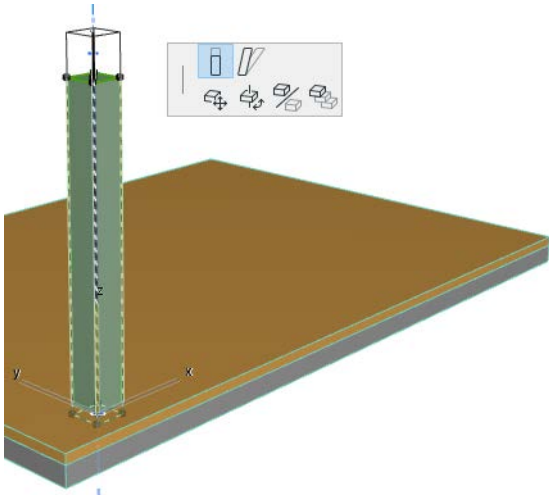
- If all the nodes of a polygon are inside the Marquee area, the polygon will be dragged instead of stretched.
- Stretching nodes in a Marquee area is not available in the 3D Window.
- **Columns, Objects** and **Lamps** cannot be stretched with the Marquee. If any of their hotspots fall inside the Marquee area, the whole object/column will be dragged along when the Marquee area is repositioned.
- When stretching **Arcs** or curved **Walls** with the Marquee, their central angle (i.e. the arc/chord ratio) will remain unchanged.



Stretch Height

In the **3D, Section/Elevation and IE windows**, you can modify the height of selected elements graphically with **Stretch height** command of the pet palette.

Click a node of a selected element and choose the vertical stretch icon in the pet palette. A ghost contour of the element follows the cursor. Click to set the new height of the element.



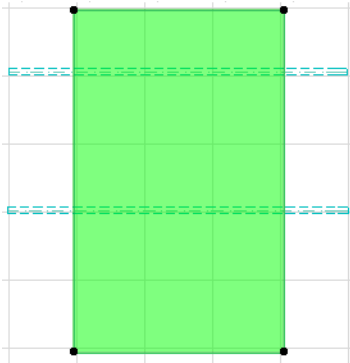
Resize (Enlarge or Reduce) Elements

The **Edit > Reshape > Resize** command lets you enlarge or reduce selected elements using numeric or graphical input.

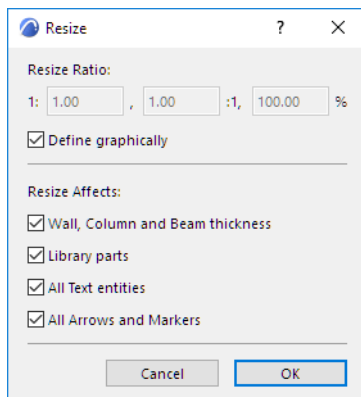
This feature is available only in the Floor Plan and 3D windows, and - for 2D drawing elements only - in the Section/Elevation, 3D Document and Detail/Worksheet windows.

The Resize dialog box offers additional options for resizing the thickness of Walls/Columns, GDL Objects, text elements, arrowheads and markers.

1. Select the elements you wish to transform and choose **Resize**.



2. In the appearing dialog box, enter a value into any one of the resize ratio fields. (Make sure the **Define graphically** checkbox is unchecked.)



- **Resize ratio:** Use any one of the three ways to define the resize ratio you want. (If you enter a value in one field, the other two will be filled in automatically.)
 - **Resize wall, column thickness:** For walls, columns and beams, the thickness will be resized as well as the length/height.
 - **Resize library parts:** Any selected library parts will be resized.
 - **Resize all text entities:** All texts and labels are resized.
 - **Resize all arrows and markers:** The size of all arrows/markers are modified.
3. Click OK.
 4. Click in the window to define which point or edge of the elements should remain in its original location.

To perform the operation graphically:

1. Open the **Resize** dialog box.
2. Check the **Define Graphically** checkbox.
3. Draw a transformation vector to define both the resizing ratio and the location of the resized elements.

Note: Resizing does not affect grouped elements. To resize grouped elements, ungroup them first or choose **Suspend Groups**.

See [Group Elements](#) for more information.

Splitting Elements

You can split many selected elements (Walls, Beams, Lines, Slabs, Roofs, Meshes, Fill and Zone Polygons, Lines, Arcs, Polylines and Splines) along a line segment, arc or element edge.

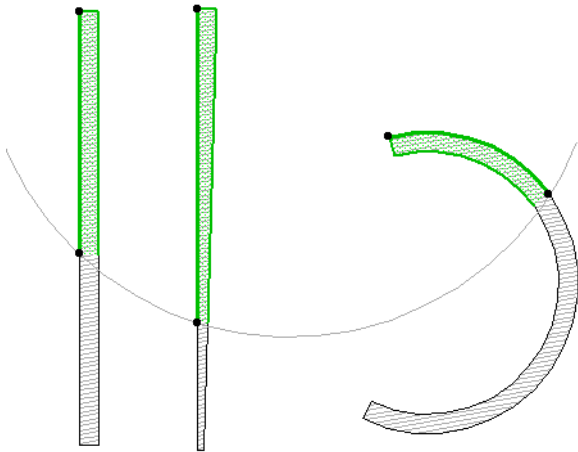
The **Split** command is available in the Floor Plan and 3D Window, and - for drawing elements only - in Section/Elevation/IE and 3D Document windows, and Details and Worksheets.

Note: You cannot split a polygon - such as a Roof element - with an arc.

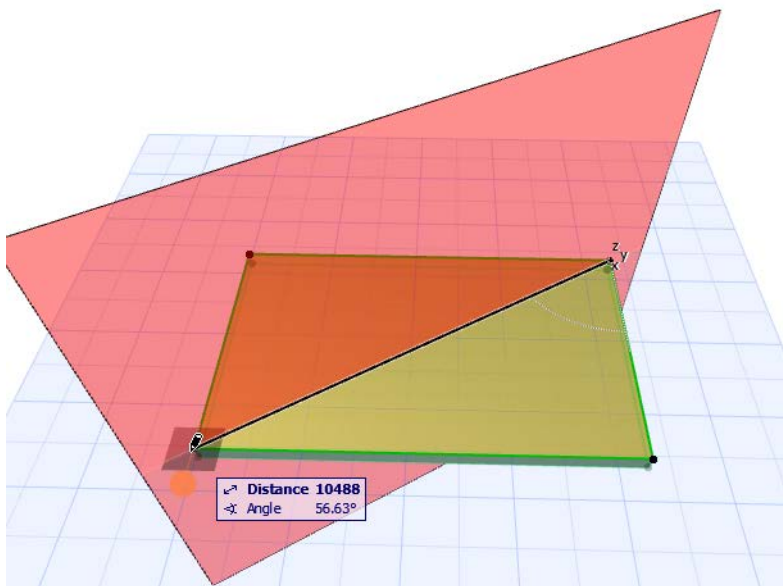
The splitting plane is always perpendicular to the Floor Plan.

Exception: If you are splitting a Morph, you have more freedom: the splitting plane can be set perpendicular to the current editing plane.

1. Select the elements you wish to split.
2. Choose the **Edit > Reshape > Split** command, or the hatchet icon from the Info Box.



3. Draw a temporary splitting line, or click on an existing Line, Arc, Wall or polygon edge. In the 3D window, a red splitting plane is shown as feedback.

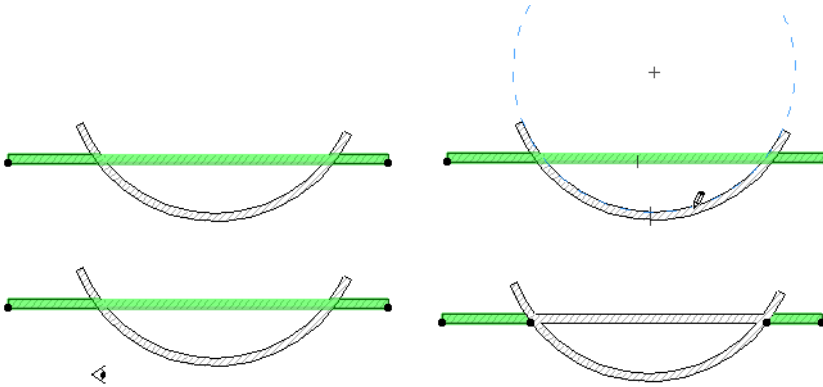


4. Click on either side of the splitting line/arc/edge with the Eyeball cursor.

Note: When splitting walls, the split takes place at the point where the splitting line intersects with the reference line(s) of the selected elements.

5. Elements on the clicked side will remain selected, while elements on the other side will be deselected. The selected group of elements can be edited in the usual ways.

If the selected element is intersected by an element in several locations, the **Split** command will split the selected element at every intersection point.



Splitting is also available in the 3D Window. You can split the selected element with a vertical plane into two segments. This feature works the same way as on the Floor Plan. The splitting line is drawn in the plane of the User Origin.

Note: When no elements are selected, the **Split** command will let you split a wall at the clicked location along any of its sides or reference line.

Related Topics

[Split Intersecting Curtain Walls](#)

[Split Stair Structure](#)

[Read More](#)

[Watch the Video](#)

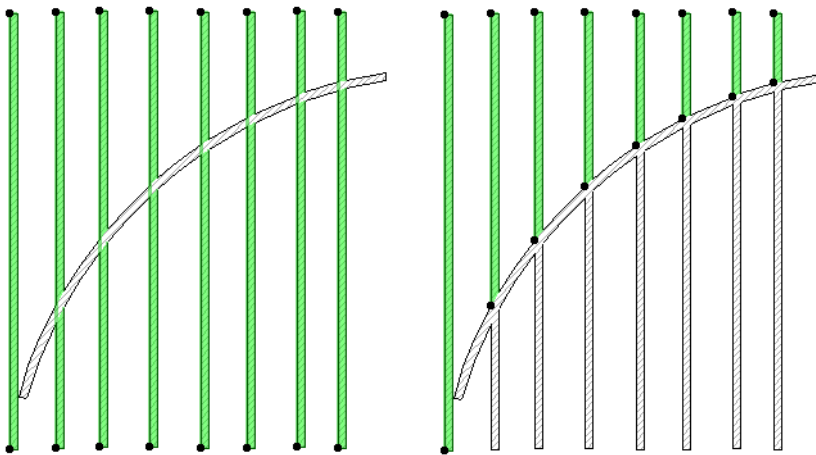
Adjusting Elements

Use the **Adjust** command to extend or trim the endpoints of selected Walls, Beams, Arcs and Lines to a Line, Arc segment or element edge.

Note: It is also possible to adjust Roof planes to each other or to other elements.

See [Intersect Single-plane Roofs and Cropping with a Remote Roof](#).

1. Select the elements you wish to adjust.
2. Choose the **Edit > Reshape > Adjust** command.
3. Draw a line segment, or click an existing line, wall, polygon edge or arc/circle. The endpoints of the selected walls and lines will be adjusted (lengthened or shortened) to meet the drawn or clicked line or curve. Only those elements will be affected that intersect (or would intersect) with the chosen line/arc/edge.



Note: If the clicked edge is that of a **Slab**, the workflow is the same as the **Adjust Elements to Slabs** workflow.

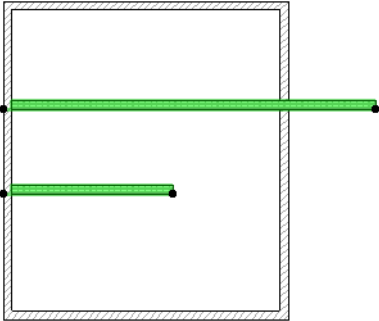
See [Adjust Elements to Slabs](#).

The **Adjust** command is available in the Floor Plan and 3D Window, and - for drawing elements only - in Section/Elevation/IE and 3D Document windows, and Details and Worksheets. In 3D view, you can adjust the selected Wall or Beam to a vertical plane.

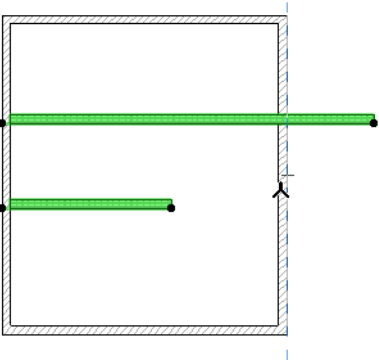
Split and Adjust Shortcut

A simple shortcut lets you adjust short elements and split long ones by the same edge.

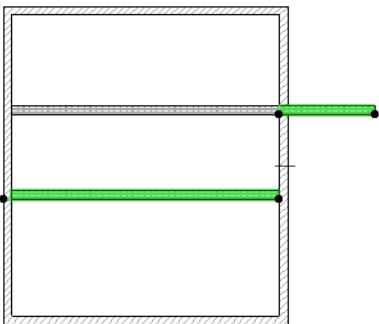
1. Select all the elements that you wish to split or adjust. The active tool must be one of those that created the selected elements.



2. Press the Ctrl (Windows) or Cmd (Mac) key and click the edge you wish to use for splitting/adjusting.



3. The operation is immediately performed. All transformed elements will remain selected.



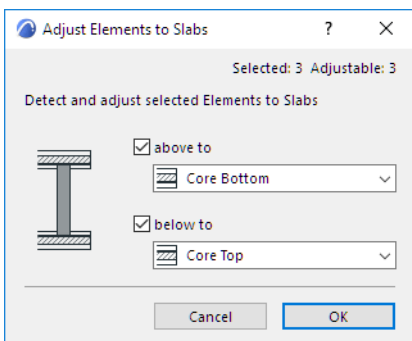
Adjust Elements to Slabs

Use this function (**Edit > Reshape > Adjust Elements to Slabs**) to easily adjust elements (Walls, Columns, or horizontal Beams) to specific levels of one or more Slabs that are located above and/or below them.

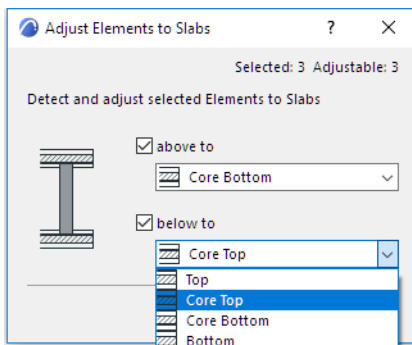
1. Select the element(s) (Walls, Columns or horizontal Beams) to be adjusted.

Note: If there is more than one Slab above or below the selected element(s), then you should also select the particular Slab that you want to adjust to. Otherwise, the closest Slab will be automatically considered the target.

2. Use the **Edit > Reshape > Adjust Elements to Slabs** command.
3. In the appearing dialog box, define whether you want to adjust the selected elements to the Slab that is **above** or **below** the element(s), or both. Again, if you select Slab(s), then the adjustment will target the selected Slab(s) only.



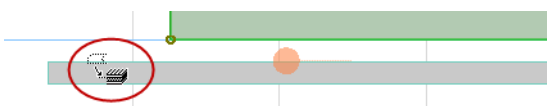
4. From the drop-down menus, choose the level of the Slab (top or bottom, or - for composites - Core Top or Core Bottom) to which to adjust the element(s).



5. Click **OK**.

Note: If a Door/Window is linked to the bottom of a Wall, the Adjust Elements to Slabs command will automatically recalculate the offset to retain the opening's position.

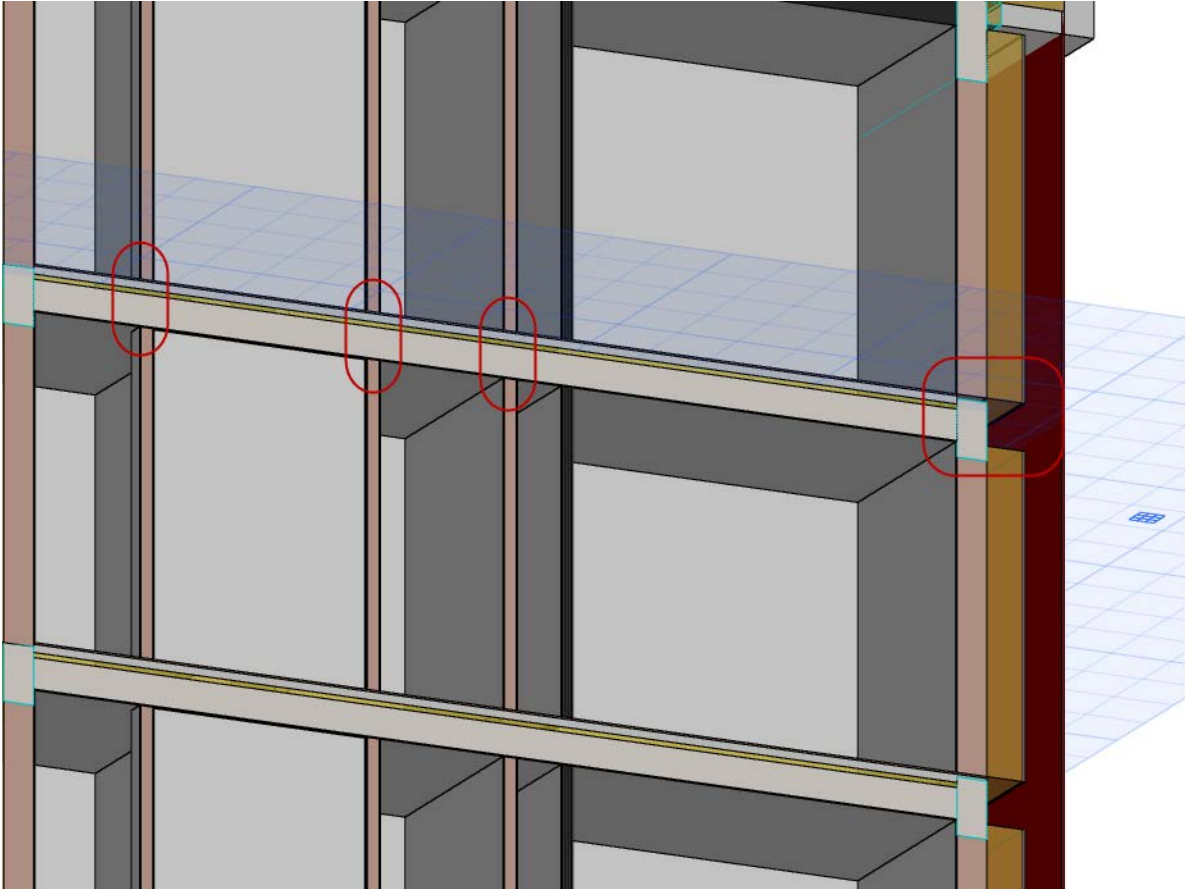
Note: The same workflow is available by using the **Edit > Reshape > Adjust** command. In that case, you must select the elements to be adjusted, then click on the targeted Slab. (The cursor indicates that you are clicking a Slab, as illustrated below.) The **Adjust Elements to Slabs** dialog box appears, and the workflow continues as described above. (Elements are adjusted to the clicked Slab only.)



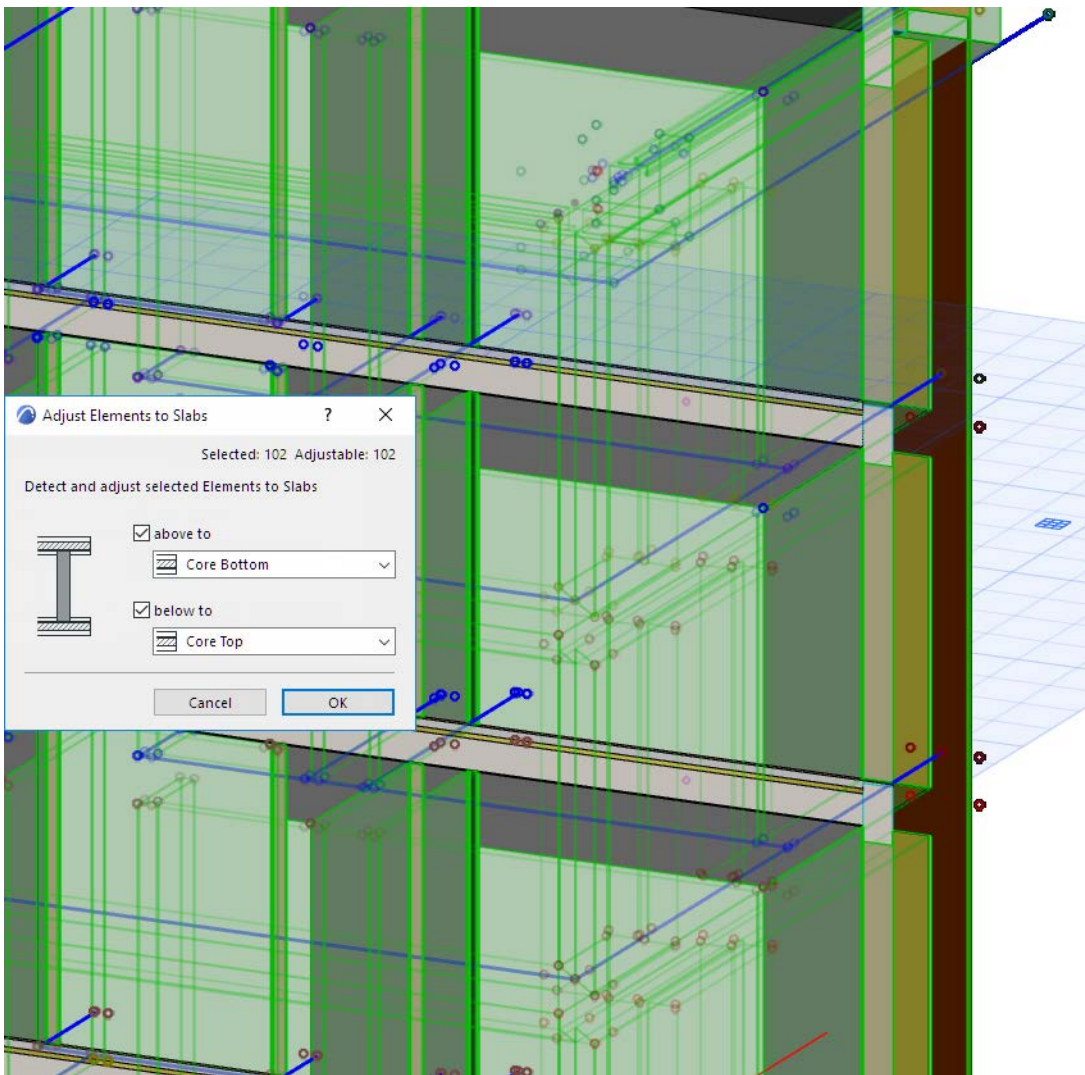
Adjust Elements to Slabs: Example

In this project, there are problems in the junctions of Walls to Slabs: the internal Walls only extend to the top of the carpeting, while the external (profile) Walls have gaps.

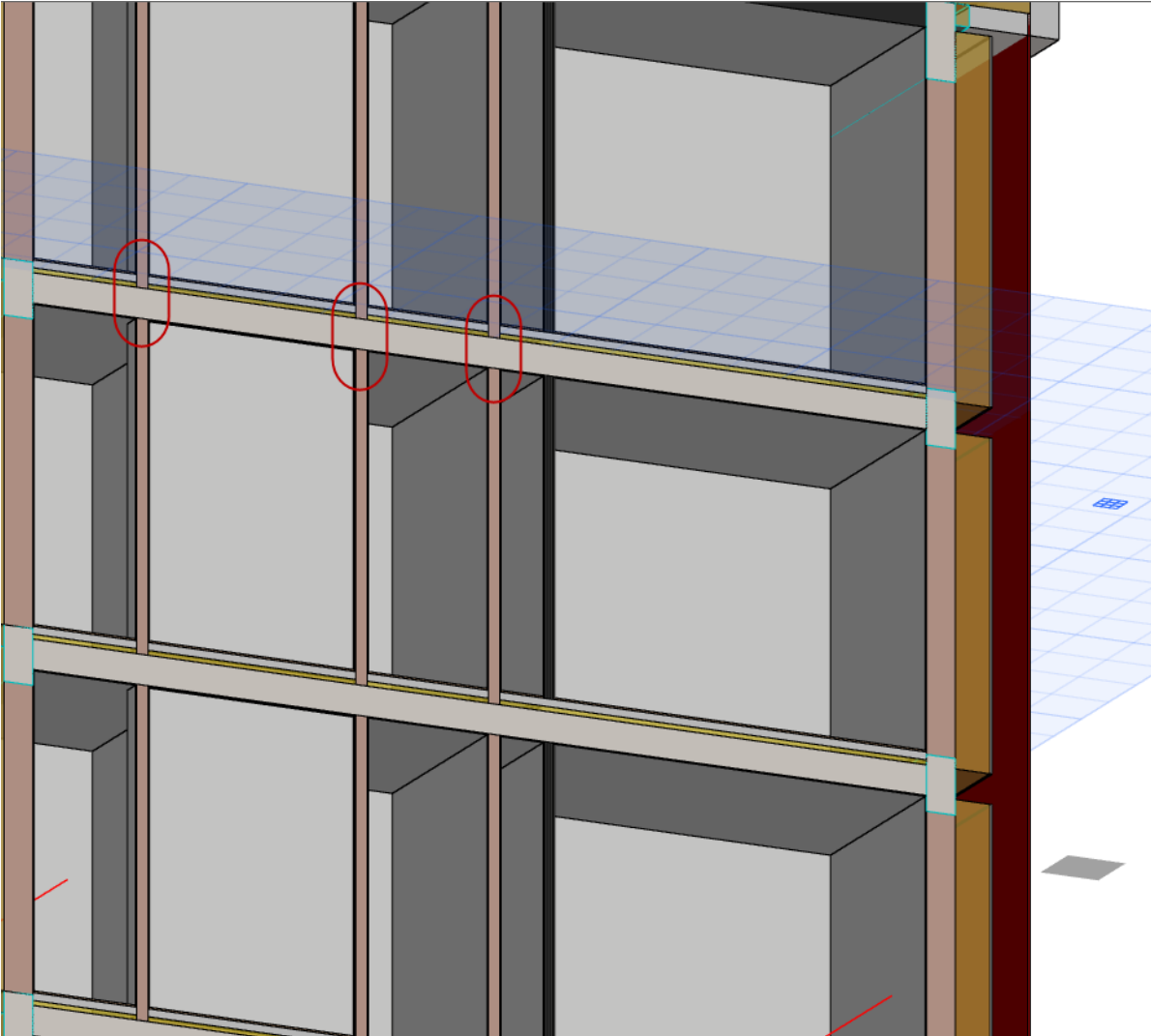
The Adjust Elements to Slabs command can help.



1. Select all Walls.
2. Use the Adjust Elements to Slabs command and set the options to “Core Bottom” and “Core Top”. This way, the Walls on top of the Slabs will be extended to reach to the Slab core.

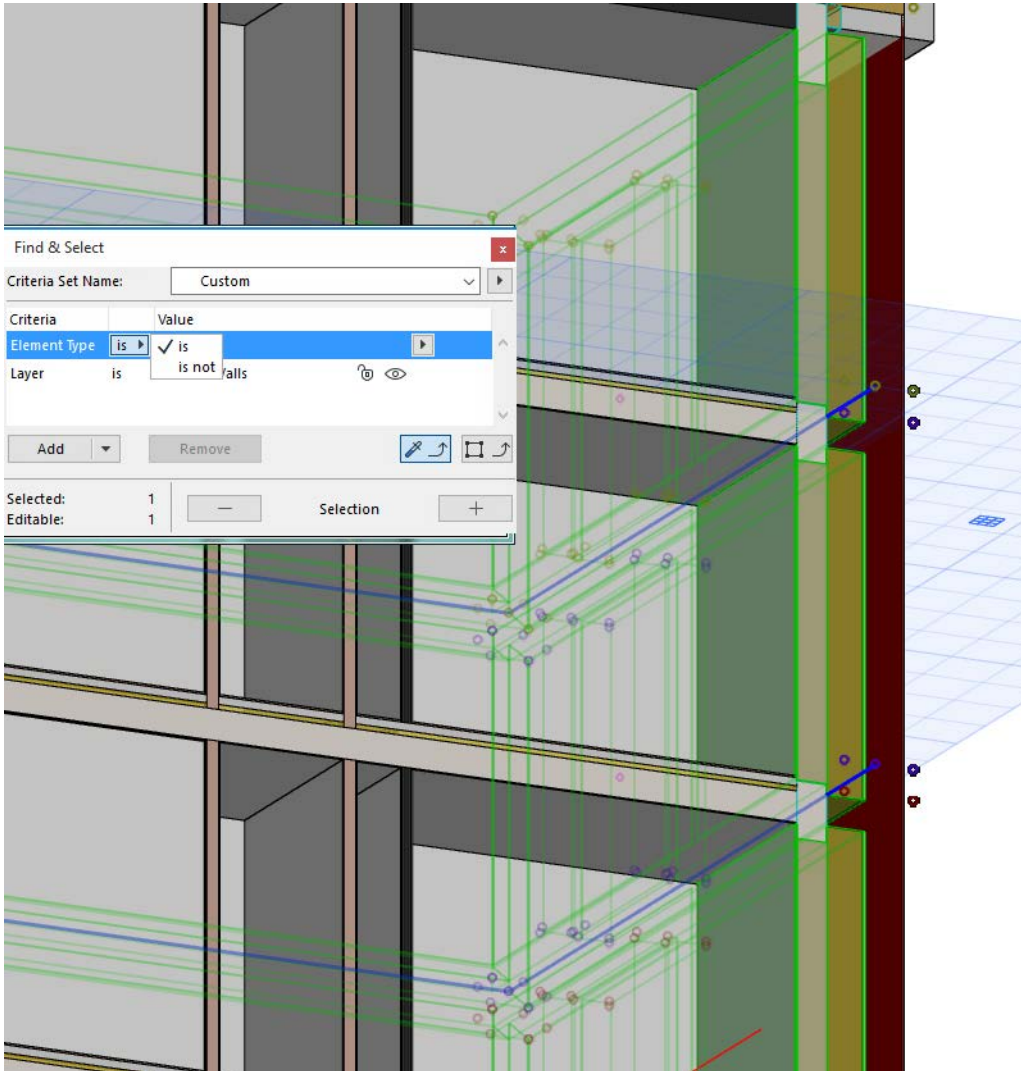


3. As a result, the internal dividing Walls are now correctly joined to the Slabs.



The external, profiled Walls are still a problem: each needs to meet the Wall above it without stopping at the bottom of the Slab core.

4. Use Find and Select to select the external Walls only.



5. Again, use Adjust Elements to Slabs, but this time, all of the selected Walls should extend all the way up to the Slab core top, to eliminate the gaps.

Create a Fillet or Chamfer

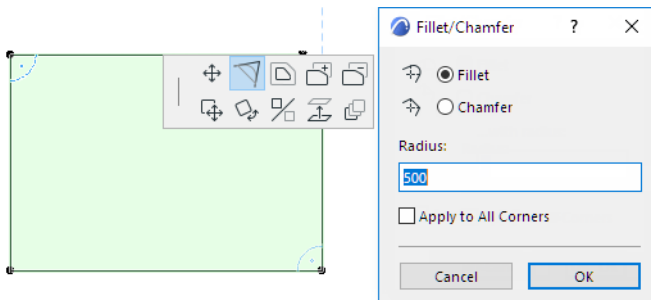
Use the **Edit > Reshape > Fillet/Chamfer** command to fillet or chamfer the intersection of two straight lines/polylines, walls, or the corner(s) of a polygon element.

For special fillet/chamfer options available with Morphs, see [Basic Morph Editing: Fillet/Chamfer a Morph](#).

To open the Fillet/Chamfer dialog box, select the two lines, or the polygon element.

Then do one of the following:

- Choose the **Edit > Reshape > Fillet/Chamfer** command, or
- If the selected element is a polygon, choose the Fillet/Chamfer icon from the pet palette.



Choose either Fillet or Chamfer:

- **Fillet** joins the endpoints of two straight segments endpoints with an Arc.
- **Chamfer** joins the endpoints of two straight segments with a straight segment, cutting off corners.
- Enter a radius for the fillet/chamfer. (The chamfer does not have a radius, but it will be drawn as the chord of arcs of the specified radius.)

Note: If filleting the arc with the current radius would extend any of the neighboring vertices, the radius will be limited automatically so that the filleting arc contains the closer neighboring vertex.

- Click **OK**.

Note: The **Fillet**, **Chamfer** and **Intersect** commands are not effective on grouped elements, unless Suspend Groups is on.

See [Suspend Groups](#).

Apply to All Corners

To apply the Fillet or Chamfer to all corners of the selected polygon, check the **Apply to All Corners** checkbox.

Note: This checkbox is applicable only if the selected element is a polygon.

If you have selected a polygon and then the **Edit > Reshape > Fillet/Chamfer** command, the checkbox is selected by default and uneditable; the fillet/chamfer will apply to all nodes.

To apply the Fillet/Chamfer to a single node of the polygon, click on that node and use the pet palette command to open the Fillet/Chamfer dialog box.

Trim Elements Using Scissors

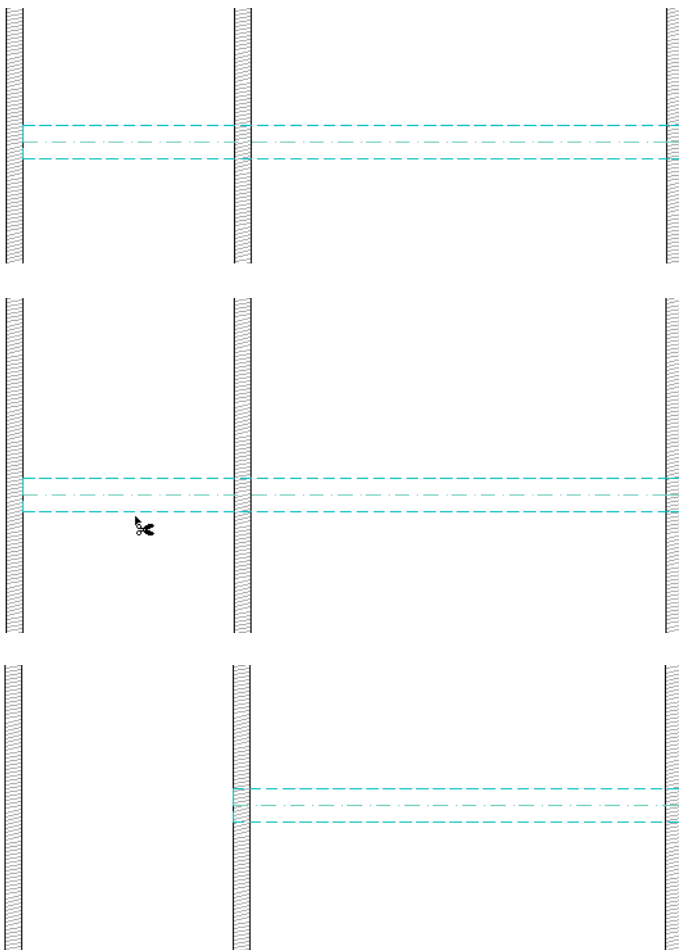
Use the Trim command to remove the part of an element extending beyond its intersection point with another element, or to cut out part of the element between two intersection points.

Trimable elements: Walls, Beams, Lines, Circles, Arcs, Polyline and Splines.

Do the following:

- Choose the **Edit > Reshape > Trim** command, or press the **Ctrl (Windows)** or **Cmd (Mac)** key.
- The Scissors cursor appears. Move it to a trimmable element and click the segment you want to trim.
Note: In the 3D window, you can also click on a Wall or Beam surface for trimming.
- The clicked part of the element between the two nearest intersection points will be deleted.

For example, here we will trim part of the beam - the part that falls between the first two walls on the left.



Trim Elements to Roof or Shell Cropping with a Remote Roof

Adding Element Nodes

You can add new nodes to elements using the pet palette.

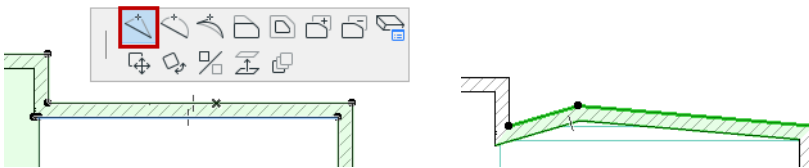
Adding a node to a straight linear element creates an additional element.

Adding a node to a polygon-type element increases the number of its edges.

In the 3D Window, you can add nodes only to polygonal elements.

To add a node to an element:

1. Select the element.
2. Click an edge of the element to bring up the pet palette.
3. Choose the Insert new node icon.
4. Click to define the location of the new node.



To insert a new node without moving it, double-click on a polygon edge. (The Insert node option must be active in the pet palette.)

Note: You cannot add a new node to a curved element or segment; if you click **Insert new node** on a curved element, the curved element will be straightened out.

Move Nodes

With the **Move node** icon of the pet palette, you can reposition the node of a polygon type element.



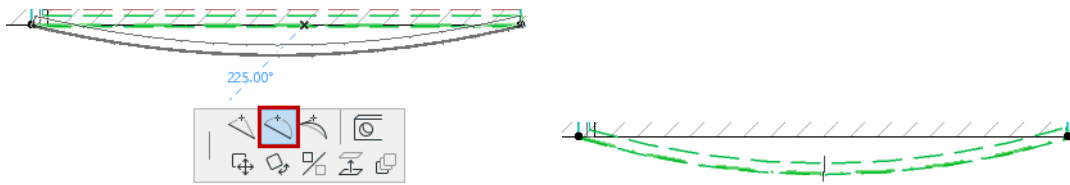
Also, the Move Node command is used to stretch objects, doors and windows.

See [Stretching Objects](#).

- You can eliminate a node by merging it into one of its neighbors.

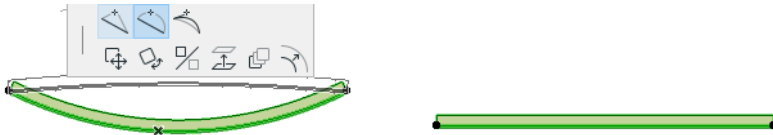
Curve/Straighten Element Edge

With the pet palette's **Curve edge** function, you can curve a segment of a straight element - such as a wall or beam - by dragging its edge or reference line.



Note: If the bent segment of a polygon intersects any of the other polygon edges, Archicad will regularize the polygon's shape, which may result in cutting the polygon into several parts.

To straighten a curved element edge, select the curved element, then select the **Insert New Node** command from the pet palette.



For more information on editing curved elements, see [Stretching Curved Elements](#).

Explode into Current View

This command is available in different forms depending on the selected items:

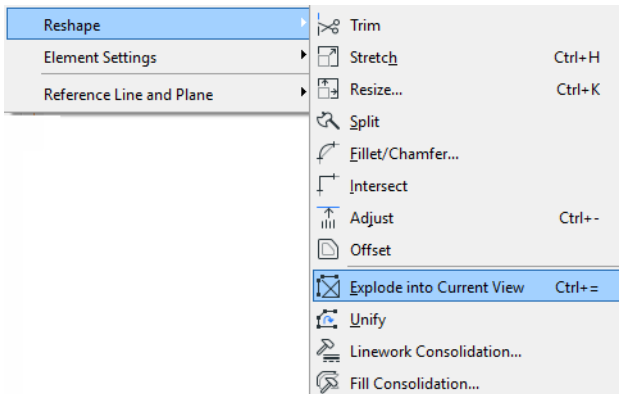
For Selected Construction Elements

For Placed Drawings (Source is not a PDF)

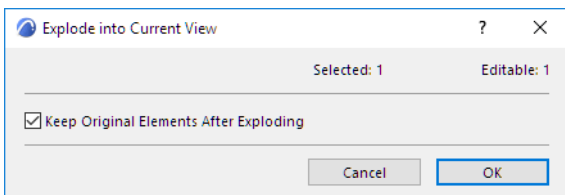
For Placed Drawings (Source is a PDF)

For Selected Construction Elements

Choosing **Edit > Reshape > Explode into Current View** will turn selected elements (Floor Plan and other 2D windows only) into 2D drawing primitives (Lines without arrowheads, Circles, Arcs, Elliptic arcs, Splines, unframed Fills, single-line Texts).



In the appearing dialog box, you have the option to keep original elements. If you uncheck this box, the original elements will be deleted, and only the exploded primitives will remain in the project.



Note: The Explode command will not affect grouped or locked elements. Turn on **Suspend Groups** before exploding grouped elements.

Note: If you explode a construction element with the “Keep Original Elements” box unchecked, then that element will no longer exist as a construction element. Consequently, it will disappear from other model views (model-type Sections, Elevations, IEs, 3D Documents).

Some examples:

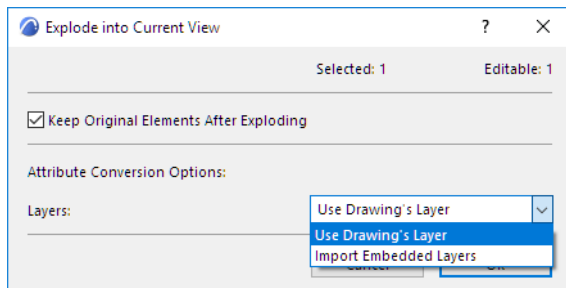
- Slabs, Roofs and Meshes are exploded into lines
- Walls and Columns are exploded into Lines and Fills (Doors and Windows inserted into Walls are also exploded)
- Doors and Windows (if selected without the Wall they are placed into) are exploded into 2D elements and replaced by empty hole openings
- Dimensions are exploded into Lines, Texts and the basic elements of arrowheads (Lines, Circles, Arcs, Fills)
- Polylines are exploded into Lines and Arcs

- GDL Objects are exploded into basic 2D drawing elements

Explode has no effect in the following cases:

- In Section/Elevation/IE and 3D Document windows, cut construction elements cannot be exploded
- Section/Elevation/IE lines and Cameras
- Basic 2D elements

For Placed Drawings (Source is not a PDF)



This dialog box appears if you select and explode a Drawing that has layers (that is, if it is a DWG file or if it is derived from an Archicad view), and does not have a PDF document as its source.

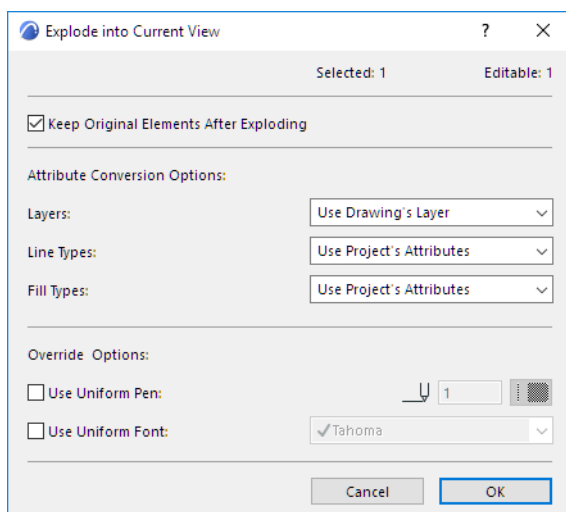
Attribute Conversion Options

Note for Teamwork users: If you choose an option that creates new attributes, you must first have the needed access right (e.g. Create Layers). If you do not have the right, a warning appears, and the option is grayed out.

Choose the Layer on which to place the drawing primitives that will result from the Explode:

- **Use Drawing's Layer:** All resulting drawing primitives will be placed on the same Layer as the Drawing is on.
- **Import Embedded Layers:** This option means that the drawing primitives will be placed on layers corresponding to their layer assignment in the Drawing source (e.g. DWG file). Those embedded layers will be imported into your Archicad project.

For Placed Drawings (Source is a PDF)



This dialog box appears if you select and explode a Drawing that has a PDF document as its source. The Attribute Conversion and Override options allow you to control the appearance of the resulting elements.

However, be aware that the elements from the PDF might not look exactly the same after Explode - e.g. what appears to be a fill in the PDF document may be composed of lines, and will be exploded as lines in Archicad.

Attribute Conversion Options

Note for Teamwork users: If you choose an option that creates new attributes, you must first have the needed access right (e.g. Create Layers). If you do not have the right, a warning appears, and the option is grayed out.

Layers: Choose the Layer on which to place the drawing primitives that will result from the Explode:

- **Use Drawing's Layer:** All resulting drawing primitives will be placed on the same Layer as the Drawing is on.
- **Import Embedded Layers:** This option means that the drawing primitives will be placed on layers corresponding to their layer assignment in the Drawing source (the PDF document). Those embedded layers will be imported into your Archicad project.

Lines: Choose a method for defining the drawing primitives' lines.

- **Use Project's Attributes:** No new Lines will be imported from the Drawing's source file; the project's existing lines will be used. If there is no perfect match, then the drawing primitives will use the one of four basic Lines - solid, dashed, dotted-and-dashed, or dotted - which corresponds to the exploded element's original Line.
- **Substitute With Similar:** New Lines are imported only if the drawing primitive does not correspond to any of the four basic Lines.
- **Import All:** All Lines from the PDF source are imported and added to the Archicad project attributes.

Fill Types: Choose a method for defining the drawing primitives' fill types.

- **Use Project's Attributes:** No new Fill Types will be imported from the Drawing's source file. Any fills in the drawing primitives that have no counterpart among the project's attributes will be transformed into solid fills.
- **Generate Symbol Fills:** For any fills in the drawing primitives that have no counterpart among the project's attributes, a new vectorial fill attribute will be created.
 - Vectorial fill patterns will be transformed to symbol fills without any text or image
 - Text or image content and gradient fills will be transformed into image fills
- **Generate Image Fills:** For any fills in the drawing primitives that have no counterpart among the project's attributes, a new image fill attribute will be created.

Note: If an image fill is generated, then an image file is added to the project's Embedded Library. If you are working in a Teamwork project, you must have the needed access right: Library Parts - Create.

Override Options

- **Use Uniform Pen:** Check this box to use a single pen defined here for all texts, fill contours and lines.
- **Use Uniform Font:** Check this box to use a single font defined here for all texts.

Fine-Tune View After Explode

After an Explode function, you may wish to fine-tune the view before final output.

You will find that the "exploded" lines and fills often contain superfluous elements (extra line segments, overlapping or superfluous fills) that make such editing difficult. To make editing easier, first use the **Linework** and **Fill Consolidation** functions on selected items in the window.

For more information, see [Consolidate Lines and Fills in Drawing Windows](#).

Duplicate Elements

You will often need to create exact duplicates of a particular element. Often, the simplest way to duplicate is to use Copy/Paste.

[See *Basic Editing Operations*.](#)

Copy/Paste Elements Between Stories

The Story Settings dialog box provides a quick way to copy elements from one story to another without having to redraw them for each story.

[See *Story Settings Dialog Box*.](#)

It is also easy to use graphical editing to drag, rotate, or mirror one or more copies of a selected element, or to use the multiply function. These are described in the following sections:

[Drag, Rotate, Mirror Element Copies](#)

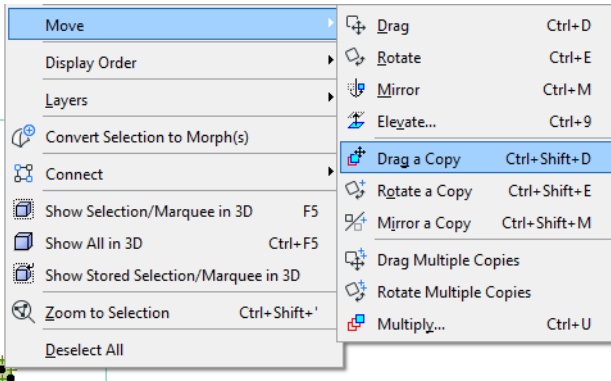
[Multiply Elements](#)

[Offset All Edges](#)

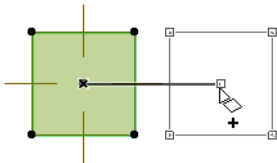
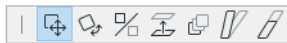
Drag, Rotate, Mirror Element Copies

To **Drag**, **Rotate**, or **Mirror one copy** of an element, select it, then do one of the following:

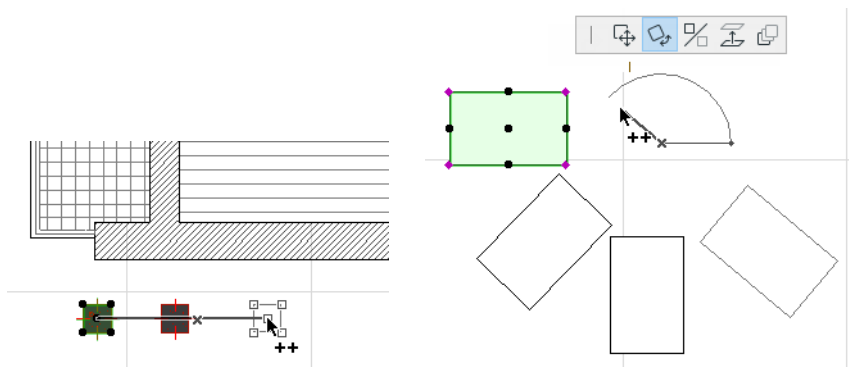
- Choose the desired command from the **Edit > Move** menu or the context menu.



- Choose Drag/Rotate/Mirror from the pet palette, then press **Ctrl** (Windows) or **Alt/Opt** (Mac). (This adds a small + sign to the cursor; press Ctrl again to undo the Copy feature). Click to place the copy, which will now be selected instead of the original.



- To **Drag** or **Rotate multiple copies** of an element, select the element, choose the desired command from the **Edit > Move** menu or the context menu (or use **Ctrl + Alt** (Windows) or **Cmd + Opt/Alt** (Mac) as a shortcut). You will see a “++” sign appear. Click the element to drag or rotate its copy to its new position. You can place any number of copies. Double-click to finish placing copies.



Note: When rotating copies, they will all be placed along the same rotation axis.

This feature works in the Floor Plan, Section/Elevation/IE windows and the 3D Window. In Detail Drawings, you can use it on any drawing elements. In the 3D Window, Drag/Rotate/Mirror is available only for added Drawing elements.

To create one or more copies of an element that are offset from (rather than exact duplicates of) the original:
[See *Offset All Edges*](#).

Multiply Elements

Multiply creates any number of exact copies of selected elements, according to the pattern and method defined by you. Set up the options in the Multiply dialog box.

To open the Multiply dialog box, select the element(s), then do one of the following:

- **Move > Multiply** command from the pet palette
- **Edit > Move > Multiply**

Topics on this page:

[Choose An Action \(Drag, Rotate, Elevate, Matrix\)](#)

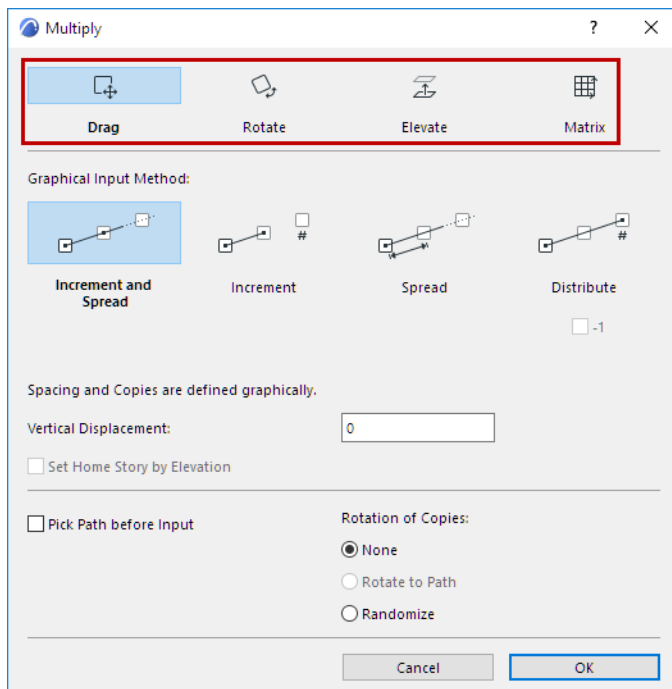
[Choose Graphical Input Method](#)

[Pick Path/Arc before Input](#)

[Vertical Displacement](#)

Choose An Action (Drag, Rotate, Elevate, Matrix)

At the top of the Multiply dialog box, choose a multiplying action:



- **Drag** multiplies the copies along a drawn straight path or clicked polyline
- **Rotate** multiplies the copies along an arced path
- **Elevate** stacks the copies with a vertical displacement. (Available only in Floor Plan and 3D Window.)
- **Matrix** places the copies in a rectangular matrix defined by two perpendicular reference lines.

Below, define an optional Vertical Displacement between the multiplied copies.

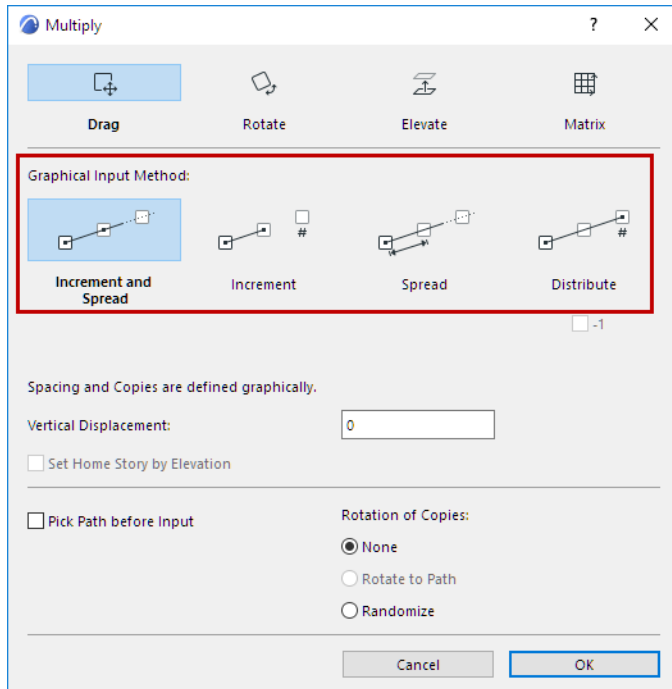
[See *Vertical Displacement*.](#)

With Drag and Rotate, you also have the option to pick a path/arc along which to place the multiplied copies.

[See *Pick Path/Arc before Input*.](#)

Choose Graphical Input Method

Choose one of the four input methods for placing your multiples. The input method determines the number of copies and the space between them (increment).

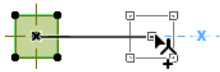


Increment and Spread

With this method, both spacing and copies are defined graphically.

Note: Not available for Elevate in the Floor Plan.

1. Click **Increment and Spread**, then OK to close the dialog box.
2. The first click defines the start point.
3. The next click defines the increment (distance between each multiplied copy).



4. Next, drag the cursor for any distance. Each copy will be placed at the defined increment.

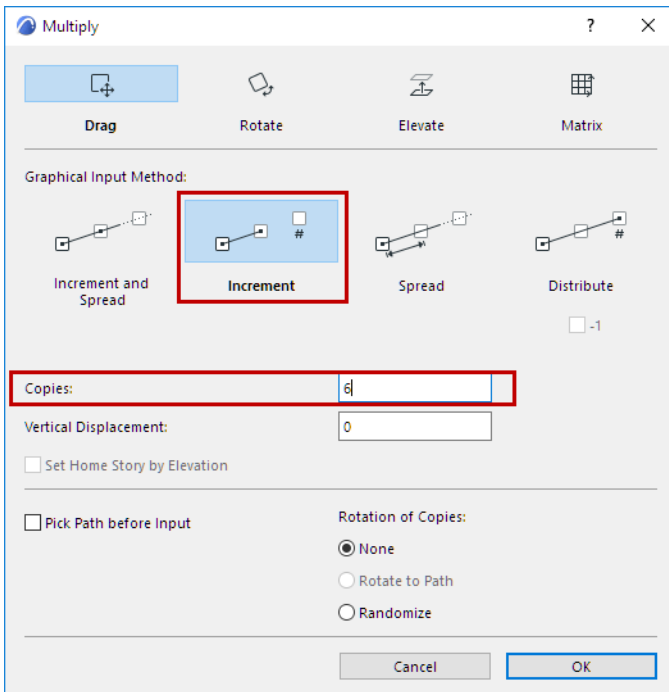


5. Click to complete the operation.

Increment

With this method, you predefine the number of copies, but you define their spacing graphically.

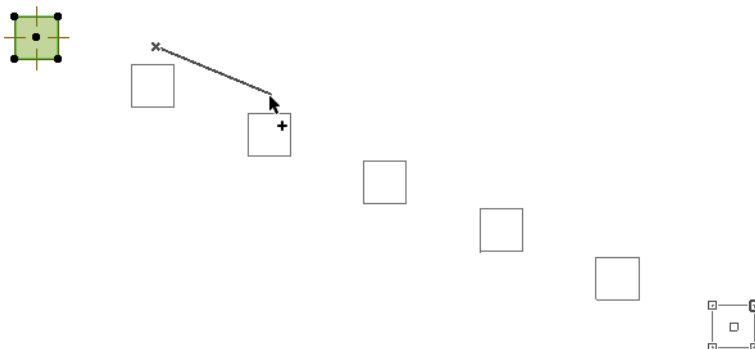
1. Click **Increment**, then enter the number of copies you want to make, then OK to close the dialog box.



2. The first click defines the start point.



3. Drag to define the spread and direction - the distance for Drag, the angle for Rotate - of the set number of copies.

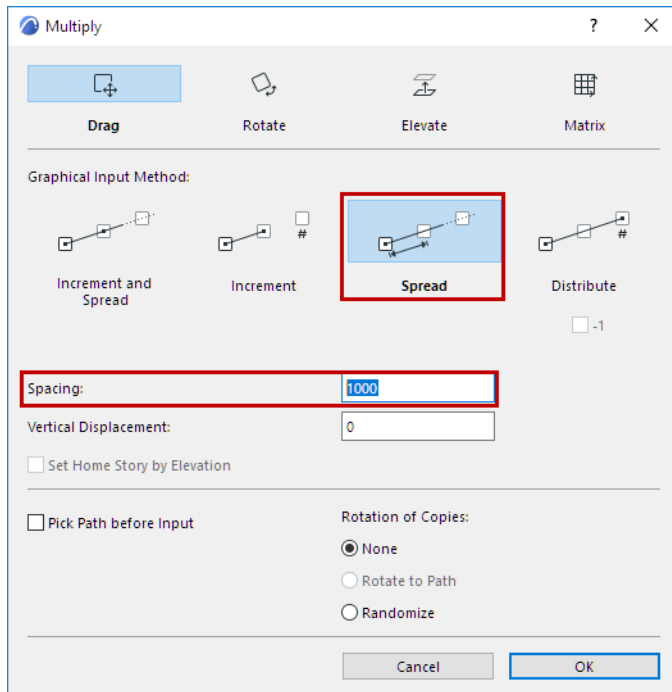


4. Click to place the multiplied copies.

Spread

With this method, you predefine the spacing, but you define the number of copies graphically.

1. Click **Spread**, then enter the fixed spacing between each multiplied copy, then OK to close the dialog box.



2. The first click defines the start point.
3. Drag to begin placing any number of copies, at the specified distance (for Drag) or angle (for Rotate).

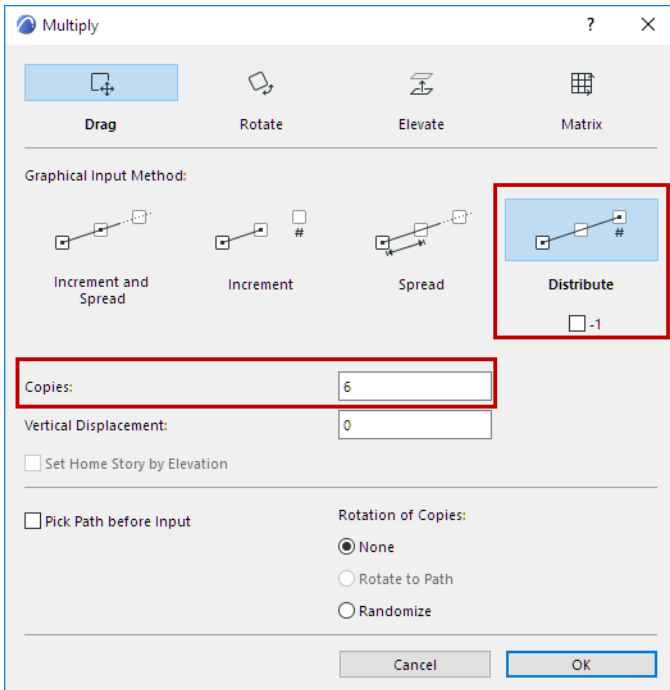


4. Click to place the multiplied copies.

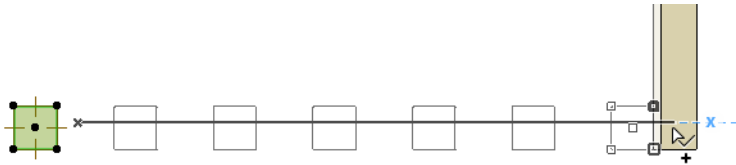
Distribute

With this method, the number of copies is predefined, and they are spaced evenly up to the endpoint you define graphically.

1. Click **Distribute**, then enter the number of copies to place between two points. Optionally, activate the **-1** checkbox to add an additional space within the same distribution distance (no copy will be placed at the end).
Click OK to close the dialog box.



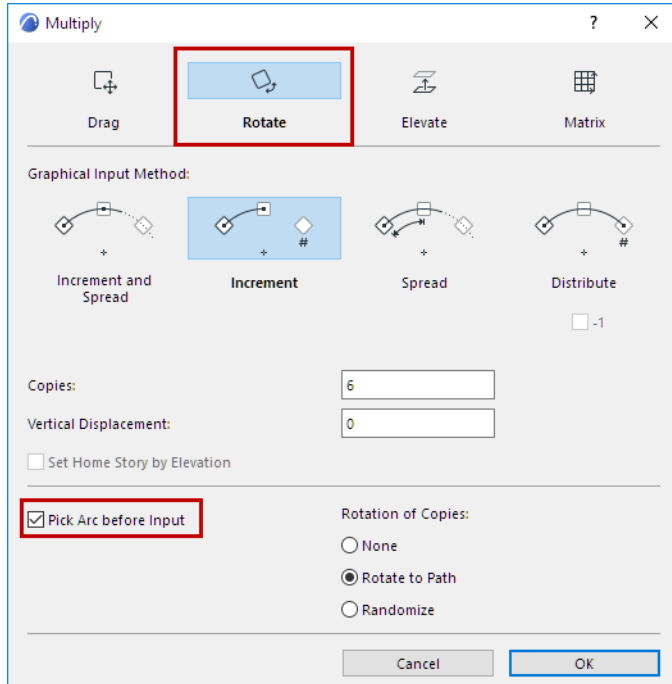
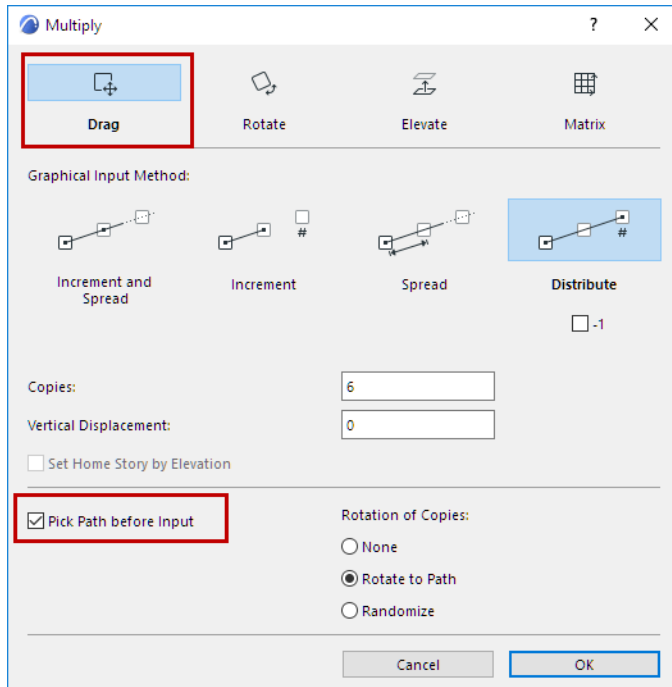
2. The first click defines the start point.
3. Drag to evenly distribute the set number of copies to the desired point in the project, then click to place.



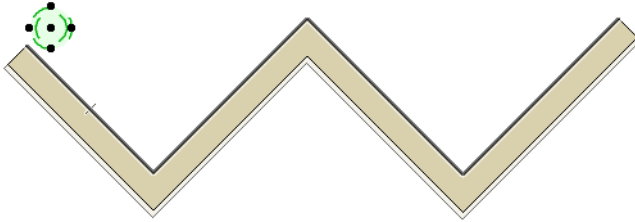
Pick Path/Arc before Input

This option is available in the Multiply dialog box with the Drag or Rotate methods.

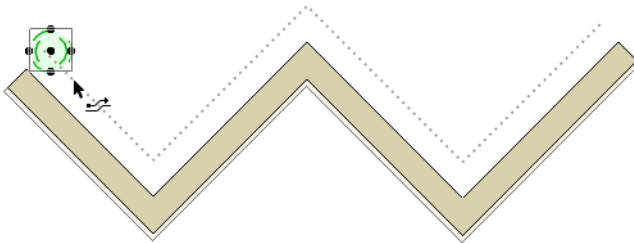
1. From the Multiply dialog box, activate the **Pick Path** or **Pick Arc** checkbox, then OK to close the dialog box.



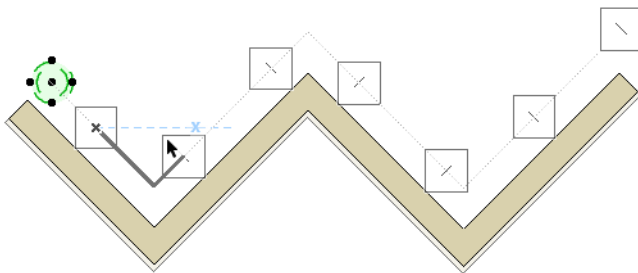
2. Hover with the cursor over any edge to auto-detect and highlight the path.
Press spacebar inside a closed area to auto-detect the boundaries.
3. Then click, defining the path or arc along which to place the multiplied copies.
(If no path is found, the input will proceed without a predefined path - as if the checkbox were off.)



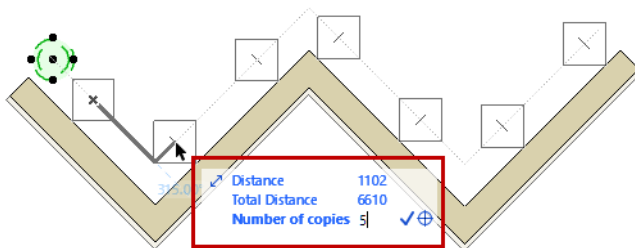
- Click on or near the selected element to multiply: this defines the rotation and reference point of the elements, and the start point of the multiply from this point.



- Drag along the path/arc to input the copies as usual (defining increments and/or copies according to the Graphical Input Method you chose in the Multiply dialog box).



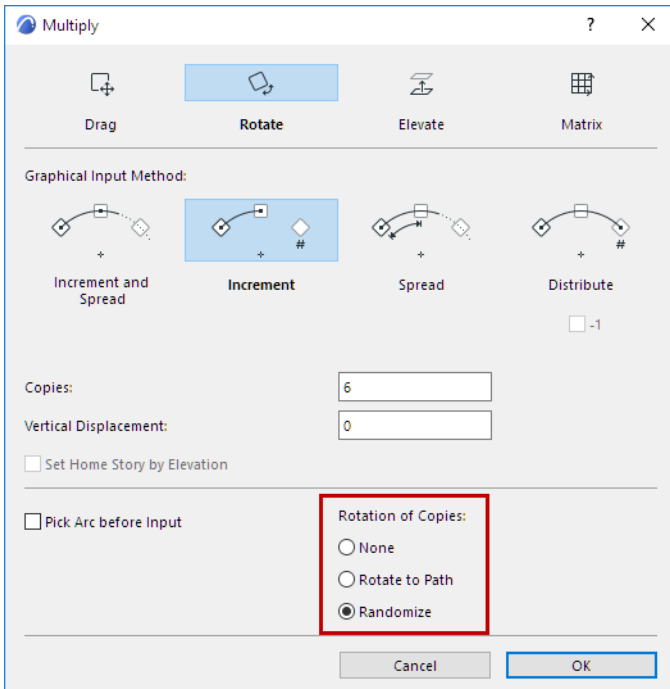
- You can still modify the final Distance, Spacing or number of copies in the Tracker before you click to complete.



Note: For a closed polygon path, the start point can be used as the finish point if the cursor has moved around the entire path.

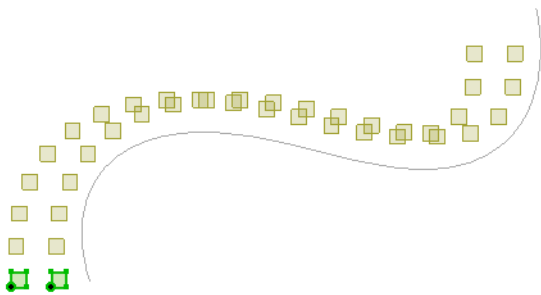
Rotation of Copies

With the Drag or Rotate actions, set a preferred option for rotating each copy:



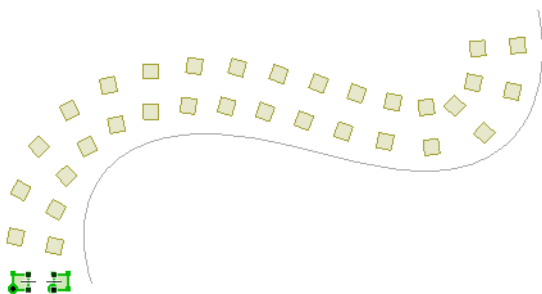
None

No rotation will be applied to the copies, every copy will have the same orientation and rotation as the original element(s).



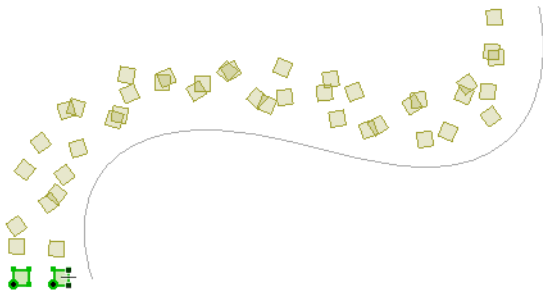
Rotate to Path

For the Picked Path or Rotate input methods, each copy will rotate the original selected element(s) to be perpendicular along the path/arc in relation to the start point of the input.



Randomize

Each copy will randomly rotate the original selected element(s) along the path/arc in relation to the start point of the input.

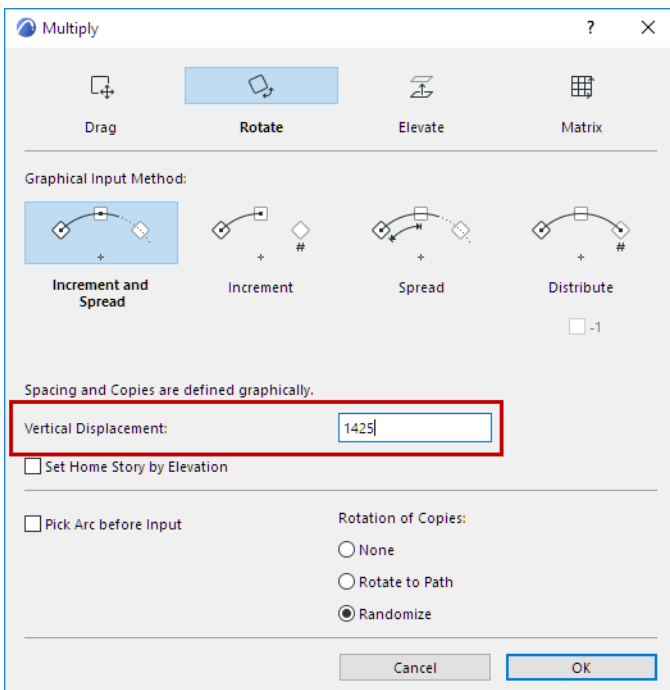


This can be useful for multiplying landscape elements, such as trees.

Vertical Displacement

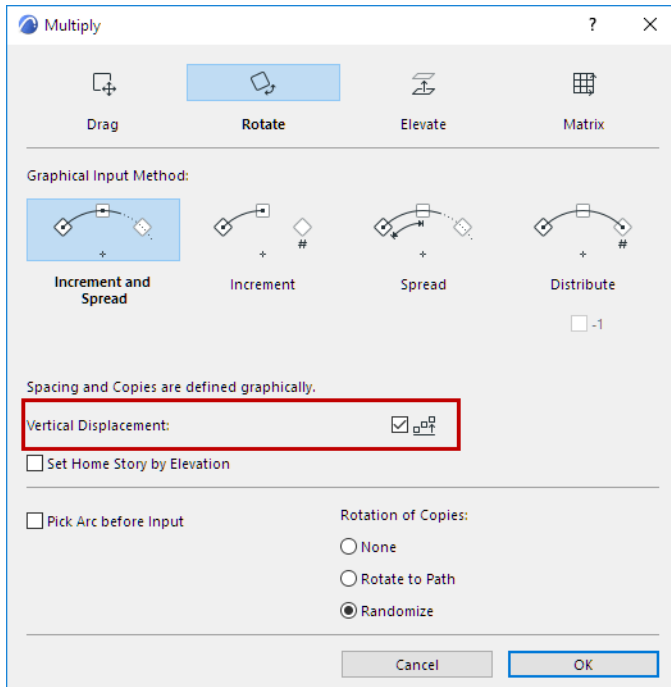
In the 2D window:

Use this field to enter a value that is added to the elevation of each subsequent copy of the multiplied object.

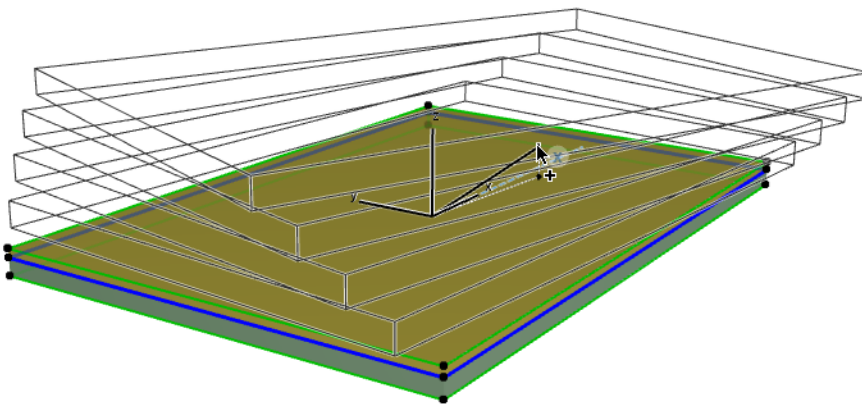


In the 3D window:

Optionally check the Vertical Displacement box.



When you close the Multiply dialog box and then carry out the Multiply operation, you will define the vertical displacement vector graphically, as shown with this slab:

**Set Home Story by Elevation**

Check this to assign a Home Story to newly created elements based on their respective story locations. If you do not check this box, new elements will have the same Home Story as the original element you are multiplying.

[See *Change Home Story by Elevation*.](#)

[Watch the Video](#)

Drag & Drop

Dragging and Dropping Text

The following possibilities exist for transferring text:

- From any Archicad Text Window or from a Text Window of any other Drag & Drop-capable application to any other Archicad Text Window
- From any Text Window to any 2D Window (Floor Plan, Section/Elevation/IE, 3D Document, Detail/Worksheet, Library Part 2D Symbol) with the default parameters
- Moving or copying text within the same Text Window
- Creating text type clipping files in the File Manager
- Pasting clipping files into a text type Window
- Dropping plain text files into a text type Window (with some restrictions)

Dragging and Dropping Drawings

The easiest way to add a Drawing from an external Archicad file is to open the external project structure in the Navigator, select a View or Drawing in the Navigator, and drag it onto the Layout in the Layout Window of the current project.

For more information, see [Place Drawing on Layout](#).

Dragging and Dropping Pictures

The following possibilities exist for transferring picture type data:

- Dropping a picture file into a GDL Object Preview Window
- Dropping a picture file into the Floor Plan to paste it as a Figure
- Creating picture type clipping files in the Mac Finder
- Pasting clipping files into the GDL Object Preview Window

Dragging and Dropping GDL Object Files

You can drag and drop GDL Object files from the Mac Finder or the file manager directly into Archicad Projects, the Object Settings dialog box or the Embedded Objects folder of the Library Manager.

Dropping an Object into the Floor Plan activates the corresponding tool in the Toolbox and the newly placed element becomes the default element for the given Object type.

- It is possible to place several GDL Objects at the same time using Drag & Drop.
- Windows and Doors can only be dropped into a Wall.

Dragging and Dropping Floor Plan Elements

If there is a selection on the Floor Plan defined either with a Marquee area or through individual selection with the **Arrow**, pressing the mouse button and dragging the cursor out of the Archicad Floor Plan Window will initiate a Drag & Drop operation.

The following can be copied:

- Floor Plan elements (in module format)
- Selected elements on the Floor Plan in picture format (for example, into the Project Preview Window)

See [Project Preview](#).

- The 2D and/or the 3D script of the selected elements in text format

3D GDL script is only generated if the drop is made into the 3D Script window. The 2D script of the selected element can be dropped into any other text type window.

Dragging and Dropping DWG/DXF Objects

Drag & Drop is available as a way to download DWG/DXF objects from the web.

Parameter Transfer

Use this function to pick up the parameters of one element and transfer them to another element of the same type.

Note: Line, Arc, Polyline and Spline elements are of the same type.

Certain basic parameters are always transferred.

You can also control the set of additional settings (“Transfer Set”) which should be transferred: configure sets at **Element Transfer Settings**, and apply the Set you need during Parameter Transfer (see below).

Related Topics:



[Element Transfer Settings](#)

[Parameter Transfer Shortcuts](#)

[Pick Up/Inject Parameters](#)

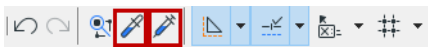
[Pick Up/Inject Morph Parameters](#)

Parameter Transfer Shortcuts

To do this:	Click this key:
Pick Up Parameters 	Alt
Inject Parameters 	Ctrl+Alt or Alt/Opt-Command (Mac)

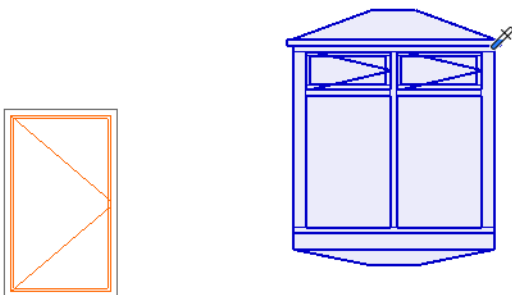
Parameter Transfer commands are also available:

- From the Standard Toolbar
- At **Edit > Element Settings > Pick Up Parameters** and **Inject Parameters**.



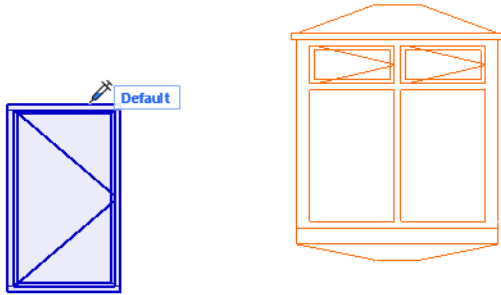
Pick Up/Inject Parameters

1. Move the cursor over the element whose parameters you wish to transfer. (The Element Information Highlight tells you which element you are focusing on.)
2. Press and hold **Alt**. The cursor changes to **Pick Up** (eyedropper) form.

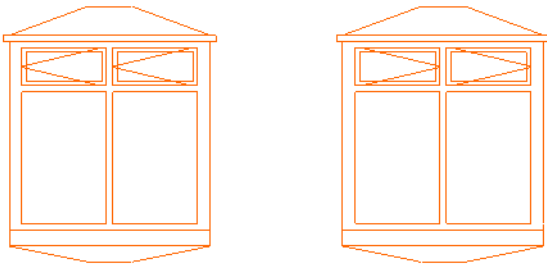


Alt to Pick Up Parameters

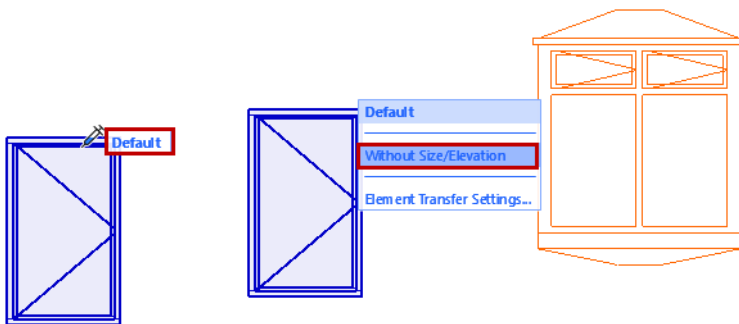
3. Click the element. All its parameters are now the default for that element type.
4. Move the cursor to the target element.
5. Press and hold **Ctrl+Alt** (Alt+Cmd on Mac).
 - The cursor changes to the **Inject** (syringe) cursor.
 - The name of the last used Transfer Set appears: this Set defines which parameters will be transferred when clicking to inject.

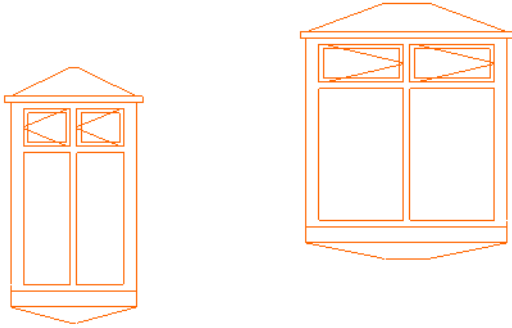


Ctrl/Cmd + Alt to Inject Parameters



6. Before injecting parameters, change to a different Transfer Set, if needed:
 - Click on the Set name to bring up the full list of Transfer Sets, then click on another Set to apply it.





- The Set you choose here becomes the default Transfer Set for the next parameter transfer operation.
- Define and manage Transfer Sets here: [Element Transfer Settings](#).

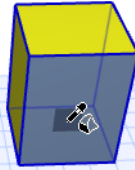
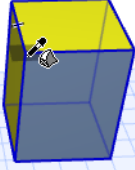
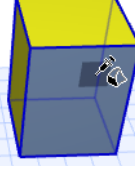
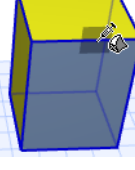
Notes:

- The transferred attributes are loaded into the Find & Select Palette's corresponding fields, provided that the Find & Select Palette is open during the parameter transfer.
- The transferred parameters will be the default setting when opening any of the corresponding Attributes dialog boxes (Lines, Pens & Colors, Fills, Surfaces, Composites, Zone Categories).
- When injecting a GDL object, the transferred line type attributes will take effect only if you are *overriding* the object's GDL attribute definition. That is, make sure you have checked the **Override Object's Lines** checkbox in the Floor Plan and Section panel of its Settings dialog box.

[Watch the Video](#)

[Watch the Video](#)

Pick Up/Inject Morph Parameters

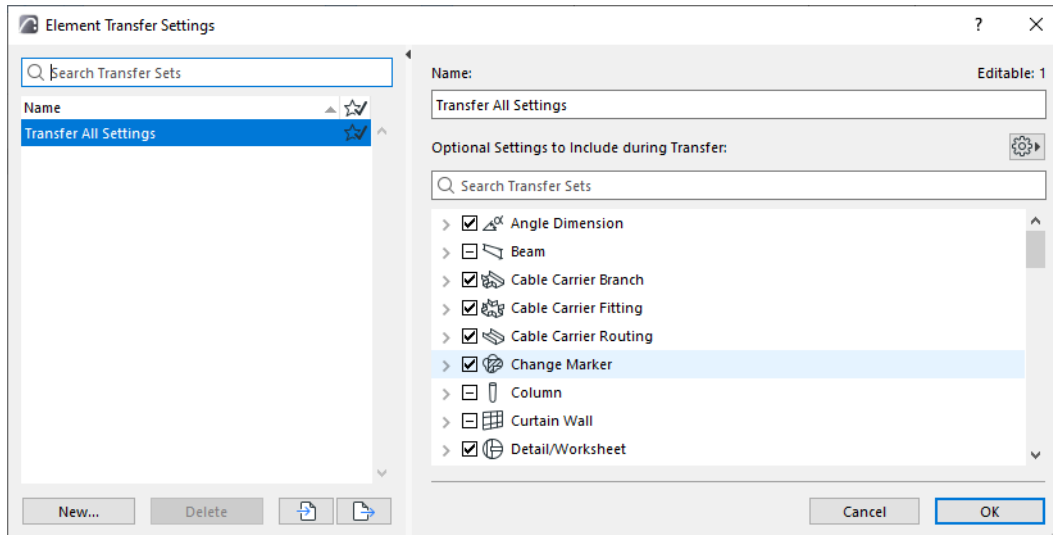
To do this:		Click this key:
<p>Pick Up Morph Face Parameters</p>		<p>Alt+click on a Morph Face</p>
<p>Pick Up Morph Default Parameters <i>Custom Morph Edge parameters cannot be transferred.</i></p>		<p>Alt+click on a Morph Edge or Node</p>
<p>Inject Morph Parameters to Morph Face</p>		<p>Ctrl+Alt+click on a Morph Face</p>
<p>Inject Morph Parameters to Entire Morph</p>		<p>Ctrl+Alt+click on a Morph Edge or Node</p>

[Watch the Video](#)

Element Transfer Settings

Use the **Element Transfer Settings** dialog box (Edit > Element Settings) to customize which settings should be injected

- when using Parameter Transfer (when using the syringe)
- when applying Favorites



The left side of the dialog box lists the named Transfer Sets. During Parameter Transfer, and when applying a Favorite, you can choose any one of these sets.

One of the sets is tagged as the default to use when applying Favorites.

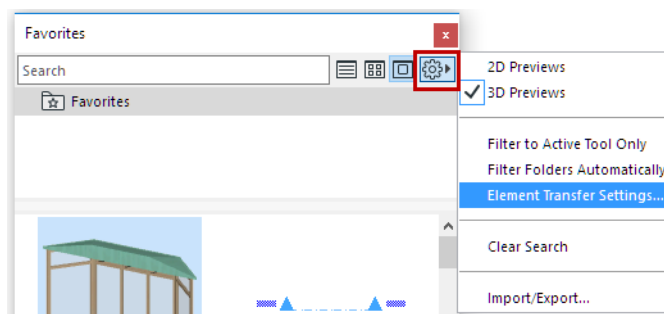
See [Set as Default for Favorites](#).

Note: Element Transfer Settings and the named Transfer Sets are stored with the project. However, the Favorite default is a user preference and stored locally.

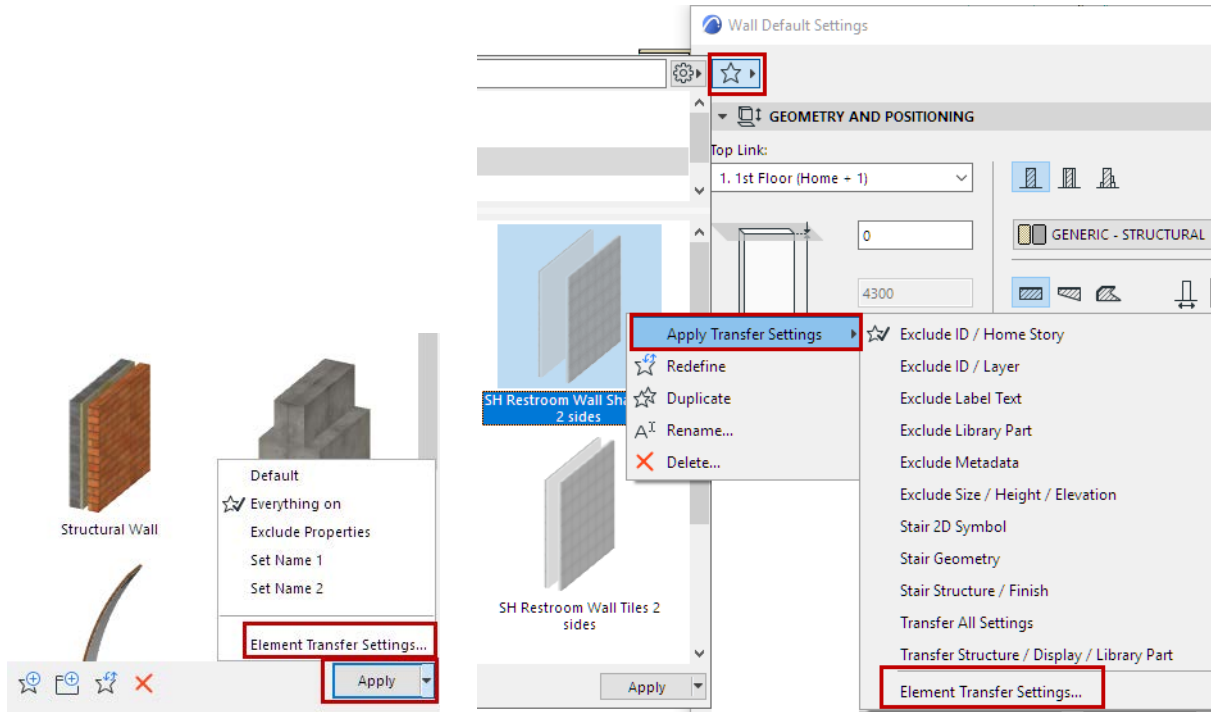
Open Element Transfer Settings

The dialog box is available from:

- **Edit > Element Settings > Element Transfer Settings**
- All locations where you can access Element Transfer sets
 - The Favorite Palette's Additional Settings pop-up menu

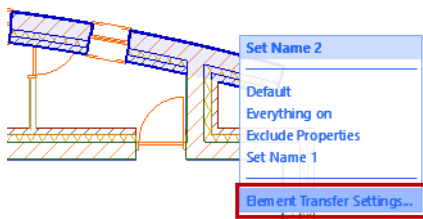


- From a Favorite's Apply button (in the Favorites interface of Tool Settings)



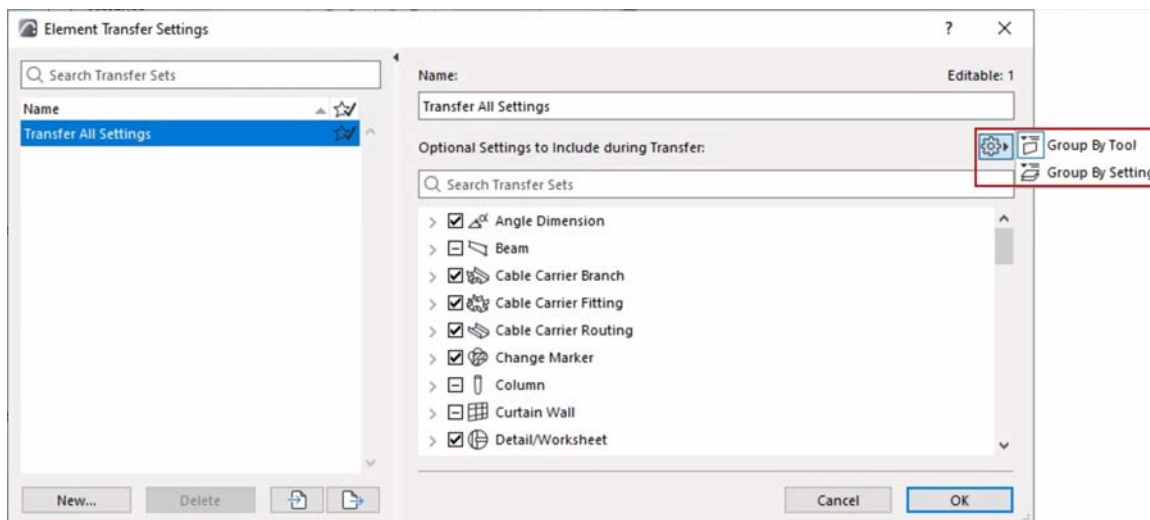
- When injecting parameters with the syringe cursor.

See [Pick Up/Inject Parameters](#).

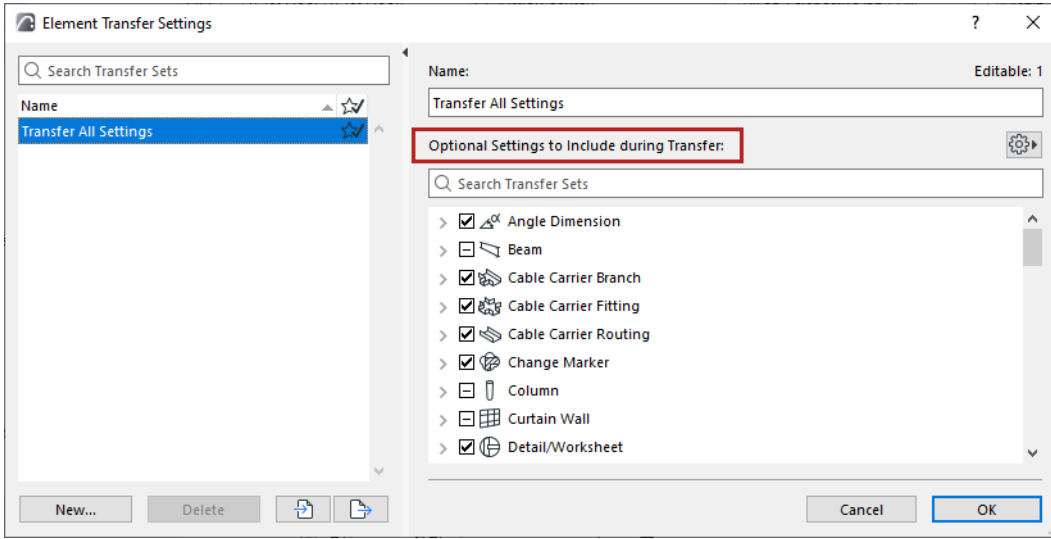


Settings to Include for Parameter Transfer

Sort the list by either Tool or by Setting:



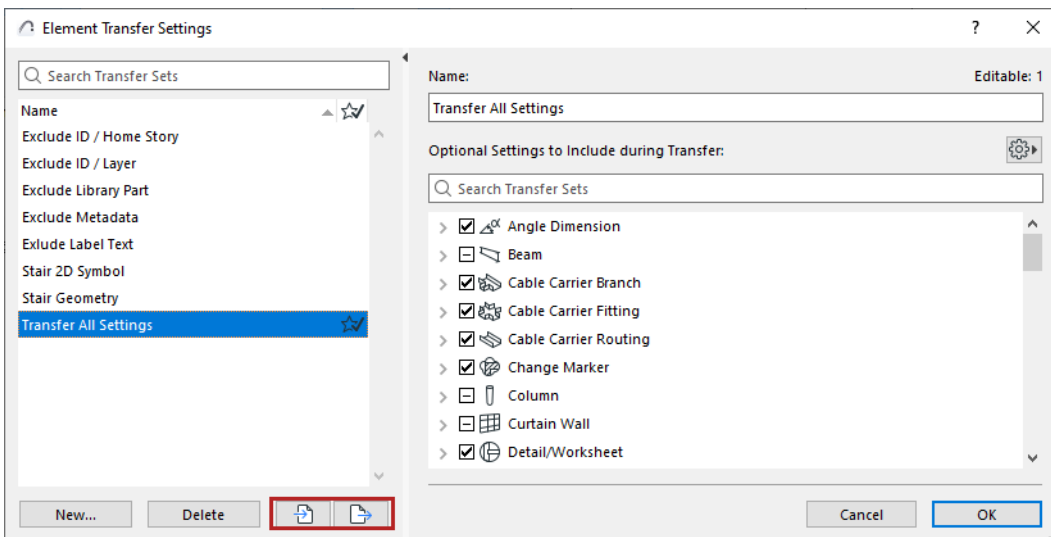
Check the settings or groups that you want to be transferred during Element Transfer and when applying Favorites. Settings *not* listed here will always transfer.



Manage Sets

Import/Export

Use the Import/Export buttons at the bottom of this dialog box to transfer one or more selected sets of Element Transfer Settings from one project to another, in XML format.

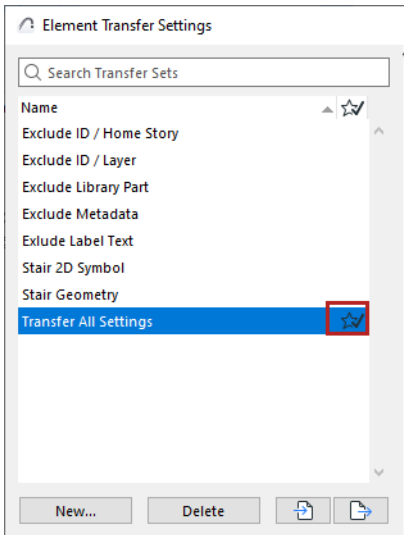


New

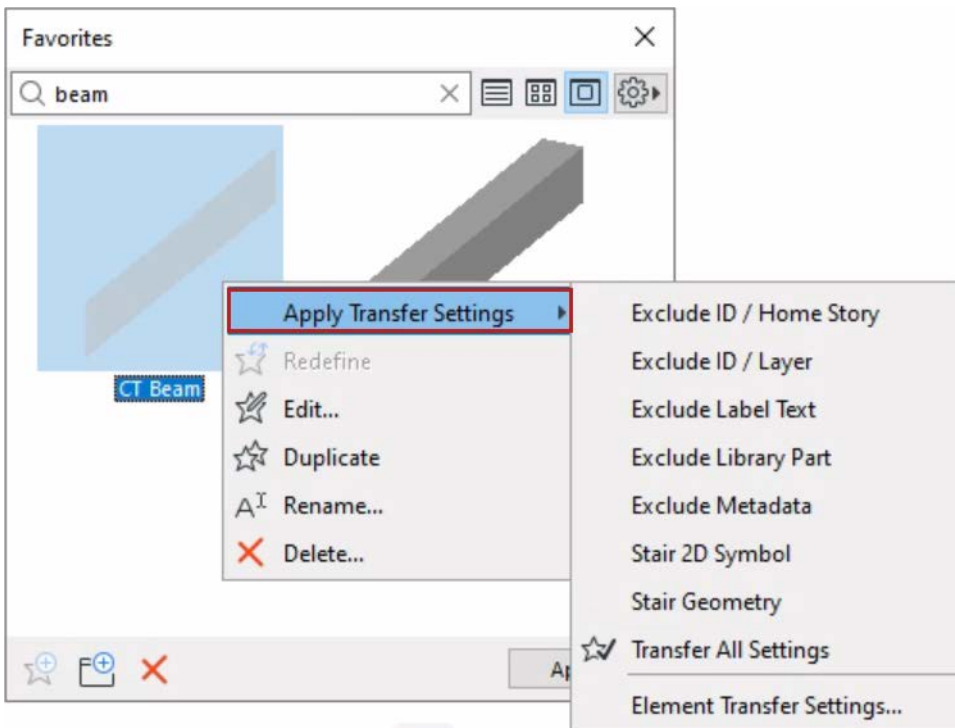
Use the New button at the bottom to define a new set of Element Transfer Settings.

Set as Default for Favorites

Choose one of the sets to use as the default when applying a Favorite: click in the right column next to the set you want:



This set will be used when you apply any Favorite (double-click the Favorite, or click the **Apply** button). You can see this preference if you click the **Apply Transfer Settings** pop-up in the Favorites interface:



The Favorite default is a user preference and stored locally.

Of course, when applying a Favorite, you can click any other set in the list to apply that set of preferred parameters.

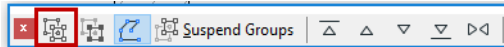
[Watch the Video](#)

[Watch the Video](#)

Group Elements

To create a Group, select elements and do one of the following:

- Use the **Edit > Grouping > Group** command
- Use the shortcut: **Ctrl/Cmd + G**
- Click the **Group** command from the **Arrange Elements** toolbar.

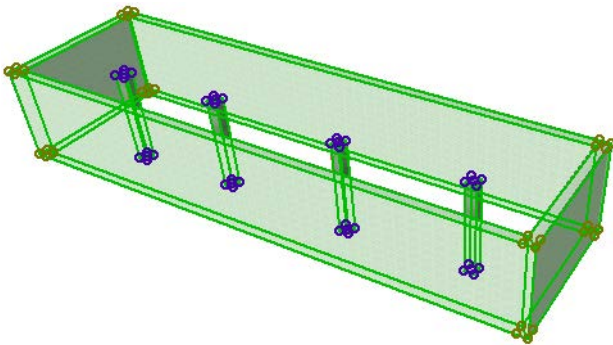


Several groups can, in turn, be grouped together into a single higher-level group.

Grouped elements can then be selected and modified as a group, (unless you temporarily suspend the group to allow editing of individual elements).

[See *Suspend Groups*.](#)

Grouped elements are distinguished by their large, empty selection dots. If you select several groups simultaneously, each group's selection dots will have a different color.



The following element types *cannot* be grouped: any of the Dimension types, Zones, Labels, Section/Elevation/IE lines, Cameras. Doors and Windows can only be grouped along with the Wall they are placed into.

Suspend Groups

You may need to perform an operation on only one particular element in a group. In this case, ungrouping them all is burdensome, since you will have to select all the elements again to recreate the group after you have finished the operation.

The solution is to temporarily suspend groups by choosing the Suspend Group toggle icon in the Standard Toolbar, or at **Edit > Grouping > Suspend Groups** (shortcut: Alt/Opt+G).



(The Suspend Groups icon is also available in the Control Box.)

By activating Suspend Groups, you can select and modify grouped elements separately. Activating Suspend Groups means that ALL groups are temporarily rendered inactive: single elements can be selected and edited individually, even if they are part of a complex group hierarchy.

To re-activate the Group function, just toggle Suspend Groups to OFF again.

Note: To change the settings of a single element in a group *without* using Suspend Groups, use Parameter Transfer. This parameter transfer will affect only the clicked element, and not the rest of the elements in the group.

See [Parameter Transfer](#).

Ungroup

To make all elements constituting a group independent again, choose **Edit > Grouping > Ungroup** (shortcut: **Ctrl/Cmd+Shift+G**).

Ungroup is also available from the **Arrange Elements** toolbar.)

- Clicking **Ungroup** with the Suspend Groups toggle ON will detach any selected elements from their groups and break down all groups *into single independent elements*, no matter how complex the nested grouping is.
- Clicking **Ungroup** with the Suspend Groups toggle OFF, for a selection consisting of lower-level groups, the selected group will first be divided *into the sub-groups it comprises*.
Thus, you may need to repeat the **Edit > Grouping > Ungroup** command several times before an element can be selected and edited individually.

Autogroup

Edit > Grouping > Autogroup, a toggle command (shortcut: **Alt+G**), lets you group elements at the same time you create them (instead of creating elements first and then grouping them).

The **Autogroup** toggle command is also available in the Standard Toolbar and in the **Arrange Elements** toolbar.

If the Autogroup command is on, chained polygonal and rectangular elements will automatically be created as a group. Components of exploded elements will also be grouped.

Autogroup is activated by default.

Operations on Grouped Elements

The following operations can be performed simultaneously on all the elements constituting the group:

- Selection
- The following **Edit** menu functions: **Drag, Rotate, Mirror, Multiply**
- Delete

Note: If any elements in the selected group are on a hidden layer, they will also be deleted!

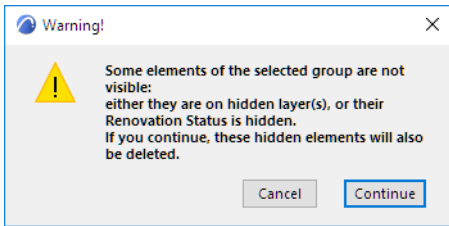
Other operations (such as Unify and Fillet/Chamfer) can be performed on individual elements only, after the elements are ungrouped or the group is suspended.

Groups and Layers

Grouping elements does not change their attribute assignments, which means that each element remains on its own Layer.

If some elements of a Group are on a hidden Layer:

- Elements on the hidden Layer will be invisible.
- When editing the group (by drag, rotate, mirror, multiply, delete), elements on hidden layers will also change their position - or will be deleted - along with the rest of the group.



If some elements of a Group are in a locked Layer, [see *Lock/Unlock Elements*](#).

- Elements on the locked Layer will be visible on the Floor Plan, but they will cause the entire group to be locked. The group will not be affected by drag, rotate, mirror and multiply.
- When selecting the group, all of its elements will be selected, but with gray dots. Use the **Suspend Groups** command to edit those grouped items that are not in a locked layer.

Lock/Unlock Elements

The **Edit > Locking > Lock** command locks the selected items to prevent them from being accidentally modified. Locked elements can still be selected and used for relative construction; Snap Guides are still available; and you can pick up their settings through parameter transfer.

See [Parameter Transfer Between Objects](#).

Note: Locking a layer will have the same effect - it locks all elements which are located on that layer.

The **Edit > Locking > Unlock** command unlocks the selected item(s).

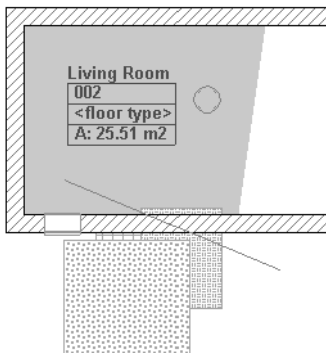
With the **Unlock All** command, you can unlock all locked elements, even if they are not selected. You can also use the relevant buttons of the **Arrange Elements** toolbar for these actions.



Display Order

When drawing a Project with Archicad, overlapping elements will be drawn according to a specified stacking order. By default, elements are stacked in an order typically used in architectural drawings.

By default, stacking order is determined by element classes, of which there are six. Regardless of the sequence in which elements are placed, those in the first class will be placed in the foreground, second class in the level behind it, and so on.



The element classes are in descending order of class:

1. Annotation (Text, Labels, all Dimension types, Zone Stamps)
2. 2D artwork (Lines, Circles, Splines, Hotspots)
3. Library Parts (Objects, Lamps, Stairs)
4. 3D structures (Walls, Beams, Slabs, Doors, Windows, Columns, Roofs, Meshes)
5. 2D Fills
6. Zones
7. Figures

Display Order of Overlapping Elements

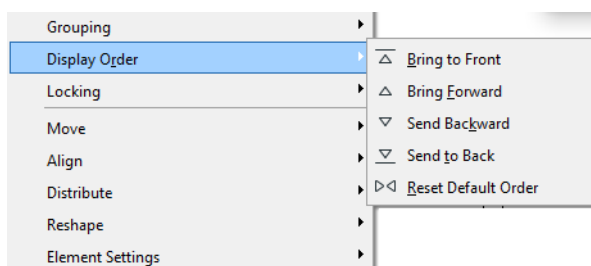
Overlapping within the same class depends on the order in which you place the elements, although walls with the same fills will display clean intersections.

When selecting overlapping elements within the same class, you can select them in sequence using Tab.

[See *Selecting Elements*.](#)

Custom Stacking Order

Archicad's default stacking order is appropriate in most cases, but you may wish to place elements in a different overlapping position. For instance, for a Fill to overlap a 2D element or an Object, a set of commands in **Edit > Display Order** hierarchical menu or the relevant buttons of the **Arrange Elements** toolbar let you override the default stacking order element by element.



On newly placed elements, the commands work as follows:

- **Bring Forward:** On this command, selected element(s) will overlap all the unchanged elements of their own and any lower classes, but will still stay below the elements of higher classes.
- **Bring to Front:** On this command, selected element(s) will overlap all other existing elements.
- **Send Backward:** On this command, selected element(s) will be overlapped by (sent behind) all the unchanged elements of their own and higher classes, but will overlap the elements of lower classes.
- **Send to Back:** On this command, selected element(s) will become overlapped by all other existing elements.
- **Reset Default Order:** This command restores the default overlapping order described previously.

Mechanism of the Stacking Order

Both the default stacking order and the customization commands are based on two ordering mechanisms: classes and stack levels.

Archicad places elements in 14 stack levels. Each level can contain any of the element types. The final stacking order you see on screen and on printouts is based on stack level and element class. All elements on a level will overlap elements on lower stack levels, regardless of class.

New elements are always placed into the preferred stack level of their own class. Stack level preference is as follows:

- 1-4: Void by default
- 5: Annotation
- 6: 2D Artwork
- 7: Library Parts
- 8: 3D Structures
- 9: 2D Polygons
- 10: Figures
- 11-14: Void by default

By default, the upper and lower 4 stack levels are empty. You can bring or send elements of any type into these stack levels with the **Edit > Display Order** commands detailed above.

The **Bring to Front** and **Send to Back** commands move the selected elements until they overlap (or are overlapped by) all of the existing elements. This is not necessarily the 1st or the 14th level, which means that in most cases it will still be possible to place other elements above or below them.

Text or Dimensions brought up to the top stack level cannot be overlapped. For example, if you want a Fill to overlap them, you will have to send the Text one level backwards.

When moving several elements placed at different stack levels forward or backward, each element will only move one level.

When bringing to front or sending to back elements of different types, they will overlap (or be overlapped by) all existing elements, and will retain their stacking order in relation to each other.

Zones and Fills are compound elements consisting of components of two different classes (Zone Polygons and Zone Stamps, Fills and their area texts). These two components always keep their order according to their classes, for instance area texts always overlap their fills. Although you can change their overlapping order separately, Fills and Zone Fills can never overlap their own area texts and zone stamps and vice versa, forcing the other component to move with them if necessary.

In DXF/DWG output, all elements will be transparent, meaning that hidden or partially overlapped elements will be entirely visible.

Virtual Trace: Using References to Edit and Compare Model Views and Drawings

In Archicad, you can opt to display two different views concurrently in any model or layout window:

- the Active Content - the actual model or drawing you are currently working on (“Active”)
- the Trace Reference (“Reference”), optionally placed alongside, on top of or underneath the currently Active content. This acts as a kind of underlay, making it easy to compare multiple model views/drawings on the same screen.

The Trace Reference is essentially a temporary editing aid that is tied to the currently active window. The chosen Reference appears in a predefined default position (appropriate to the type of the Active Content and the Reference item).

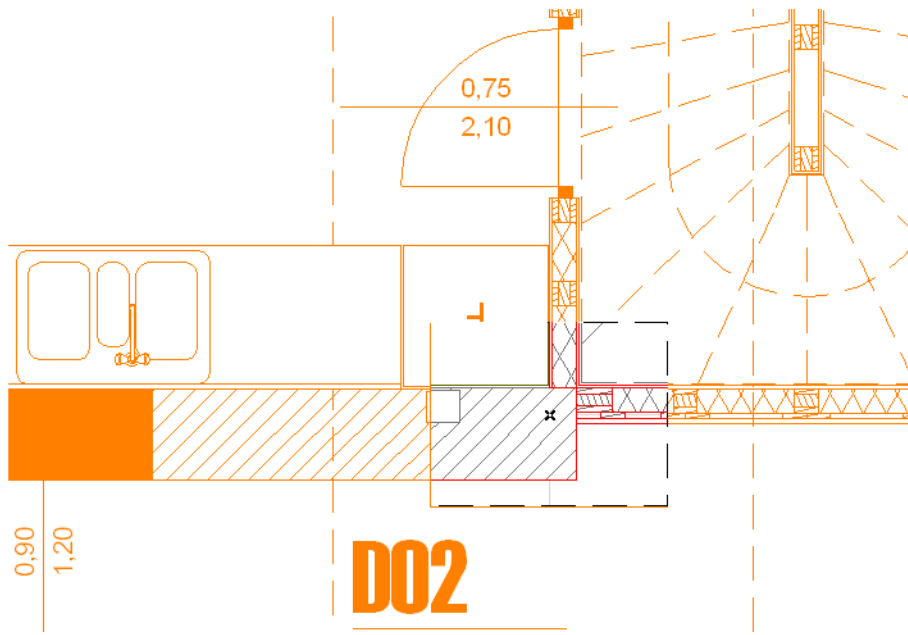
Three item types can be chosen as a Reference:

- 1. Viewpoints.** Any item in the Navigator’s Project Map (except 3D window items and list-type items) can be set as a Reference. The Reference Viewpoint will always reflect the current settings of the active window.
- 2. Views.** Any view in the Navigator’s View Map (except 3D window items and list-type items) can be set as a Reference. The Reference’s view settings are independent of the active window. For example, if you change the layer combination in the active window, the Reference won’t change. If a view is deleted from the view map, its Reference is no longer available.
- 3. Layouts** (and their drawings).

For example, you can draw a roof detail in a Detail window (the Active Content) while seeing the relevant wall section as a Reference.



A Section shown as a Reference next to another Section



A Floor Plan shown as a Reference beneath a Detail

Any given window can display only one Reference at a time. However, as you navigate among your project windows, you can display a different Reference in each:

- The Floor Plan window has a single current Reference for all stories
- Each Section, Elevation, Interior Elevation, 3D Document, Worksheet, Detail, Layout and Master Layout can have its own unique Reference.

Reference settings are *saved along with the window* (for example, each Section in your project can have a Reference of a different color and filter different elements).

References are also *saved with the project file*. All user-defined Reference-related parameters, such as the list of recent References and recently chosen Reference colors, are saved together with the project file.

When using the **Print** command to output a model window or a Layout, you can opt to print the currently visible Reference as well.

When you place a Drawing onto a Layout, the **Drawing content** will *not* include the Reference.

The user can offset and rotate the Reference as needed, though logical default Reference settings for each combination of Active/Reference views will usually align them correctly for the particular context.

On-Screen View Options (View menu) affect the Reference just as they affect the Active.

The cursor is sensitive to elements in the Reference. Consequently:

- You can use the **Pick Up Parameters** command (**Alt +C**) on Reference elements, then transfer the parameters to an element in the Active.
- You can use the **Trim** command, the **Magic Wand**, and **snap points** with elements in the Reference to edit Active elements.

Topics in this section:

[Choose a Trace Reference](#)

[Show/Hide Trace Reference](#)

[Set Color of Reference Elements](#)

[Show/Hide Elements in Reference](#)

Move/Rotate/Reset Reference

Switch Reference with Active: Access Elements within the Reference for Editing or Copying

Activate Reference

Rebuild Trace Reference

Compare Reference with Active

Choose a Trace Reference

Use one of the following methods to choose a Trace Reference.

Show Navigator Item as Trace Reference

Browse for Trace Reference

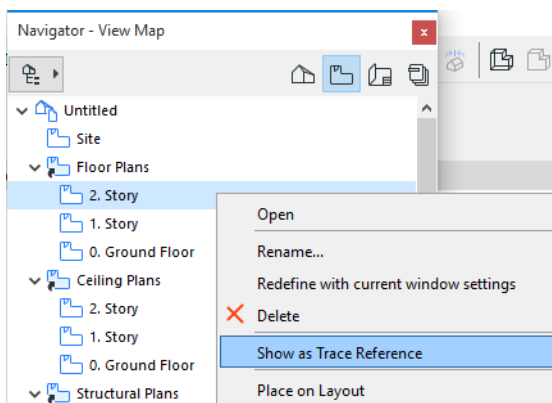
Choose Trace Reference from Menu

Choose Reference from Trace & Reference Palette

View Context of Hotlinked Module while Editing Module Content

Show Navigator Item as Trace Reference

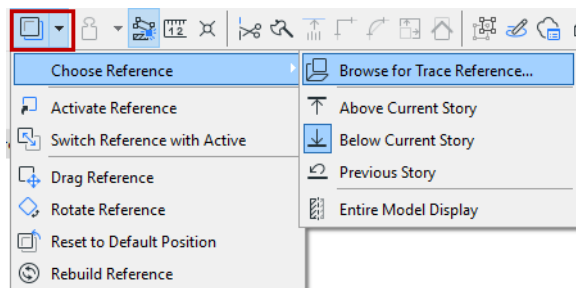
1. Select the item from the **Navigator** and right-click to bring up its context menu.
2. Choose **Show as Trace Reference**.



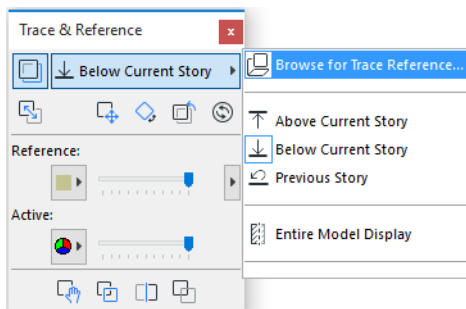
Browse for Trace Reference

Click the “Browse for Trace Reference” sub-command from one of these locations:

- The “**Choose Reference**” command in the Trace Reference drop-down menu of the Standard Toolbar

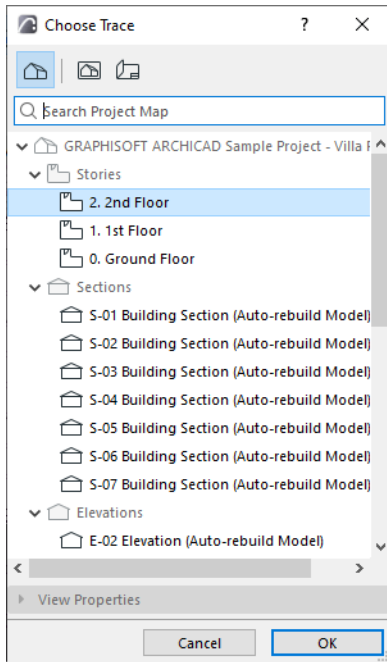


- The chooser button at the top of the Trace & Reference Palette



- View > Trace Options > Choose Reference > Browse for Trace Reference

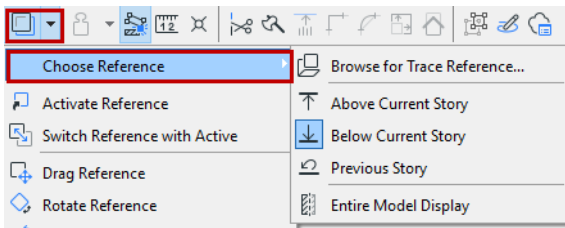
The **Choose Trace Reference** dialog box appears. This is a Navigator-like structure, with only those items available which are eligible to use as a Trace. (In the View Map, View Properties are also displayed.) Choose the desired Trace Reference and click OK to display it as a Trace.



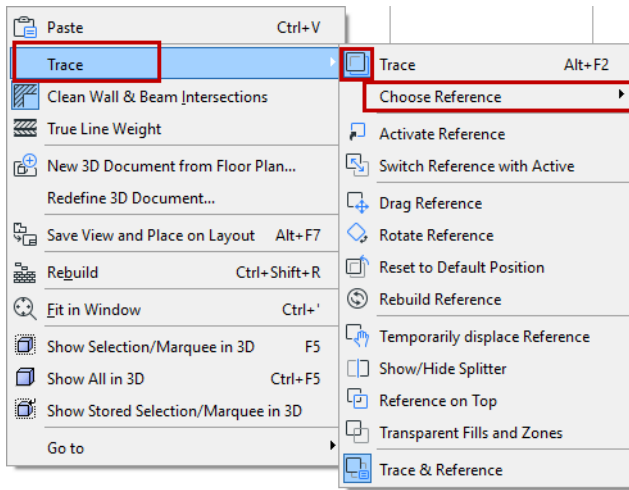
Choose Trace Reference from Menu

Do one of the following:

- From the Standard Toolbar, activate the Trace button, then open the drop-down menu to choose an item from the **Choose Reference** list.



- From the context menu in any model window, click Trace, then choose an item from the **Choose Reference** list:

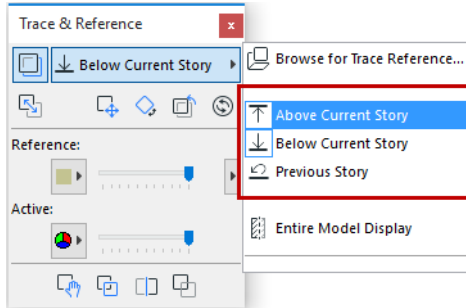


- The same commands are available from **View > Trace Options**.

The “Choose Reference” list is dynamic. It includes recently used references, and also depends on the type of the current tab/window.

Choose Reference from Trace & Reference Palette

Click the button atop the Trace & Reference Palette to access a list of possible Reference items. Click the one you need to show it as a Trace Reference.



View Context of Hotlinked Module while Editing Module Content

You may wish to edit a Hotlinked Module at its source, then update it in the host file.

While you edit the source file, it can be helpful to see the module’s Floor Plan context in the host file. In this case, use the “Trace” function:

- In the host file, save the Floor Plan (in the host file) as a view and place it onto a Layout.
- In the source file, use the Organizer to access the host file and copy the Layout into the source file.
- In the source file, select this Layout from the Navigator and choose “Show as Trace” from its context menu.

Show/Hide Trace Reference

Click the Trace toggle button to show or hide the Trace Reference in the current window.

The Trace toggle button is available:

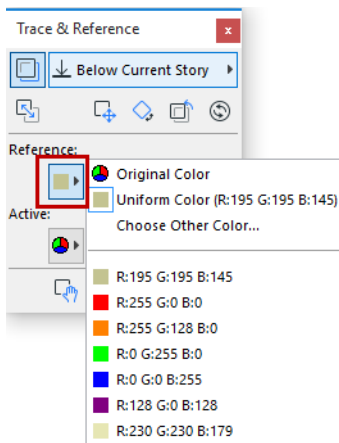
- in the Standard Toolbar



- in the context menu of any window - right-click in an empty space
- in the Trace & Reference Palette
- in the View menu

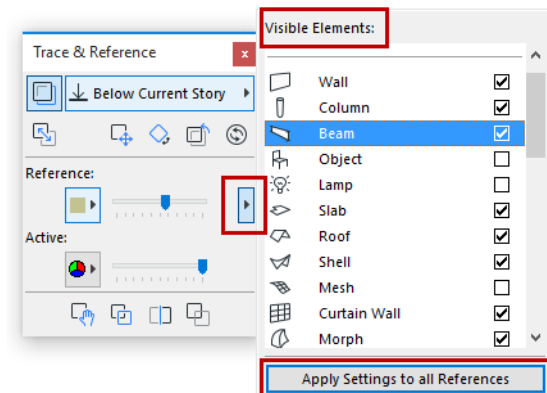
Set Color of Reference Elements

1. Open the Trace & Reference Palette. (Use **Window > Palettes > Trace & Reference**, or choose **Trace & Reference** from the list of **Trace Reference** commands.)
2. Click the pop-up to choose a uniform color, if desired, for Reference elements.



Show/Hide Elements in Reference

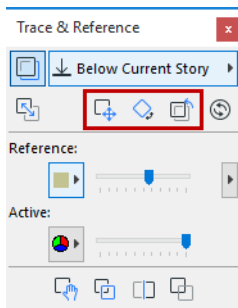
1. Open the Trace & Reference Palette. (Use **Window > Palettes > Trace & Reference**, or choose **Trace & Reference** from the list of **Trace Reference** commands.)
2. Click the pop-up to open a checklist of element types. Uncheck any that you do not wish to display in the Reference.



Change you make here apply to the Reference of the current window only. However, to apply the settings in this pop-up to all References in the project, click the **Apply Settings to All References** button.

Move/Rotate/Reset Reference

These three command icons are available from the Trace & Reference palette:

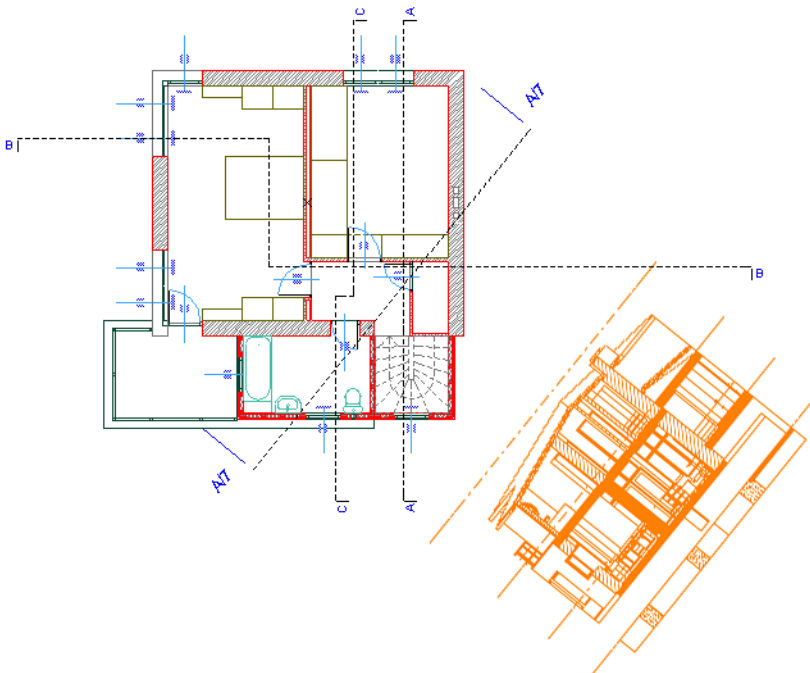


From left:

- **Move:** Click **drag** the Reference to another location.
- **Rotate:** Click to **rotate** the Reference.
- **Reset:** Use this icon if you want to **reset** the Reference to its original default position after you have moved it.

Note: The same commands are available from the **context menu** by right-clicking into any window showing a Reference; and from the Standard Toolbar's pop-up list of Reference commands.

In the image below, a Section viewpoint is displayed as a Reference and rotated alongside the Floor Plan that contains the Section marker:



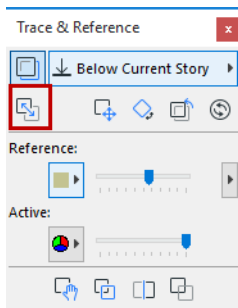
Drag/Rotate commands applied to the Reference will be reflected in the Tracker's coordinate data (coordinate data are calculated according to the Active's coordinate system, not the Reference's.)

Switch Reference with Active: Access Elements within the Reference for Editing or Copying

Elements on the Reference cannot be selected or edited.

However, by switching the Reference and Active views temporarily, you can select and edit elements in the Reference. For example, you can use the **Switch Reference with Active** command if you need to paste certain Reference elements into the Active view:

First, use the **Switch Reference with Active** command (from the list of Trace Reference commands, or the Trace & Reference palette.)



This changes the Reference into the Active (now you can edit its elements), and the Active into a Reference.

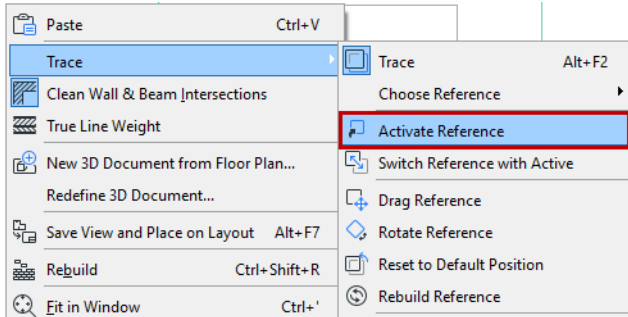
Note: This command is not available if a Drawing Section is active and its own Model-based Section is the Reference.

3. Select, edit and/or copy the required elements.
4. Once again, use the “**Switch Reference with Active**” command and return to the original Active content.
5. If you copied elements to the clipboard, now you can paste these into the Active.

Note: When pasting into a drawing-type window (such as a Worksheet), construction elements will be “exploded” into their 2D components.

Activate Reference

The **Activate Reference** command is available from **View > Trace Options**, and from the list of **Trace Reference** commands (from the Standard Toolbar or the context menu of an Archicad window.)



Activate Reference will change the active window to the Viewpoint that is currently the source of the Trace Reference. This active window, in turn, will use the same Trace Reference definition that was previously set, IF it applies and makes sense.

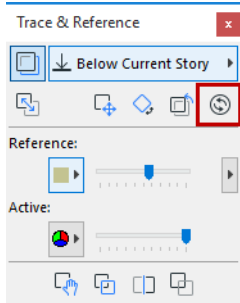
For example: Ground Floor is active and Floor Above (First Floor) is set to be the Trace Reference. If you activate the Reference, then the First Floor will be the active, and Second Floor will be shown as Trace Reference.

If the current Reference definition has no meaning in the activated viewpoint, then no Reference will be shown.

For example: The South Elevation is shown as Trace Reference under the Ground Floor Plan. If you use the **Activate Reference** command, the program will switch to the South Elevation Viewpoint and no Reference will be shown.

Rebuild Trace Reference

Use the **Rebuild** command (available from the Trace & Reference Palette or among the **Trace Reference** commands) to rebuild the Reference to reflect any changes in the source view.



In some cases, the Reference will be rebuilt automatically as a result of certain other user actions, such as navigating to another window and panning inside a window.

However, editing a model view will not necessarily rebuild the Reference; you must use the Rebuild Reference button.

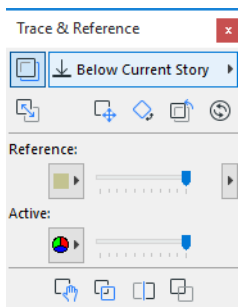
Note the following:

- If the Reference is an **Auto-Rebuild Model**, it is possible that you make changes in the Active, yet these are not reflected in the Reference, even after you navigate or pan, because - to avoid slowing down the program - you have unchecked "Update Autorebuild Model Viewpoints continuously" in **Options > Work Environment > Model Rebuild Options**. Use the Rebuild Reference button when needed.
- If the Reference contains an **Auto-update Drawing**, such a Reference will not be continuously rebuilt, to avoid slowing down the program. Use the Rebuild Reference button when needed.

Compare Reference with Active

Archicad provides several functions to help you identify and understand differences between the Reference and the Active content, especially when the Reference is on top of the Active content, or vice versa, and you need additional help to see the differences clearly. The best way to compare the two will depend on your particular task. You can use several of these functions at the same time to get the most accurate information. All the Compare functions can be used “on the fly” while editing your Active content. They are on-screen only functions, with no effect on output.

To access these functions, use the Trace & Reference Palette, from **Window > Palettes > Trace & Reference**.



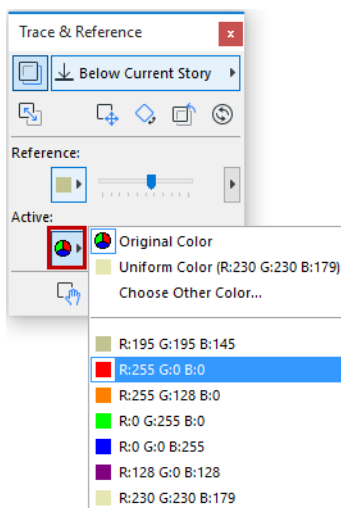
Each of the techniques below can be useful in visually comparing the Reference with the Active. Try them out to see which works best.

Tip 1: Use Separate Colors for Active and Reference

You can set separate **display colors** for the Reference and the Active content: a custom color for each, or else retain the original colors.

You may want to set two sharply different, custom colors for each of the views - say, red and blue - to make it easy to distinguish one from the other.

To set the colors, use the pop-up icons (one each for Reference and Active) in the middle of the Palette.

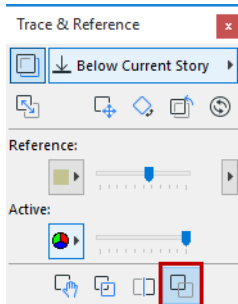


Note: Even if you've set separate colors, the background fills of one view may obscure the underlying elements in the other view.

Tip 2: Make Fills and Zones Transparent

Note the **Make Fills and Zones Transparent** toggle at the bottom of the Trace & Reference Palette: activate this to make fills and zones transparent in both the Reference and Active views. This way you will “uncover” information that otherwise might have been covered up by a fill in the top view.

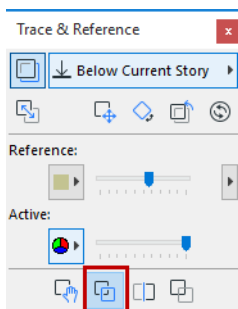
This switch has only a temporary effect and does not affect the settings of the model elements.



Tip 3: Reverse Display Order of Reference and Active

When comparing the Reference to the Active, it may help to switch their display order.

Click the “Reference on Top” button at the bottom of the Trace & Reference Palette.

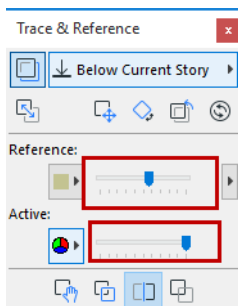


If your Reference was previously underneath the Active, this command will change them around.

Tip 4: Optimize Intensity of Reference vs. Active

A simple way to make an initial visual comparison is to use the Intensity sliders of the Reference and Active.

Note: This method works best if the Active is shown in its original colors, and the Reference in a different color.



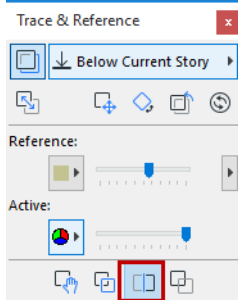
Pull the Reference intensity slider back and forth. On screen, the effect will be to flash the Reference “on” and “off”, allowing you to identify places on screen where there are differences from the Active View.

Then you can zoom in on these locations and analyze the differences.

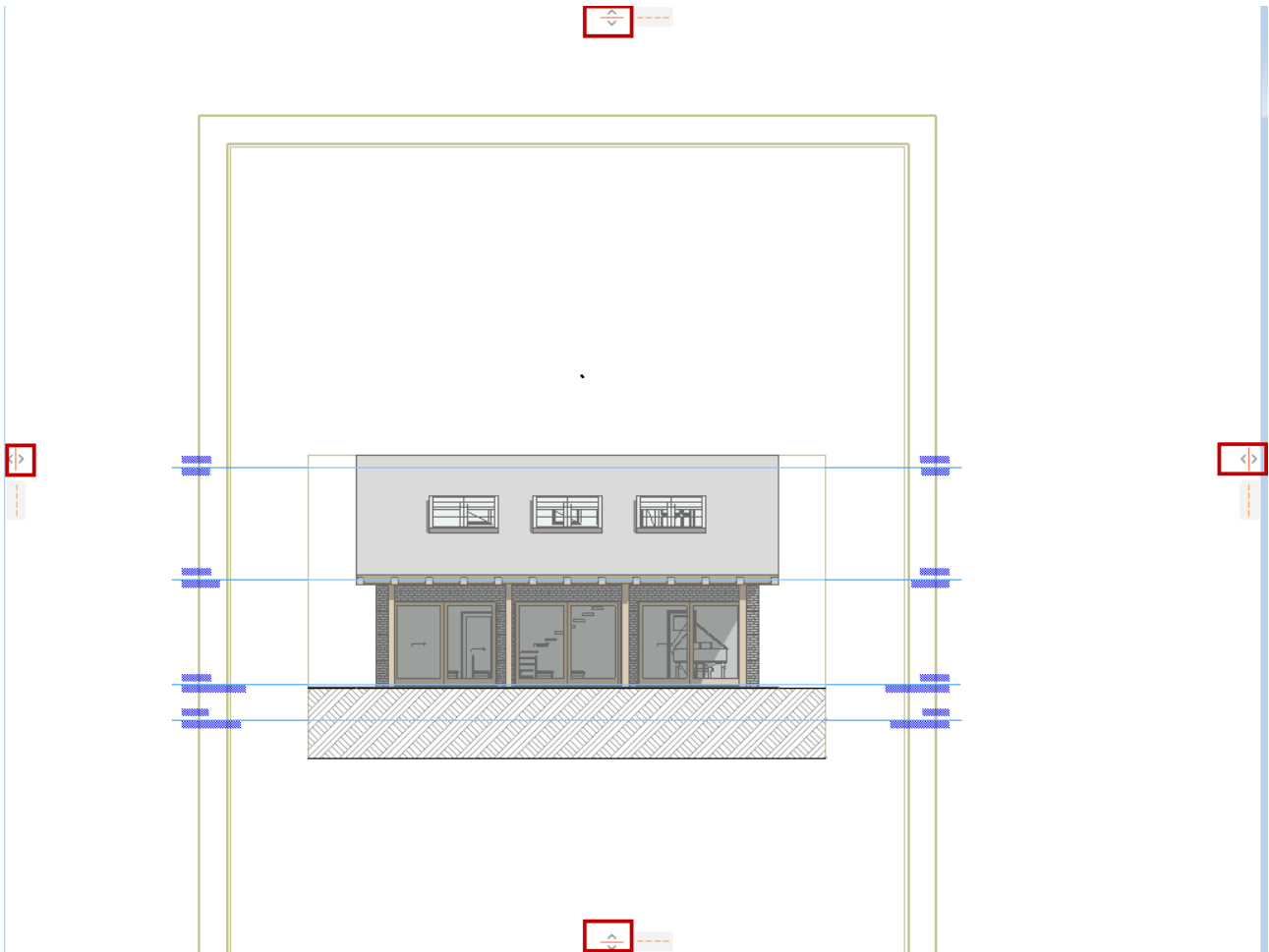
Tip 5: Use the Splitter Bar to “Turn the Page”

This function is useful for identifying differences between the Reference and Active when they overlap. You will drag a Splitter bar across the window, with the Reference on one side and the Active on the other. The effect is like turning the “page” of an overlay to see what is beneath.

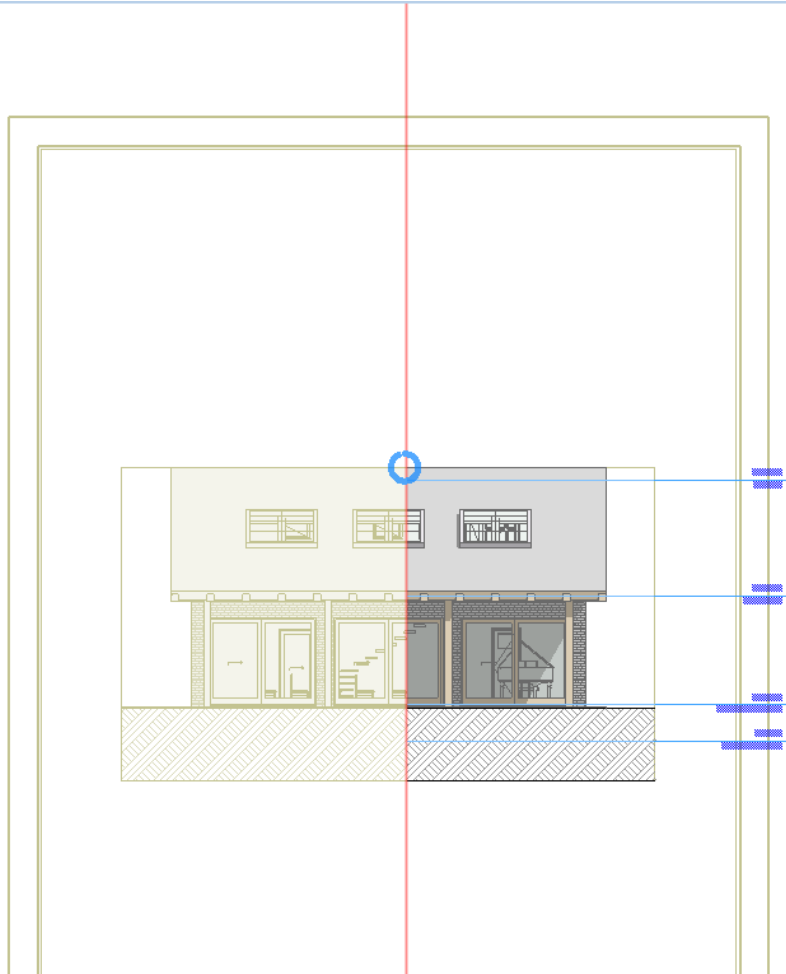
To activate the Splitter function, click the Splitter icon from the Trace & Reference Palette.



Four “splitter handles” appear, one on each edge of the screen. Choose any one of them and move it in a perpendicular direction, to create either a horizontal or vertical splitter bar.



As you drag it, the content on either side of the splitter changes dynamically.

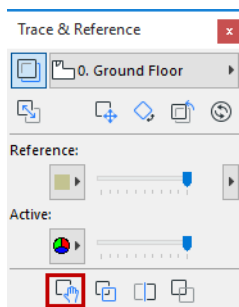


Once you let go of the mouse button, the splitter bar returns to its original position.

Tip 6: Move the Reference Over Temporarily (Displace Reference)

Use this function when you have zoomed in to an area where the two views are different, and you want to quickly check what is on the view underneath.

Click the **Temporarily Displace Reference** button in the Trace & Reference Palette.



Click in the window. The cursor changes to the hand shape, allowing the user to nudge the Reference temporarily (i.e. to move it “out of the way”).

Click again, and the Reference jumps back to its original position.

Consolidate Lines and Fills in Drawing Windows

Several Archicad features involve the creation of “exploded” elements. Model-based Details, Worksheets and Drawing Sections create 2D elements (lines, fills) out of the model construction elements. Also, imported DWG drawings appear in Archicad in 2D form, as a large collection of lines and fills.

In these Drawing-type windows, you may wish to fine-tune the view before final output. You will find, however, that the “exploded” lines and fills often contain superfluous elements (extra line segments, overlapping or superfluous fills) that make such editing difficult.

To make editing easier, first use the **Linework** and **Fill Consolidation** functions on selected items in the window.

Line and Fill Consolidation are separate functions and are executed independently of each other. Select the elements, then execute either the Linework Consolidation or the Fill Consolidation command. (**Edit > Reshape > Linework/Fill Consolidation**). The two commands are briefly described below.

Linework Consolidation

In the window, select at least two line-type elements you wish to include in the Linework Consolidation process. Line-type elements are straight-lines, polylines, arcs and circles.

Note: Linework Consolidation does not affect any line-type element having an arrowhead.

The Linework Consolidation command (**Edit > Reshape > Linework Consolidation**) brings up the Linework Consolidation Wizard.

This command is also available from the Edit Elements toolbar.



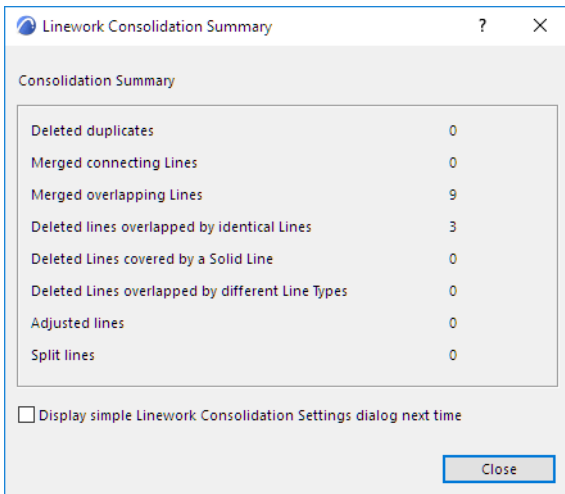
Now page through the wizard (click “Next” at the bottom of every screen) and check the boxes of every Line Consolidation option that you wish to execute as part of the consolidation process.

For details, see [Linework Consolidation Settings](#).

On the last screen, click “Consolidate” to execute Linework Consolidation.

Following the linework consolidation, a report appears, listing the number of elements (if any) that have been deleted or merged for each type of line consolidation.

At the bottom of this page, a checkbox gives you the option of using the simplified **Linework Consolidations Settings** (rather than the Wizard) the next time you issue the command.



Linework Consolidation Settings contains the same options as the Wizard, but all options are accessible in a single dialog box.

Fill Consolidation

In the current window, select at least two fills that you wish to include in the Fill Consolidation process.

The Fill Consolidation command (**Edit > Reshape > Fill Consolidation**) brings up Fill Consolidation Settings.

[See details at *Fill Consolidation Settings*.](#)

[Read More](#)

Check for Linear Overlaps - Experimental Feature

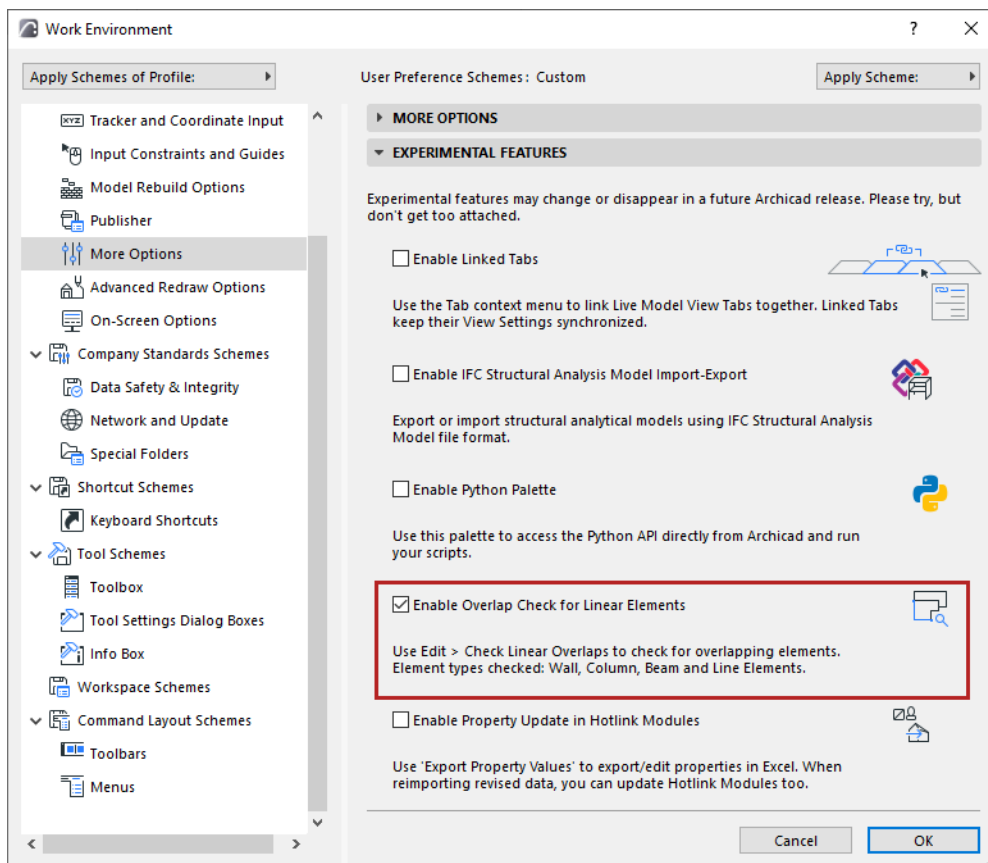
Improve model quality before creating documentation.

The add-on detects pairs of overlapping elements: if their Reference Lines overlap partially or fully, they are considered overlapping. Such pairs are then highlighted as Issues, so you can examine and correct each Issue as needed.

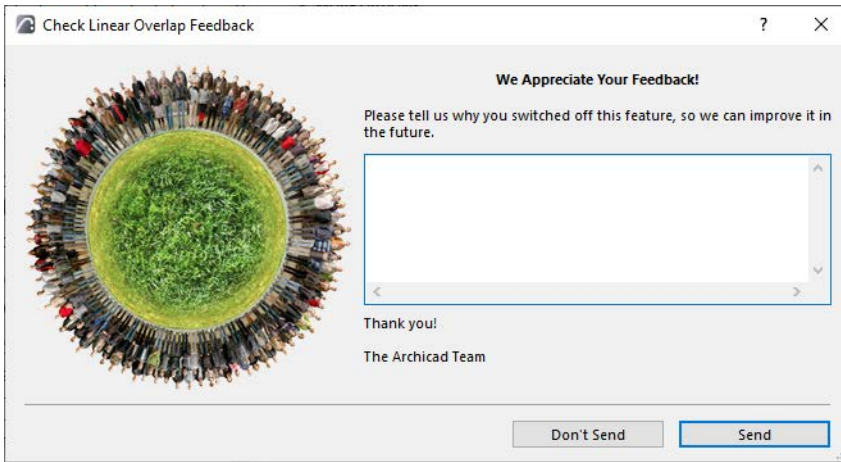
Note: This is an experimental feature which *does not work on curved elements*.

Enable the feature

This is an experimental feature. Make sure you enable it at Options > Work Environment > More Options > Experimental Features:

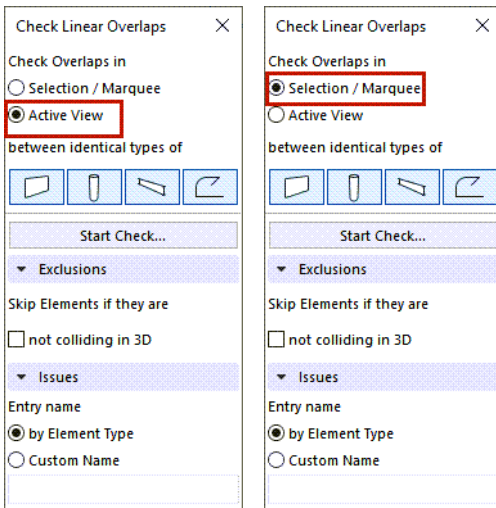


To provide feedback, try the feature, then uncheck the 'Enable Overlap Check for Linear Elements' feature and send your comments. We appreciate your feedback!



Suggested workflow

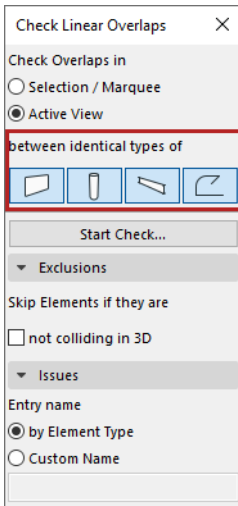
1. Go to Edit > Check Linear Overlaps to open the Check Linear Overlaps palette.
2. Choose which part of the model to check: either the Active View, or Selection/Marquee



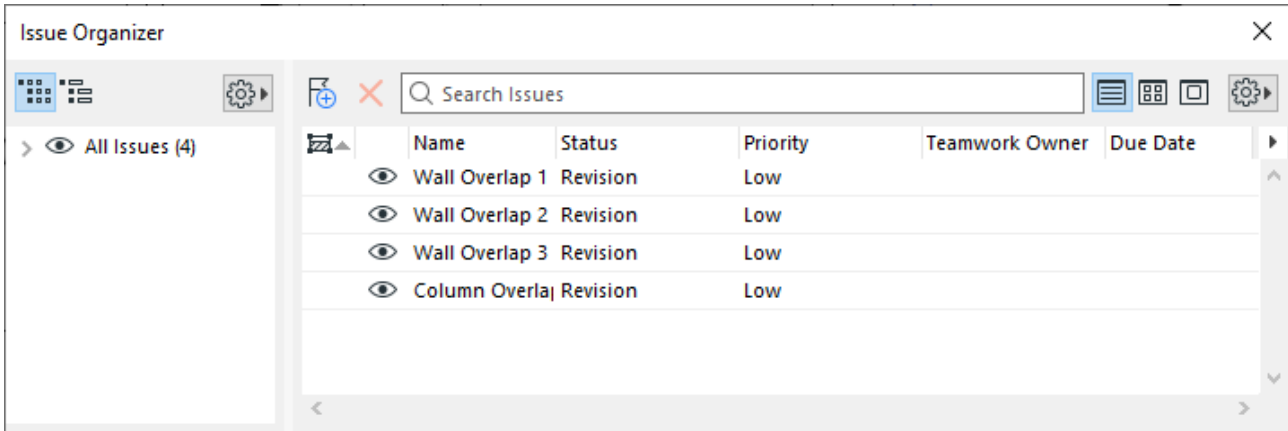
Active View: Checks all elements that are visible in the currently active view.

Selection/Marquee: Check only elements that are currently selected or Marqureed.

3. Activate the tool icons to define which element pairs to check: Walls vs Walls, Columns vs Columns, Beams vs Beams, and/or Line Elements vs Line Elements.

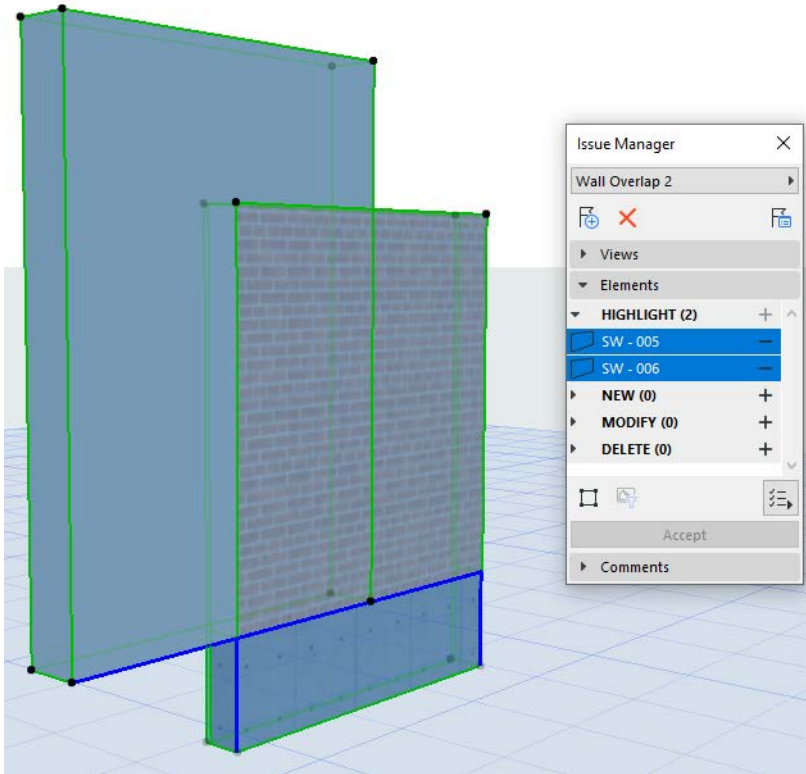


4. Click 'Start Check...'
5. To see the results, open Issue Organizer (Window > Palettes > Issue Organizer).

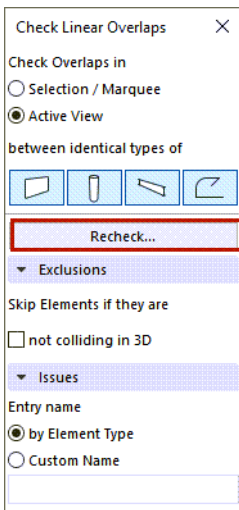


By default, all Issues are visible, have an open status and low priority.

6. Double-click an Issue to see it in Issue Manager. Zoom to the highlighted elements and adjust the model as needed.



7. Click Recheck to run it again. Any resolved Issues will be removed from Issue Manager.

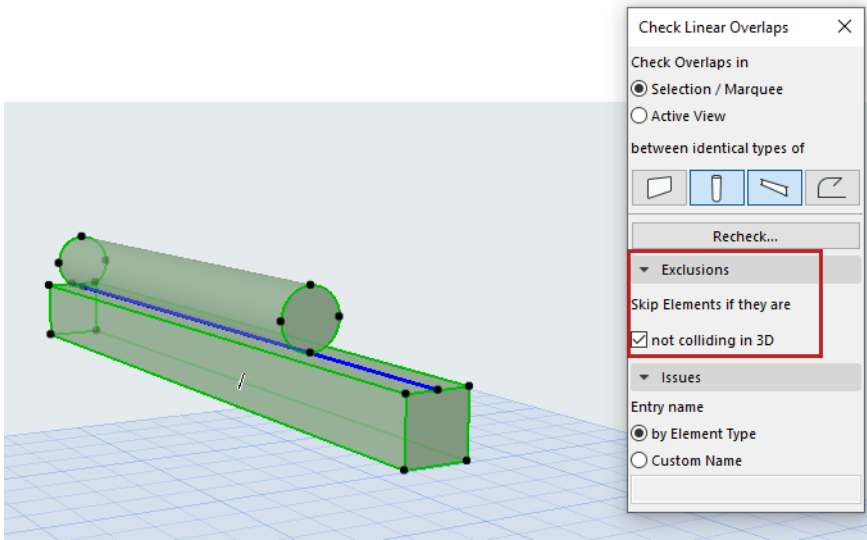


Note: The command remains "Recheck" until you close the project.

Further Options

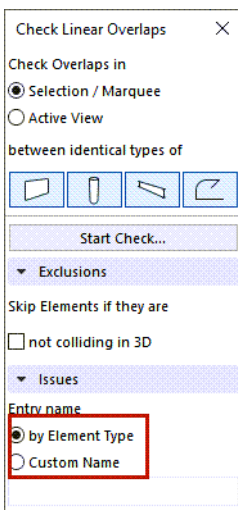
Skip Non-colliding Elements

In some cases, two elements might have overlapping Reference Lines, but they do not actually collide in 3D. To ignore such issues, check 'Skip Elements if they are not colliding in 3D'.



Reference lines overlap, but no 3D collision. These elements are skipped, no Issue found.

Automatic Issue Naming



Issue names are created based on element type by default. They are also numbered.

However, by choosing 'Custom Name', you can choose another name for the Issue in the Issues panel. This way, all created issues will share the same custom name, but they will be numbered.

You can modify the default names one by one later in the Issue Organizer.

Elements of the Virtual Building

This section of the Archicad Reference Guide describes each of the Archicad construction elements found in the Archicad Toolbox: their parameters, characteristics and variations, and how they work with each other.

This chapter also describes Parametric Objects, which are placed as library parts from your loaded or project-embedded libraries, and Openings, which provide an easy-to-use way to handle engineering breakthroughs in certain elements.

Construction Elements

Parametric Objects

Dedicated Object Tools: Doors, Windows, Skylights, Wall Ends

Openings

TrussMaker

Solid Element Operations

Properties and Classification Systems

Component Data in Archicad

The Virtual Building refers to the physical, architectural model. Archicad also includes tools and workflows for efficient collaboration with structural engineers and analysis programs. Please see details here:

Structural Analytical Model

Construction Elements

Construction Elements are the virtual equivalents of actual three-dimensional building components. Each of these elements has a dedicated tool in the Archicad Toolbox.

[Place a Construction Element in Archicad](#)

[Home Story](#)

[Define Toplink for Wall, Column, Zone or Stair](#)

[Element Display in Floor Plan and Section](#)

[Walls](#)

[Columns](#)

[Beams](#)

[Complex Profiles](#)

[Roofs](#)

[Shells](#)

[Slabs](#)

[Meshes](#)

[Zones](#)

[Curtain Wall](#)

[Stairs](#)

[Railings](#)

[Morphs](#)

[Openings](#)

[Model Element Connections](#)

[Element Intersections](#)

Place a Construction Element in Archicad

Use the Toolbox

The typical steps in element creation using the **Toolbox** are:

1. Select the element's tool in the Toolbox and open its settings dialog box.
2. Adjust the settings offered by default to suit your particular situation or purpose. You can choose a Favorite - a predefined element configuration - using the button on top of the Tool Settings dialog box, or from the Toolbox.

[See Favorites.](#)

3. Use the panels of the element Settings dialog box (or Info Box) to assign parameters.
4. Where applicable, choose a Geometry Method from the Info Box allowing you to create plain, curved, chained, rotated or polygonal variations of the elements.
5. Draw or place the element in the Floor Plan or 3D window.

Other Ways to Create an Element

Other methods of element creation include:

- Use the **Favorites Palette**: double-click on stored Favorite settings for the relevant tool, then click to place an element with those settings.
- Use the **Magic Wand** to create new elements that automatically match the shape of an existing element (e.g. create a Roof that fits exactly over your exterior Walls).

[See Magic Wand.](#)

- Create duplicates of existing elements.

[See Basic Editing Operations.](#)

- **RoofMaker** and **TrussMaker** from the **Design > Roof Extras** or **Design > Design Extras** menu.

[See TrussMaker and Create Special Roof Objects with RoofMaker.](#)

Gravitate Elements onto a Slab, Roof, Shell or Mesh

To make placement more accurate, you can use the Gravity function to place Wall, Column, Beam and Object type elements directly onto a Slab, Roof, Shell or Mesh.

[See Gravity.](#)

Home Story

A new element placed in a 2D model window is at the level of its Home Story (plus or minus any optionally defined offset).

Each construction element and object-type element is physically linked to its Home Story and moves together with it.

- On the **Floor Plan**, a new element's Home Story is the one defined in its Tool Settings dialog box: by default the current story, but you can choose a different Home Story. In any case, the element will be placed onto its Home Story.
- If you place the element in the **3D Window**, the Home Story depends on the element's elevation: the closest story below the element's bottom elevation, at the time the element is placed, will be its Home Story. (If the 3D Window is active, the Home Story control and the offset from Home Story field in the Default Tool Settings dialog are grayed out. However, you can still set the bottom elevation from Project Zero in the Tool Settings dialog boxes.)
- If you redefine an element's Home Story (e.g. from 0. Story to 1. Story), the element will change its position and move onto its new Home Story (from 0. Story to 1. Story).

Note: For reasons of compatibility with older Archicad versions, you can redefine the Home Story in such a way that the element stays in place. [See *Relink Home Story*](#).

- If you move the element onto a different story (e.g. with the Elevate command), its Home Story will, by default, change to match the element's new location. ([See *Change Home Story by Elevation*](#).)
- If you modify the Home Story itself (e.g. redefine the story level in Story Settings), the linked element will move along with it (while respecting the defined offset, if any).
- If you delete an element's Home Story, that element will be deleted!

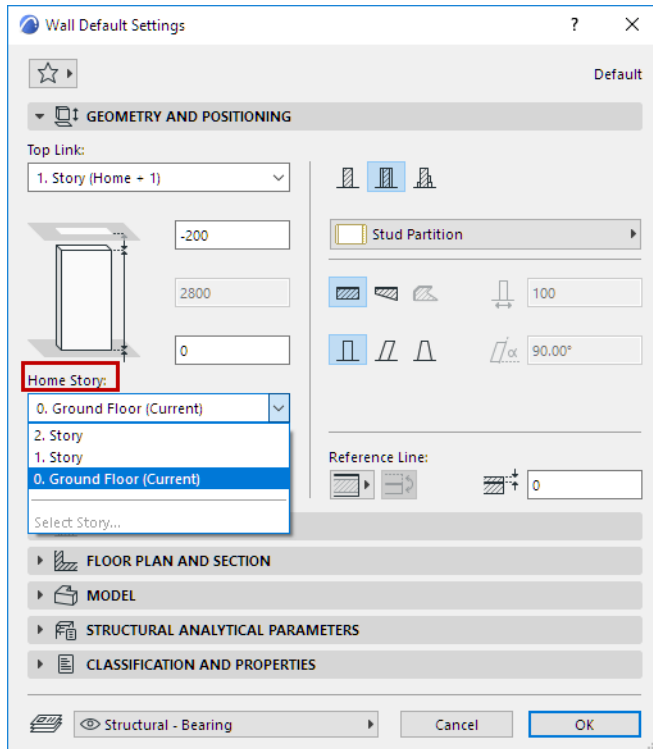
Other uses for the Home Story:

- The Home Story helps define which part of an element will be displayed, according to the **Show on Stories** control in the Tool Settings dialog box. ([See *Show On Stories*](#).)
- The Home Story can be used in listing criteria.

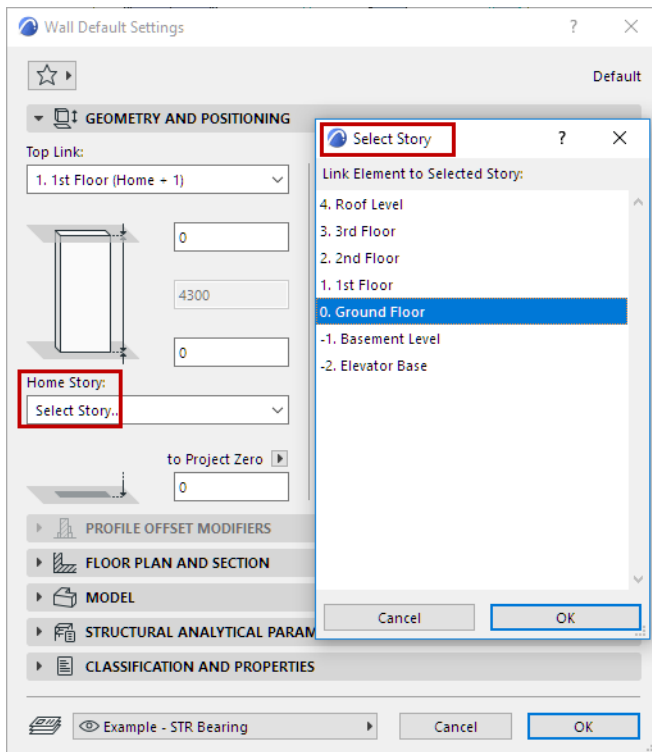
Define Home Story (Elements Placed on Floor Plan)

Before placing a new element on the Floor Plan:

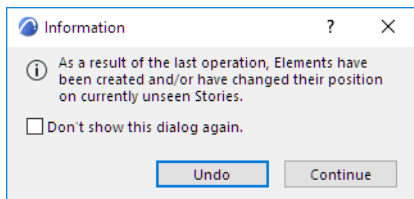
In the element's Tool Settings dialog box, or its Info Box, choose a Home Story onto which to place the element on the Floor Plan.



- Default: The element's Home Story will be the current story.
- Alternatively, choose a different story from this pop-up, or (if there are too many stories to list here) click **Select Story** to bring up the full list of stories. Choose one onto which to place the element: this will be its Home Story.



If the current story is the Ground Floor, yet you choose the 2. Story as the Home Story, the new element will be placed onto its Home Story (that is, 2. Story) - you will not see it on the current story. You will get a message telling you this.

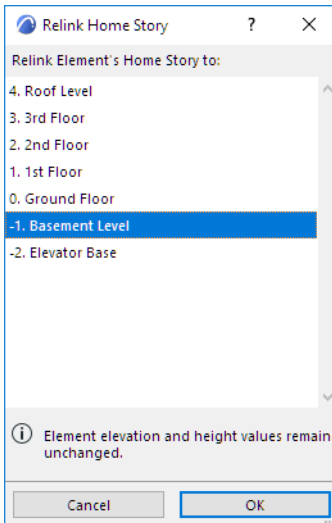


Relink Home Story

Although elements “follow” their home story - if you change an element’s Home Story, the element will move onto its new Home Story - there is a way to relink the element to a different Home Story so that the element still remains physically located on its original story.

Use the **Edit > Element Settings > Relink Home Story** command to redefine a selected element’s Home Story. The element will stay in place even as its Home Story changes.

For selected elements, this command is also available in the context menu.



[Watch the Video](#)

Home Story in the 3D Window

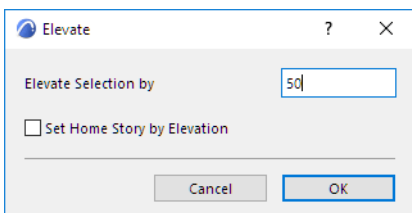
If you place the element in the **3D Window**, the Home Story will be defined *not* according to the Settings dialog box (where the control is grayed out), but rather by elevation: that is, the story beneath the element's reference point, at the time the element is placed.

Note: For a Shell, this reference point is the first point used to draw the Shell (not the bottommost point).

Change Home Story by Elevation

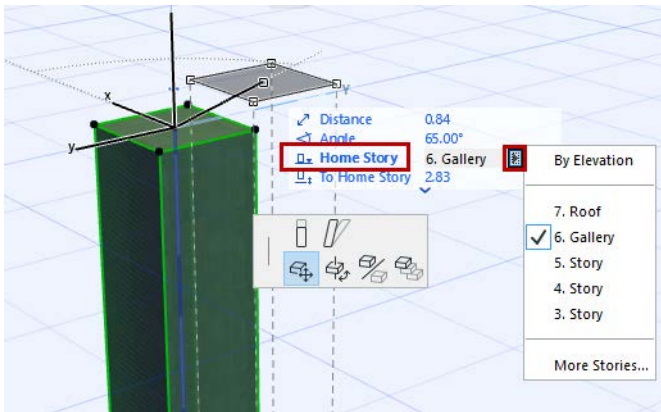
If you move a placed element to a different story, the element's Home Story can change to match the element's new location.

As you elevate the element, the "By Elevation" text in the Tracker indicates that the Home Story changes to match the element's new elevation:

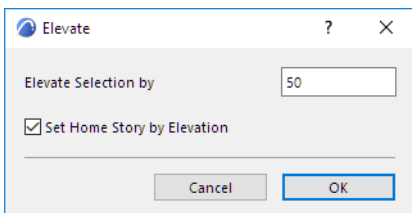


Notice that a Guide Line appears marking the story positions, to make it easier to place the element.

Alternatively, use Tracker controls to choose a different Home Story for the selected element, regardless of its elevation:



Similarly, in either 2D or 3D windows, you can automatically reset an element's Home Story as its elevation changes if you use the **Elevate** or **Multiply** commands, by checking the **Set Home Story by Elevation** box:



If you uncheck the box, the element's Home Story as set in the Tool Settings (Info Box) will remain in effect regardless of the elevation change.

See also [Elevating Elements](#) and [Multiply Elements](#).

Define Toplink for Wall, Column, Zone or Stair

You have the option to link the top of certain elements (Wall, Column, Zone, Stair), to a story. This top link is defined relative to element's Home Story (Home+1, Home+2, etc.).

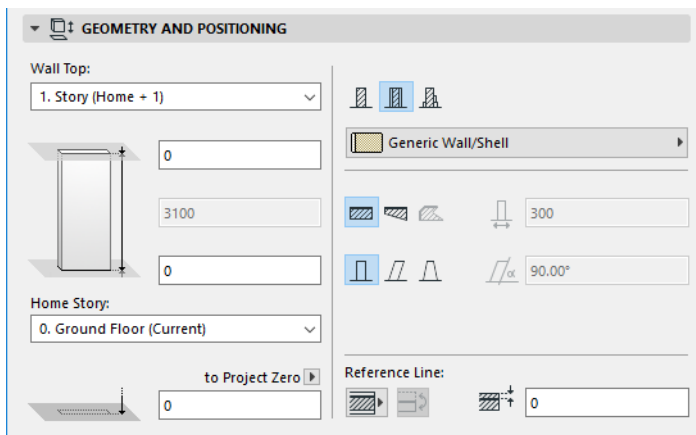
(By default, the element top is linked to one story above its Home Story).

If you later modify the positions and heights of stories in your projects, the heights of any top linked elements will follow automatically. Elements that you have *not* top linked are therefore fixed-height elements.

Stairs have some additional Toplinking options.

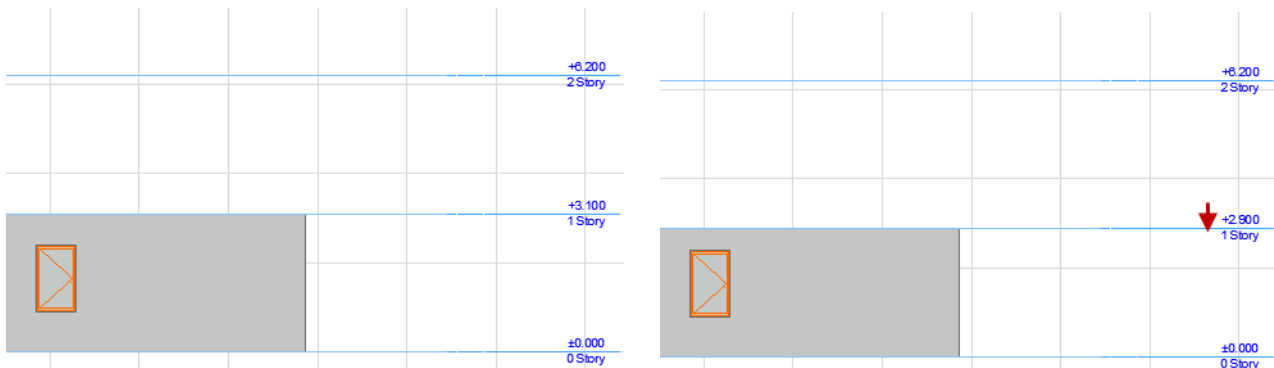
[See Stair Geometry Controls.](#)

Top Link controls are located in the Geometry and Positioning panels of Tool Settings for the Wall, Column, Zone and Stair (as well as the Info Box).



- If you don't want to top link, choose "Not linked".
- Optionally, define an offset for the element top from the element's top linked story: the element height will change accordingly, but the link to a particular story remains.

If you top link an element to a particular story, and if you later modify this story's position, the height of the linked element will automatically change accordingly:



If you delete the story which is the top link, then the story above takes its place.

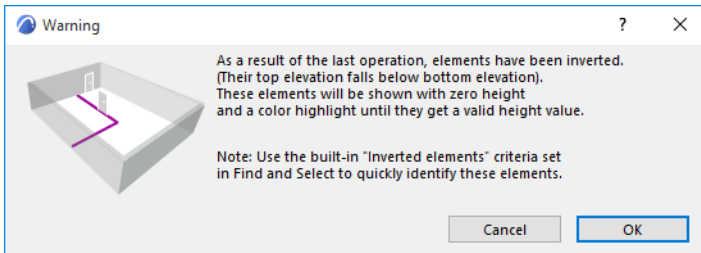
[Watch the Video](#)

[Watch the Video](#)

[Watch the Video](#)

Inverted Elements

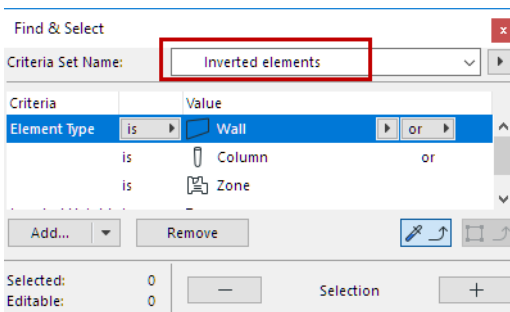
Changing story heights may result in inverted elements: their top elevation will fall below the bottom elevation. This can occur, for example, if your project contains stories of varying heights, and you are placing a series of top linked walls with certain offsets which may not “fit” onto a particular story.



Such elements are displayed as having zero height (e.g. such a wall is displayed as a straight line in section view), and are highlighted so that you can easily find and correct them.

Find Inverted Elements

Or use the built-in Inverted elements criteria set from **Edit > Find & Select** to find and select any inverted walls, columns and/or Zones.



Note: Use the **Check for Inverted Elements** command to find and list any inverted elements in your project. This command is not available by default in the Archicad Work Environment. To use this command, customize your Work Environment menus to include it.

[See Menu Customization Dialog Box.](#)

Inverted Elements Highlight Color

To set the highlight color of inverted elements, use the control at **Options > Work Environment > On-Screen Options**. (This color is also used for invalid Stairs.)

[Watch the Video](#)

Walls

In Archicad, you can draw walls that are straight, curved, trapezoidal and polygonal.

Wall structure can be basic (of a single component), or composite (composed of several components). You can also create complex walls (Profile Wall) of any custom shape and with a combination of components. Like other construction elements, a Wall's essential structure derives from its Building Material.

Certain parametric objects such as Doors, Windows, and Wall Ends can only be placed into Walls.

A Curtain Wall is constructed with a separate, dedicated Curtain Wall tool.

[See *Curtain Wall*.](#)

Wall parameters are set in Wall Tool Settings. The bottom of every Wall (like other elements) is linked to its Home Story; you can also top-link the Wall to any story above it in the project. As needed, define offsets from its top link or Home Story; this will affect the Wall Height. Alternatively, just enter a fixed Wall height, without a top link.

Certain Walls can be included in a Structural Analytical Model generated in Archicad.

[For details, see *Structural Analytical Model*.](#)

Display of Walls

Use Wall Settings to choose attributes for the display of your wall in the various 2D and 3D windows, and to define its Building Material.

Use the Floor Plan Cut Plane, combined with element projection preferences, to determine which parts of the placed wall should be displayed.

While a Wall's surface is defined as part of its Building Material, you can override a Wall's surfaces on the Wall's "outside" (reference line side), edge, and "inside" surfaces, using the controls in the Model Panel of Wall Settings.

Some aspects of Wall display depend on the current settings in **View > On-Screen View Options**.

- **Clean Wall & Beam Intersections** can be set on and off.
- Fill patterns can be shown as **True Fill Appearance** or bitmap.

To use Graphical Overrides for Wall fills:

[See *Graphic Override Rules*.](#)

Related Topics:

[Wall Reference Line](#)

[Create a Straight Wall](#)

[Create a Curved Wall](#)

[Create a Chain of Walls](#)

[Create a Rectangle of Walls](#)

[Create a Trapezoid Wall](#)

[Composite Structures](#)

[Create a Polygon Wall \(PolyWall\)](#)

[Stretching Walls](#)

[Create a Slanted or Double-Slanted Wall](#)

[Modify Slanted Wall](#)

[Create a Log Wall](#)

Complex Profiles

Walls and Other Elements

Modify Wall Reference Line

Invert Wall Direction

Wall Tool Settings

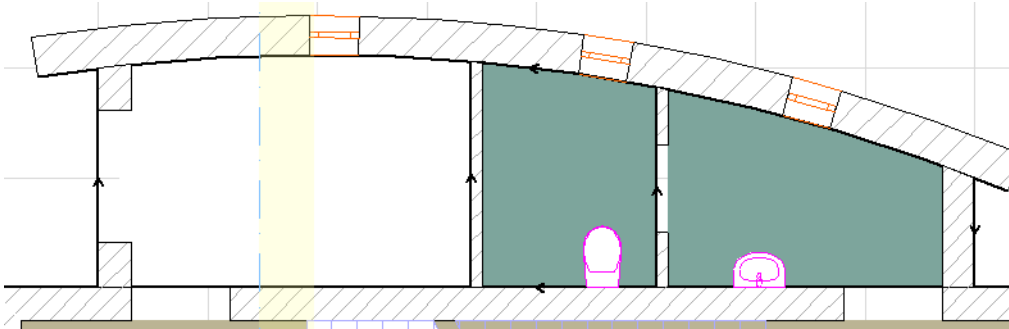
Define Toplink for Wall, Column, Zone or Stair

Structural Analytical Model

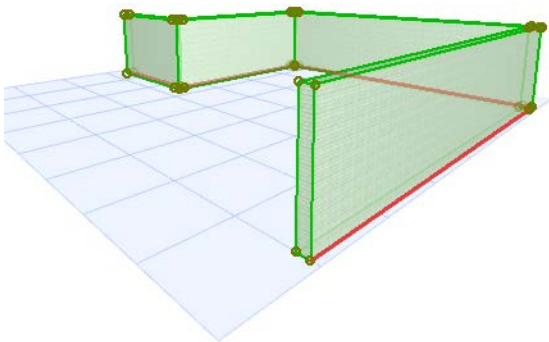
Wall Reference Line

Each Wall has a **reference line** and a direction. The Wall reference line is visible on the Floor Plan if you switch on:

View > On-Screen View Options > Walls & Beams Reference Lines



In the 3D window, the reference line of a selected Wall is shown in a separate color: you can change this color using the “Reference Lines in 3D” setting at **Options > Work Environment > Selection and Element Information**.

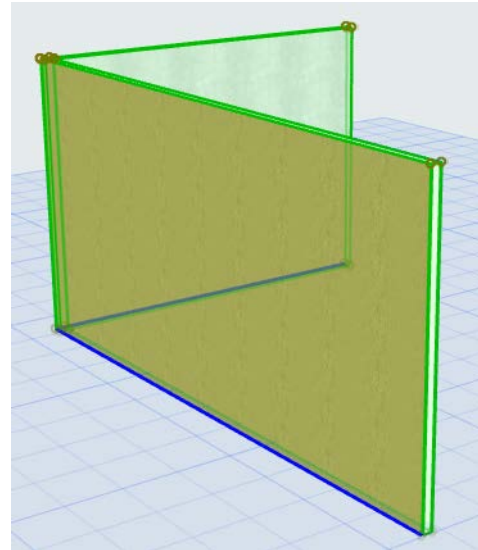
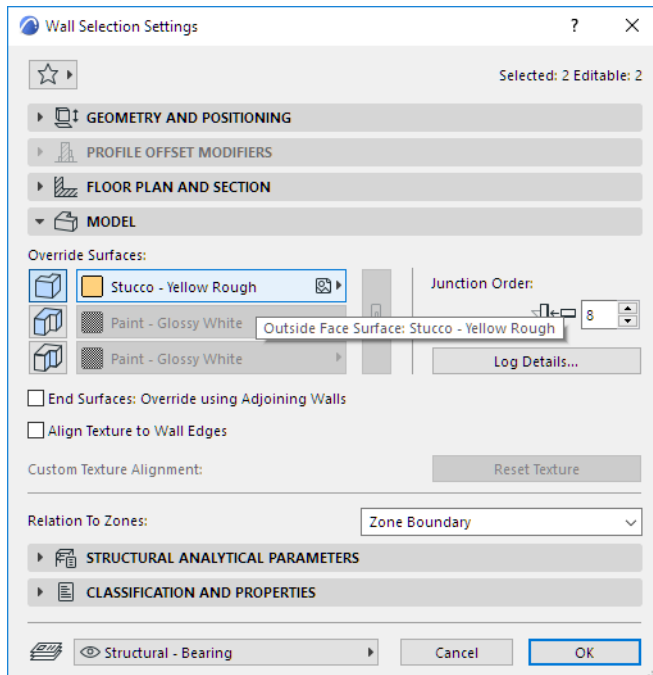


Uses for Reference Line

- The reference line helps you achieve the precise connection of Walls for clean intersections. Associating the reference line to the core allows finishes to be edited without moving the core’s position.
- The reference line establishes hotspots and edges for selecting, moving and transforming Walls. Some Pet Palette editing commands are accessible only by clicking on the reference line of the selected Wall, as shown in the two images here:



- The reference line position determines the Wall's "outside" and "inside" surfaces. Separate surface materials can be applied to the Wall's "outside" and "inside" surfaces, using the controls in the Model Panel of Wall Settings.



Wall Direction

The **direction** of the Wall, indicated by the arrow on the reference line, is defined by the order in which you define the wall's endpoints.

Note: The directions of curved Walls and polygon Walls are fixed and cannot be reversed.

Note: With the commands at **Edit > Reference Line and Plane**, you can modify the Reference Line position and the direction of selected Wall type elements.

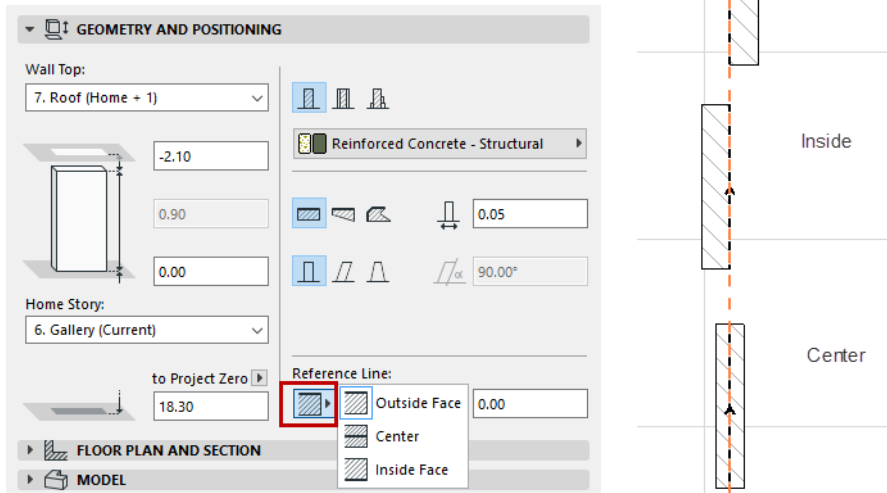
See [Modify Wall Reference Line and Invert Wall Direction](#).

Change Wall Reference Line Position

Use the Info Box or Geometry and Positioning Panel of Wall Settings to choose a reference line position. Click the "C" shortcut to cycle through the reference plane options in the Wall Info Box.

Reference Line of a Basic Wall

A basic Wall has three possible reference lines: running along the outside face, inside face, or the center. By default, the reference line runs along the "outside face".



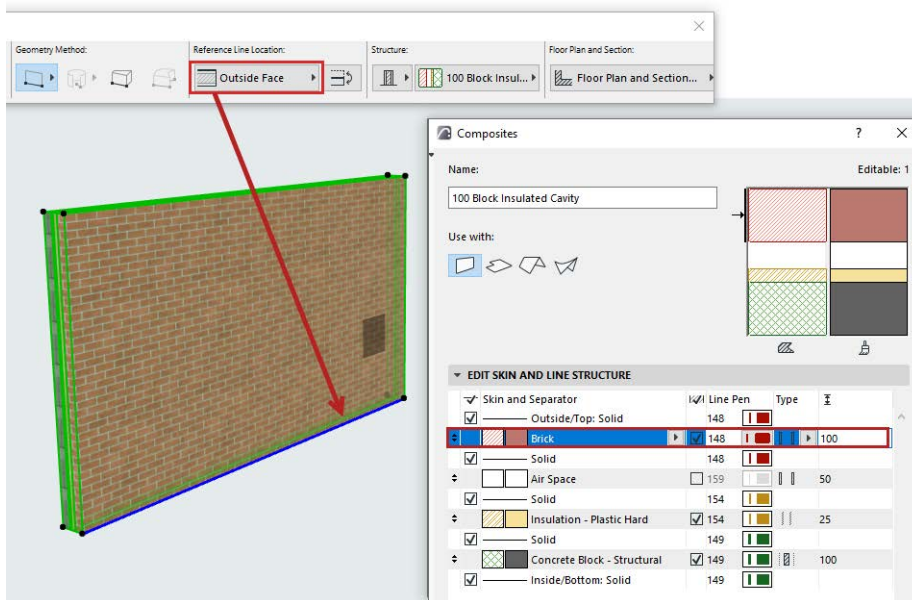
Reference Line of a Composite Wall

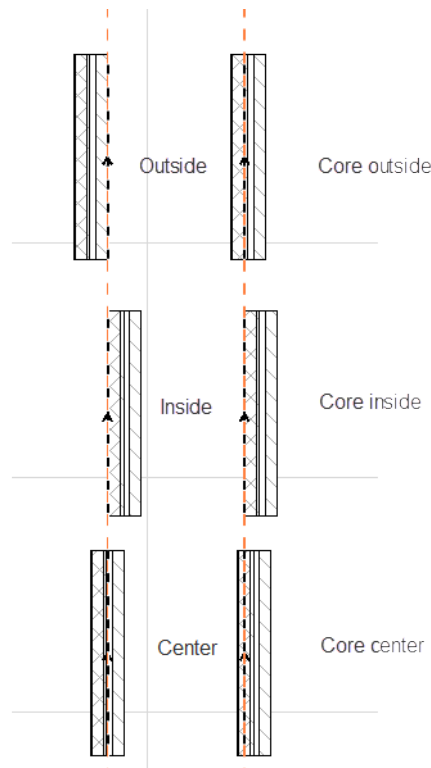
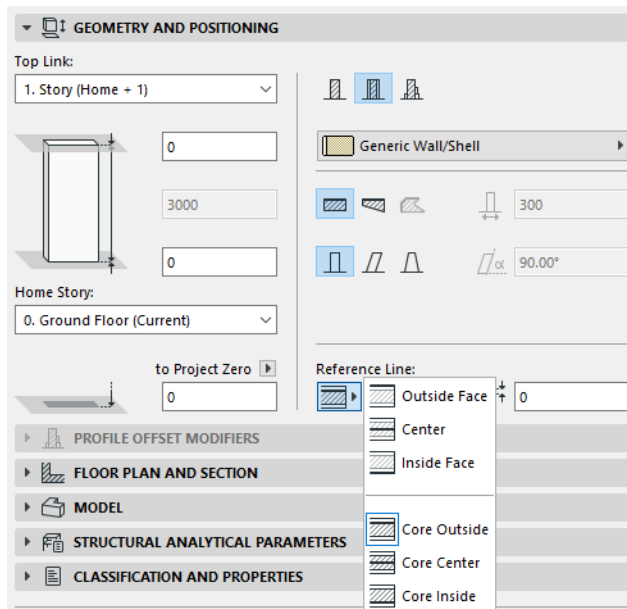
A **composite Wall** has six possible reference lines: Outside/Center/Inside of the Wall, plus Outside/Center/Inside of the Wall core.

Note: If the chosen composite structure has no core, then the core-related Reference Line choices are grayed out.

By default, the reference line runs along the outside face.

For a composite, the “outside” surface is the first listed skin of the composite, in Composite Settings.





[Watch the Video](#)

Reference Line of a Trapezoid Wall

A Trapezoid wall has two possible reference line locations: Outside Face or Core Outside (if it has a core).

See also [Create a Trapezoid Wall](#).

Reference Line of a Polygon Wall

Reference line controls in Wall Settings do not apply to Polygon walls. (Polygon wall reference lines can be moved graphically via the pet palette.)

Reference Line of a Profile (Complex) Wall

The reference line of a Profile (complex) wall depends on the user origin defined in Profile Manager/Editor.

Reference Line Offset

If using an “outside” or “inside” reference line, enter a positive offset to move the reference line toward the center (or center of core) of the Wall. A negative offset will move the reference line away from the center (or core center). (No offset is available if the reference line is in the center.)

As you adjust the reference line location, notice that the body of the Wall moves laterally. The reference line stays in place.

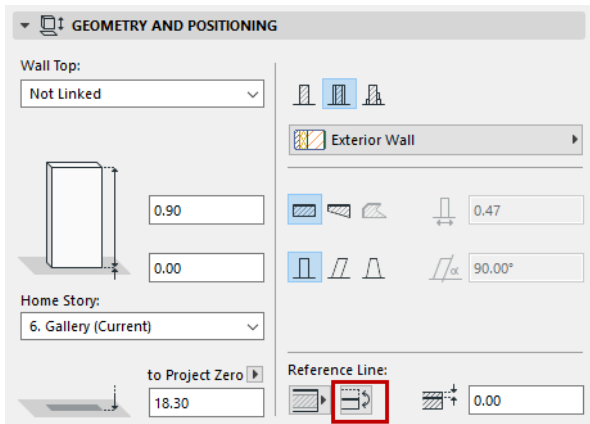
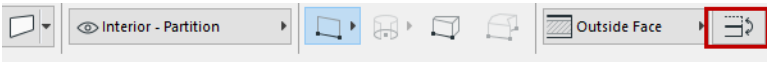
Note: In contrast, to move the reference line while retaining the Wall’s position intact, and/or mirror the openings, use the Modify Wall Reference Line command.

See [Modify Wall Reference Line](#).

Flip Wall Along Reference Line

This command flips (or mirrors) a selected Wall along its reference line. The reference line stays in place, but the body of the Wall flips to the other side. Notice that the openings do not change their opening direction.

The command is available in the Wall Info Box, and in the Geometry and Positioning Panel of Wall Settings.

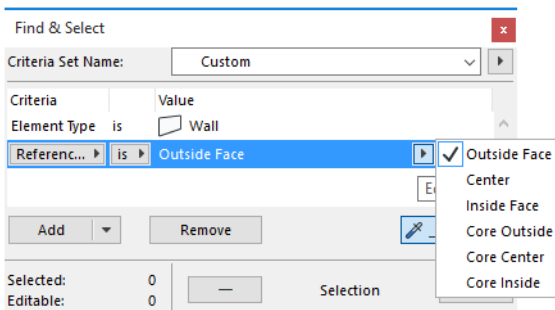


Note: To move the reference line while retaining the Wall's position intact, and/or mirror the openings, use the Modify Wall Reference Line command.

[See *Modify Wall Reference Line*.](#)

Reference Line in Interactive Schedule/Find & Select

In the Interactive Schedule, you can Find & Select the Walls in your project, or list them in the Interactive Schedule, based on the location of their reference lines:

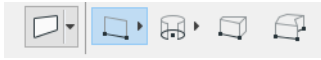


Wall List		
Area [m2]	Reference Line Location	Perimeter [m]
2.64	Outside Face	18.99
2.76	Inside Face	
6.37	Outside Face	

Create a Straight Wall

The Single Straight Wall method produces one straight wall element at a time.

With the Wall Tool selected, choose the Straight Wall Geometry method from the Info Box and draw the wall segment on the plan.



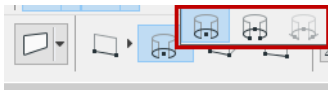
In both Floor Plan and 3D, you define the length of Wall segments by clicking at their endpoints.

Note: This CAD-like drawing method is the default method of drawing segments in Archicad, but you can change it in **Options > Work Environment > Input Constraints & Guides**.

Walls can be stretched using either menu commands or pet palette commands.

Create a Curved Wall

With the Wall Tool selected, choose one of the three Curved Wall Geometry Methods from the Info Box.



- Centerpoint and radius
 - Click to define centerpoint.
 - Move the cursor to define the radius, then click a second time.
 - Move the cursor to draw the length of the Wall, then click to complete.
- Circumference
 - Click on three points of the Wall's circumference (e.g. other elements or intersections, or special snap points).
 - Click a fourth time to define the length of the Wall.
- **Tangential:** Define a full circular Wall based on three **tangential edges** or points.
 - Select three initial points: these can be a tangent edge, a node, or a free-floating point.

Note: Not available in 3D window.

The next step depends on the geometric situation.

- If there is only one solution, the circular Wall is automatically drawn.
- If there are multiple solutions, the Eyeball cursor appears and the ghost contour of the Wall flips from one position to the other as you move the cursor around. Click when it is at the right place to complete the circular Wall.
- If there is no solution, (for example, if you define three parallel edges for tangency), no circle will be made.

Note: Since Walls need to have two endpoints, the fully closed circular Wall that you draw will in fact consist of two half-circles. Walls in the forms of full ellipses, elliptic arcs or splines cannot be created directly, but you can trace the shapes of plain drawing elements with the Magic Wand.

Related Topics:

[Stretching Curved Elements](#)

[Curve/Straighten Element Edge](#)

[Board in Curved Wall](#)

[Board in Polygon Wall](#)

[Doors/Windows in Curved Wall](#)

Create a Chain of Walls

A chained wall is a sequence of connected straight or curved wall elements.

With the Wall Tool selected, choose the Chained Wall method from the Info Box. (Pop out the small black arrow on the Straight Wall Method button.)



As soon as you start drawing the Wall, a pet palette appears with geometry options for each segment:



- A straight segment
- A curved segment tangential to the previous one (cannot be used for the first segment)
- A curved segment tangential to a line you define before actually drawing the segment
- A curved segment passing through two points
- A curved segment defined by its centerpoint, radius and length

Click to place each successive segment using any of the geometry methods.

Double-click to complete the Wall.

- Click Backspace to undo just the last input segment.
- Click Cancel or Delete to undo the entire Wall.
- If you change the Wall's attributes in the **Info Box** during the creation process, all created Wall segments will have the modified attributes (reference line position, line type, etc.).

This input method is analogous to creating a chained Beam.

[See *Chained Beam*.](#)

Create a Rectangle of Walls

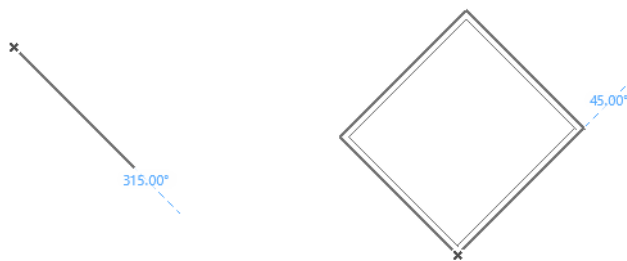
With the Wall Tool selected, choose the **Rectangle Wall** method from the Info Box. (Pop out the small black arrow on the Straight Wall Method button.)



Draw the diagonal of the rectangular wall and click to define its length. The rectangle wall is created.

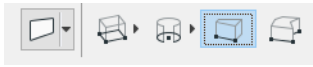
Its four sides are always aligned orthogonally with the Normal Grid and are not affected by the use of a Rotated Grid.

The **Rotated Rectangle Wall** method produces four wall elements as with the previous method, except that you first define a rotation vector for the rectangle's base reference line. Then drag the cursor to define the rectangle.

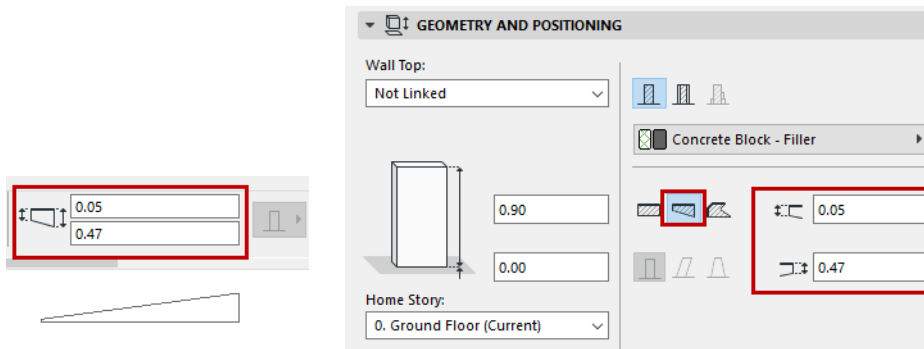


Create a Trapezoid Wall

The Trapezoid or Non-Parallel method allows you to draw straight Walls whose thickness is not constant.



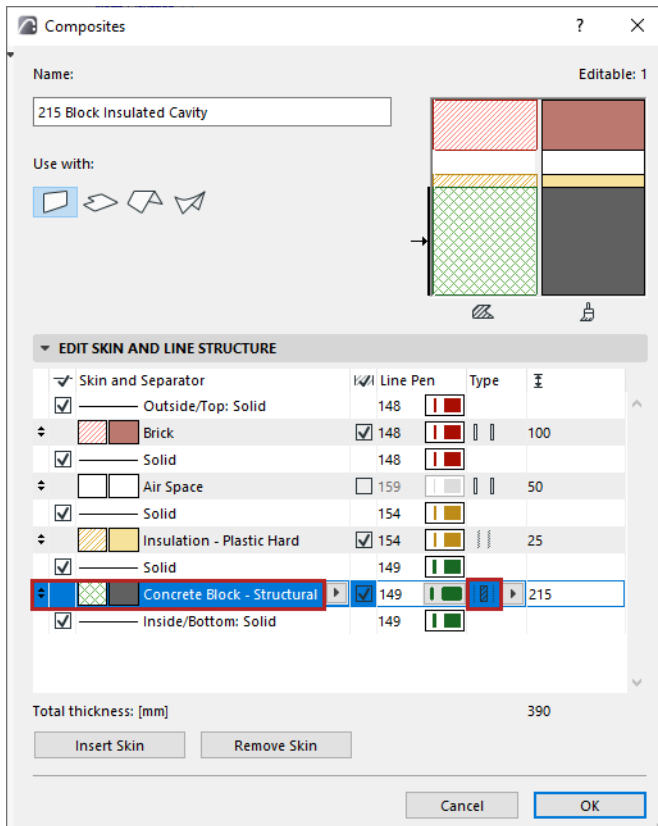
You draw this type of Wall as a straight Wall, except that you will first enter the thickness of the Wall at its two endpoints, using the Wall Thickness fields in the Info Box or in Wall Settings.



Composite Trapezoid Wall

When drawing a trapezoid Wall with a composite fill, only the *last* Core skin's thickness reflects the trapezoidal shape of the Wall. In the image below, two of the skins are marked as Core, but only the Concrete Block core skin - the last Core skin listed in the dialog box - has a trapezoid shape.

[See also Reference Line of a Trapezoid Wall.](#)



If the composite structure has no core, then the composite's thickest skin gets the trapezoid shape. (If several skins are equally thick, then the last of those skins listed in the dialog box gets the trapezoid shape.)

See also [Composite Structures](#).

Create a Polygon Wall (PolyWall)

To define a polygonal wall block of free shape, choose the *Polygon Wall* Geometry Method in the Info Box. Polygon Walls can contain Doors or Windows. For Polygon Walls, only simple fills are available (no composites).



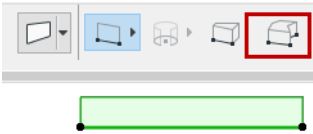
Note: If your polygon wall contains a window with a board, an option in Window Settings lets you set the board edge to either follow the wall polygon (Parallel to Wall Polygon) or to stay parallel with the window.

See [Board in Polygon Wall](#).

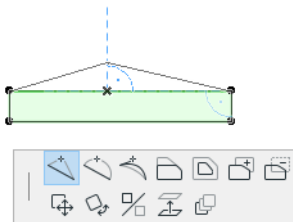
Draw the contour of the Polygon Wall, using pet palette segment options as needed (as when drawing a chained wall). Both straight and curved edges are allowed.

Once you have closed the polygon, click with the hammer cursor to complete the wall.

You can also transform a regular wall into a polygon wall: select the wall, then choose the Polygon Wall geometry method.



Now use the **Pet Palette** commands to model the wall shape as desired.

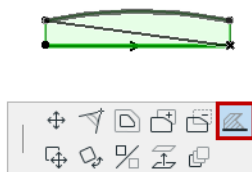


If you define a polygon whose edges intersect themselves, Archicad will warn you about this, but the wall will still be drawn.

The wall's edges will include one **reference edge** (like a straight wall's reference line).

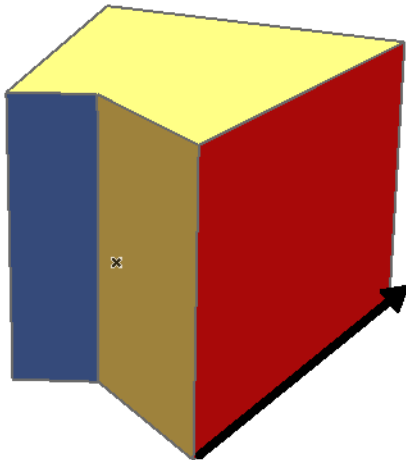
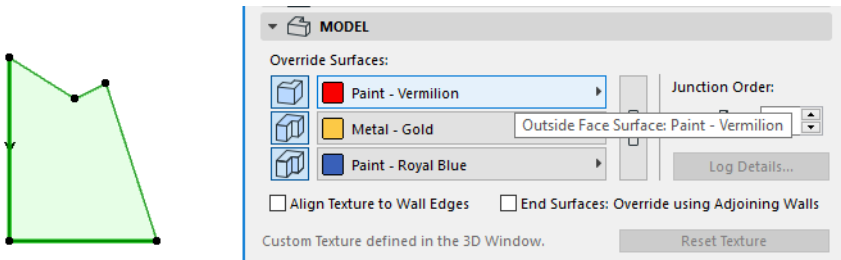


To make another edge the reference edge, select an endpoint of the current reference edge. The pet palette contains a command with which you can move the reference edge endpoints to other nodes of the polygon wall.



When you override surfaces of a polygon wall in the Model panel of Wall Settings, the surfaces are assigned depending on the location of the reference edge.

- The “Outside Face” is the reference-edge surface; here, in red.
- The “Edge Surface” is assigned to the two surfaces adjacent to the reference edge; here, in gold.
- The “Inside Face Surface” surfaces are assigned to all other edges (those which are opposite the reference edge), here, in blue.



To edit polygon walls, click the nodes and edges of the selected Walls and use the pet palette. When connected to other, plain walls, wall polygons keep their original contour by default.

For more information, see [PolyWall Corners](#).

Stretching Walls

To stretch/shrink the length of straight Walls with the menu command:

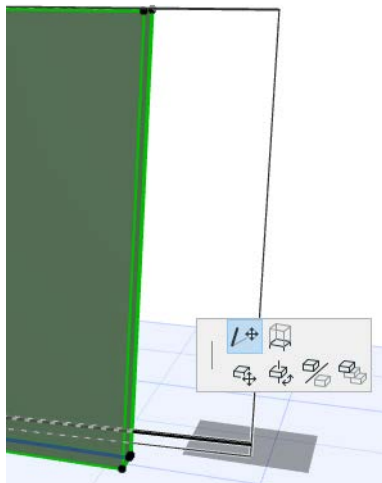
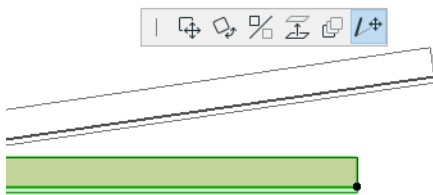
1. Select a wall.
2. Choose the **Edit > Reshape > Stretch** menu command, or the **Stretch** command from the pet palette.
3. Click the reference line endpoint and move it.
4. Click again to either define the new endpoint as an extension (or reduction) of the previous length or define an entire different new location. The wall is stretched or shrunk and, if needed, rotated according to the new endpoint. The other endpoint will remain at its original position.

Note: Multiple Walls, Lines and Arcs with overlapping endpoints can all be stretched at the same time with the **Stretch** menu command, or by using the Marquee.

For more information, see [Stretching with the Marquee Tool](#).

When using the pet palette:

1. Select a wall.
2. Click one of the wall's reference line endpoints. The pet palette appears. Choose the stretch icon.

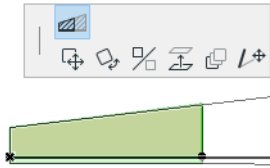


Click to define the new endpoint.

3. You can stretch a curved wall by using a special angular or radial stretch commands.

See [Stretching Curved Elements](#).

To stretch a Trapezoid wall, you can use the special **Stretch Trapezoid Wall** command from the pet palette: this will increase/decrease the width of the wall proportionally as you stretch it from either end.

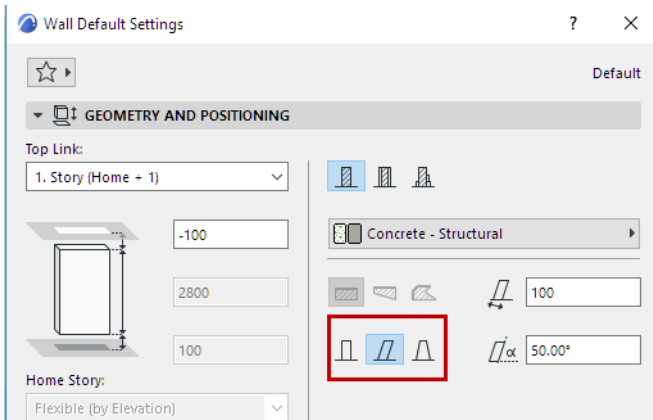


See also [Stretching or Shrinking Lines](#); [Stretching Objects](#).

Create a Slanted or Double-Slanted Wall

To draw a wall whose vertical axis is slanted on either or both sides:

1. Go to Wall Settings (Geometry and Positioning panel).
2. Click the desired Slanted Wall Icon.
3. Enter the desired wall angle(s).

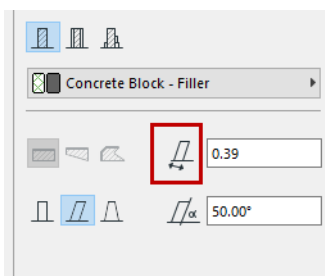


A slanted wall tilts in one direction; its two faces are parallel to each other. A double-slanted wall has faces which are not parallel.

The height of either type of slanted wall is the distance between its lowest and highest point. When setting the angles of a double-slanted wall, you are constrained by the minimum thickness of the top of the wall. For a basic structure, this can be zero. The top of a composite wall, however, has to be wide enough to accommodate all of the skins.

When entering a height value of a double-slanted wall, you face the obvious constraint that the wall's width at the top cannot be negative (i.e. once the two faces meet, you can't go any higher.)

The thickness of a slanted wall is its thickness perpendicular to its slant vector (as indicated by the icon in the Info Box):



The thickness of a double-slanted wall equals the width of the wall base.

If you switch from one kind of slant to another, the wall is transformed as follows:

- If you transform a single-slanted into a double-slanted wall, the new wall's width (i.e. its wall base) will take on the width of the old, single-slanted wall.
- Similarly, if you have a double-slanted wall and change it to either a vertical or single-slanted wall, the new wall will have the same width as that of the old, double-slanted wall.

Double-Slanted Wall in Section view

In Section view, a double-slanted wall is trapezoid in shape.

If the wall has a composite structure, its skin geometry corresponds to that of a Composite Trapezoid Wall.

[See Composite Trapezoid Wall.](#)

[See also Modify Slanted Wall.](#)

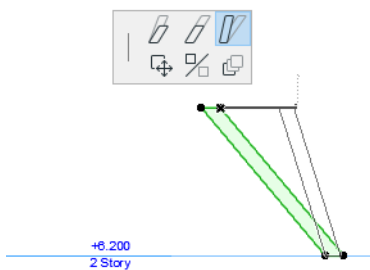
Modify Slanted Wall

Slanted walls can be stretched and/or their slant angle can be modified with pet palette commands in 3D and Section views.

In 3D, you can:

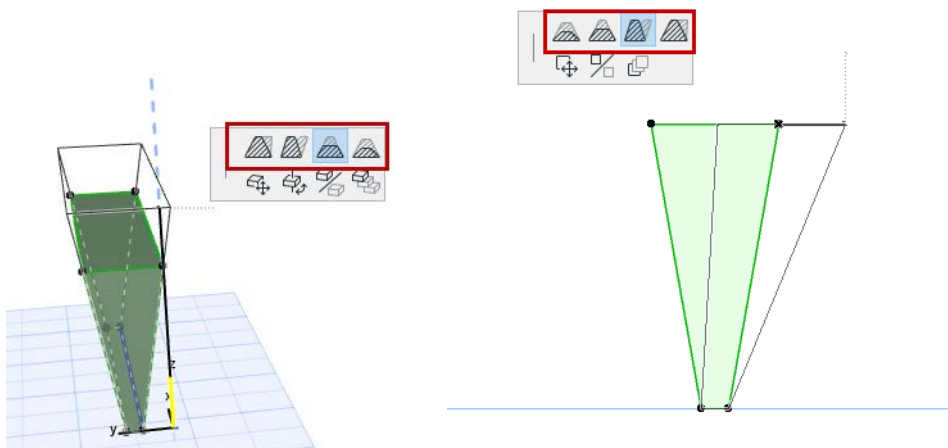
- Slant a wall or change the slant angle by selecting its corner point and dragging it along the horizontal plane. (Use **Modify Angle** in the pet palette.)
Note: You cannot slant a wall whose reference line is centered.
- Stretch a slanted wall by moving its top or bottom edge up or down along the wall plane (Use **Stretch slanted height**.)
- Stretch a slanted wall vertically. (Use **Stretch height** in the pet palette.) As you change the wall height, the wall thickness remains the same but its slant angle is adjusted accordingly.

In Section view, you can use the same functions. (“Modify Slant Angle” is possible only if the wall’s reference line is perpendicular to the section line.) In Section view, the Tracker’s “Distance” field provides a way to edit the wall inclination using its offset (e.g. Distance) value.)



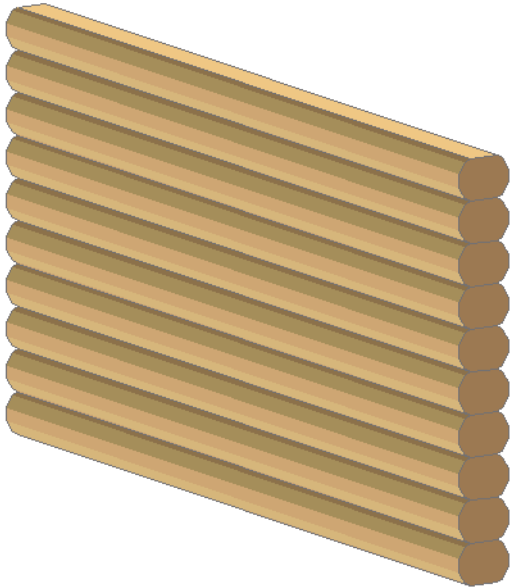
Double-Slanted Wall Editing

You can edit double slanted walls in 3D or Section in four different ways:



Create a Log Wall

You can create straight walls constructed from logs. Log details appear in both the 3D Window and Section/Elevation/IE Windows.



Clicking the **Log Details** button in the Model panel of Wall Settings provides a number of geometry and texture options.

Note: Log construction is only available for basic straight walls. When determining the log's height, remember that if you enter a value that is larger than the wall's width parameter, the logs will not rest on each other.

See [Log Details Dialog Box](#).

Walls and Other Elements

For information on how intersections (junctions) work among construction elements:

[See *Element Intersections*.](#)

The following other features also affect Wall intersections:

- Columns and Walls

[See *Columns and Walls*.](#)

- The **Connect** commands can trim Walls to Roofs or Shells.

[See *Trim Elements to Roof or Shell*.](#)

- You can also crop a Wall to a Single-plane Roof.

[See *Crop Elements to Single-plane Roof*.](#)

- The **Adjust Elements to Slabs** function lets you adjust Walls to specific levels of one or more Slabs that are located above and/or below them.

[See *Adjust Elements to Slabs*.](#)

- Walls can be connected to Curtain Walls using the **Design > Connect > Join Wall(s) to Curtain Wall**. The same command is available under **Design > Modify Curtain Wall**.

[See *Join Wall to Curtain Wall*.](#)

- **Relation to Zones:** In the Model Panel of Wall settings, define how the Wall should behave in relation to Zones: as a Zone boundary (delimiter), or whether its area/volume should be subtracted from the Zone it is in, or whether it should have no effect at all on a Zone.

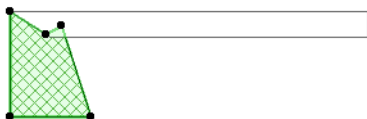
[See *Relation to Zones*.](#)

PolyWall Corners

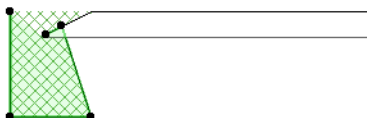
When connected to other, plain walls, wall polygons keep their original contour by default.

You can override this default setting.

In the following example, a Single Wall is connected to a Wall Polygon.



Now, check the **PolyWall Corners Can Change** box in the **Geometry and Positioning** section of the **Wall Settings** dialog box (available when a PolyWall is selected).



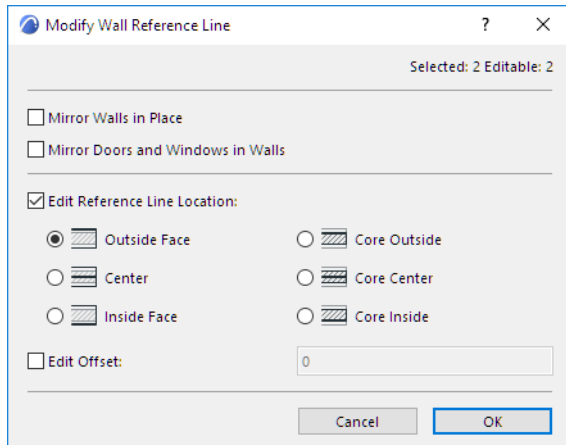
This effect is visible if you turn on **View > On-Screen View Options > Clean Wall & Beam Intersections**.

Modify Wall Reference Line

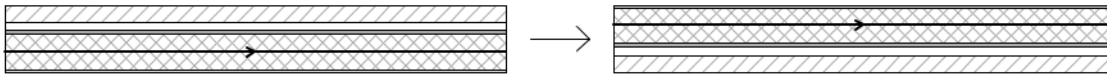
Use the commands of the **Reference Line and Plane** hierarchical menu to modify the Reference Line position and the direction of selected Walls. By default, this menu is at: **Edit > Reference Line and Plane**.

This command moves the reference line *without changing the Walls' position* on the Floor Plan

The **Modify Wall Reference Line** command brings up the following dialog box:

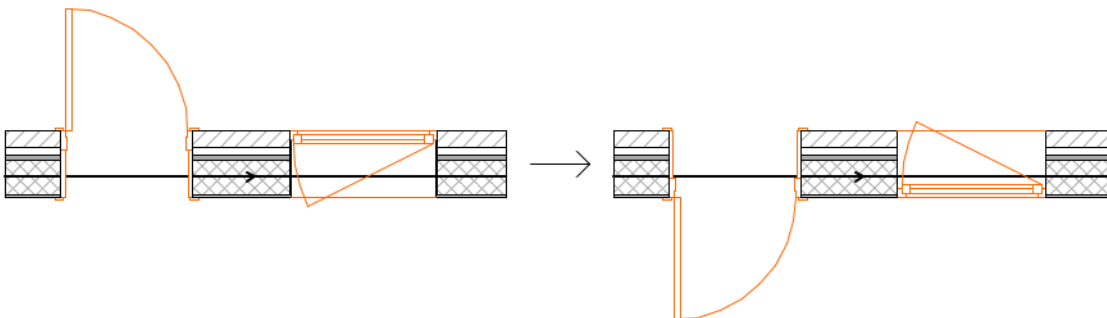


- **Mirror Walls in Place:** Check this box to flip the wall's reference line around the central axis of the wall (direction remains unchanged).










Doors and Windows remain unchanged (unless you also check the following box):

- **Mirror Doors and Windows in Walls:** Check this box to reverse the opening direction of the Wall's Doors and Windows.

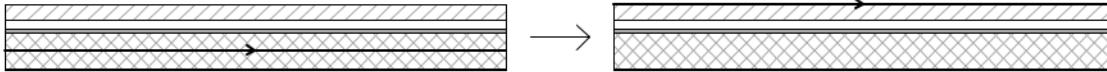


Edit Reference Line Location: Check this box, then use the controls to move the reference line of the selected Walls without changing the Walls' position on the Floor Plan.

Edit Reference Line Location: 

 Outside Face
  Core Outside
  Center
  Core Center
  Inside Face
  Core Inside

Edit Offset:



Note: Modifying the reference line location here has no effect on Complex Profile Walls.

Edit Offset: Check this box, then enter an offset distance (if any) of the reference line from its defined location. A positive number moves the reference line toward the center of the Wall core; a negative number moves it away. (The Wall itself does not move.)

Offset is not available if the reference line is at the Center or Core Center.

[Watch the Video](#)

Invert Wall Direction

With the **Invert Wall Direction** command (**Edit > Reference Line & Plane**), you can change the direction of the reference line. (The Wall direction is visible when the **Wall & Beam Reference Line** command toggle is **On**.)

Note: The directions of curved Walls and polygon Walls are fixed and cannot be reversed.

See also [Wall Reference Line](#).

Columns

[Place a Column](#)

[Create a Slanted Column](#)

[Create a Multi-Segment Column](#)

[Define Column/Segment Structure](#)

[Rotate Join Angle](#)

[Tapered Column or Segment](#)

[Editing Columns](#)

[Edit Mode for Multi-Segment Columns](#)

[Core and Veneer](#)

[Columns and Other Elements](#)

[Column Tool Settings: Preview and Positioning](#)

[Column End Cuts and Reference Axis](#)

[Column Model Display](#)

[Column Display on Floor Plan and Section](#)

Related Topics:

Columns can be included in a Structural Analytical Model generated in Archicad.

For details, see [Structural Analytical Model](#).

For information on the Structural Analytical Parameters Panel of Column Tool Settings:

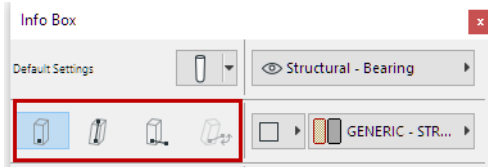
See [1D Members \(Column and Beam\)](#).

Place a Column

Before placing a Column, go to Column Settings to adjust its core geometry, height, positioning and end cuts.

See [Column Tool Settings: Preview and Positioning](#)

To place a Column, choose one of the four Geometry Methods (input methods) from the Info Box.



Simple Column



Place the Column (straight or slanted) with one click. The Column axis will be projected from the Floor Plan.

For a slanted Column: see [Create a Slanted Column](#).

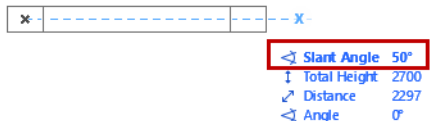
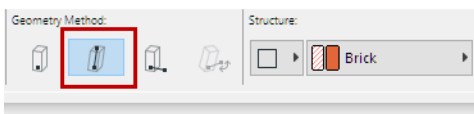
Two-Click Column Input

Use this to freely define the column slant during input.

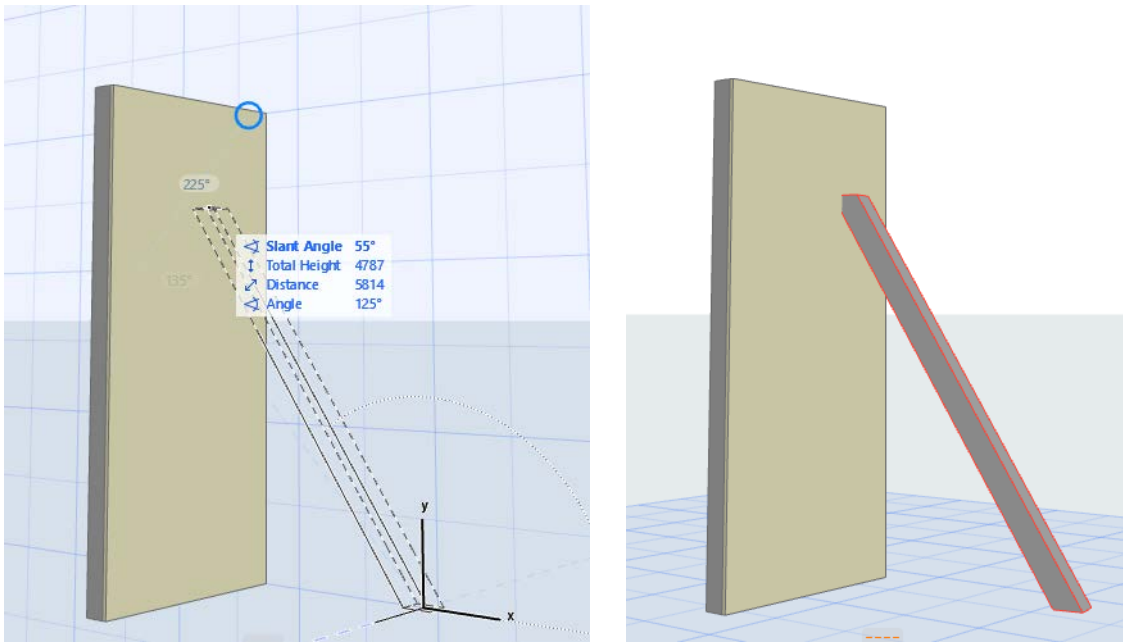
No need to define a slant angle before input.



1. Click to place the Column base.
2. Click again to complete the Column.
 - The second click defines the Column's slant angle and (in the 3D window) its height.
 - In Floor Plan, the height is fixed, and you defined the slant angle graphically (as shown in the Tracker.)



- In the 3D window, you can snap to existing elements, and/or click Shift to constrain input to an editing plane.



Rotated Column



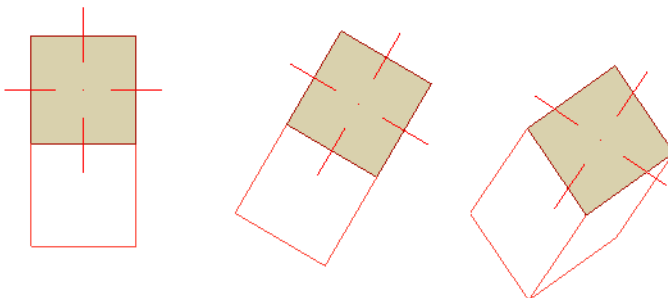
1. Click to place the Column base.
2. Click again to define its rotation angle.

Slanted and Rotated

Slanted column only.



1. Click to place the Column base.
2. Click again to define its rotation angle.
3. Revolve it around its slanted axis, and click to complete.



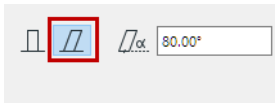
Create a Slanted Column

Use Column Settings, or use the two-point method to define Column slant graphically in the 3D window.

Set Slant Angle in Column Settings

Define Slant Angle in Settings

1. From Column Settings (Column page, Placement panel) or the Info Box, choose the Slanted icon.
2. Enter a slant angle in degrees.



3. Choose a Placement Method from the Info Box, then place the column.

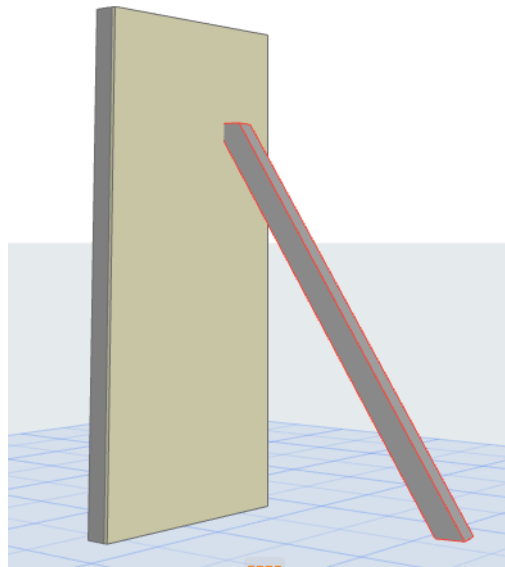
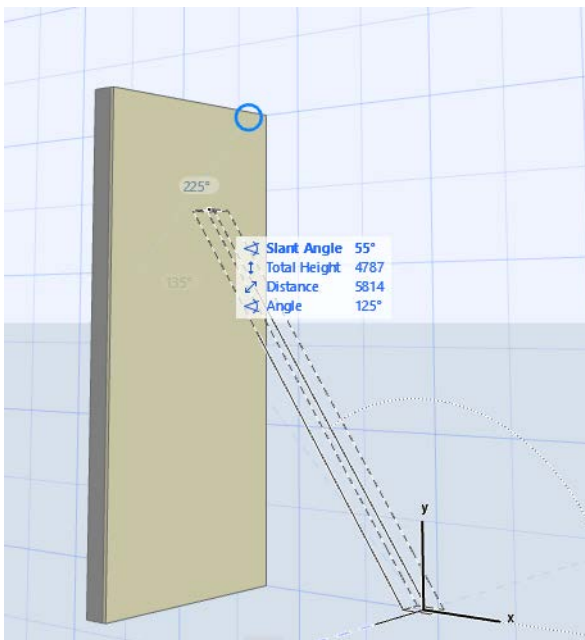
Two-Click Column Input

Use this to freely define the column slant during input. This is easiest in the 3D window.

Do not define a slant angle before input.



1. Click to place the Column base.
2. Click again to complete the Column.
 - In the 3D window, you can snap to existing elements, and/or click Shift to constrain input to an editing plane.



- The second click defines the Column's slant angle and (in the 3D window) its height.
- When creating a two-click Column on the Floor Plan, the Column height is fixed.

Related Topics:

[Place a Column](#)

Create a Multi-Segment Column

Set up a Column that consists of multiple segments, with a configurable “Join” component between them. To create Column segments, use either the Settings dialog or graphical interactions. Both methods are described below.

[Watch the Video](#)

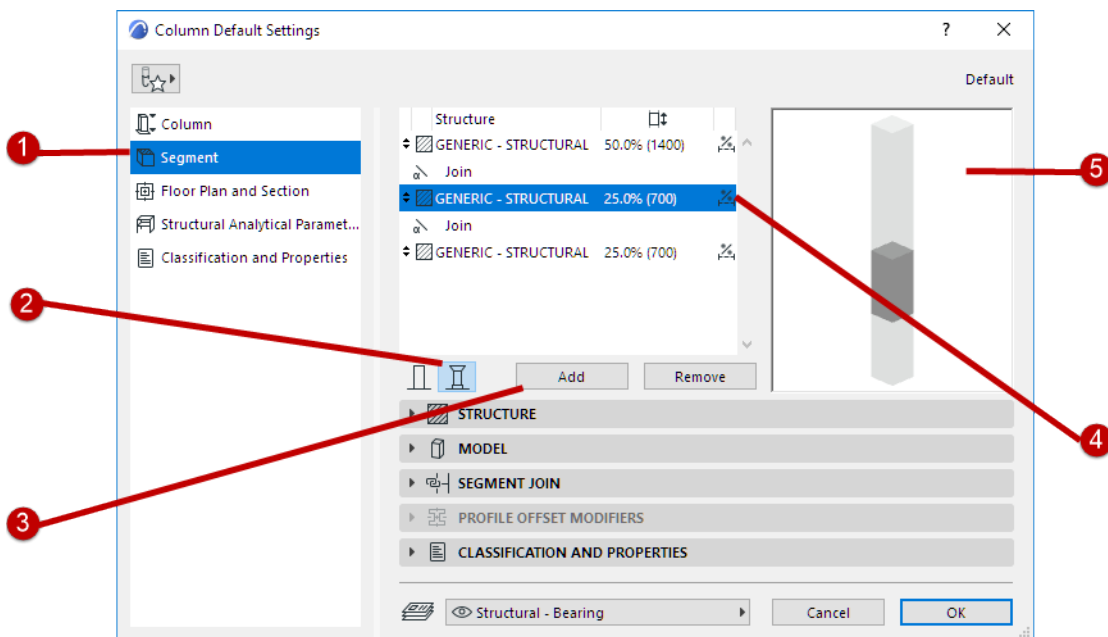
Notes:

- Each segment has its own Classifications and Properties settings and its own Element ID. However, Category definitions (e.g. Renovation status) are inherited from the element level.
- Find & Select, Labels, Info Box, and Interactive Element Schedule fields can use segment-specific data

[Watch the Video](#)

Use Column Settings

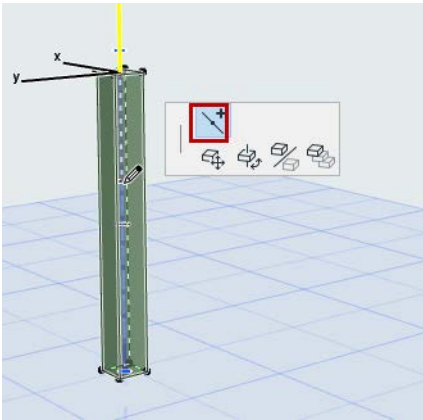
1. Open Column Settings and go to the Segment page.
2. Click the Multi-Segment column icon.
3. Click **Add** to add as many segments as needed.
 - A “Join” component is automatically added between each segment.
4. View the Structure list to see segments and Joins. Select or multi-select to edit.
5. View the preview. The segment currently selected in the list is highlighted.



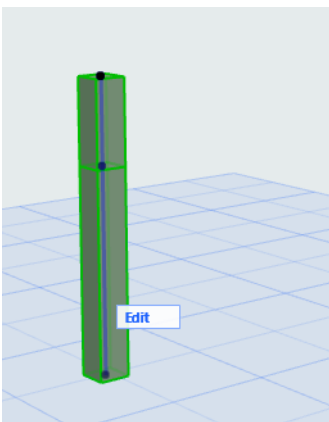
Multi-Segment Column

Create Graphically

1. Select a Column in the 3D window.
2. From the pet palette of the Column axis, choose **Split Segment**.



3. Click anywhere on the Column axis: the Column is split at that point into segments. Repeat to create more segments as needed.



- To edit the segments individually and graphically, you can use Edit mode.
[See *Edit Mode for Multi-Segment Columns*](#).
- To define attributes for each segment, use the Settings dialog.
[See *Define Column/Segment Structure*](#).

Related Topics:

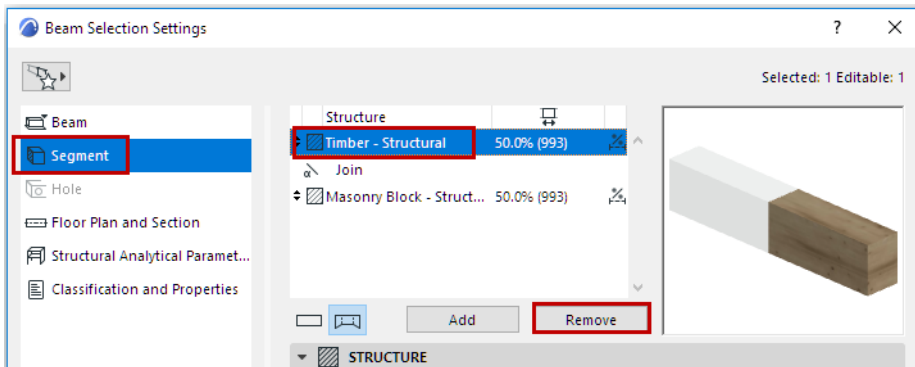
[Delete Column Segment or Beam Segment](#)

Delete Column Segment or Beam Segment

For a multi-segment Column or Beam, delete a particular segment:

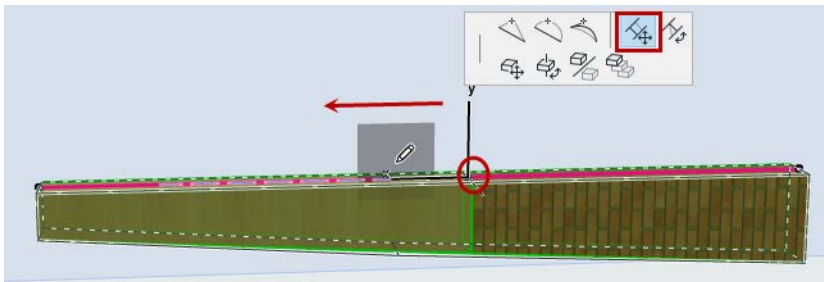
Use Element Settings

1. Select the element.
2. Open Column or Beam Settings - Segment page.
3. Select a segment and click **Remove**.



Remove Segment Graphically

1. Select the element.
2. Click the Join hotspot (on the reference axis or reference line).
3. From the pet palette, use **Move Segment Join**.
4. Drag the join until you reach the end of the segment.

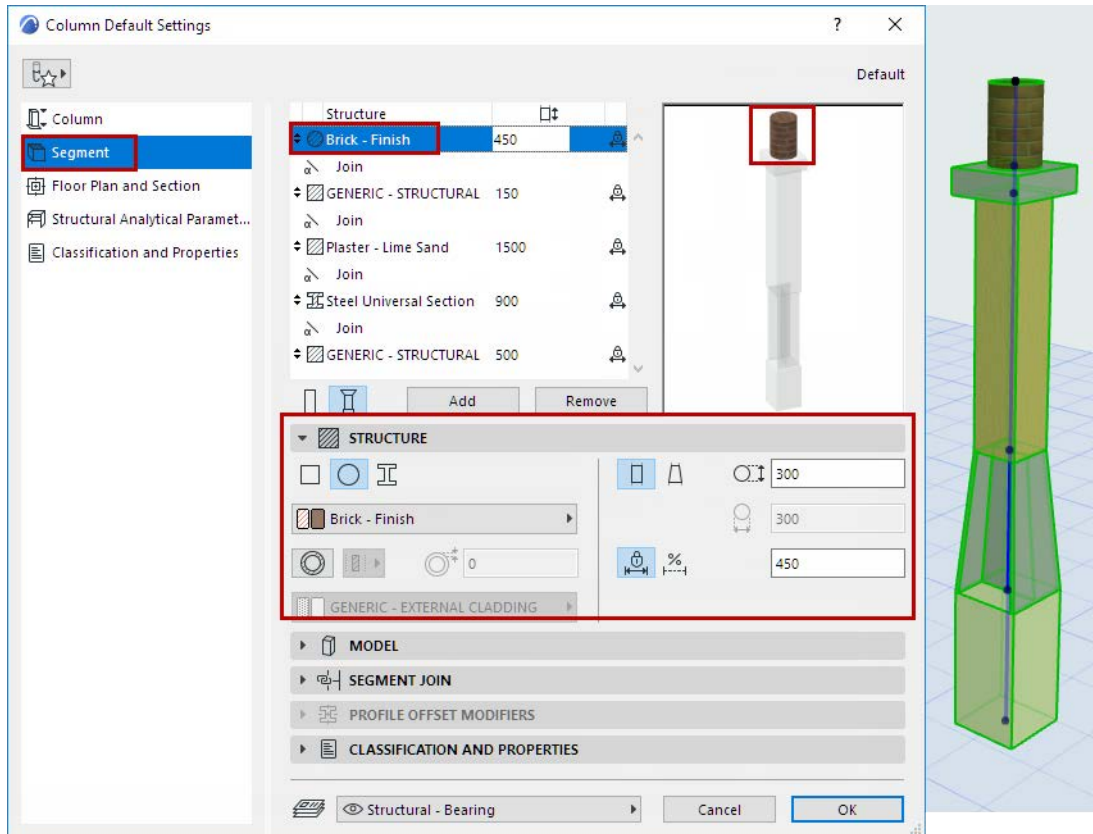


Define Column/Segment Structure

Select a Column or Column segment.

Use the Structure panel of Column Settings (Segment page) to define its structure properties and length (fixed or flexible).

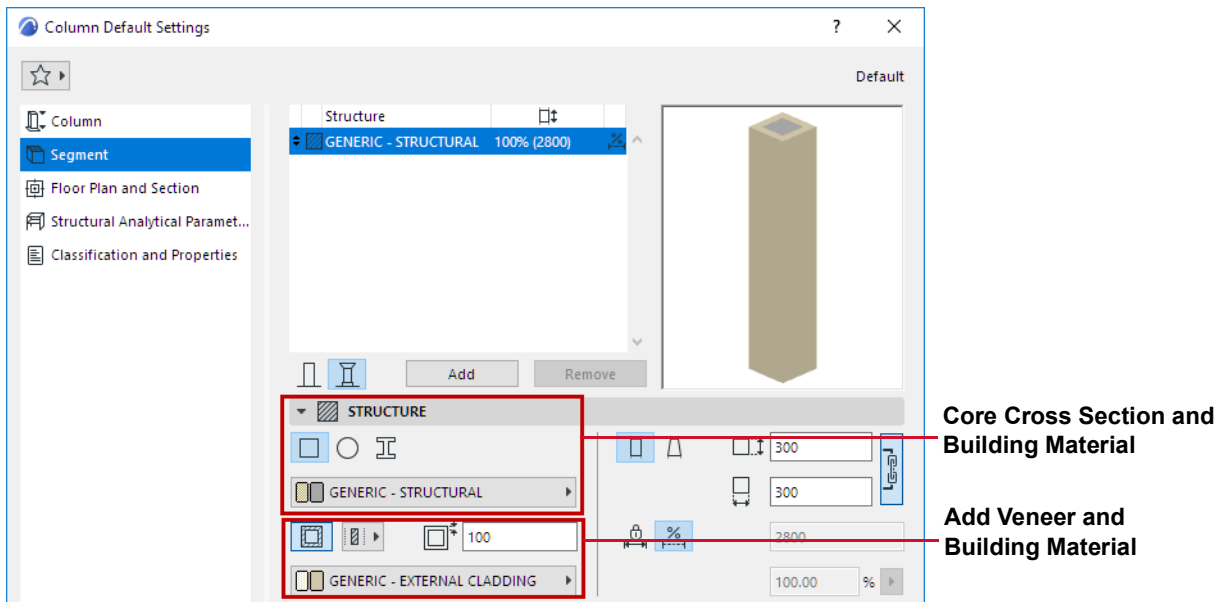
For a selected segment: these Settings are also available in Column Edit mode.



Cross Section

Click an icon to define segment cross-section.

- Rectangular
- Circular
- Complex Profile



Core Cross Section and Building Material

Add Veneer and Building Material

Building Material

For a **Rectangular** or **Circular** Column:

- Use the pop-up to choose its Building Material.
- Add an optional veneer. ([See also Core and Veneer.](#))

For a **Complex Profile** column:

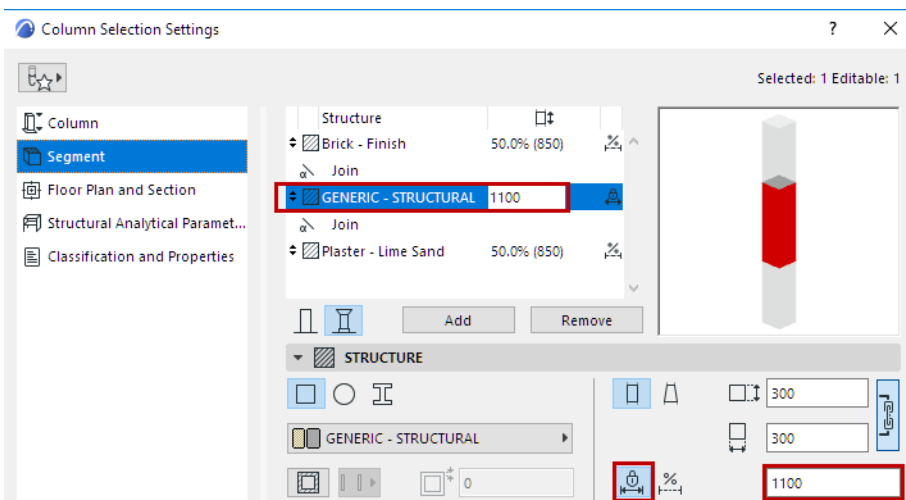
- Use the pop-up to choose its Profile.

Note: This pop-up shows only those Profiles whose “Use With” control includes Columns. Profiles are defined at Options > Element Attributes > Profile Manager.

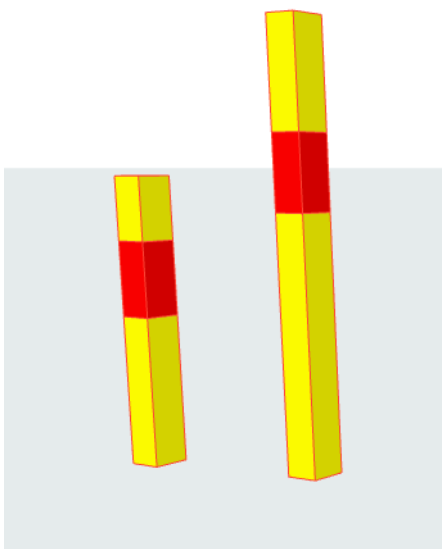
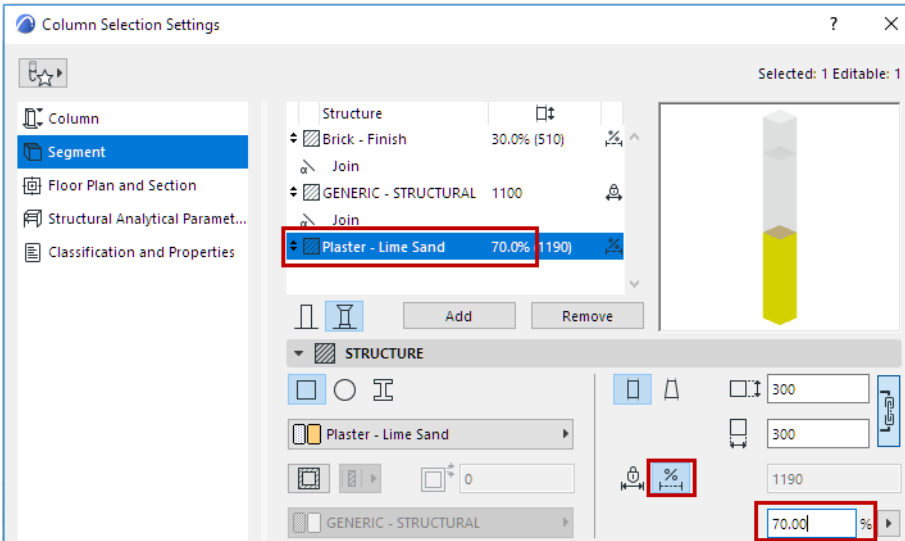
Segment Length (Fixed or Flexible)

For each selected segment: Define its length as either **fixed** or **flexible**, by clicking the corresponding icon in the Structure panel.

- For fixed-length segment: Define length. This length will remain unchanged even if the Column stretches. (You can also enter a selected segment’s length in the Structure list.)



- For a flexible segment: Define its length as a percentage of all the flexible segments. (The flexible segments together add up to 100%.) Segment length will stretch accordingly if the Column stretches.



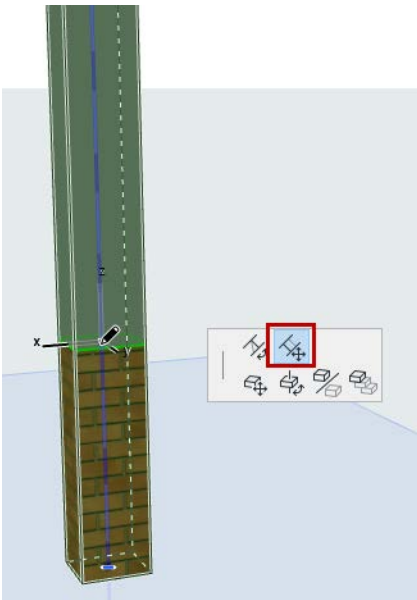
Flexible (yellow) segments stretch with the column.

Fixed (red) segment does not stretch.

Segment Length - Edit in 3D

1. Select the Column in the 3D window.
2. Click the Join hotspot (on the Column reference axis).
3. From the pet palette, use **Move Segment Join** to adjust segment length.

Note: If you drag the hotspot onto adjoining hotspot segment, this eliminates the segment entirely.

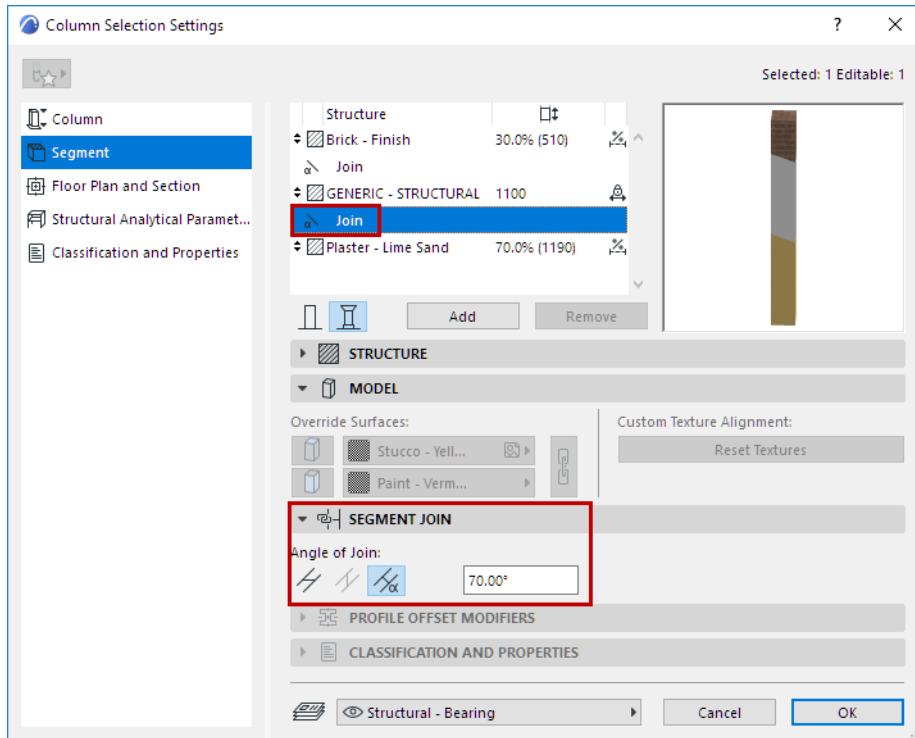
**Related Topics:**[Tapered Column or Segment](#)[Delete Column Segment or Beam Segment](#)[Rotate Join Angle](#)[Core and Veneer](#)

Rotate Join Angle

In Column Settings:

For a Join component selected in the list (at the top of Column Segment Settings): use the **Segment Join** panel (below) to define its angle as horizontal, vertical or custom.

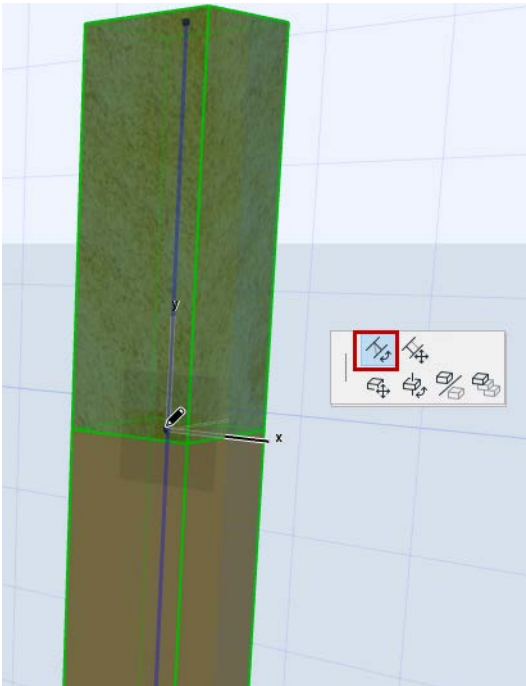
Note: Vertical Join available for slanted Column only.



In 3D Window:

Joins can be edited graphically:

1. Select the Column in the 3D window.
2. Click the Join hotspot (on the Column reference line).



3. From the tool palette, use **Rotate Join Angle** to adjust the Join angle

Related Topic:

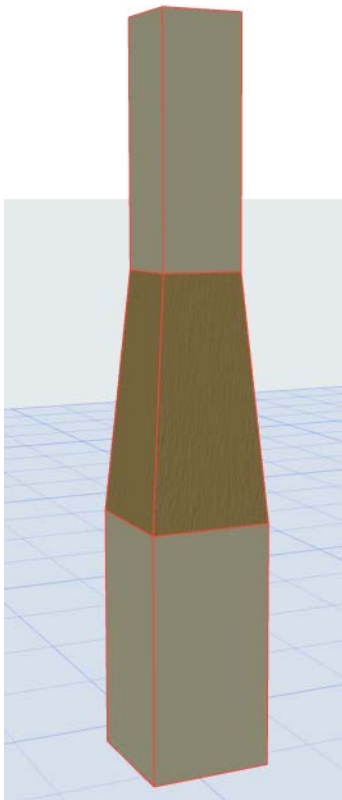
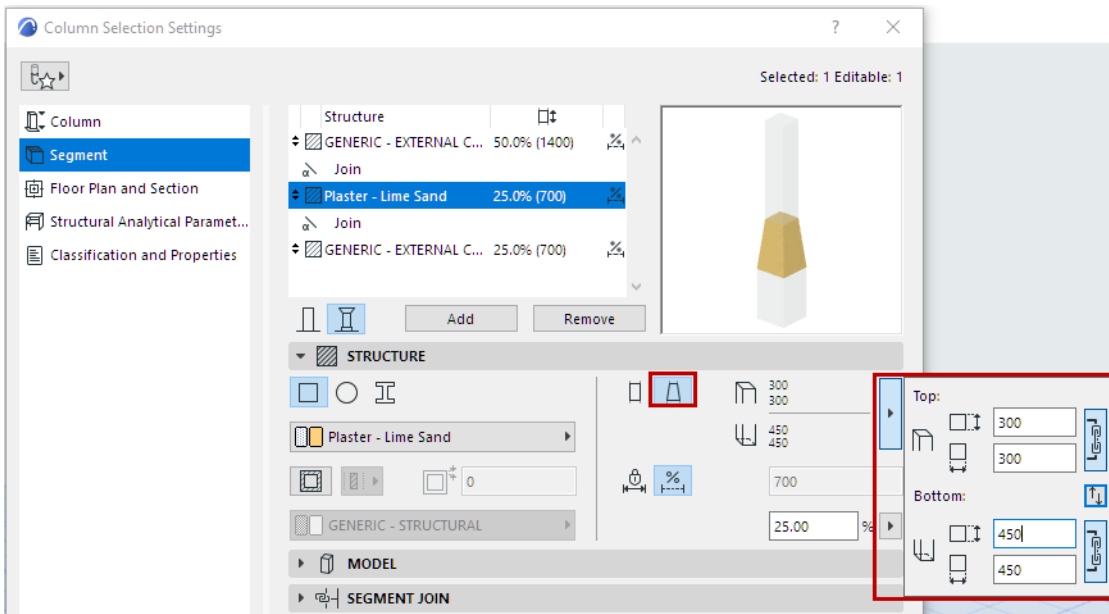
[Define Column/Segment Structure](#)

Tapered Column or Segment

You can create a tapered Column, or taper any single Column segment.

Use the Segment page of Column Settings. (For a multi-segment Column, select the segment to taper.)

1. In the Structure panel, choose Tapered geometry.
2. Use the pop-up to define the cross-section dimensions of the tapered Column (segment) at its top and at its bottom.
 - Use the chain icon to make the height/width dimensions equal
 - Use the arrow button to reverse the tapering direction



Related Topic:

[Define Column/Segment Structure](#)

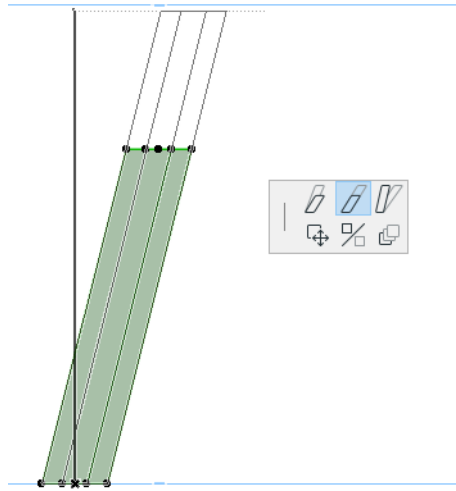
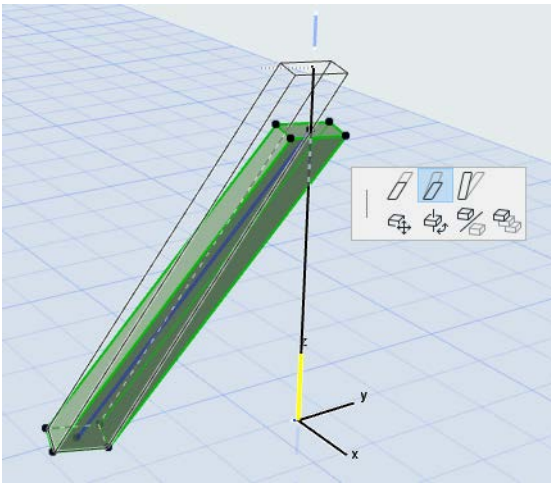
Editing Columns

Stretch Height/Modify Angle

In the 3D or Section window and/or Column Edit mode:

Use the Pet Palette commands at the Column's Reference Axis endpoint. Available commands depend on Column geometry.

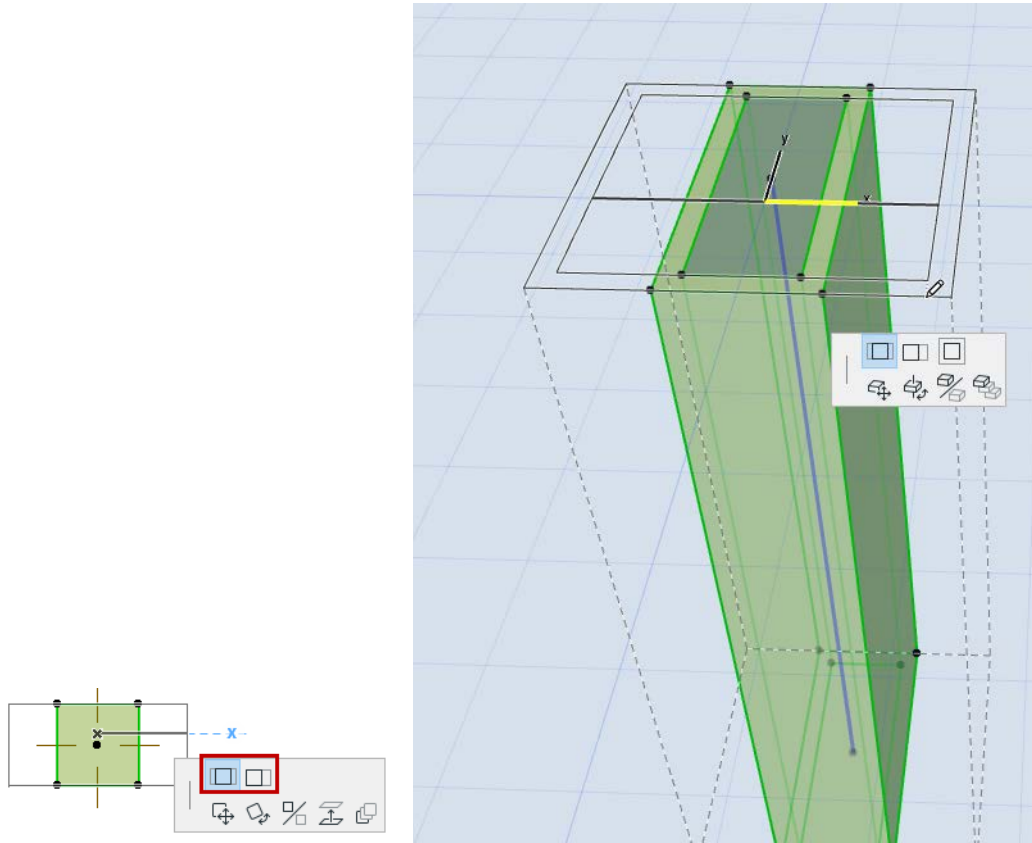
- **Stretch height**
- **Stretch height with fixed angle**
- **Modify angle**



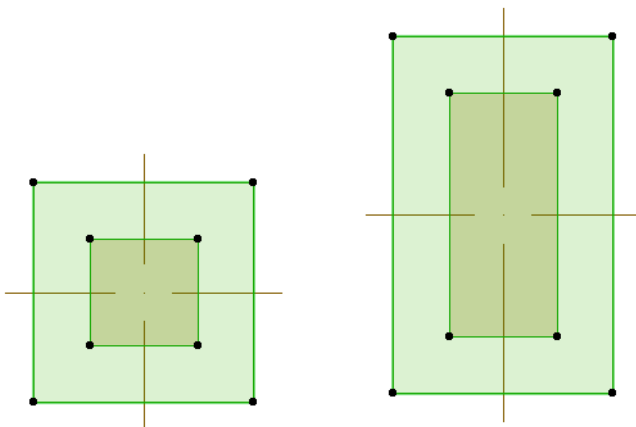
Stretch Edge(s) of Column Profile

In the Floor Plan or 3D window, use the Pet Palette commands to stretch a Column by its edges - either the core, or the veneer (if any).

Depending on the position of the Reference axis, you can stretch the profile edge in both directions:



Stretching the core will not affect the veneer thickness.



Related Topics:

[Edit Mode for Multi-Segment Columns](#)

[Watch the Video](#)

Edit Mode for Multi-Segment Columns

For a multi-segment Column, use Edit mode to edit each segment separately. Edit mode is ideal for fine-tuning segment geometry graphically.

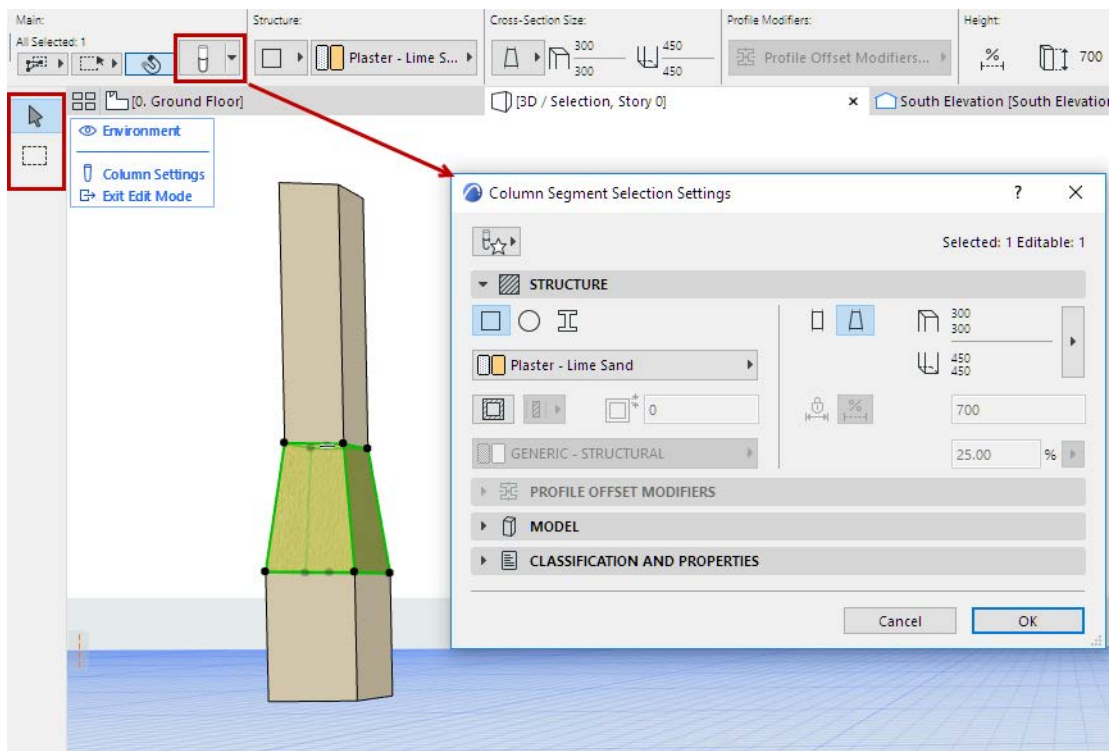
Enter Edit Mode

Select a multi-segment Column in any model view, then do one of the following:

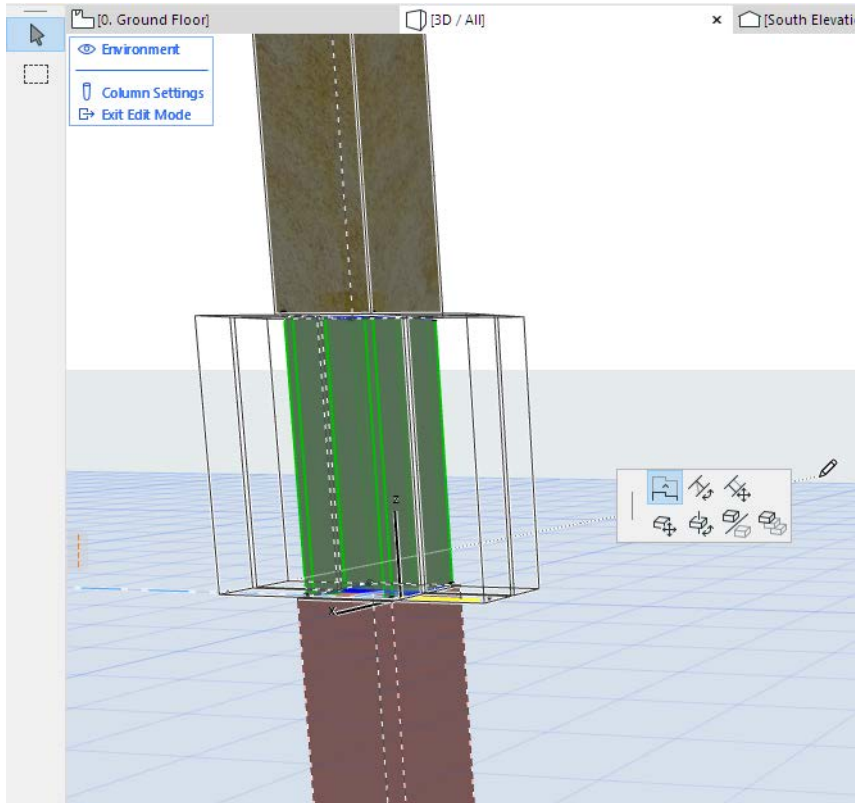
- Click the **Edit** button which appears.
- Use **Design > Enter Column Edit Mode**.

Using Edit Mode

- In Edit mode, the regular Archicad toolbox is replaced by just selection tools and a display palette.
- Use Environment toggle in the display palette to show/hide the rest of the building model (i.e. everything apart from the selected Column). In Edit mode, you cannot edit the rest of the model.
- From the display palette, the full Column Settings dialog is available.
- From the Info Box, you can open a limited, segment-specific Settings dialog.



For example, edit the Profile Modifiers of a profiled Column segment:



Exit Edit Mode

Do one of the following:

- Click **Esc**
- Click **Exit Edit Mode** from the display palette.
- Use **Design > Exit Column Edit Mode**

Click Column Settings from the display palette to access the settings dialog for the selected segment.

See [Define Column/Segment Structure](#).

Core and Veneer

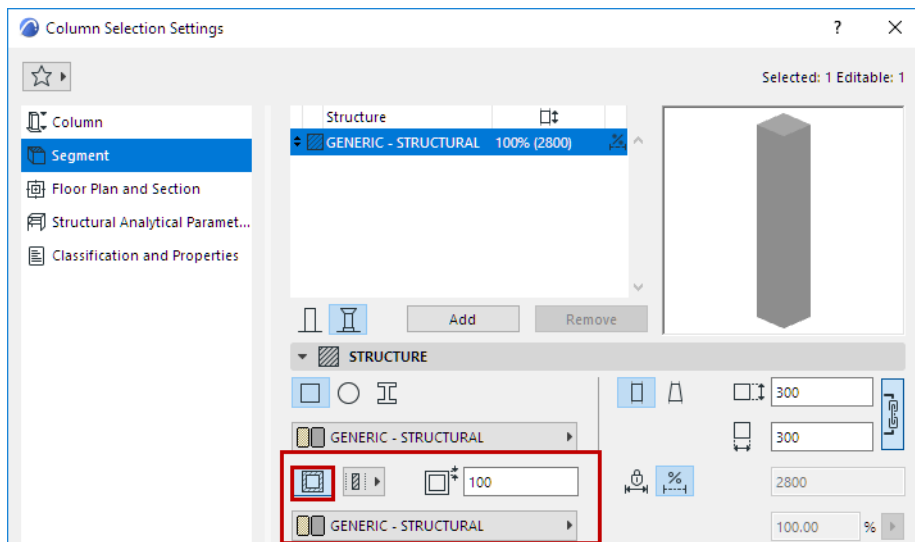
Columns in Archicad are made up of two components: the load-bearing core and the optional veneer, used to simulate fire proofing or any kind of sheathing around the core. The veneer has its own building material, and thus its own intersection priority.

Set a Veneer for any Column or Column segment.

Note: Veneer not available for Profile Columns (segments).

Add/Remove Veneer

1. Go to the Structure Panel of Column Settings (Segment page).
2. Use the **Add/Remove Veneer** toggle button.

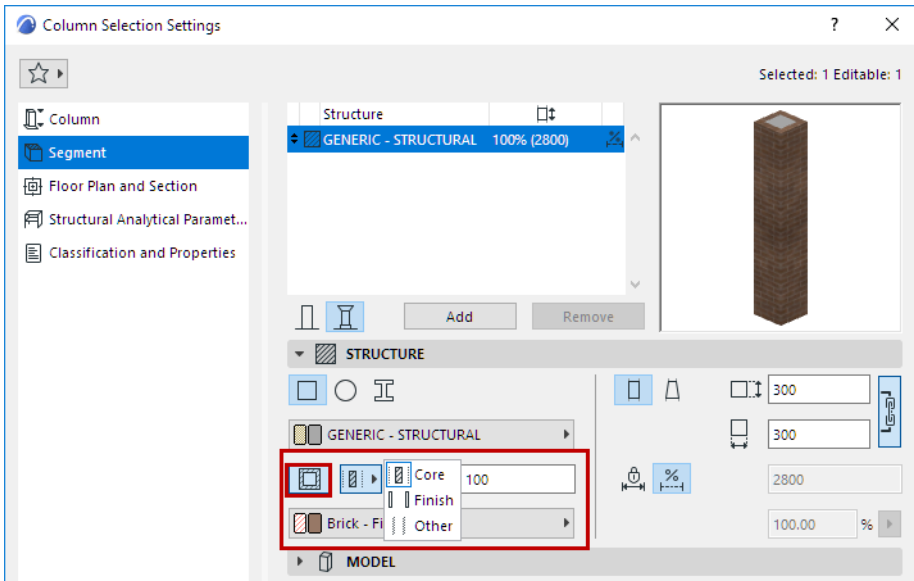


3. Define a uniform veneer thickness in the field to the right.
4. Use the pop-up to choose a Building Material for the veneer.

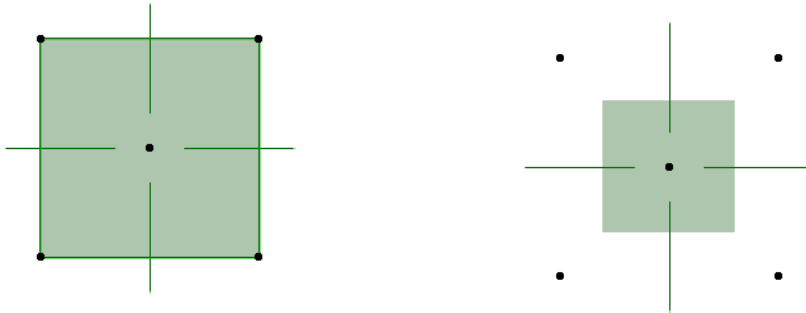
Veneer Structure: Core, Finish, Other

The veneer component can be defined as either “core”, “finish” or “other”.

Note: The Finish option is not available for wrapped columns. ([See Columns and Composite Walls: Wrapped or Unwrapped.](#))



Partial Structure Display settings takes the Veneer structure into account, in both 2D and 3D views.



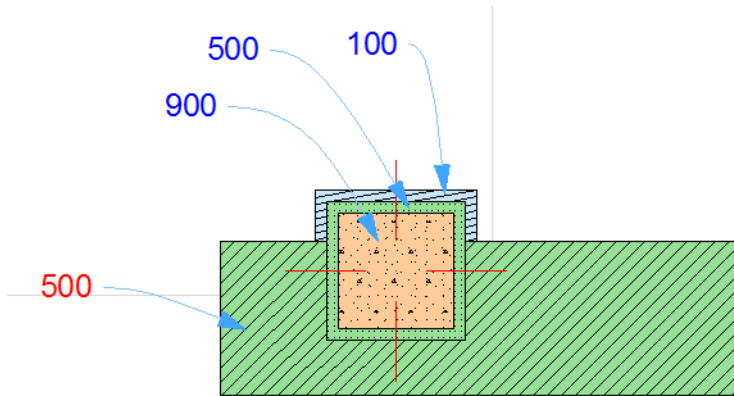
Column with a veneer, with varying Partial Structure Display settings. This veneer is defined as a “Finish”
[See Partial Structure Display.](#)

Columns and Other Elements

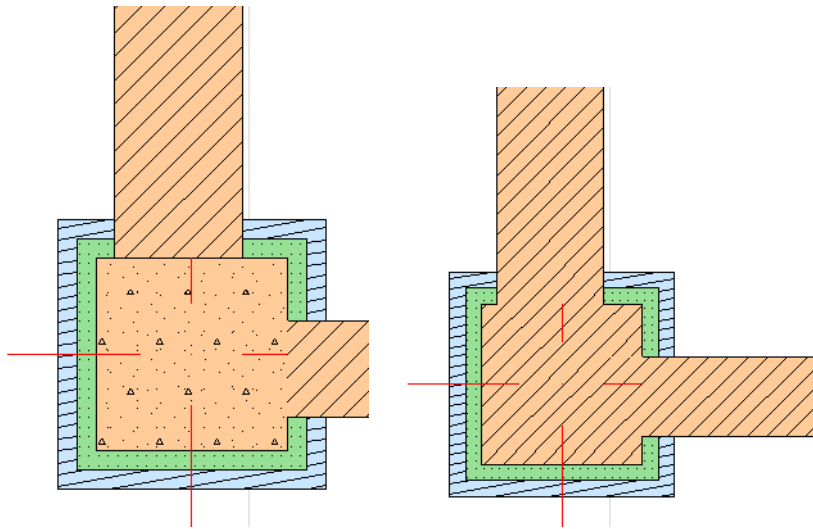
Columns and Walls

The display of their intersections depends on the Intersection priority of the Building Materials: the higher number will prevail.

Note: Each Building Material has a separate Intersection Priority, set in the Building Materials dialog box.



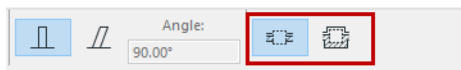
- If two identical Building Materials meet, the separator line will be hidden.



See also [Element Intersections](#).

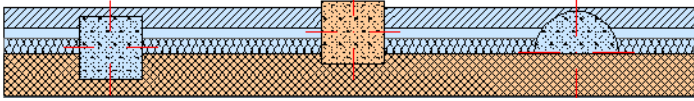
Columns and Composite Walls: Wrapped or Unwrapped

Use the wrapped/unwrapped icons from the Column Info Box or the Column Settings dialog box to determine how the columns interact with composite walls.



Unwrapped

Unwrapped (freestanding) Columns do not modify the shape of the Wall - they cut out a place for themselves, breaking through all Wall skins.

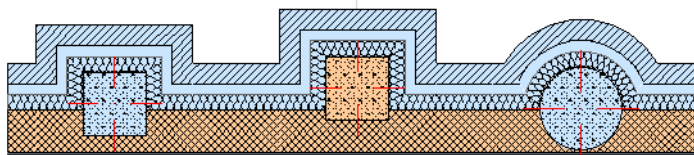


The separator line between Wall skins and the Column are removed where their Building Materials match.

Wrapped

Wrapped Columns break through the core skins of straight, vertical, composite Walls.

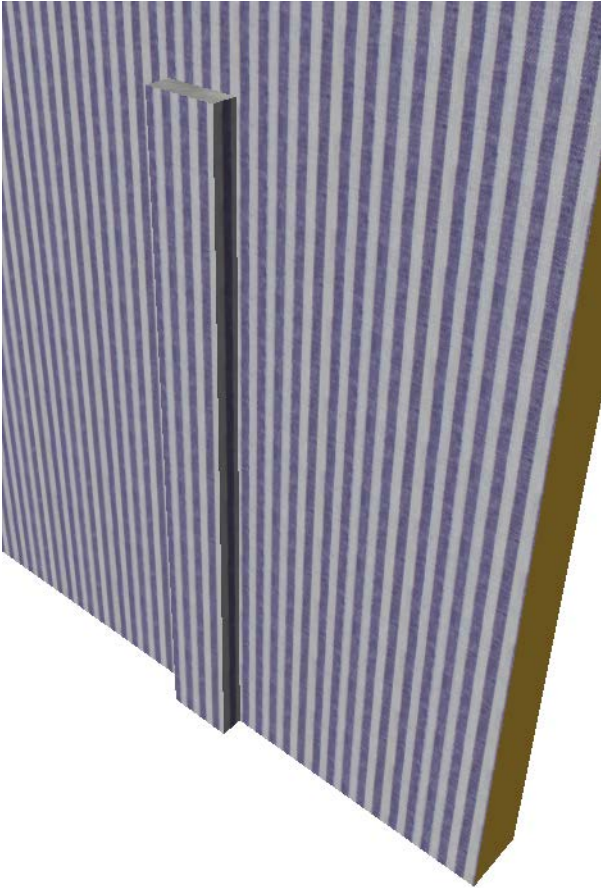
The other (non-core) Wall skins wrap around the Column, thus adding this thickness to the column.



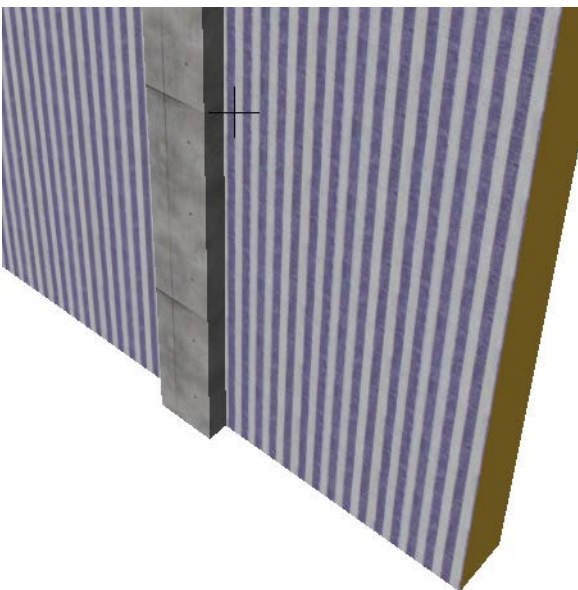
- The composite Wall must have at least one skin that is core and one skin that is non-core (which will wrap around)
 - The Column must intersect the core skin(s) of the composite Wall
- Note:** For composite Walls, the core skins are defined in **Options > Element Attributes > Composites**.
- Wrapping is *not* available for:
 - Columns that are slanted, profiled, tapered, or multi-segmented
 - Columns whose veneer is defined as “Finish” ([see Veneer Structure: Core, Finish, Other](#))
 - Columns that intersect with curved, slanted, or polygonal Walls
 - Columns with an End that is angled

3D Display of Wrapped Columns

In 3D view, Columns placed with the Wrapped Construction Method will inherit the Surface of the connecting Walls, even if the wall is not a composite wall, and even if there are no wall skins configured to actually wrap around the Columns.



Unwrapped columns retain their own surfaces in 3D.



Related Topics:

Columns can be trimmed to Roofs or Shells.

[See *Trim Elements to Roof or Shell*.](#)

You can crop a Column to a Single-plane Roof.

[See *Crop Elements to Single-plane Roof*.](#)

Use **Adjust Elements to Slabs** to adjust Columns to specific levels of one or more Slabs that are located above and/or below them.

[See *Adjust Elements to Slabs*.](#)

For Columns inside a Zone:

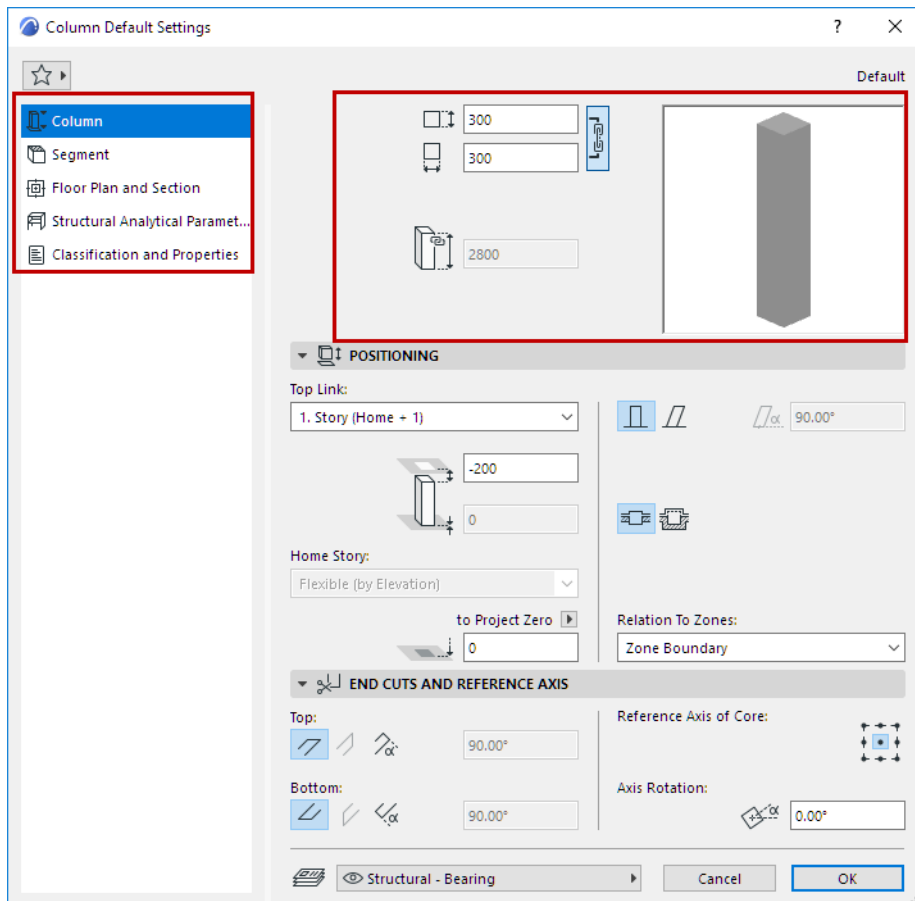
[See *Relation to Zones*.](#)

Column Tool Settings: Preview and Positioning

The Column Settings dialog box is divided into five pages.

The Preview section remains visible on all pages, and reflects your settings modifications.

Navigate among the pages using the tree structure on the left:



Core Width/Height

For a simple (single-segment) Column, enter the core's horizontal and vertical dimensions.

Note: For Columns with a rectangular cross-section, use the **chain** icon to constrain them to the same value.

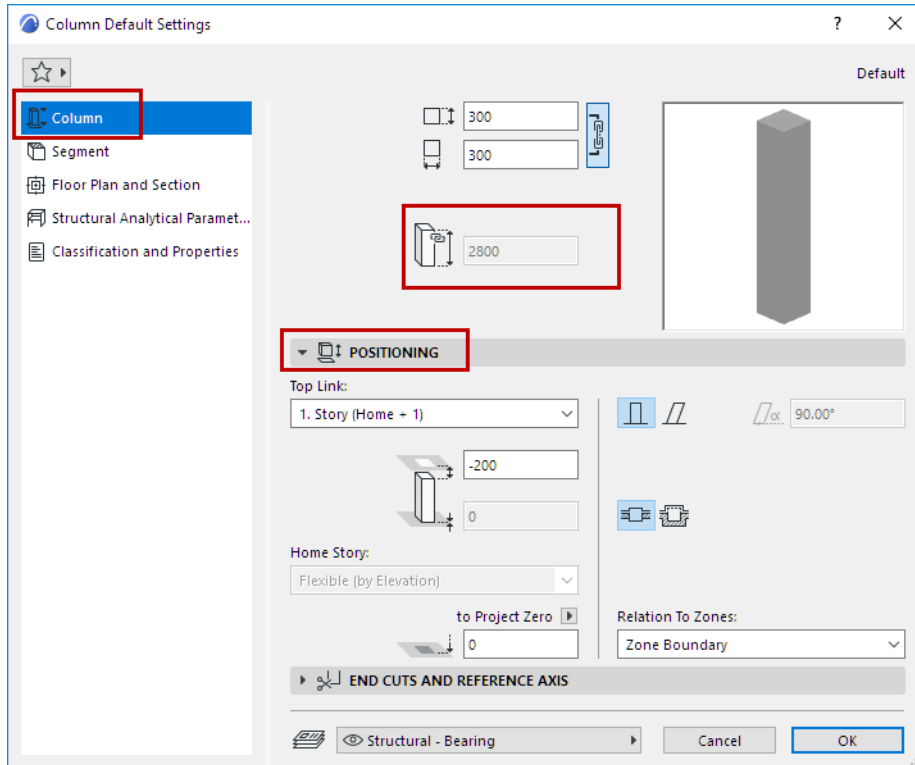
For a circular column, enter a diameter.

Column height

Shows the Column's total height.

- For an unlinked Column, set its height here.
- For a top linked Column: this field is not editable. Column height depends on the settings in the Positioning panel below.

Column Positioning Panel



By default:

- The Column top is linked to one story up
- The Home Story is the current story.

Top Link

Use this control to do one of the following:

- Top link the Column relative to its Home Story (Home+1, Home+2, etc.)

If you later modify the positions and heights of stories in your projects, the heights of any linked Columns will follow automatically.

- Choose **Not linked** to make it a fixed-height Column. Define its height in Column height field above.

Offset from Top Linked Story

You can **offset** the Column top from its top linked story.

- The offset value can be positive, negative or zero.
- The offset field is not available if the Column has no top link.

You can also **offset** the Column bottom from the Home Story, in either direction.

- The Column height value (seen in the Preview section of Column settings) is affected by the top and bottom offsets, if any.

Home Story

See [Home Story](#).

Bottom Elevation [to Reference level]

Calculates the current elevation of the Column's reference line, as measured from the Reference level (by default, this Reference level is Project Zero).

Click the pop-up arrow to change the Reference level, if needed.

Note: Reference levels are defined at **Options > Project Preferences > Reference Levels**.

For general information on levels, see [Reference Levels Preferences](#).

Note: This is a calculated value, not a link. If you change the Reference level, the Column will not change its position.

Column Slant Angle

See [Create a Slanted Column](#).

Wrapping Method

See [Columns and Composite Walls: Wrapped or Unwrapped](#).

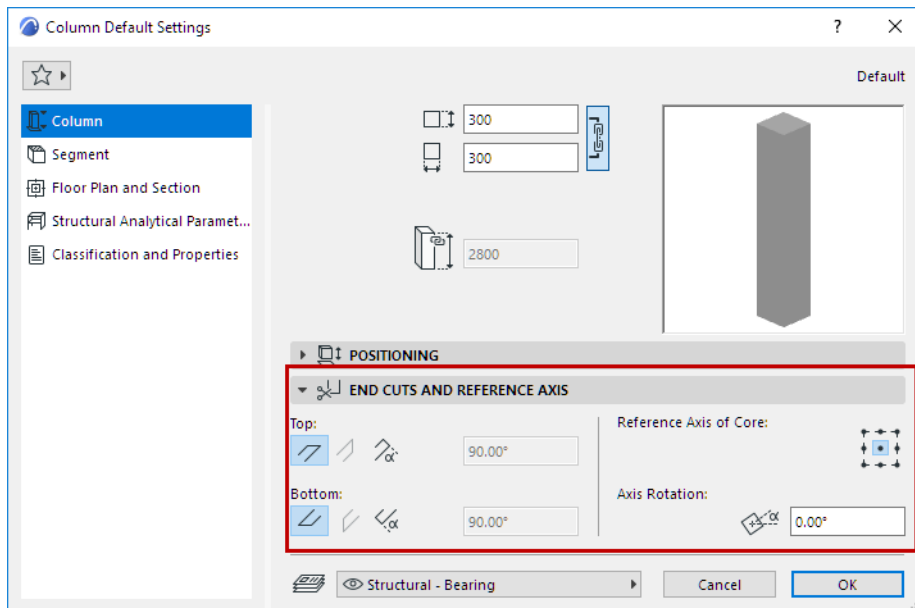
Relation to Zones

See these topics:

[Calculating Zone Area and Zone Volume](#)

[Relation to Zones](#)

Column End Cuts and Reference Axis



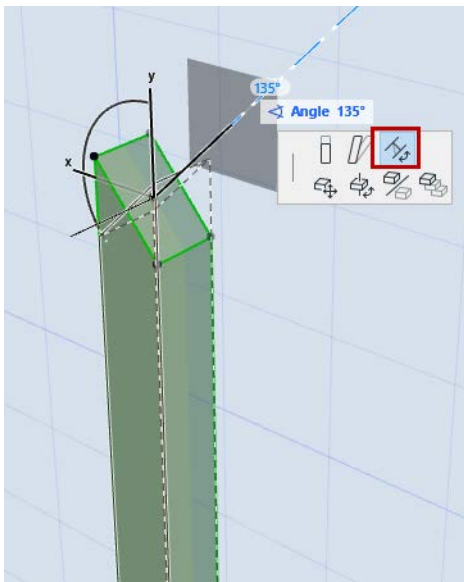
End Cuts

Define the angle of the Column's Top and Bottom surfaces: horizontal, vertical or custom.

Note: Vertical available for slanted Beam only.

You can also edit this geometry graphically:

- Select a Column in 3D. Click on its end surface, then choose **Rotate Join Angle** from the pet palette.



Reference Axis of Column Core

Choose one of the nine points to serve as the Column's Reference Axis.

If you modify the Reference axis: the axis stays in place while the Column moves accordingly.

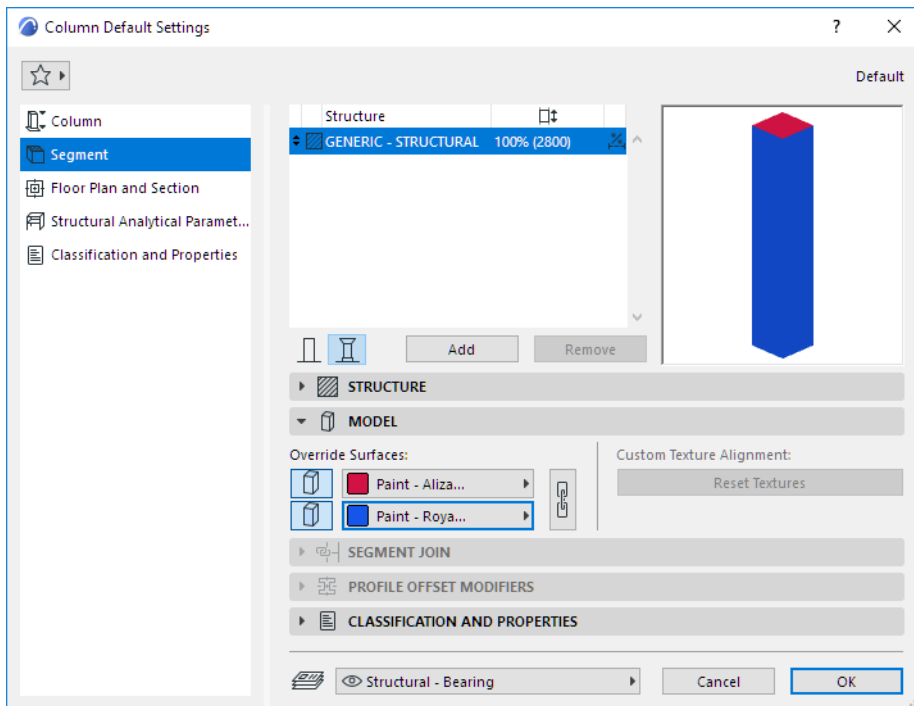
Note: Available only if at least one Column segment has a rectangular or circular cross-section. Not available for complex profiles.

Axis Rotation: Set a rotation angle for the Column around its axis.

See also [Rotating Beam and Columns](#).

Column Model Display

Use the Model Panel of Column Settings (Segment page) to override Column surface(s).



Override Surface: Click to override the surfaces assigned to this column or segment in its Building Material:

- Top and Bottom Surfaces
- Side (Extrusion) Surfaces

Note: For Columns placed with the *Wrapped* method, the choice you make here is overridden by the surfaces of the Walls that they intersect. ([See 3D Display of Wrapped Columns.](#))

Note: For a Profile Column, you can also apply a Custom surface to any edge(s) of the profile element. ([See Apply Custom Surface or Line to Profile Edge.](#))

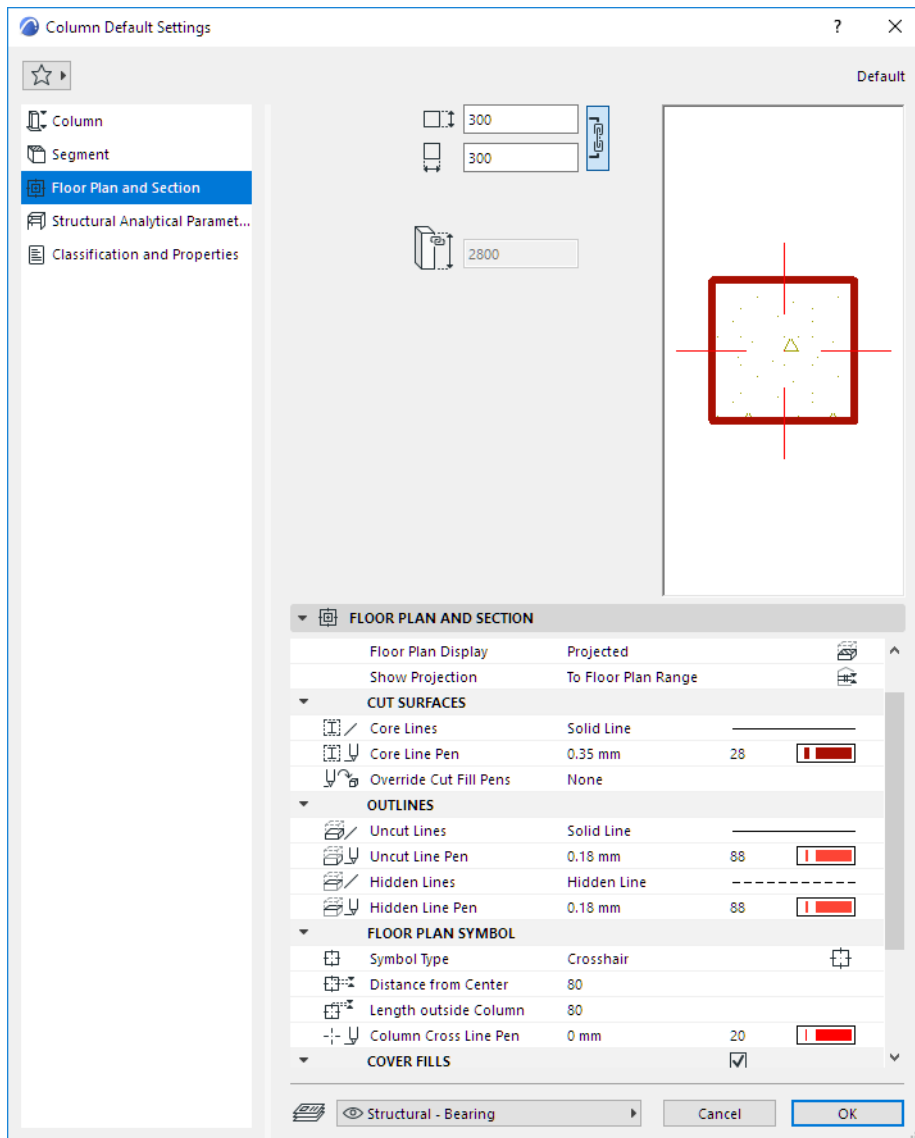
Custom Texture Alignment: If this message is activated, it indicates that the currently selected column (already placed in the project) has been assigned a custom 3D texture.

- In this case, you can click **Reset Texture** to restore the origin of the Texture of the selected column or segment.

For more information, see [Align 3D Texture](#).

Column Display on Floor Plan and Section

Use the Floor Plan and Section page of Column Settings.

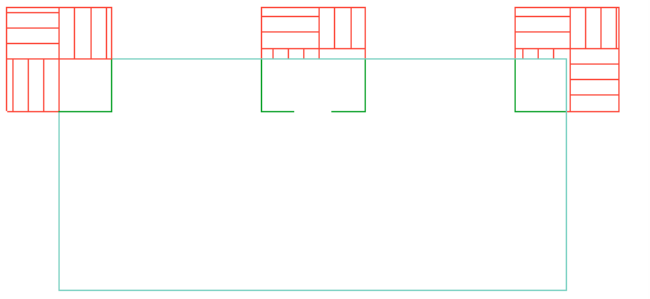


[See Element Display in Floor Plan and Section.](#)

Hidden Lines

Define Hidden Line attributes. Hidden Lines are used for Columns under Slabs if this is turned on at Model View Options.

[See Model View Options for Construction Elements.](#)



Column Underneath Slab: Use Hidden Line

Floor Plan Symbol

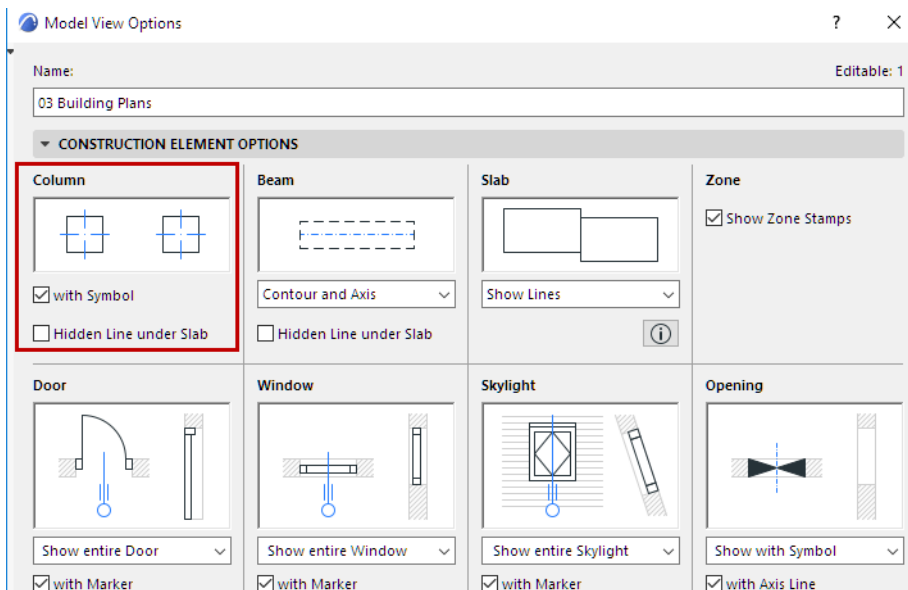
Symbol Type: Plain, Slash, X, or Crosshair

Note: For a profiled column, only the Plain and crosshair symbols are available. The crosshair symbolizes the column’s axis and will revolve along with the column cross section.

Crosshair has two additional settings: **Distance From Center** and **Length Outside Column**

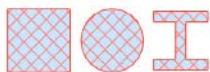
Although the crosshair is a graphic symbol and not part of the model, the crosshair will be resized (together with the column itself) if you change the drawing’s scale.

Set display (on or off) of column floor plan symbols by view at **Document > Model View > Model View Options**.



Cover Fills

See [Assign a Cover Fill](#).



Columns with Cover Fills



Slanted Column with Cover Fill (Orientation: 3D Distortion)

Beams

Archicad Beams are horizontal or slanted construction elements. Basic beams have a rectangular cross-section, but you can also use a Complex Profile.

Beams can be curved, either on a horizontal plane or vertically. Like other construction elements, a Beam's essential structure derives from its Building Material, defined in Beam Tool Settings.

Place a Beam

Create a Slanted Beam

Curve a Beam Vertically

Create a Multi-Segment Beam

Define Column/Segment Structure

Rotate Join Angle

Tapered Column or Segment

Editing Beams

Create a Hole in a Beam

Selecting and Modifying Beam Holes

Beams and Other Elements

Beam Tool Settings: Preview and Positioning

Beam End Cuts

Beam Model Display

Beam Display on Floor Plan and Section

Rotating Beam and Columns

Related Topics:

Beams can be included in a Structural Analytical Model generated in Archicad.

For details, see [Structural Analytical Model](#).

For information on the Structural Analytical Parameters Panel of Beam Tool Settings:

See [1D Members \(Column and Beam\)](#).

Place a Beam

Before placing a Beam, go to Beam Settings to adjust its core geometry, height, positioning and end cuts.

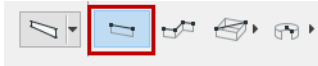
See [Beam Positioning Panel](#)

To place a Beam, choose one of Geometry Methods (input methods) from the Info Box.

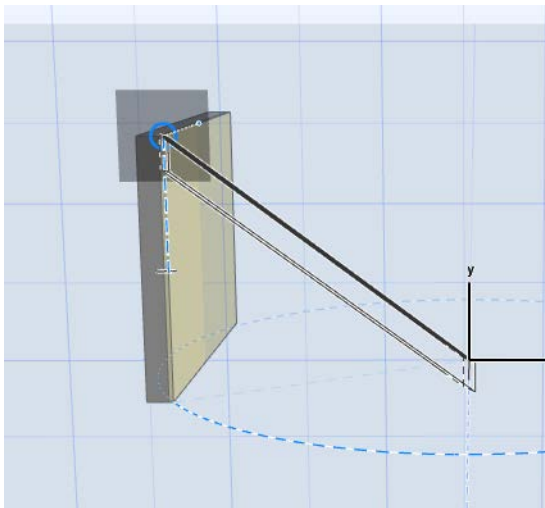
Single Beam

1. Choose the Single Beam method from the **Info Box**.

Note: If you are drawing a slanted beam, only the Single Beam method is available.



2. Click twice to place each end of the Beam.
3. In the 3D window:
 - Snap the endpoints to existing elements
 - Click Shift to constrain input to an editing plane.



Note: If the Beam already has a defined slant angle (in Settings or Info Box), the slant angle prevails, regardless of the constraint.

For a slanted Beam: see [Create a Slanted Beam](#).

Chained Beam

1. Choose the Chained Beam method from the **Info Box**.
2. Click in succession to place segments of a chained Beam.



This input method is analogous to creating a chained Wall.

See [Create a Chain of Walls](#).

In the 3D window:

- Snap the endpoints to existing elements
- Click Shift to constrain input to an editing plane.

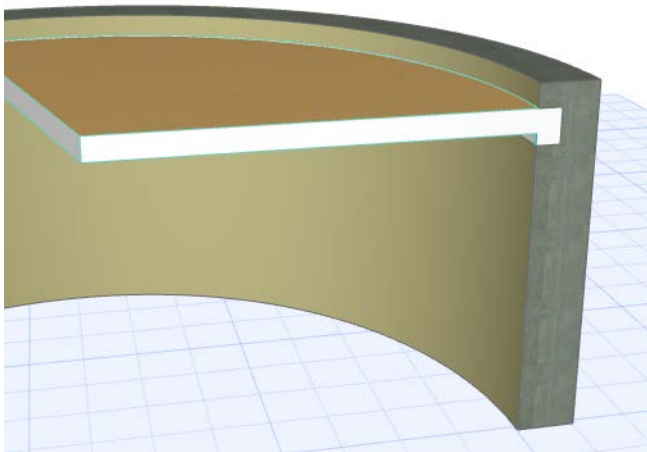
Rectangle/Rotated Rectangle Beam

Choose one of these methods from the Rectangle Beam icon's pop-up:



- For a regular rectangle beam, click twice to define its two opposite corners.
- For a rotated rectangle, click twice to define a side of the rectangle, including the rotation vector, then drag and click again to complete.

Horizontal Curved Beam



To create a horizontally curved Beam, do one of the following:

Curved Beam Input from Info Box

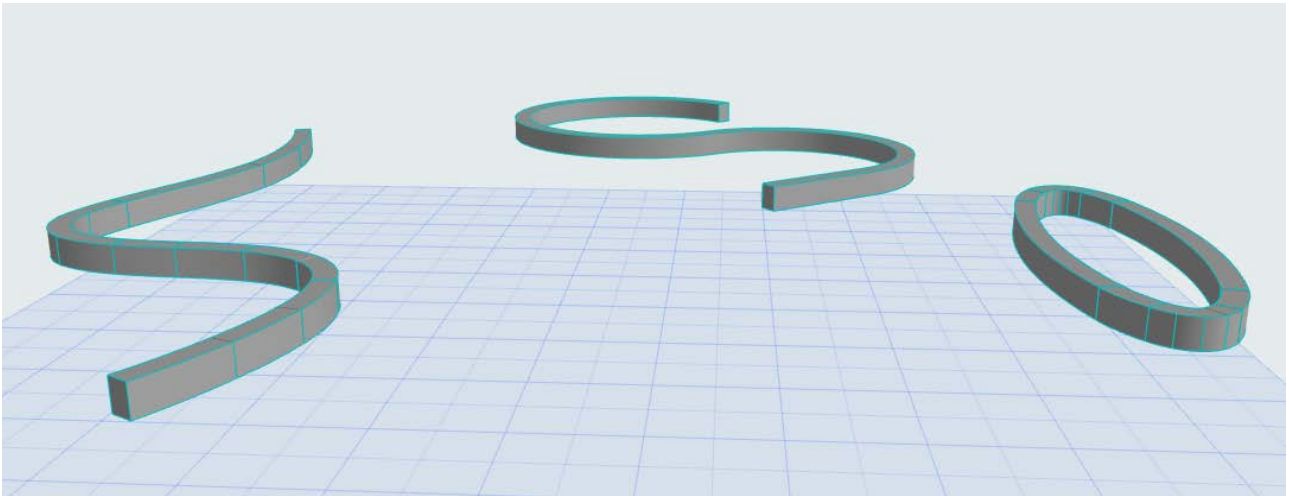


Choose one of the three curved input methods. These methods are analogous to creating a curved Wall.

[See *Create a Curved Wall*.](#)

Other Ways To Create Horizontal Curved Beams

- Create curved Beam segments as part of a Chained Beam.
- Select a straight Beam on the Floor Plan, then use the pet palette commands to curve it, or edit using tangent.
- Magic Wand: Trace the shapes of curved drawing elements with the Magic Wand.



[Watch the Video](#)

[Watch the Video](#)

Related Topic:

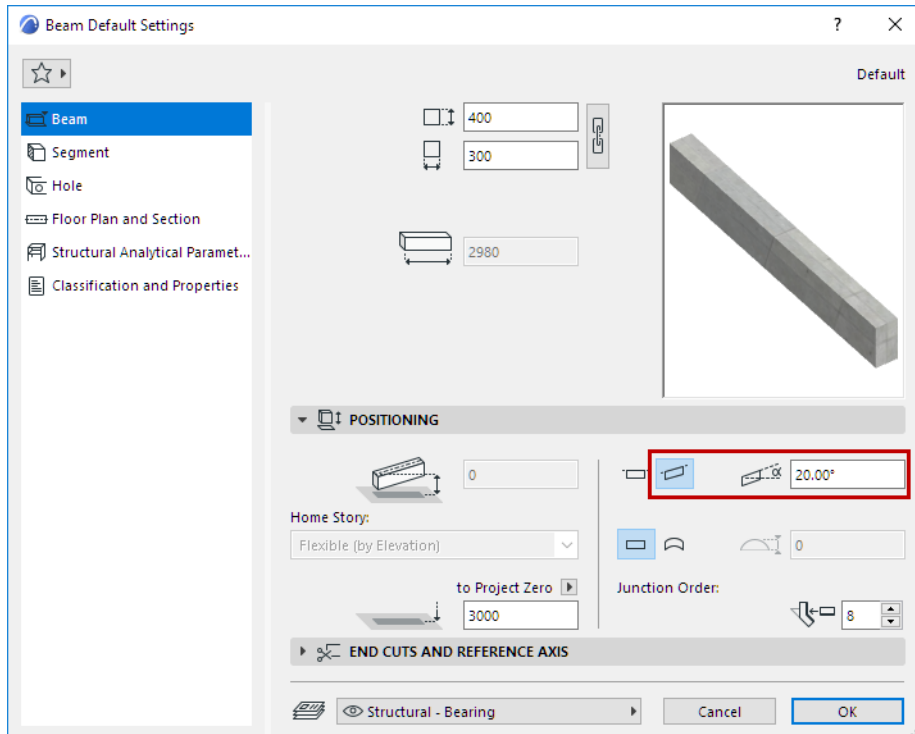
[Curve a Beam Vertically](#)

Create a Slanted Beam

Use Beam Settings, or use Beam slant graphically in the 3D window.

Set Slant Angle in Beam Settings

1. From Beam Settings (Beam page, Positioning panel) or the Info Box, choose the Slanted icon.
2. Enter a slant angle in degrees.

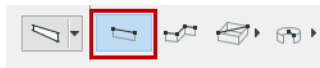


The value must be between -89° and $+89^\circ$, and is measured from the starting point of the Beam.

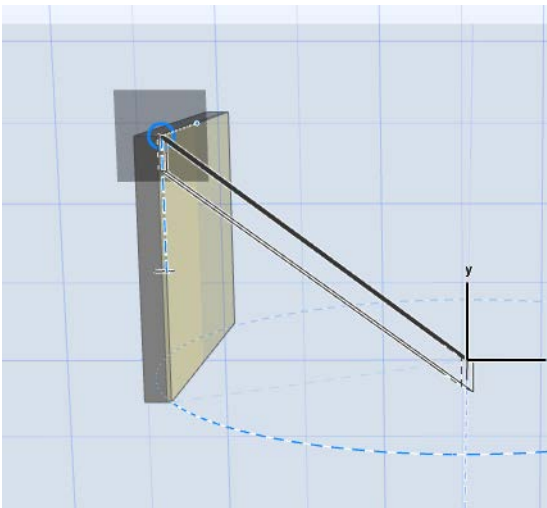
3. Choose a Placement Method from the Info Box, then place the column.

Two-Click Beam Input

1. Choose the Single Beam method from the **Info Box**.

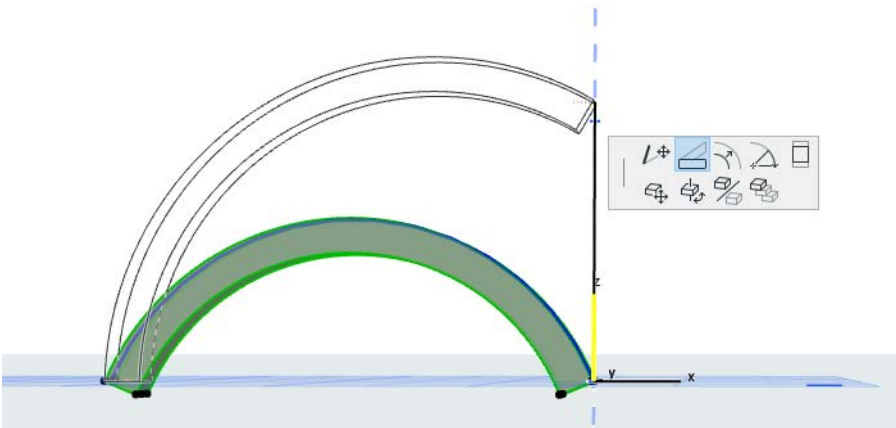


1. Click twice to place each end of the Beam.
2. In the 3D window:
 - Snap the endpoints to existing elements
 - Click Shift to constrain input to an editing plane.



Note: If the Beam already has a defined slant angle (in Settings or Info Box), the slant angle prevails, regardless of the constraint.

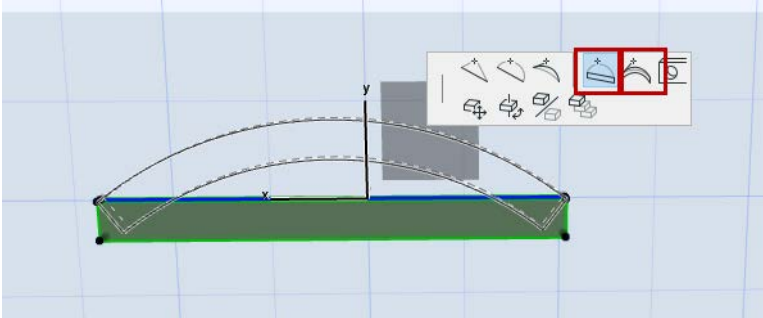
An existing, vertically curved Beam can also be slanted.



Curve a Beam Vertically

In the 3D or Section/Elevation window:

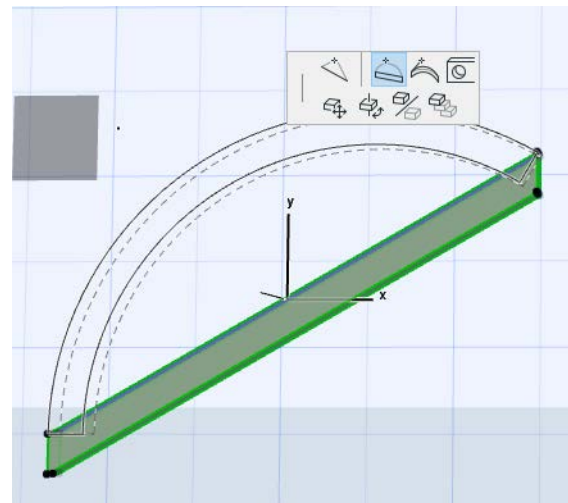
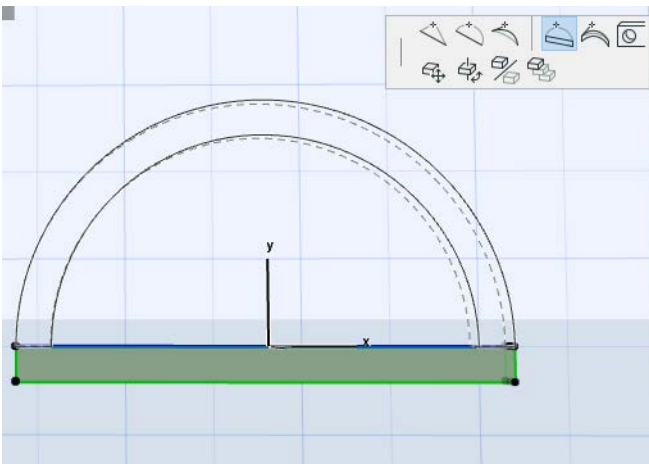
1. Select a Beam
2. Use the pet palette commands to curve the edge vertically, or edit using tangent.



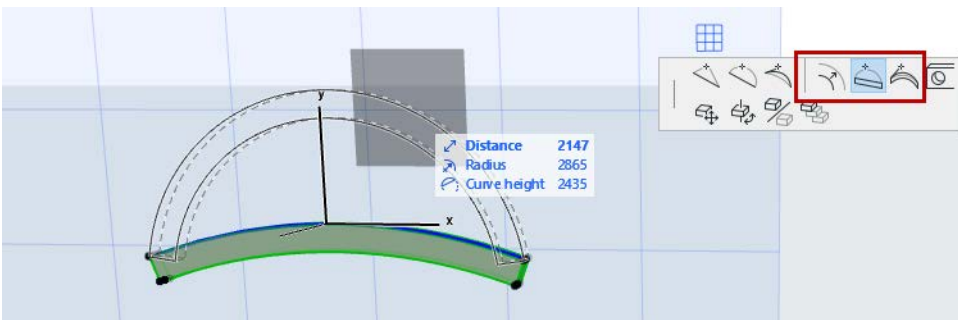
A Beam's vertical curve cannot curve in on itself.

A given Beam cannot curve in both vertical and horizontal directions.

- For a straight beam: its curve height cannot be larger than the radius of the curve.
- For a slanted beam: the arc's start and end points cannot be lower than the arc's origin

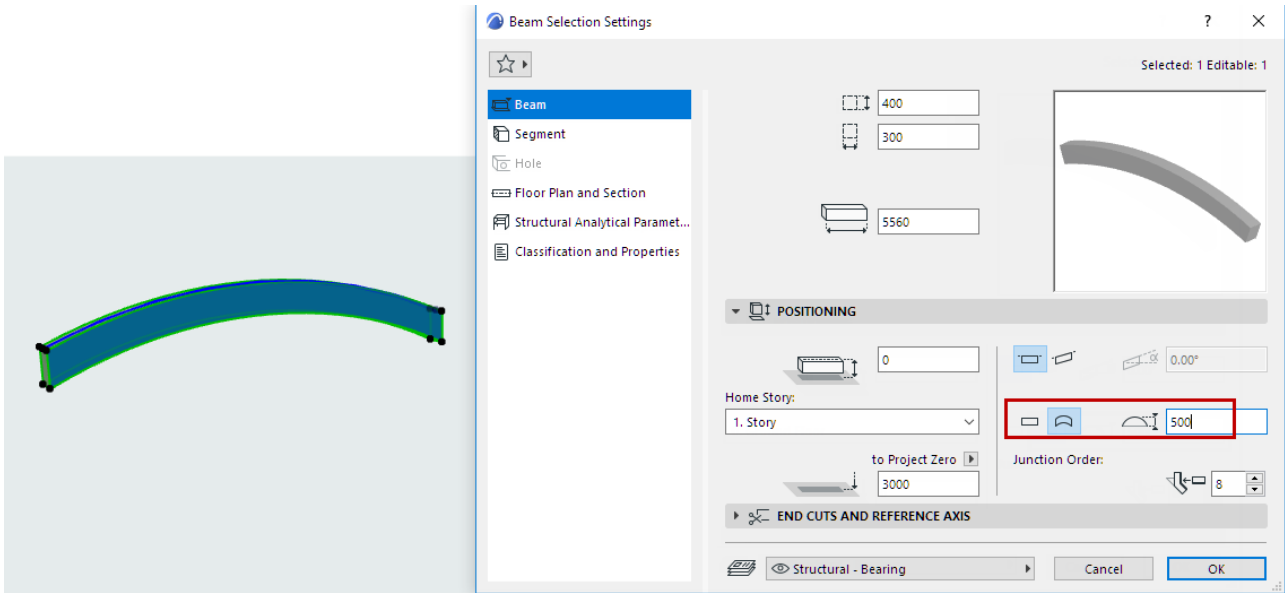


- For a vertically curved Beam, the Radial Stretch command is also available:



Alternatively, in Beam Settings or the Info Box: click the Curved icon and define the Chord/Arc Height.

- Negative value is possible



- If the initial Beam was horizontally curved, then clicking the Curved icon will assign a vertical curve and eliminate the horizontal curve.

Related Topic:

[Horizontal Curved Beam](#)

Create a Multi-Segment Beam

Set up a Beam that consists of multiple segments, with a configurable “Join” component between them. To create Beam segments, use either the Settings dialog or graphical interactions. Both methods are described below.

[Watch the Video](#)

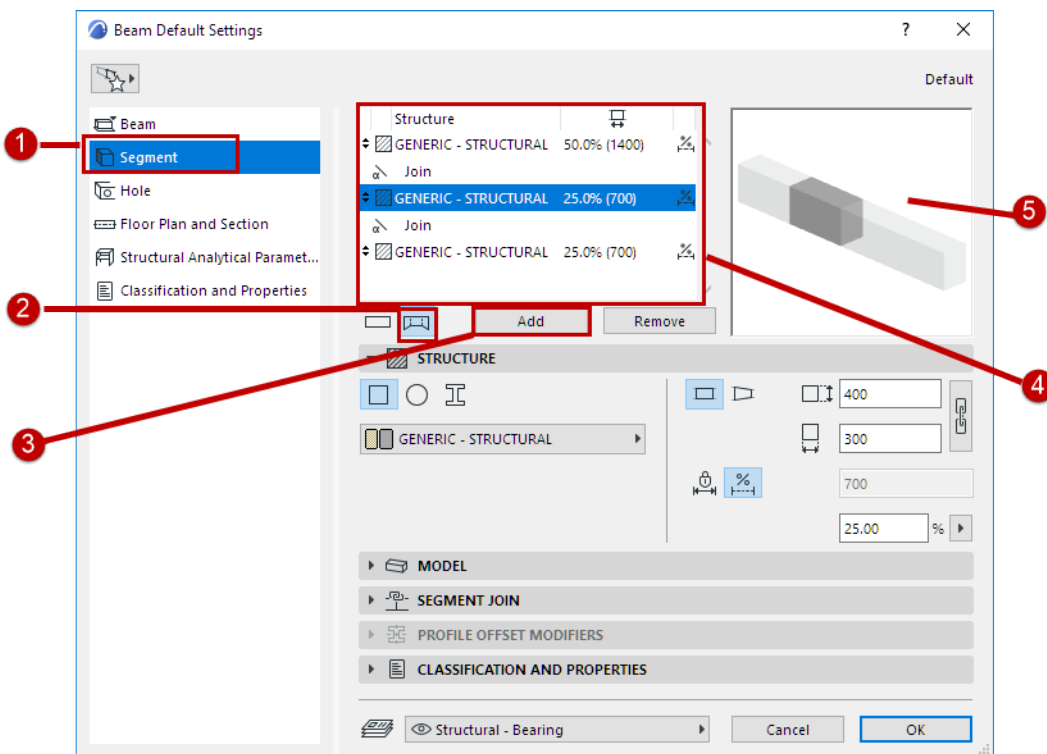
Notes:

- Each segment has its own Classifications and Properties settings and its own Element ID. However, Category definitions (e.g. Renovation status) are inherited from the element level.
- Find & Select, Labels, Info Box, Interactive Element Schedule fields and Dimensions/Labels can use segment-specific data

[Watch the Video](#)

Use Beam Settings

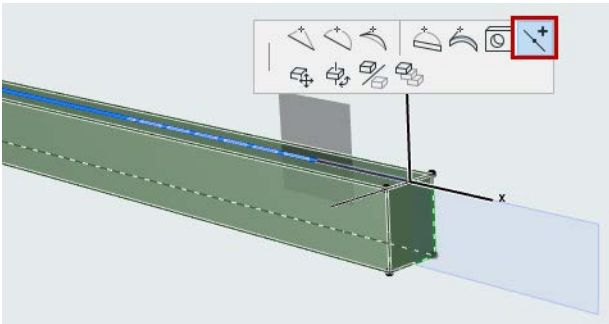
1. Open Beam Settings and go to the Segment page.
2. Click the Multi-Segment beam icon.
3. Click **Add** to add as many segments as needed.
 - A “Join” component is automatically added between each segment.
4. View the Structure list to see segments and Joins. Select or multi-select to edit.
5. View the preview. The segment currently selected in the list is highlighted.



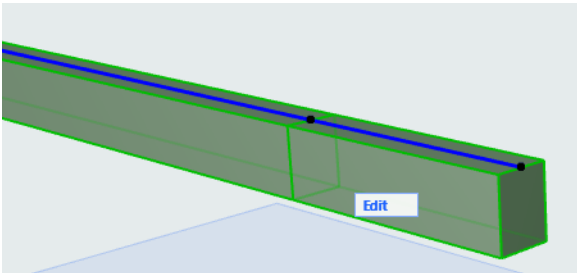
Multi-Segment Beam

Create Graphically

1. Select a Beam in the 3D window.
2. From the pet palette of the Beam axis, choose **Split Segment**.



3. Click anywhere on the Beam axis: the Beam is split at that point into segments. Repeat to create more segments as needed.



- You can adjust segment length graphically by dragging the Joins:
[See *Rotate Join Angle*](#).
- To edit the segments individually and graphically, you can use Edit mode.
[See *Edit Mode for Multi-Segment Columns*](#).
- To define attributes for each segment, use the Settings dialog.
[See *Define Column/Segment Structure*](#).

Related Topics:

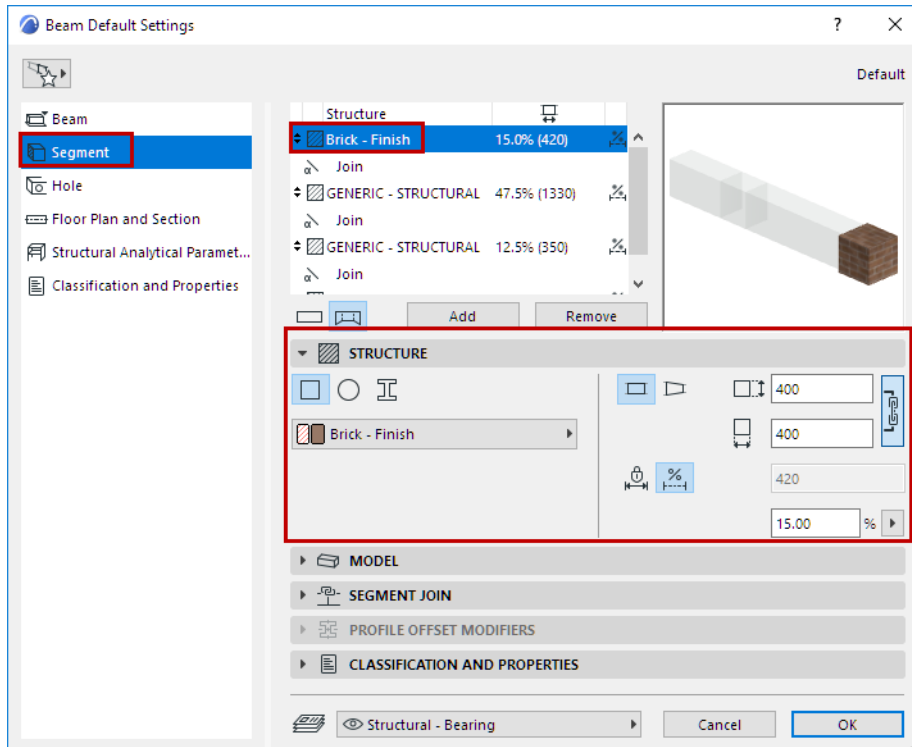
[Delete Column Segment or Beam Segment](#)

Define Beam/Segment Structure

Select a Beam or Beam segment.

Use the Structure panel of Beam Settings (Segment page) to define its structure properties and length (fixed or flexible).

For a selected segment: these Settings are also available in Beam Edit mode.



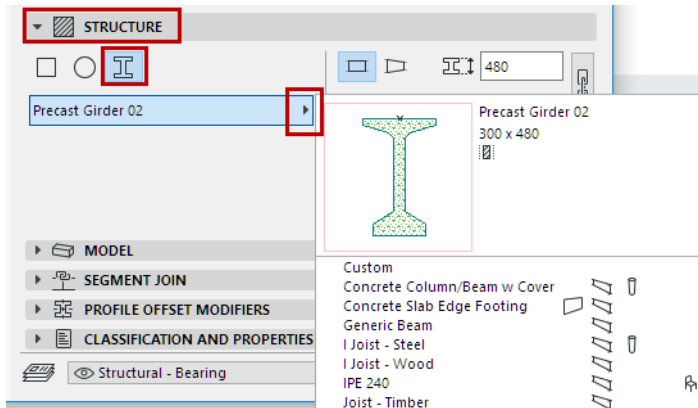
Cross Section

Click an icon to define segment cross-section.

- Rectangular
- Circular
- Complex Profile

Building Material/Profile

Use the pop-up to choose a Building Material or Profile for the Beam or Beam segment.

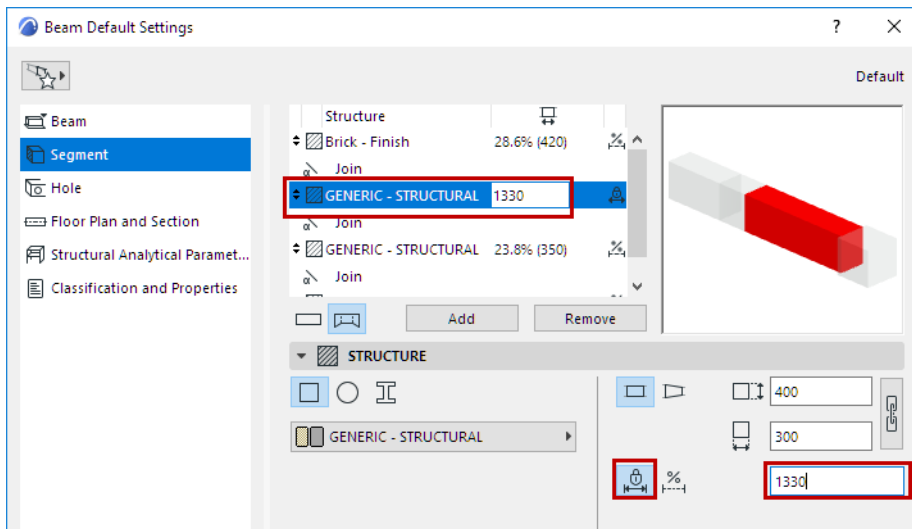


Note: The pop-up shows only the Profiles whose “Use With” control includes Beams. Profiles are defined at Options > Element Attributes > Profile Manager.

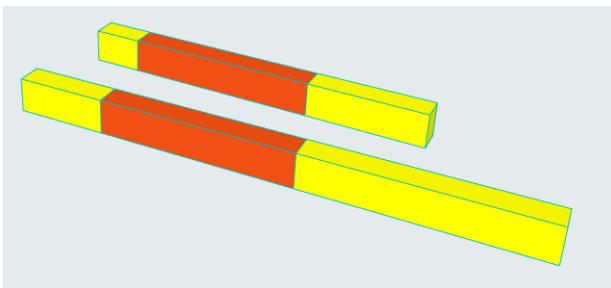
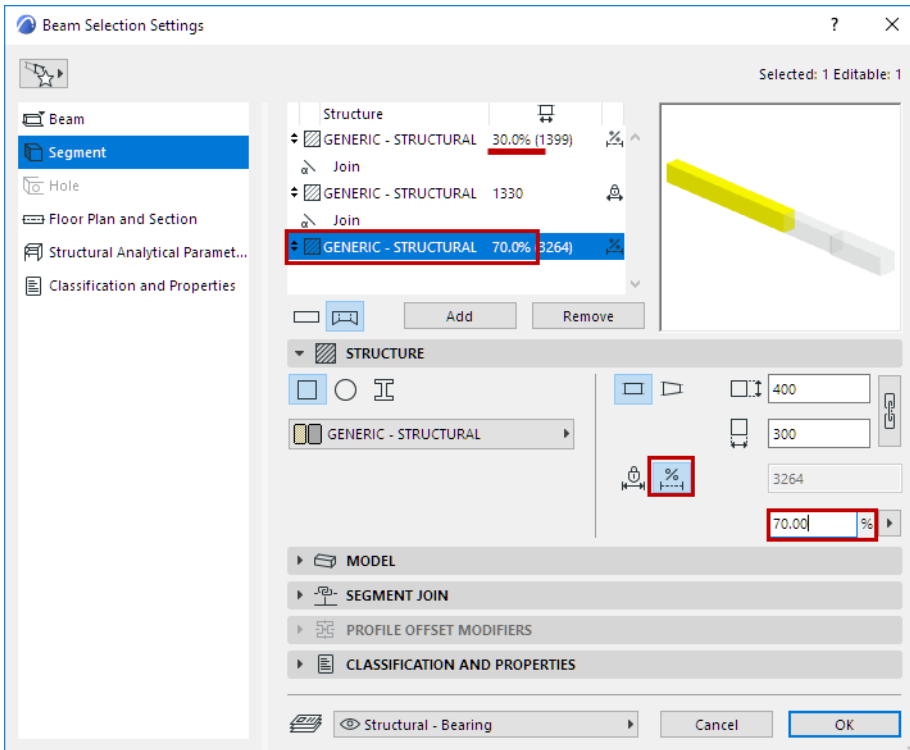
Segment Length (Fixed or Flexible)

For each selected segment: Define its length as either **fixed** or **flexible**, by clicking the corresponding icon in the Structure panel.

- For fixed-length segment: Define the length. It will remain unchanged even if the Beam stretches. (You can also enter a selected segment’s length in the Structure list.)



- For a flexible segment: Define its length as a percentage of all the flexible segments. (The flexible segments together add up to 100%.) Segment length will stretch accordingly if the Beam stretches.



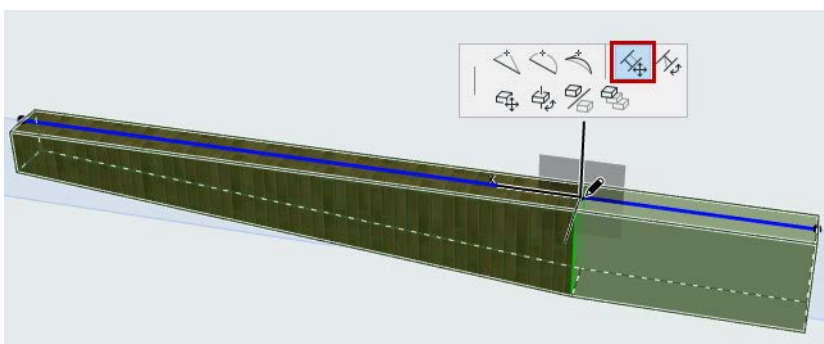
Flexible (yellow) segments stretch with the Beam.

Fixed (red) segment does not stretch.

Segment Length - Edit in 3D

1. Select the Beam in the 3D window.
2. Click the Join hotspot (on the Beam reference line).
3. From the pet palette, use **Move Segment Join** to adjust segment length

Note: If you drag the segment hotspot into the adjoining segment, this eliminates the dragged segment entirely.



Related Topics:

[Tapered Column or Segment](#)

[Rotate Join Angle](#)

[Core and Veneer](#)

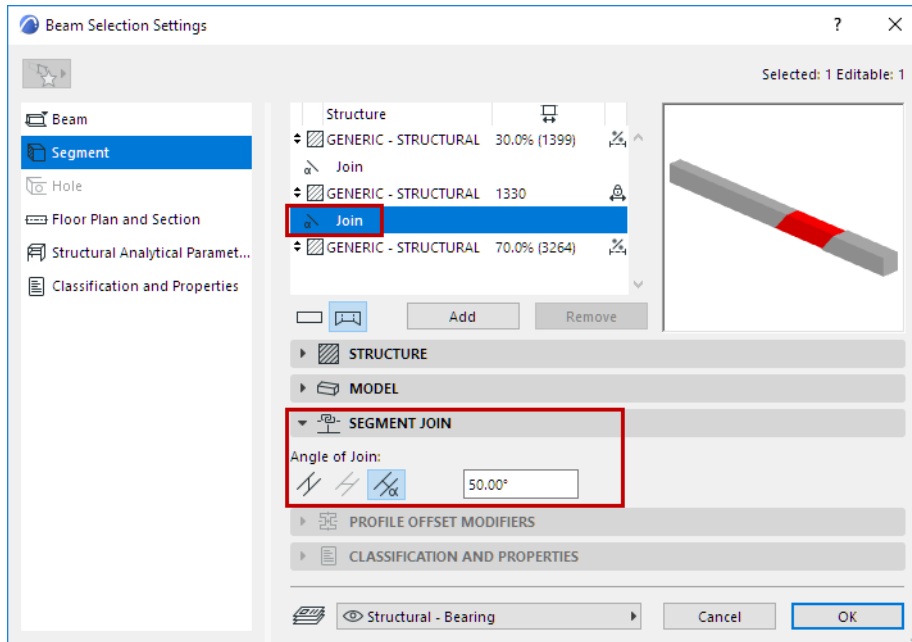
[Delete Column Segment or Beam Segment](#)

Define Join Geometry

In Beam Settings:

For a Join component selected in the list (at the top of Beam Segment Settings): use the **Segment Join** panel (below) to define its angle as horizontal, vertical or custom.

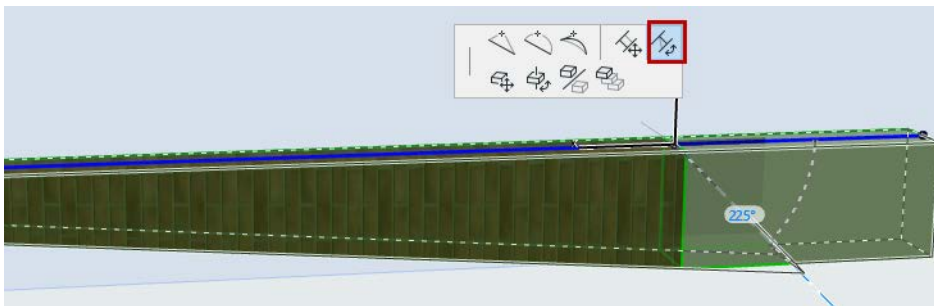
Note: Vertical Join available for slanted Beam only.



In 3D Window:

Joins can be edited graphically:

1. Select the Beam in the 3D window.
2. Click the Join hotspot (on the Beam reference line).
3. From the pet palette, use **Rotate Join Angle** to adjust the Join angle



Related Topic:

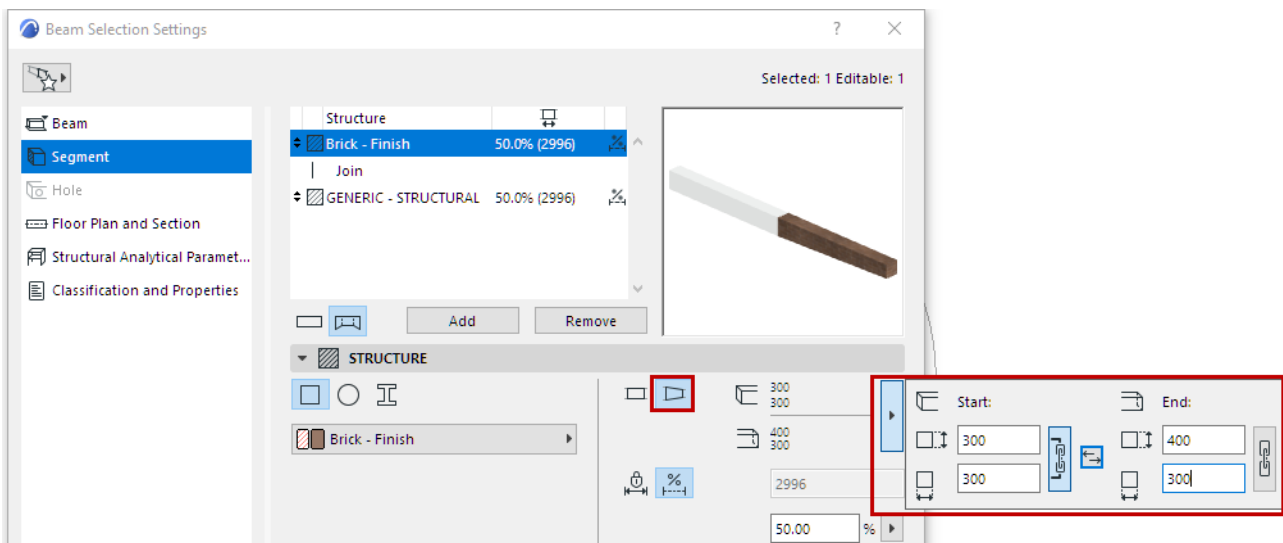
[Define Beam/Segment Structure](#)

Tapered Beam or Segment

You can create a tapered Beam, or taper any single Beam segment.

Use the Segment page of Beam Settings. (For a multi-segment Beam, select the segment to taper.)

1. In the Structure panel, choose Tapered geometry.
2. Use the pop-up to define the cross-section dimensions of the tapered Beam (segment) at its Start and End surfaces.
 - Use the chain icon to make the height/width dimensions equal
 - Use the arrow button to reverse the tapering direction



Related Topic:

[Define Beam/Segment Structure](#)

Editing Beams

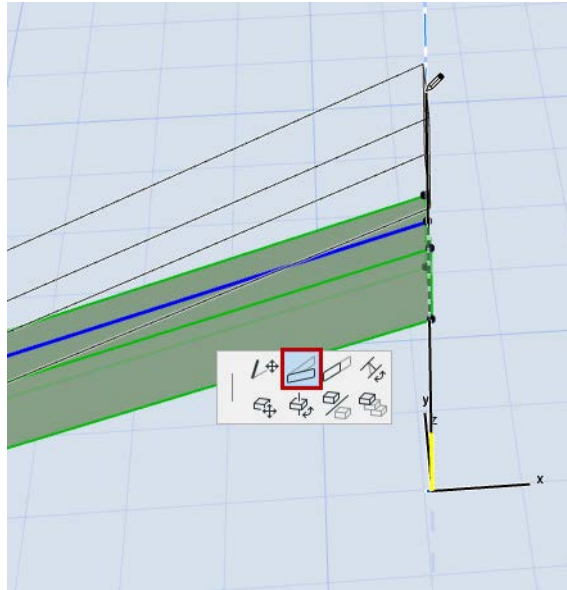
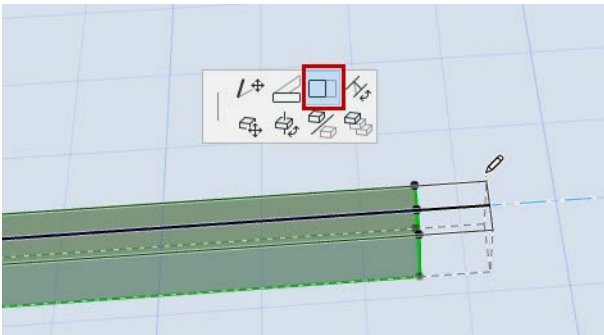
Stretch Length/Modify Angle

Use the Pet Palette commands at the Beam's Reference Axis endpoint.

- **Stretch length**

For a slanted Beam:

- **Stretch length with fixed Pitch**
- **Modify angle**



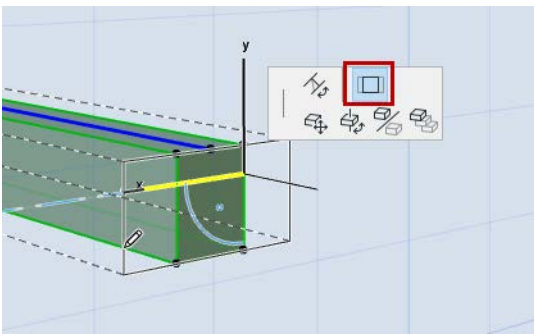
Stretch Beam Cross-Section

For a non-curved, non-segmented Beam:

In the Floor Plan or 3D window, use the Pet Palette commands to stretch the Beam's cross-section dimensions.

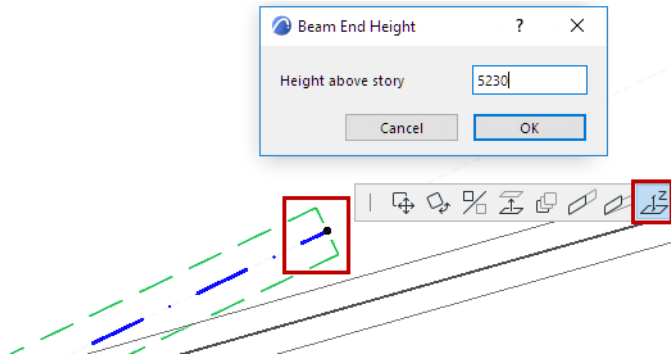
Note: Beams can also be stretched in Section view, if the Beam reference line is *perpendicular* to the Section line.

Alternatively, use **Edit > Reshape > Stretch**.



Edit Slanted Beam on Floor Plan

1. Select the slanted Beam on the Floor Plan.
2. From the pet palette of the reference line, click the endpoint height command.
3. In the Beam End Height dialog, enter the new height of the selected Beam endpoint.



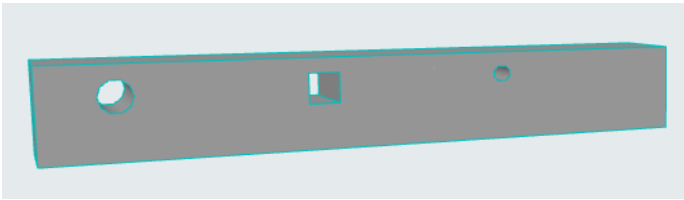
Related Topics

[Stretching Curved Elements](#)

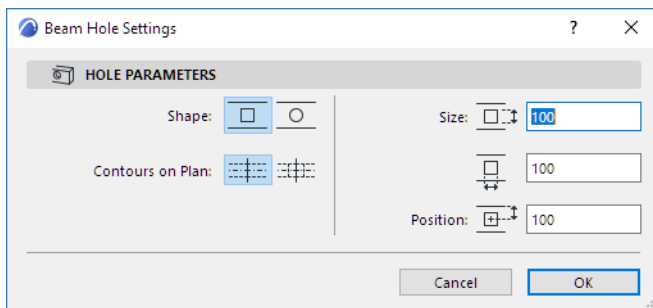
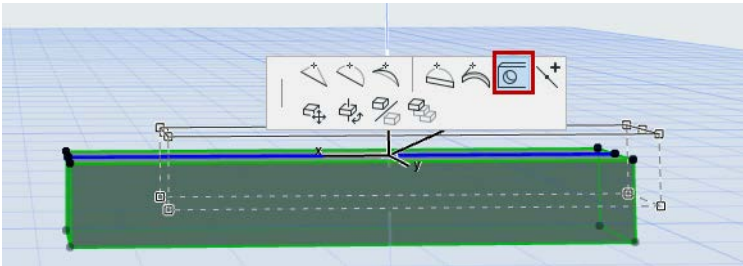
[Watch the Video](#)

Create a Hole in a Beam

Holes are defined after the Beam has been placed.



1. Select the desired Beam and make sure the **Beam** tool is active.
2. Click on an edge or the reference axis.
3. In the appearing pet palette, click the Hole icon.
4. The **Beam Hole Settings** dialog box appears.



Click OK to apply the beam hole settings; the Hole will immediately appear in the Beam.

Related Topics:

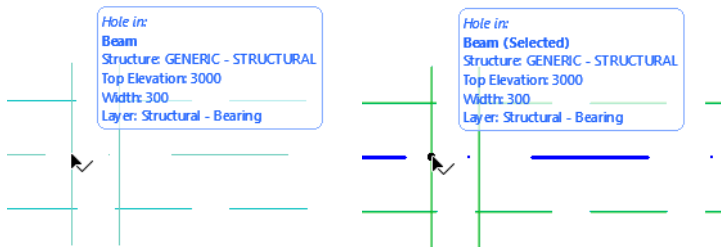
[Selecting and Modifying Beam Holes](#)

Selecting and Modifying Beam Holes

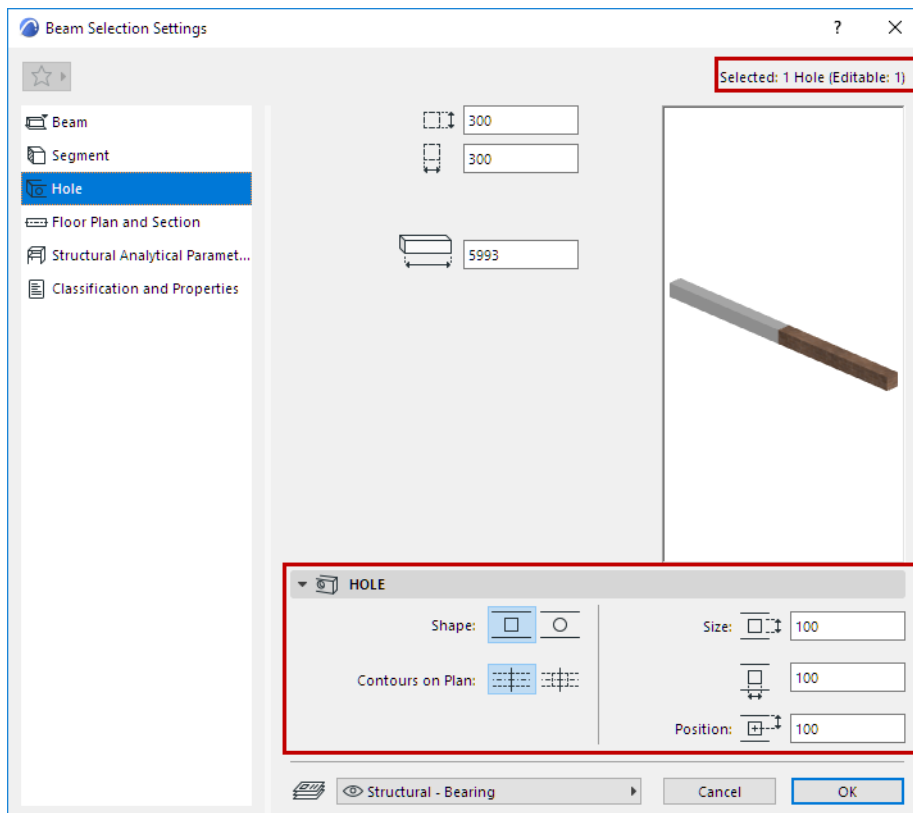
Modify Single Hole

- Do one of the following:
 - With Arrow tool active, click in the center of the hole.
 - With any other tool active, Shift-click in the center of the hole.

Now the hole is selected.



- Open Beam Selection Settings - Hole Panel to modify the selected hole's shape and size (or use the controls in the Info Box).

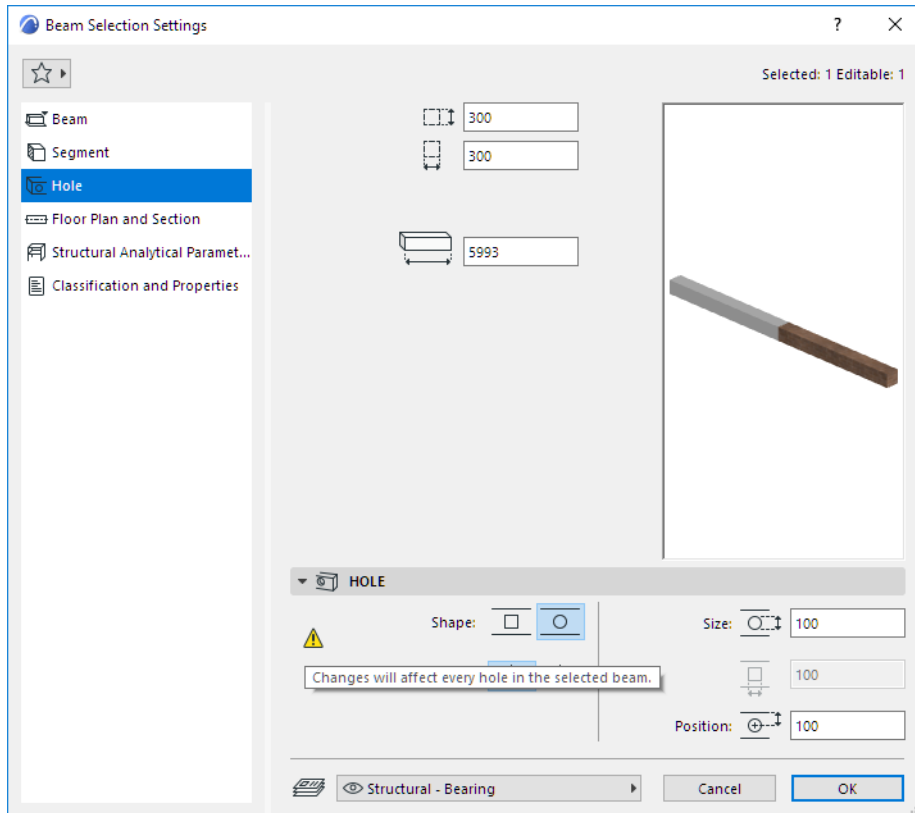


- Shape:** Rectangular or circular
- Contours on Plan:** Choose floor plan symbol: show outline, or show axis only
- Size:** Enter the horizontal and vertical dimensions of the Hole. For a circular hole, you will enter a diameter.

- **Position:** Distance of the center of the Hole from the top surface of the Beam.

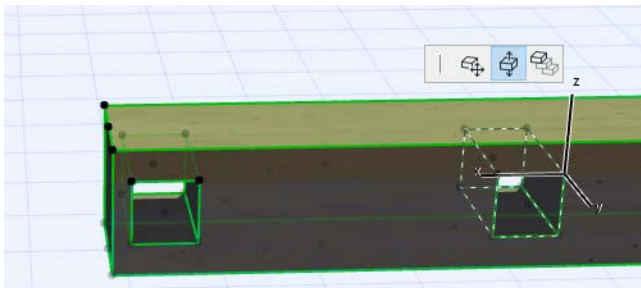
Modify all Holes

1. Select the Beam. All of the Beam holes are selected, too.
2. Open Beam Selection Settings - Hole Panel to modify all Beam holes simultaneously (or use the controls in the Info Box).



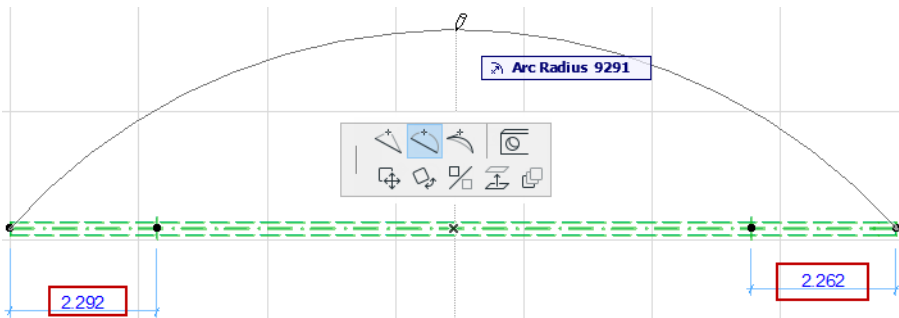
Move Beam Hole Graphically

- Move a hole laterally along the Beam (Floor Plan or 3D)
- Move a hole up and down along the Z axis (3D)

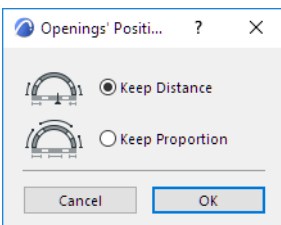


Holes in Curved Beams

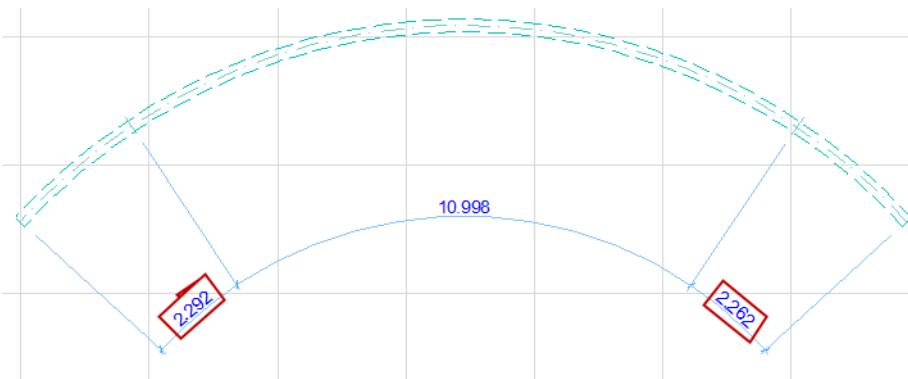
Suppose you have placed holes into a straight beam (at the given distances from either end of the beam), and then you curve the beam (as in the picture below).



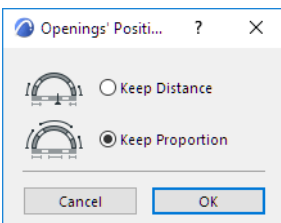
The **Beam Openings' Position** dialog box allows you to choose how to handle the holes' new position in the curved beam:



Keep distance: The holes' measured distances from the nearest beam end (as measured along the center beam arc) remain unchanged.



Keep proportion: The holes remain at the same proportional distance from the beam ends as before.



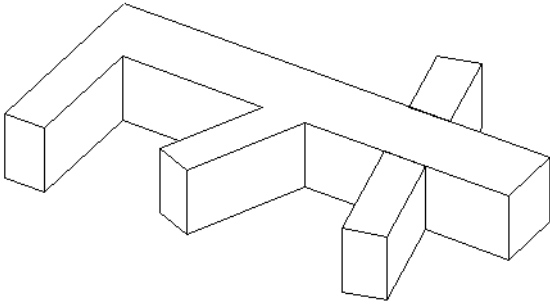
[Watch the Video](#)

Beams and Other Elements

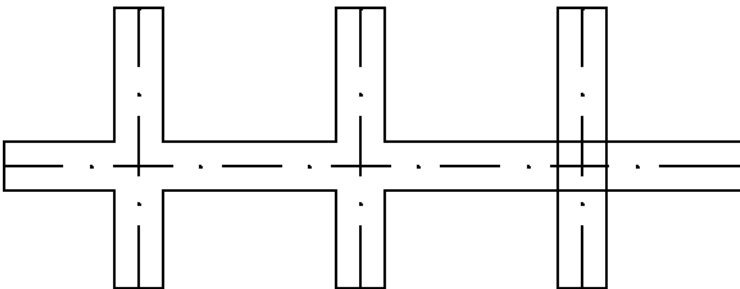
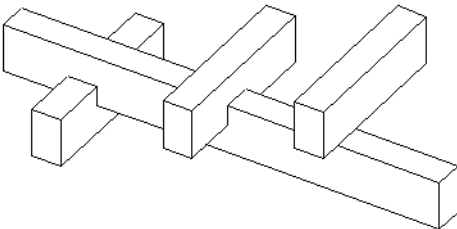
Auto-Intersection of Beams with Other Beams

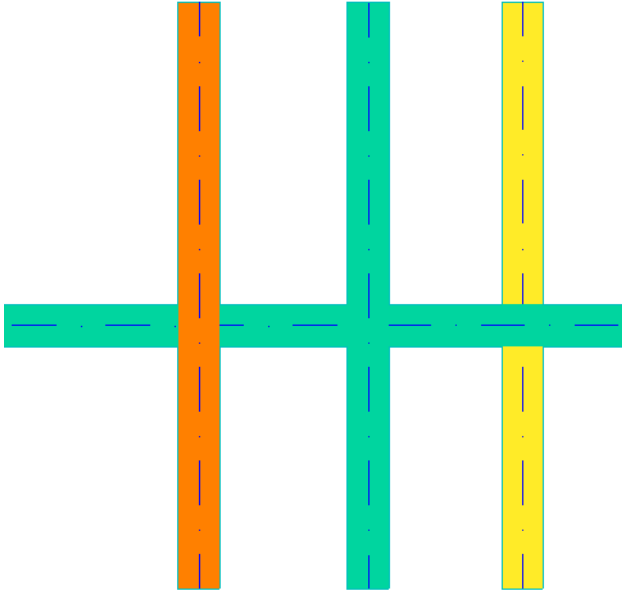
When a Beam intersects another Beam, Archicad automatically cleans crossings, **T** and **L** intersections of Beams if the **Options > Auto Intersection** feature is turned on.

If one end of the joining Beam falls within the contour of the other Beam, the end of the joining Beam will automatically adjust to the reference axis. Their appearance in 3D will be “cleaned up” accordingly, provided that the beams have the same intersection priority.



Beam intersections are also cleaned up if the crossing or joining Beams run at different elevations. If Beams pass each other in space, without intersecting reference lines, the intersection will be cleaned up depending on their respective intersection priorities.





Cover fills of beams on floor plan

See [Use layers to prevent intersections.](#)

Related Topics:

Beams can be trimmed to Roofs or Shells.

See [Trim Elements to Roof or Shell.](#)

You can crop a Beam to a Single-plane Roof.

See [Crop Elements to Single-plane Roof.](#)

Use **Adjust Elements to Slabs** to adjust horizontal Beams to specific levels of one or more Slabs that are located above and/or below them.

See [Adjust Elements to Slabs.](#)

For information on how intersections (junctions) work among construction elements:

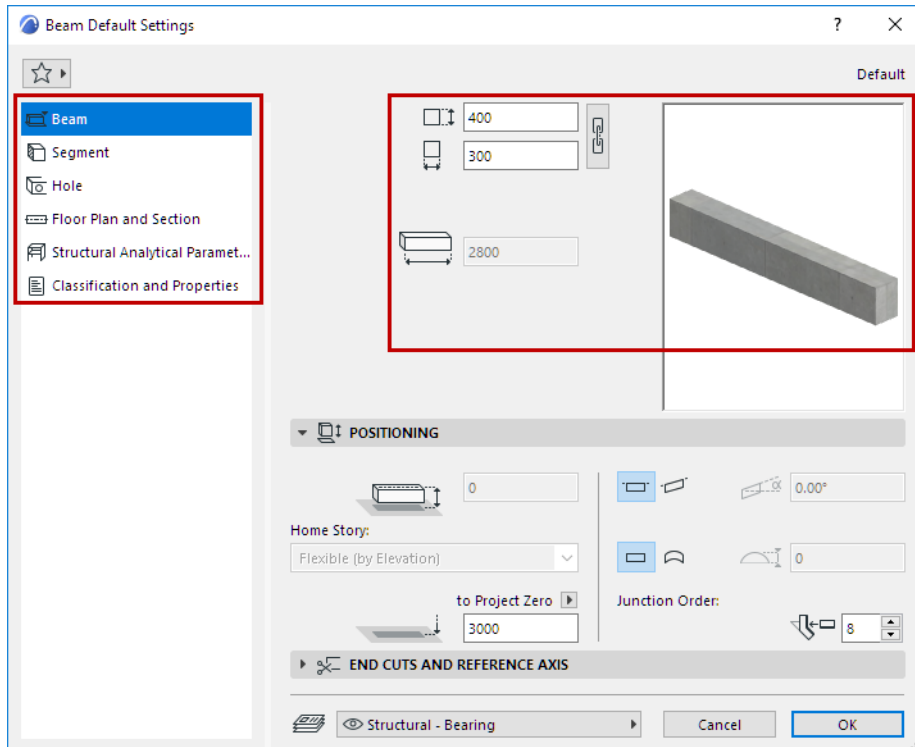
See [Element Intersections.](#)

Beam Tool Settings: Preview and Positioning

The Beam Settings dialog box is divided into five pages.

The **Preview** section remains visible on all pages, and reflects your settings modifications. For a multi-segmented Beam, the Preview section includes its Structure settings.

Navigate among the pages using the tree structure on the left:



For a simple (single-segment) Beam, enter its geometry values:

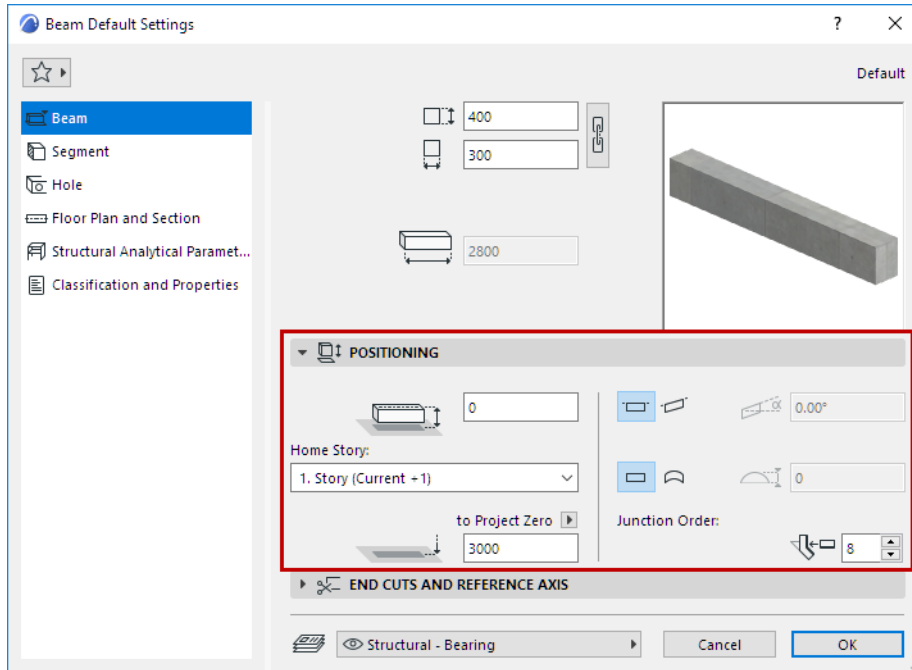
Beam Width/Height

- Use the **chain** icon to constrain them to the same value

Beam Length

- Default (segment) length

Beam Positioning Panel



Offset to Home Story

Offset the Beam's reference axis from the Home Story.

Home Story

See [Home Story](#).

Reference Line Elevation [to Reference level]

Calculates the current elevation of the Beam's reference axis, as measured from the Reference level (by default, this Reference level is Project Zero).

For slanted Beams, this elevation is measured from the starting point of the Beam's reference axis.

Click the pop-up arrow to change the Reference level, if needed.

Notes:

- Reference levels are defined at **Options > Project Preferences > Reference Levels**.
- This is a calculated value, not a link. If you change the Reference level, the Beam will not change its position.

For general information on levels, see [Reference Levels Preferences](#).

Horizontal or Slanted

See [Create a Slanted Beam](#).

Flat or Curved

Beam geometry in the Z-direction.

- For a vertically curved Beam, enter its **Arc/Chord** height.

See [Curve a Beam Vertically](#).

Junction Order

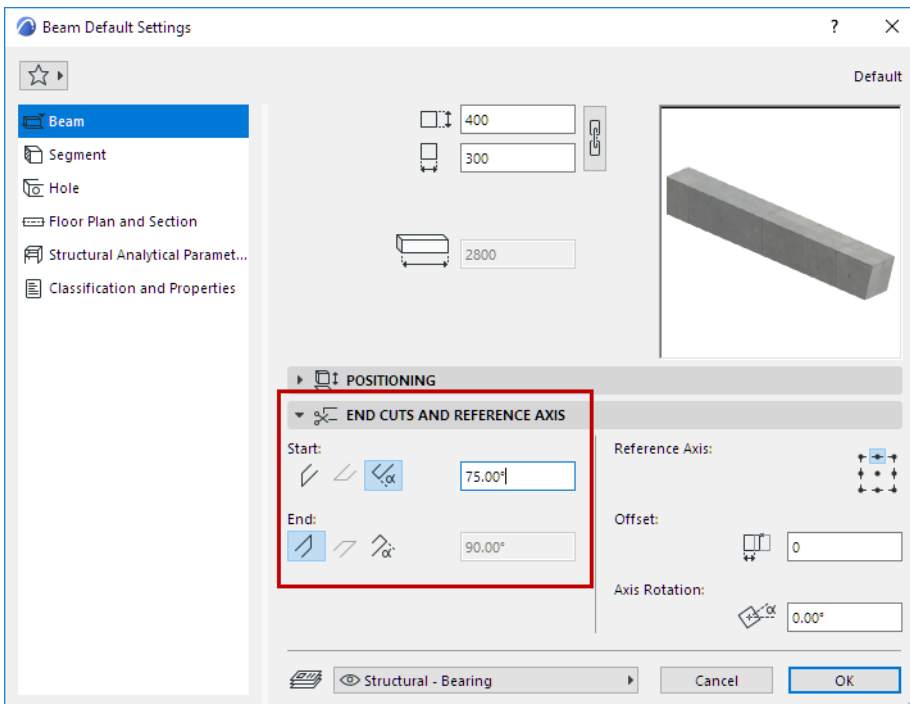
A higher Junction Order number takes priority over the lower.

Junction Order is relevant

- when two Beams meet whose Building Materials have the same priority
- when 3 or more Beams meet in a junction: Junction Order determines which two Beams are joined first

Beam End Cuts

Go to Beam Settings - End Cuts and Reference Axis Panel, or the Start/End angle controls in Beam Info Box.

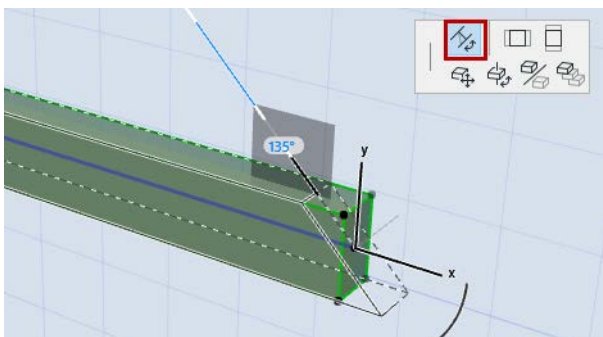


Define the Start and End edges of the Beam as angle as horizontal, vertical or custom.

Note: Horizontal available for slanted Beam only.

You can also edit this geometry graphically:

- Select a Beam in 3D. Click on its end surface, then choose **Rotate Join Angle** from the pet palette.



Beam Reference Axis

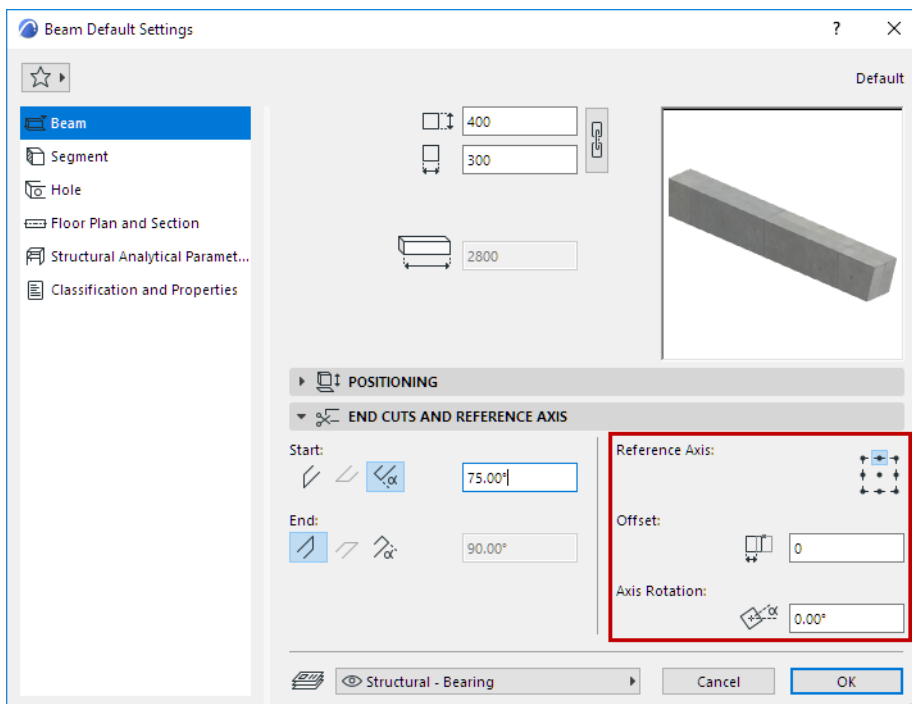
The **Reference axis** serves the precise connection of Beams for clean intersections, and establishes hotspots and edges for selecting, moving and transforming Beams. Its direction determines the beam's "left" and "right" surfaces for the purpose of applying surfaces in the Model Panel of Beam Settings.

[See *Beam Model Display*.](#)

The direction of the reference axis follows your cursor as you drag it to draw the beam. (To see the arrow on the beam that indicates the axis direction, turn on **View > On-Screen View Options > Walls and Beams Reference Lines**.)

The reference axis of a selected Beam is shown in a separate color: you can change this color using the "Reference Lines" setting at **Options > Work Environment > Selection and Element Information**.

In Beam Settings or Beam Info Box, choose one of the nine points to serve as the Beam's Reference Axis.



Note: Available only if at least one Beam segment has a rectangular or circular cross-section. Not available for complex profiles.

If you modify the Reference axis: the axis stays in place while the Beam moves accordingly.

Display/Hide Reference Axis

Use the Floor Plan and Section panel of Beam Settings. [See *Symbols*](#).

Reference Axis Offset

Optionally, enter a value to offset the Reference Axis from the middle of Beam.



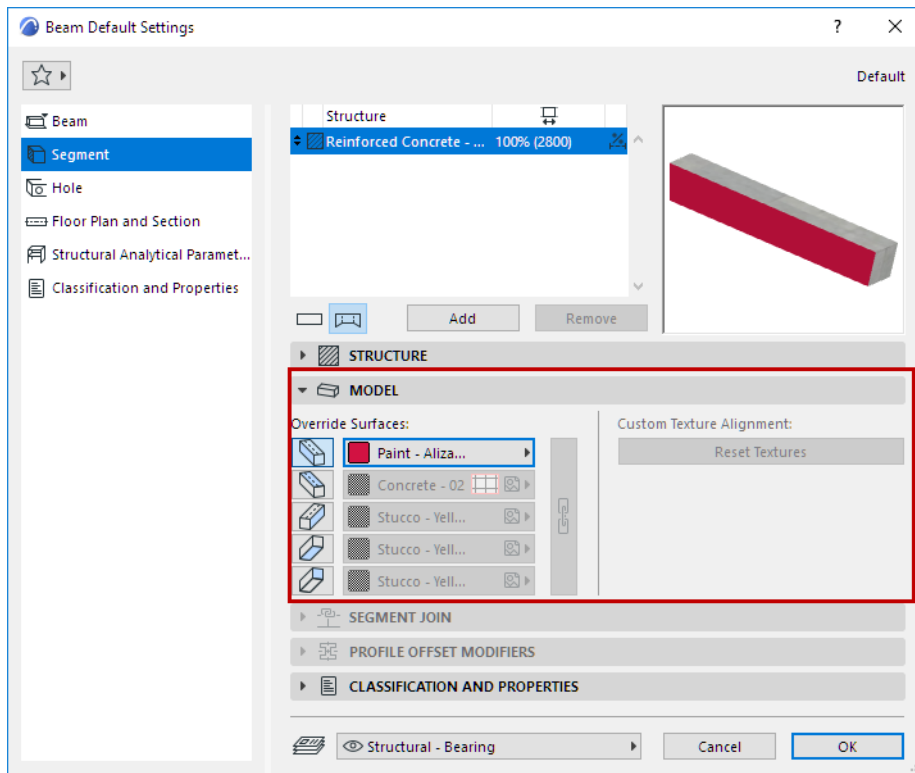
Axis Rotation

Set a rotation angle for the Beam around its axis.

See also [Rotating Beam and Columns](#).

Beam Model Display

Use the Model Panel of Beam Settings (Segment page) to override Beam surface(s).



Override Surfaces:

Each side of the Beam is listed. Click the icon of the Beam side(s) to override that surface (assigned via its Building Material).

- For a rectangular Beam: Override left side, right side, top, bottom, and/or the ends.
- For a circular Beam: You can override the Ends surfaces only.

Note: Left side is defined by the left hand, looking from the first point to the next point of the Reference Axis of the Beam.

- For a profile Beam: Override the Extrusion surface and/or the Ends surface.

Note: You can also apply a Custom surface to any edge(s) of the profile element.

[See *Apply Custom Surface or Line to Profile Edge*.](#)

Click the **chain** icon to assign the last override surface to all the selected faces/edges. Deselecting the chain icon will restore the surfaces that you originally set for each face/edge.

[For more information, see *Surfaces*.](#)

[See also *Override Surfaces With Surface Painter*.](#)

Custom Texture Alignment: If this message is activated, it indicates that the currently selected beam (already placed in the project) has been assigned a custom 3D texture.

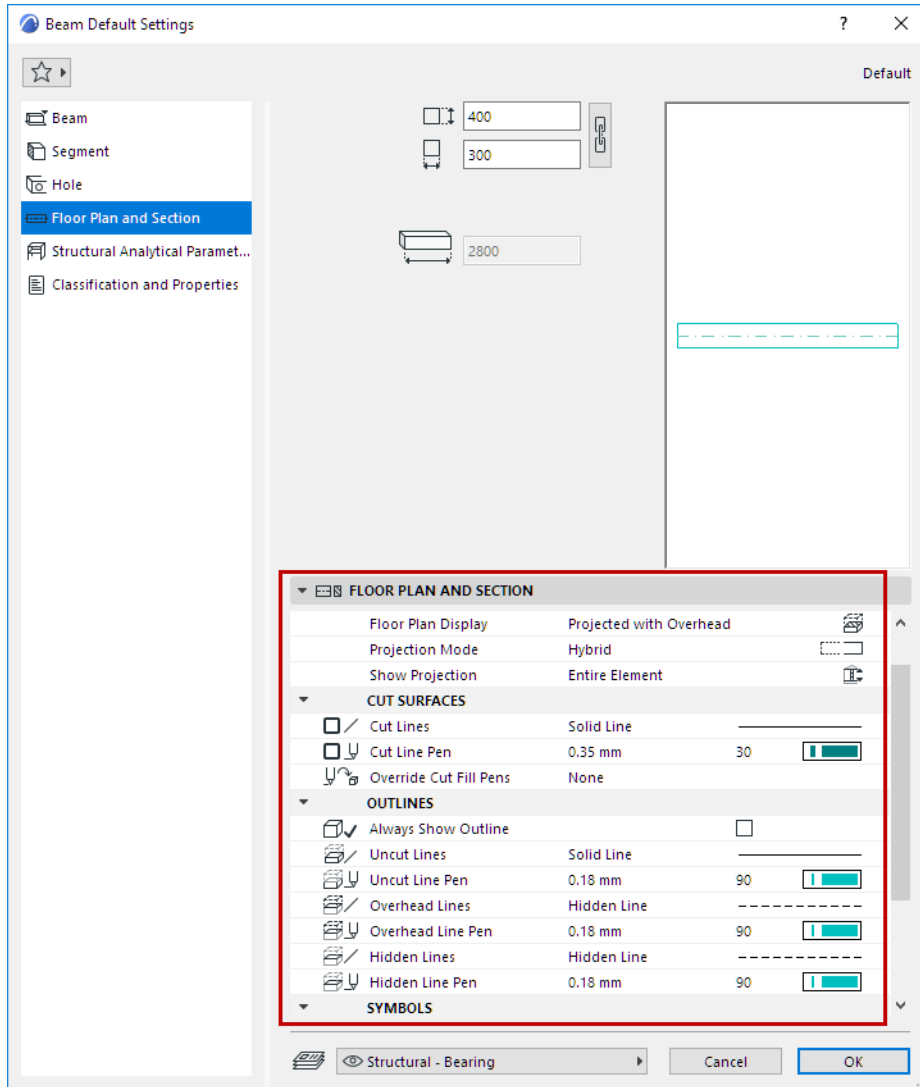
- In this case, the **Reset Texture** button is also activated. Click to restore the origin of the Texture of the selected beam.

[For more information, see *Align 3D Texture*.](#)

Beam Display on Floor Plan and Section

Use the Floor Plan and Section page of Beam Settings.

[See Element Display in Floor Plan and Section.](#)

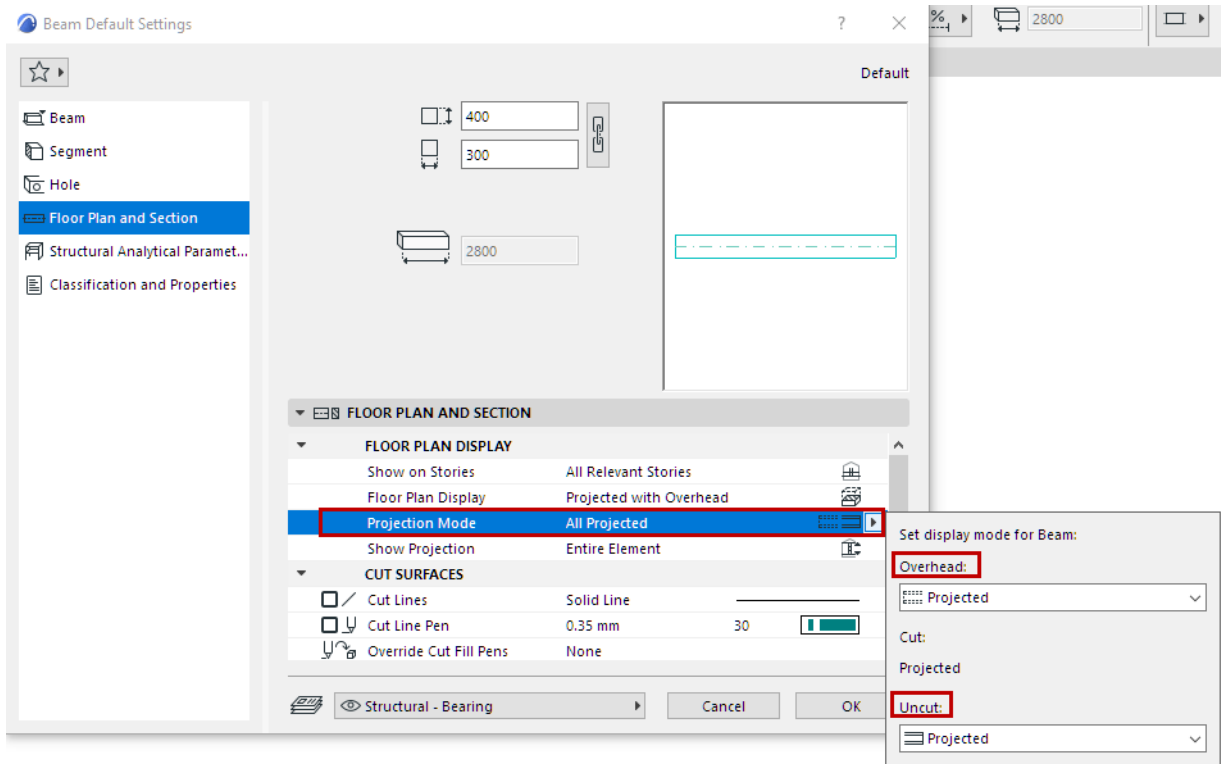


[Watch the Video](#)

Projection Mode

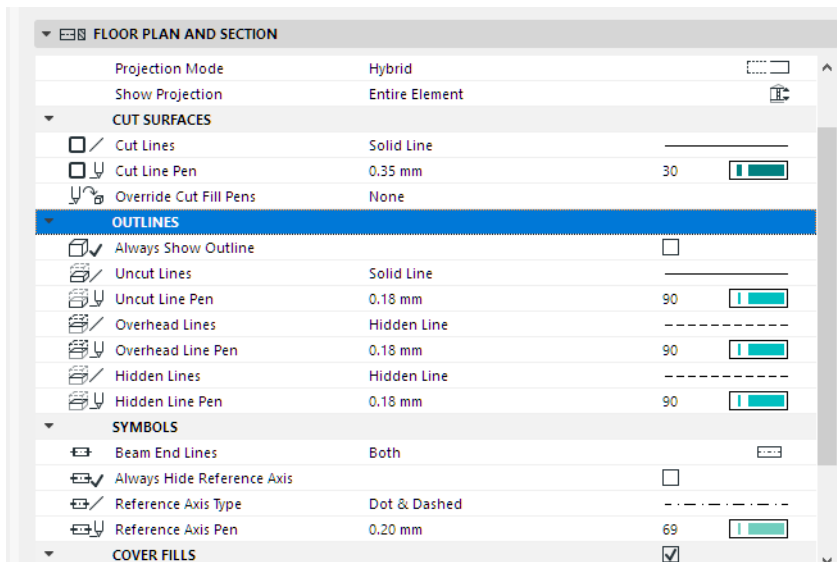
Once you have set Floor Plan Display, use Projection Mode to fine-tune each displayed component (Overhead, Uncut) of the Beam: as either Symbolic or Projected.

Cut parts of the Beam always use Projected display.



- Available options depend on the Floor Plan Display you chose above; some components may be invisible, and you cannot set a display mode for those.

Outlines

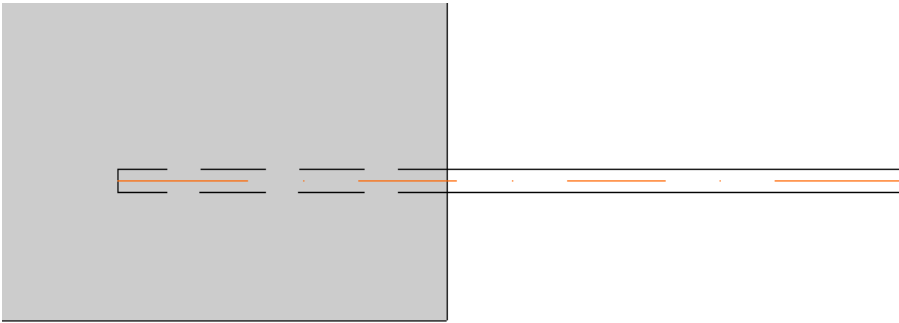


- **Always Show Outline:** Check this to display the contour of this Beam even if the Model View Option is set to hide Beam contours.

Hidden Lines

Define Hidden Line attributes. Hidden Lines are used for Beams under Slabs if this is turned on at Model View Options.

[See Model View Options for Construction Elements.](#)



Beam Underneath Slab: Use Hidden Line

Symbols

Use these controls to define settings for the beam's symbolic components.

- **Beam End Lines:** These options let you show or hide one or both end lines of the beam. (The end line is the contour line perpendicular to the longitudinal axis of the beam.)
- **Always Hide Reference Axis:** Check this to hide the Reference Axis of this Beam even if the Model View Option is set to show them.
- Choose a Line type and Pen color for the Beam Reference axis.

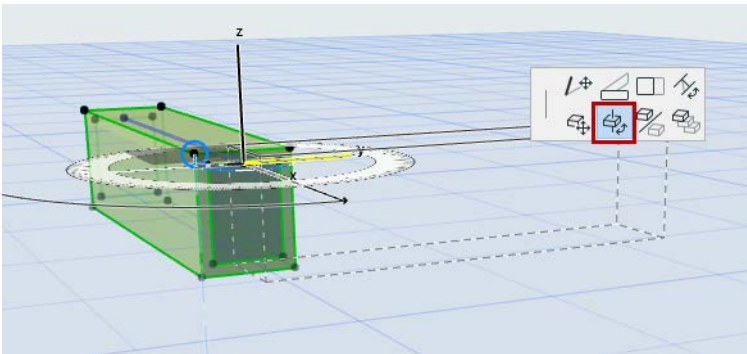
Cover Fills

See [Assign a Cover Fill](#).

Rotating Beam and Columns

Rotate Vector

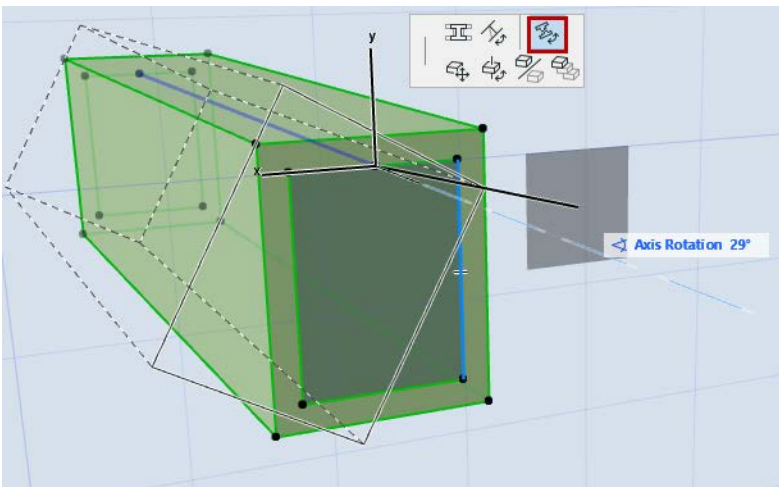
The regular Rotate command rotates the Beam/Column element together with its extrusion vector (i.e. the Beam reference line or the Column axis.)



Rotate Profile

Do one of the following:

- Use the **Rotate Profile** (Beam) or **Rotate on Axis** (Column) command from the pet palette of the selected element in the 3D window.



- Enter a value in the **Axis Rotation** field in Beam or Column Settings (End Cuts and Reference Axis panel)

Complex Profiles

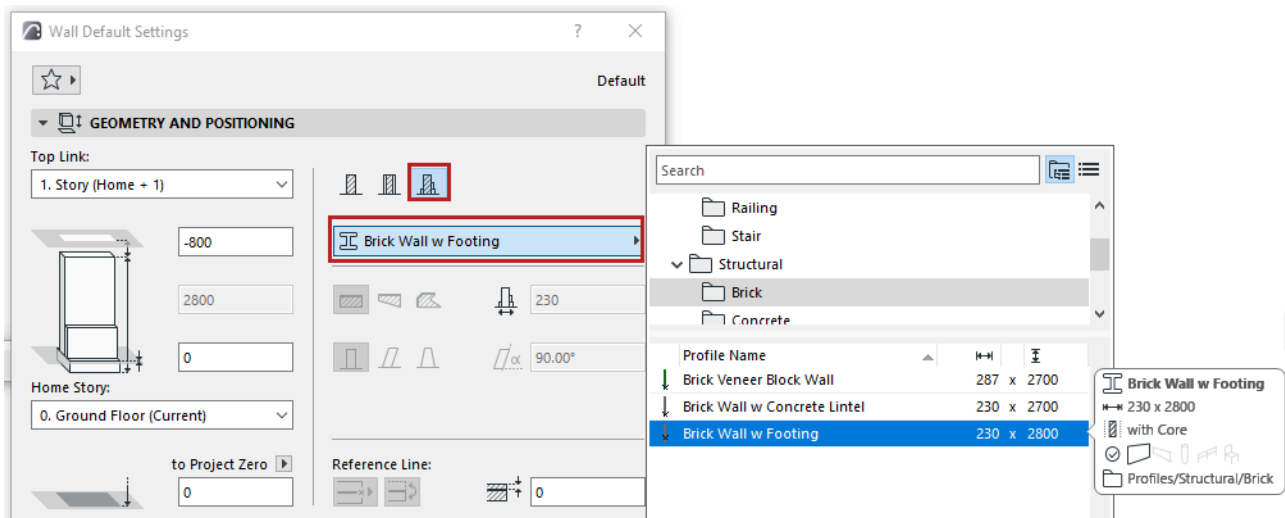
A Complex Profile element is composed of a cross-sectional profile that is extruded perpendicular to the profile's plane.

You can use or modify a predefined profile, or create a new one.

Archicad includes a predefined set of profile attributes, defined at **Options > Element Attributes > Profiles**.

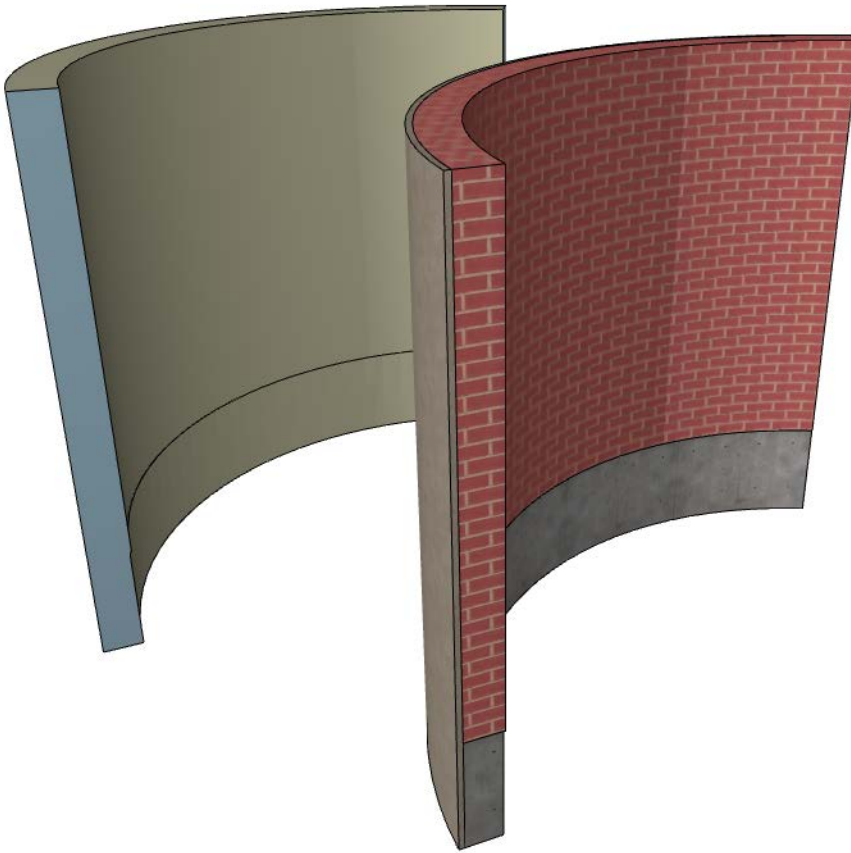
Profiles are available for Walls, Columns, Beams, Railing components, or Library Parts.

Note: Profiles for Library Parts are available only for GDL elements that include a profile selection option - such as Curtain Wall and Stair components.



Like other construction elements, a Profile's essential structure derives from its Building Material. Various surface overrides are available on an element-level basis.

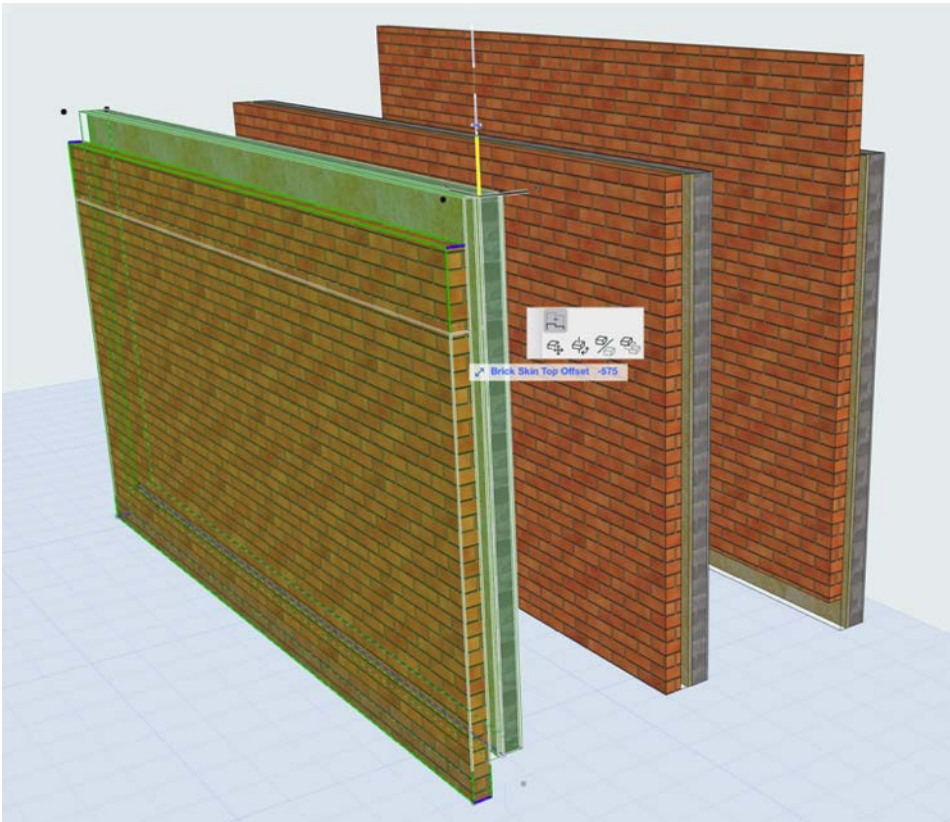
For example, the following image shows two curved walls to which the “Brick Wall with Footing” profile has been applied. The back wall uses element-level surface overrides on the extruded surfaces and ends.



A saved Complex Profile is handled as an attribute of the element (e.g. Wall or Railing). Saved profiles appear in the Settings dialog box of the respective tool.

You can apply a custom profile to any selected element in the model window on a one-time basis without having to save it as an attribute.

Wall, Column and Beam profiles can have defined Edge Offset Modifiers: these edges can be adjusted individually at the element level, even if all the elements use the same profile.



Block Wall Profile: Offset Brick Veneer for Each Placed Element

Special pet palette commands are available to stretch and rotate profile elements.

Related Topics:

[Create Complex Profile from Parallel Elements](#)

[Use Standard Steel Column or Beam Profile](#)

[Profile Manager](#)

[Edit Complex Profile](#)

[Offset Modifiers](#)

[Component Data in Archicad](#)

Apply Profile to Element

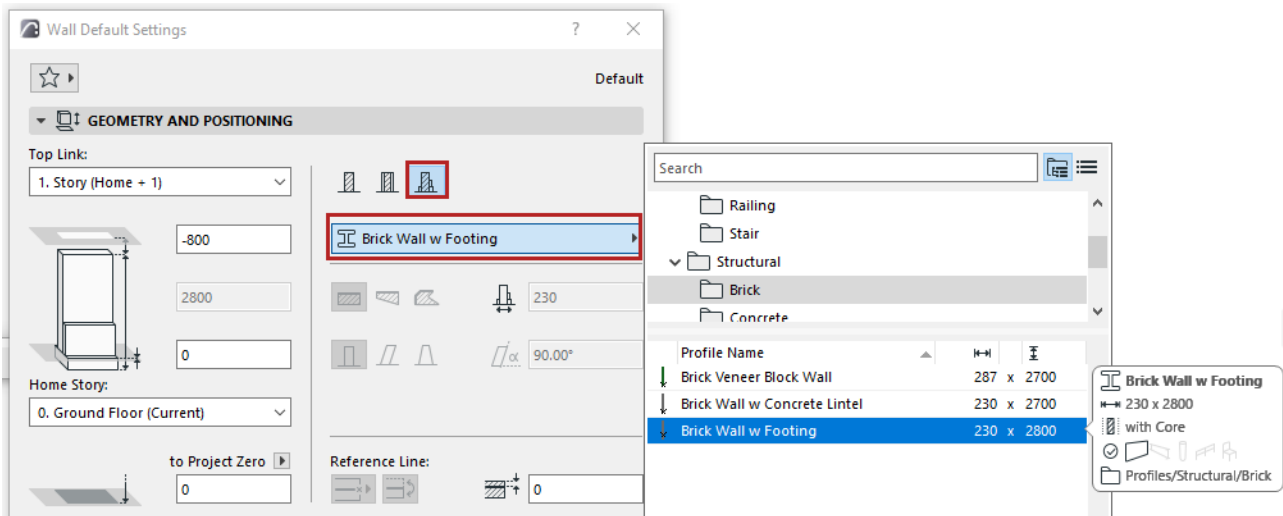
Profile attributes are available for walls, columns, beams, railing components, or Library Parts.

Note: Profiles for Library Parts are available only for GDL elements that include a profile selection option - such as Curtain Wall and Stair components.

Apply a profile to an element in element settings or from Profile Manager.

From element settings (wall, beam, column):

1. Open the element settings.
2. In the Geometry and Positioning Panel, click the “Complex Profile” icon.
3. Use the pop-up to select a profile.

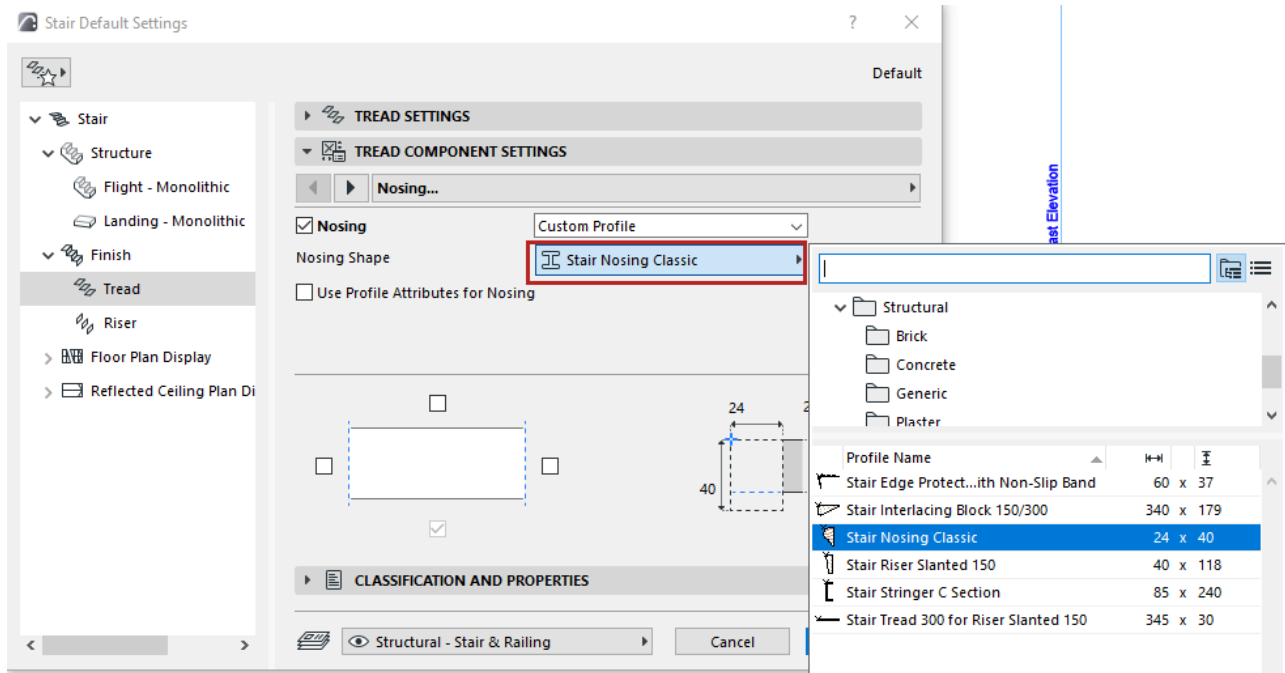


Note: This pop-up lists profiles whose “use with” setting includes your element type: see [“Use With” - define profile availability](#)

From component settings (Railing, Stair, Curtain Wall):

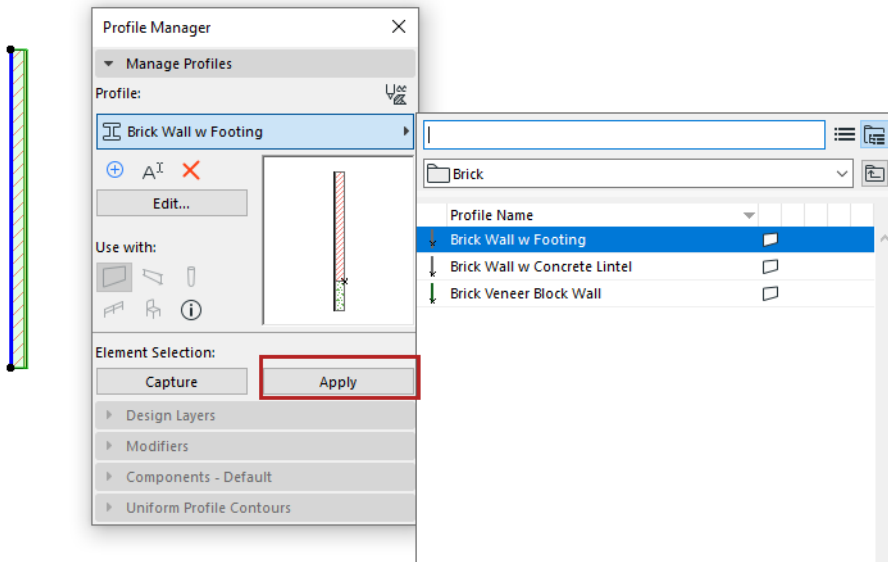
For example: choose a profile structure for railing components, Curtain Wall frame, Stair tread.

1. Open the element settings.
2. Use the component’s pop-up to select a profile.



From Profile Manager

1. Select one or more element(s) in the model window.
2. In Profile Manager, select from the **Choose Profile** pop-up.
3. Click **Apply**.



Create Complex Profile from Parallel Elements

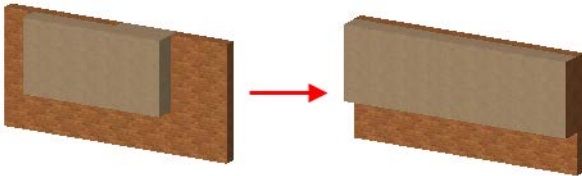
A quick way to create a complex profile is to use the **Edit > Reshape > Merge Walls/Columns/Beams**.

To use this command, draw multiple walls, columns or beams in the model window. The elements must be of the same type and must be parallel to each other. (In the case of curved walls or beams, they must be concentric.)

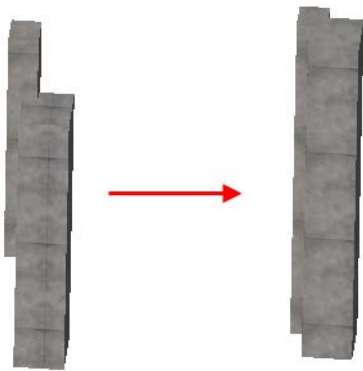
Executing the command will create a new profile element, which will replace the original elements. The new profile element will be as long as the combined length of the original elements, and will inherit the attributes of the original as well as any wall openings.

By default, the new profile is named “custom” but can be opened in the Profile Editor, modified and saved with a new name.

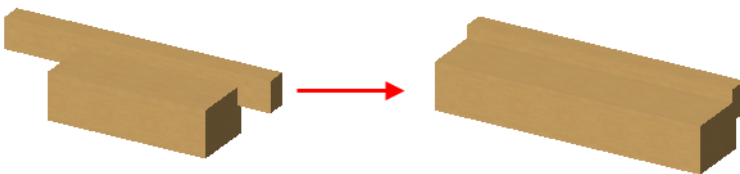
To merge two or more parallel Straight Walls, use the **Merge Walls** command (**Edit > Reshape** menu) on them. The Walls will become the same length, and the cross section of the resulting Wall will be a unity of the cross sections of the two Walls. The bodies of the two Walls do not need to touch, they may be at a distance from each other. This command does not work for Trapezoid or Polygonal Walls.



The **Merge Columns** command (**Edit > Reshape** menu) allows you to merge two or more Columns whose axes are parallel to each other. If this condition is met, all types of Columns (Vertical, Slanted, Custom Profile) may be merged. More than two Columns may be merged at the same time.



Use the **Merge Beams** command (**Edit > Reshape** menu) to merge two or more Beams (whether Horizontal, Slanted, or Custom Profile) whose axes are parallel to each other. (In the case of curved Beams, they must be concentric.)



Related Topics:

[Apply Profile to Element](#)

[Edit Complex Profile](#)

[Profile Manager](#)

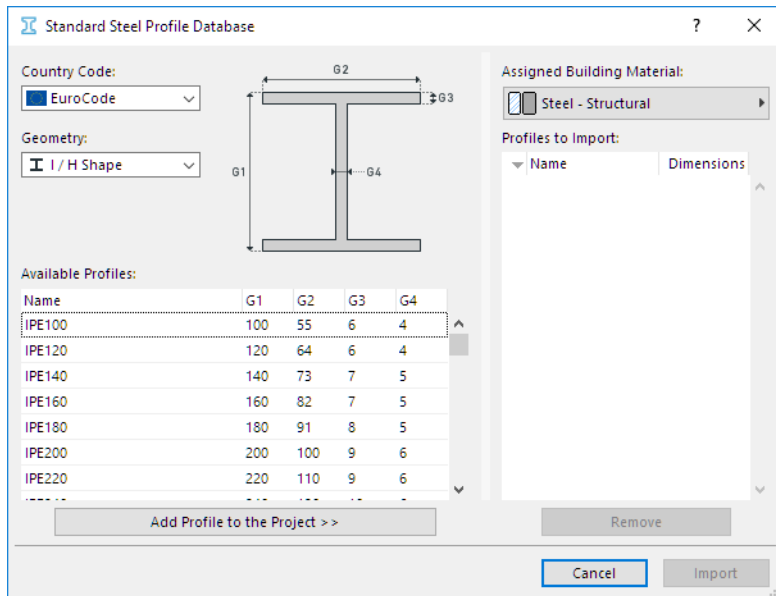
Use Standard Steel Column or Beam Profile

If you exchange data with structural softwares, it can be helpful to use standard elements as your profile Columns or Beams. During the data exchange process, those structural engineering programs will recognize and identify such standard elements accurately and completely, without requiring any settings adjustments.

Use the **Options > Complex Profiles > Import Standard Steel Profile** command.

This brings up the Standard Steel Profile Database.

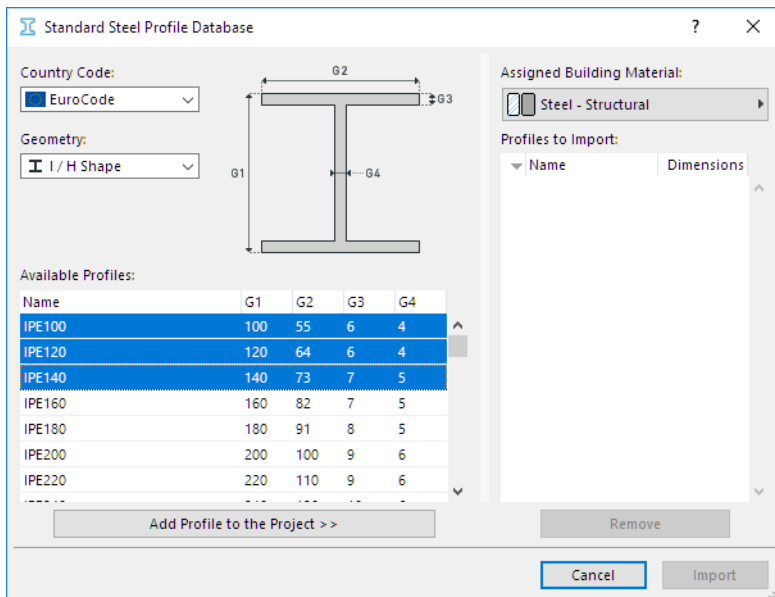
Profile Database



In this image, our list of Project Profiles (the list window on the right) is empty, so we must add selected profiles from the database.

From the non-editable Available Profiles list on the left, you can access thousands of standard profiles. (If needed, sort the list columns by Name or by specific dimension.) Use the **Country Code** and **Geometry** controls to help find the profiles you need.

Select these, then click **Add Profile to the Project**.



The selected structures now appear in the list to the right.

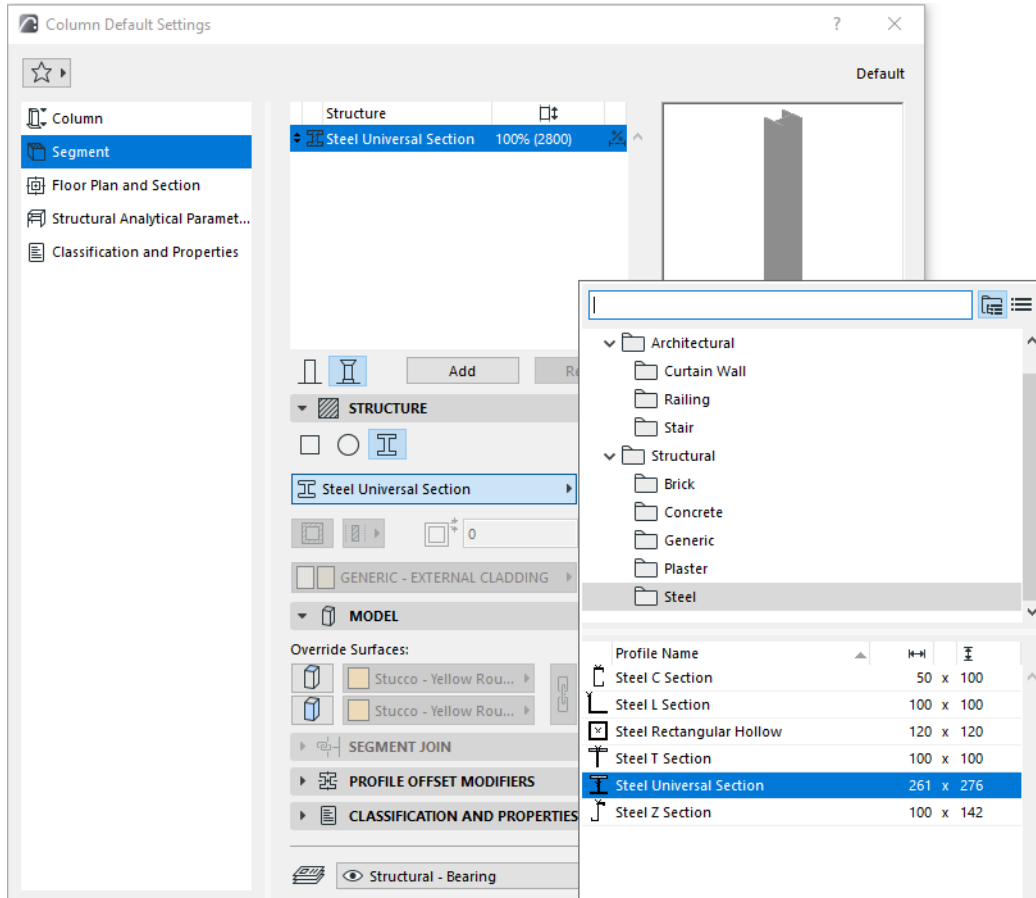
- From the pop-up at the top, assign a **Building Material** to the structures as needed.
- Click **Remove** to remove a project profile from the import list.

Click **Import**. The selected profiles are now available for use in your Archicad project.

Place Structural Profile Element

Choose the Complex geometry method in the Geometry and Positioning Panel of Column or Beam Settings.

In the Profile pop-up, choose the profile you need.



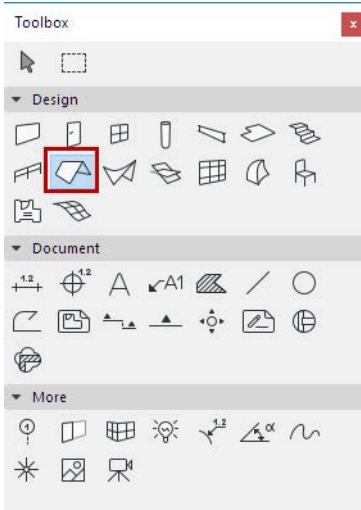
When placing the structural Column or Beam, note that its insertion point is the center of the object's bounding box. However, an additional hotspot is located at the element's center of gravity.



These predefined steel profile Beams and Columns can be listed by name in the Interactive Schedule: just include "Complex Profile" or "Building Material/Composite/Profile/Fill" in the Fields panel of Interactive Schedule Scheme Settings.

Roofs

Archicad's flexible Roofs can be used to create both standard and abstract 3D shapes meeting a wide variety of needs.



About Roofs

Composite Structures

Create a Single-plane Roof on the Floor Plan

Create a Single-plane Roof in the 3D Window

Create a Multi-plane Roof

Examples of Multi-plane Roofs

Editing Roofs

Crop Elements to Single-plane Roof

Create Special Roof Objects with RoofMaker

Trim Elements to Roof or Shell

Roof Tool Settings

Single-plane Roofs can be included in a Structural Analytical Model generated in Archicad.

For details, see [Structural Analytical Model](#).

About Roofs

Choose either the **Single-plane** or **Multi-plane** construction method to create a Roof element.

Like other construction elements, a Roof's essential structure derives from its Building Material, defined in Roof Tool Settings.

The **Multi-plane Roof** is a single element despite its multiple planes. When you edit one of the planes, the rest of the planes are automatically adjusted to the new geometry. You can edit each plane separately, changing its pitch or surface as needed. A Multi-plane Roof can have more than one level, yet such a Roof always remains a single element in Archicad even after you edit its components.

A Multi-plane Roof can be split up into Single-plane Roofs: select the Multi-plane Roof and use the **Edit > Reshape > Split into Single-plane Roofs** command or **Design > Roof Extras > Split into Single-plane Roofs**.

Connecting Roofs to Other Elements

For information on how intersections (junctions) work among construction elements, including Roofs:

[See *Element Intersections and Merge Elements: Roofs, Shells, Morphs*](#).

Roofs in Archicad (along with Shells) can serve as trimming elements: the **Connect** commands enable you to use Roofs to trim other construction elements, and then remain connected to those elements even if you edit them later.

The **Connect** commands let you trim several separate Roof elements to each other, combining them into a more complex structure.

[See *Trim Elements to Roof or Shell*](#).

Pivot Line

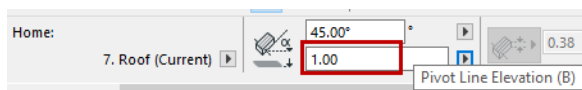
The roof's elevation is measured by the elevation of the **pivot line** or **pivot polygon**, a horizontal non-printing line that you draw when creating the roof. A Roof's pivot line or pivot polygon can be edited to graphically change Roof geometry, which is especially powerful with Multi-plane Roofs.

In the 3D window, pivot lines are shown with a specific color to make editing easier. You can change this color ("Reference lines in 3D") in **Options > Work Environment > Selection and Element Information**.

In most cases, you will want the pivot line of the roof to coincide with a Wall Reference Line or a Slab edge.

Note: You can hide all pivot lines on the Floor Plan, by disabling the Roof Pivot Lines option in **View > On-Screen View Options**.

The Pivot Line elevation is shown in the roof's info box.

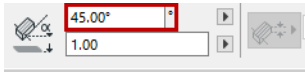


This is the same value as the Roof's Pivot Line Offset to Home Story, in Roof Settings.

Roof Pitch

The **Roof Pitch** is measured from the pivot line.

The Roof Pitch value is also shown in the Info Box.



See also [Modify the Roof Pitch](#).

Display of Roofs on Floor Plan

Use Roof Settings to define the parameters of your roof.

Use the Floor Plan Cut Plane, combined with element projection preferences, to determine which parts of the placed roof should be displayed.

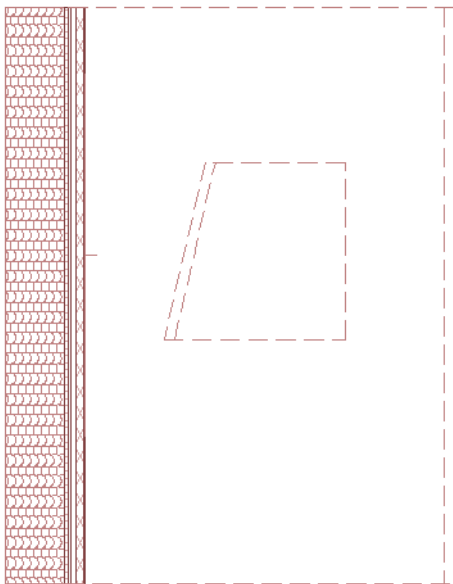
See [Floor Plan Cut Plane \(Global Setting\)](#).

For Roofs, the default Floor Plan Display option is “Projected with Overhead”.

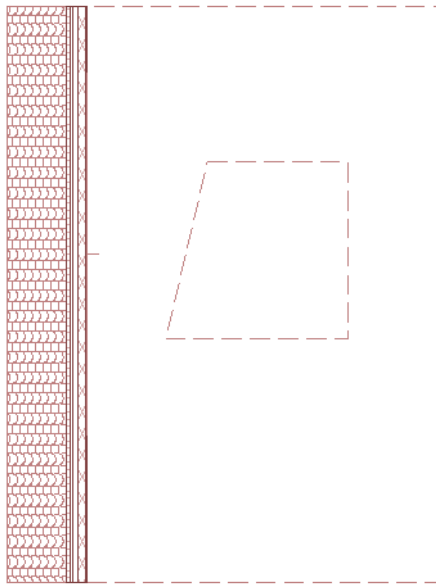
For more information on Floor Plan display of roofs, see [Element Display in Floor Plan and Section](#).

In Archicad 15 and later, all Roofs are displayed on the Floor Plan as a true 3D projection, showing all visible contours - including, for example, the edge of the hole in a pitched roof plane, as shown below.

If you prefer to display *Single-plane Roofs* as they were in Archicad 14 and earlier - that is, in Symbolic mode with their top surface only - you can switch this as a Project Preference: go to **Options > Project Preferences > Legacy**, and choose “Symbolic” as your preferred display mode. This affects Single-plane Roofs plus any Skylights they contain.



All Visible Contours



Top Surface Only

Gravitate New Elements onto a Roof

Use the Gravity control to place Wall, Column, Beam and Object type elements relative to the elevation of a Roof.

See [Gravity](#).

Create a Composite Roof

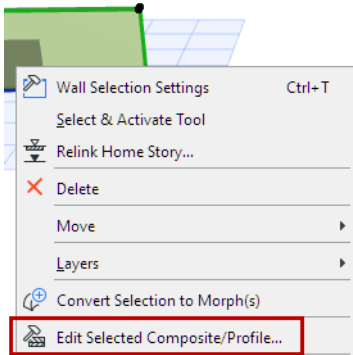
You can apply or customize a composite structure for a Roof.

For more information, see [Composite Structures](#).

(Remember that the model's Partial Structure Display settings will affect the display of composite Roofs.)

See [Partial Structure Display](#).

You can edit a selected composite by accessing its settings from its context menu:



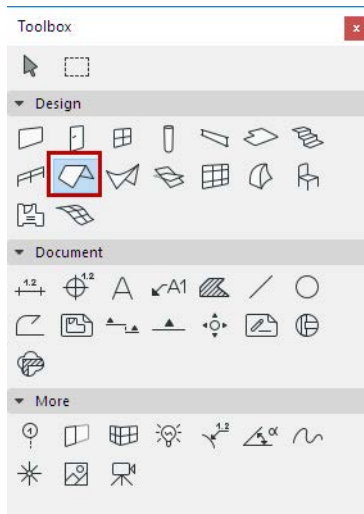
If the structure of the Roof is a composite, then the Roof's thickness is defined in **Options > Element Attributes > Composites**, and equals the sum of the skins' thicknesses.

For a composite Roof, the "top" surface is the first listed skin of the composite, in Composite Settings.

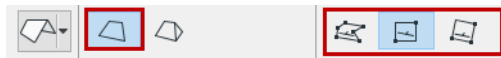
Create a Single-plane Roof on the Floor Plan

Note: Apart from drawing a new Roof as described below, you can also select a Multi-plane Roof and split it into several Single-plane Roofs, using **Edit > Reshape > Split into Single-plane Roofs** or **Design > Roof Extras > Split into Single-plane Roofs**.

1. Select the Roof Tool.



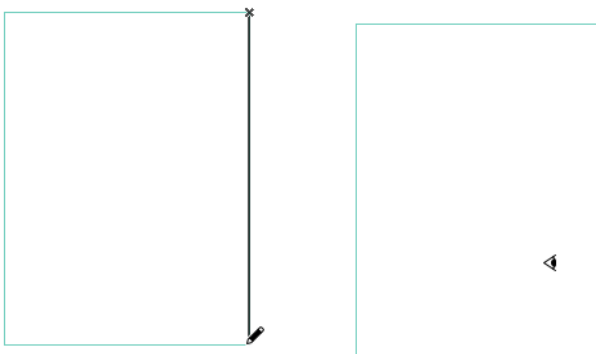
2. In Roof Settings or the Info Box, adjust the main roof parameters (e.g. pitch, surfaces).
3. Choose the Single-plane construction method from the Info Box or from Roof Settings.
4. Choose one of the three geometry methods in the Info Box (Polygon, Rectangle, Rotated Rectangle)



5. On the Floor Plan, click twice to define the Roof's **pivot line**.

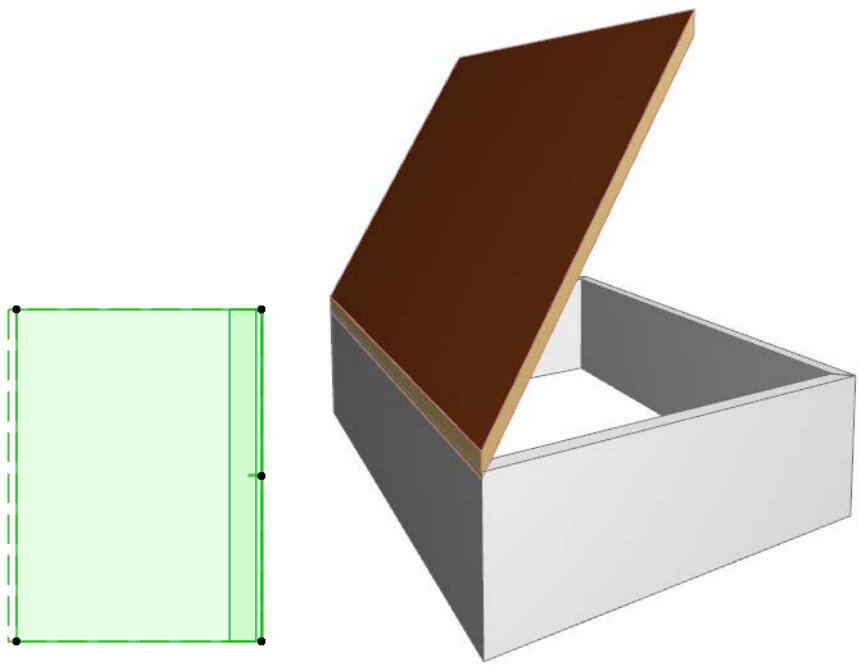
In most cases, you will want the pivot line of the roof to coincide with a wall Reference Line or a slab edge.

6. The eyeball cursor appears. Click on the side of the pivot line where you want the roof plane to rise (or, in the case of a negative roof pitch, slope downward).



7. Draw the contour of the roof by clicking on each of the new roof's nodes. Or use the Magic Wand: click on an existing element to serve as the pattern for the roof's contour.

See also [Magic Wand](#).

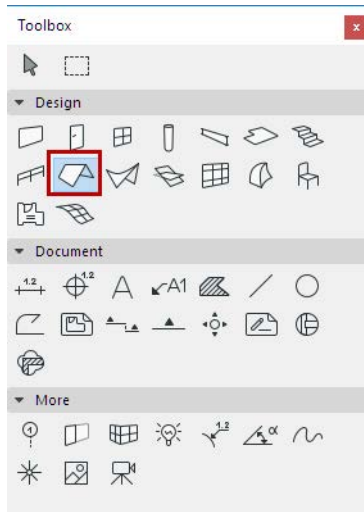


Create a Single-plane Roof in the 3D Window

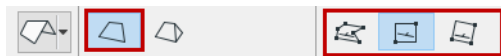
Note: Apart from drawing a new Roof as described below, you can also select a Multi-plane Roof and split it into several Single-plane Roofs, using **Edit > Reshape > Split into Single-plane Roofs** or **Design > Roof Extras > Split into Single-plane Roofs**.

Use this method if you want to place a roof that snaps to existing elements in the 3D window.

1. Select the Roof Tool.

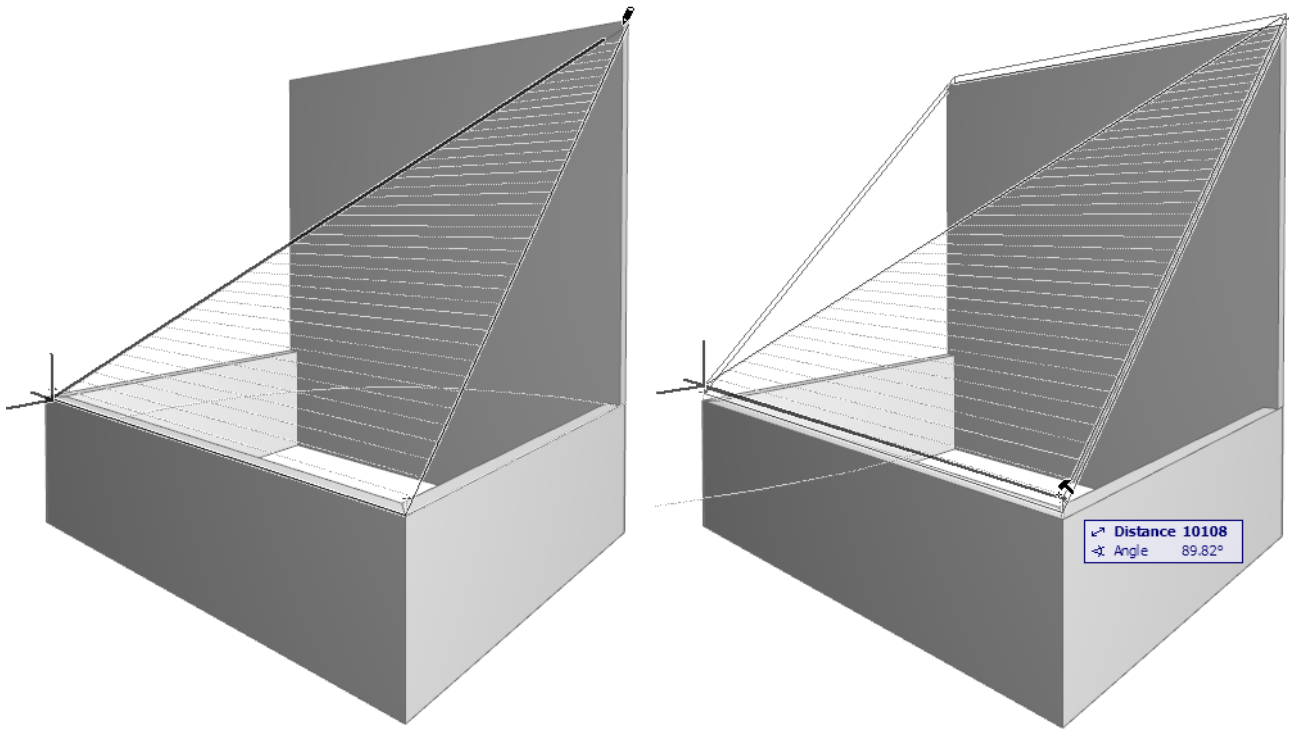


2. In Roof Settings or the Info Box, adjust the main roof parameters (e.g. pitch, surfaces, eaves overhang)
3. Choose the Single-plane Construction method from the Info Box.
4. Choose one of the three Geometry Methods in the **Info Box** (Polygon, Rectangle, or Rotated Rectangle).

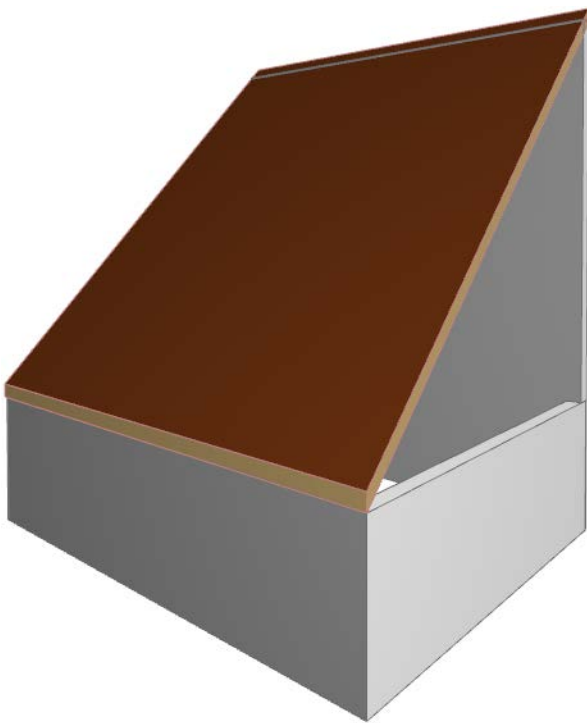


5. In the 3D window, define the roof plane by clicking on three points. The first two points constitute the pivot line.

6. Draw the contour of the roof. The Hammer cursor appears when you return to the first node. Click to close the polygon.



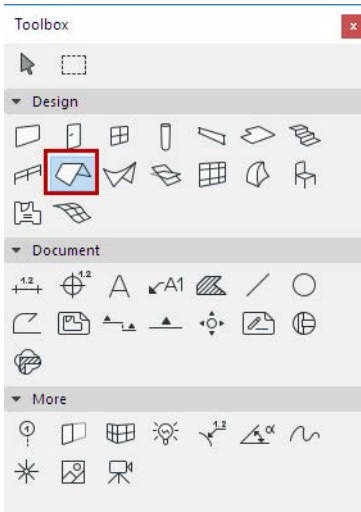
7. Archicad will generate the Roof.



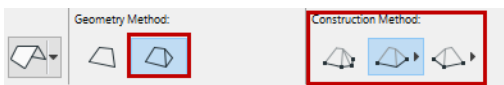
Create a Multi-plane Roof

On the Floor Plan or 3D Window:

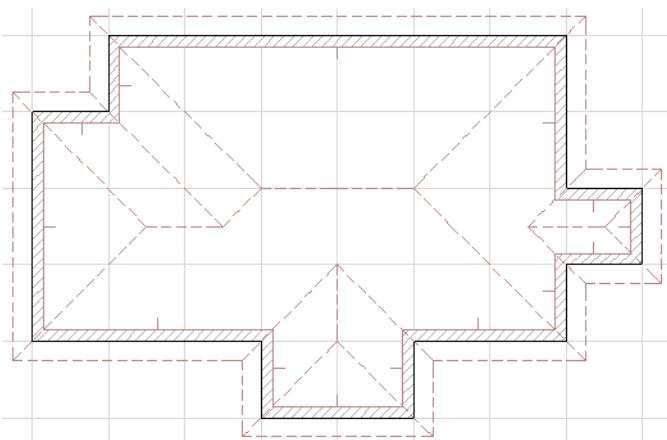
1. Select the Roof Tool.



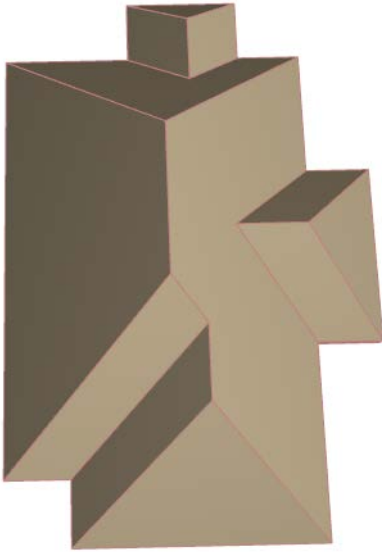
2. In Roof Settings or the Info Box, adjust the main roof parameters (e.g. pitch, surfaces, eaves overhang).
3. Choose the Multi-plane method from the Info Box or from Roof Settings.
4. Choose one of the construction Methods from the **Info Box**:
 - Complex
 - Hip
 - Gable
 - Rotated Hip
 - Rotated Gable



5. Start drawing the pivot polygon of the Multi-plane roof - usually, along the outline of your building structure.
6. Close the polygon to complete the Multi-plane roof.



7. Alternatively, use the Magic Wand (space+click) on an existing Wall to instantly place the pivot polygon and the resulting Multi-plane roof.



The result is a single roof element, consisting of multiple planes. You can further edit the roof plane by plane (for example, assign a custom surface or pitch to any plane), but the roof always remains a single element.

[See *Customize Roof Plane*.](#)

Even after the roof is created, you can easily change the roof geometry by graphically editing the pivot polygon or the roof ridge.

[See *Graphical Editing of Roof Geometry*.](#)

You can divide a Multi-plane Roof structure into multiple horizontal levels or segments, e.g. to make a mansard Roof.

[See *Add Roof Level*.](#)

Examples of Multi-plane Roofs

The following sections show you how to create the following roof types, step by step:

[Hip Roof](#)

[Complex Roof](#)

[Gable Roof](#)

[Pyramidal Roof](#)

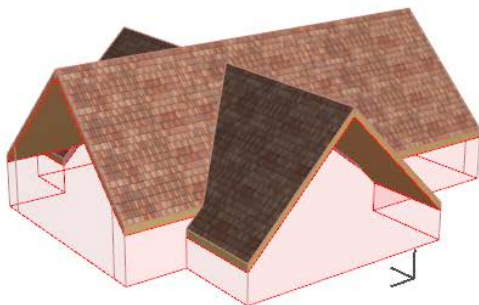
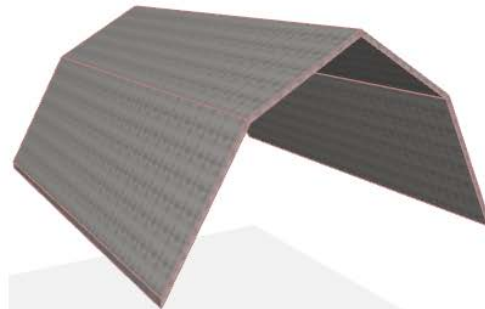
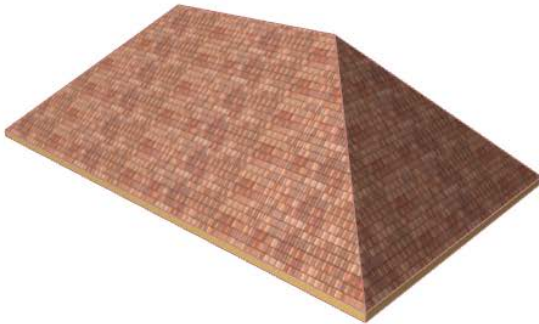
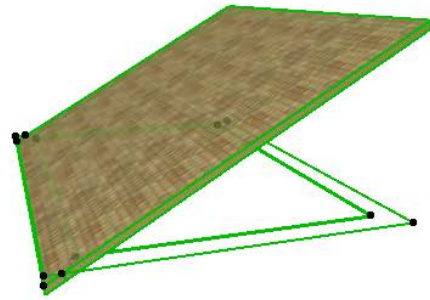
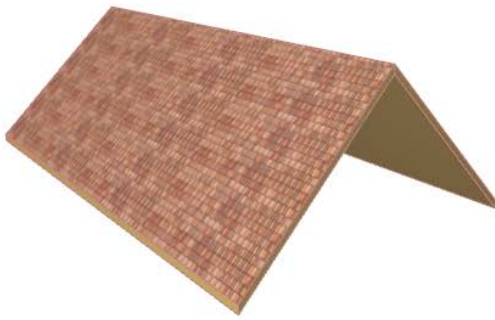
[Mansard Roof](#)

[Gablet Roof](#)

[Gambrel Roof](#)

[Add a Tower to the Roof](#)

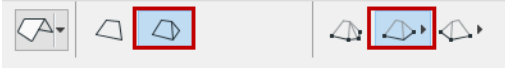
[Cross-Gable Roof](#)



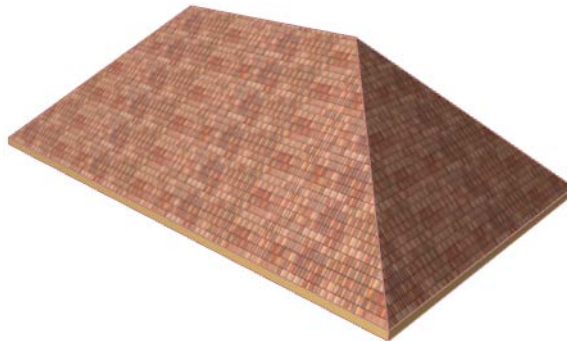
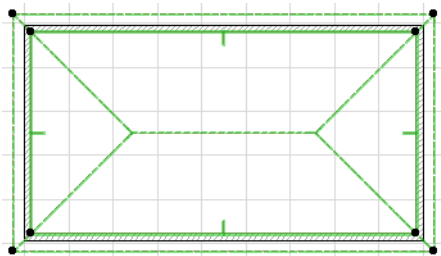
[Watch the Video](#)

Hip Roof

1. Draw the Floor Plan outline of your building (i.e. external walls).
2. Activate the Roof Tool, and use the Multi-plane Construction method and the Hip geometry method.

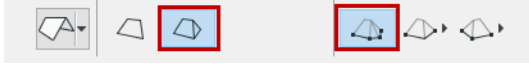


3. In Roof Settings, adjust the main roof parameters (e.g. pitch, eaves overhang)
4. Click twice to place the rectangle-shaped pivot line of the Roof, or use the Magic Wand (Space+click) on the edge of the Wall outline to quickly create a Multi-plane roof.
5. The created Roof is a single, Multi-plane roof element.

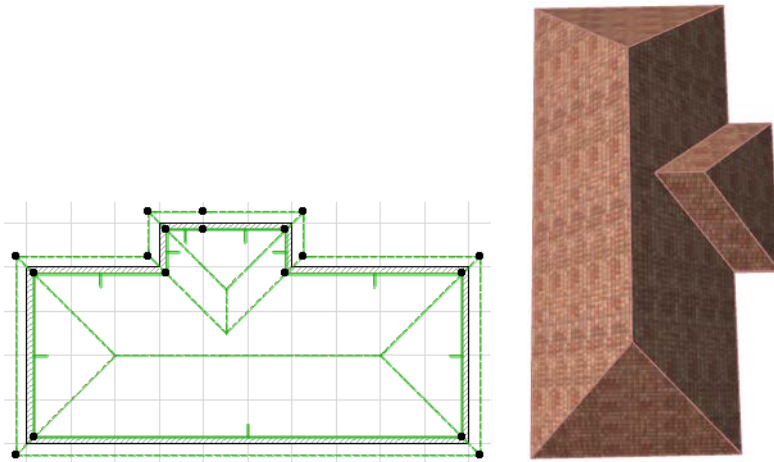


Complex Roof

1. Draw the Floor Plan outline of your building (i.e. external walls).
2. Activate the Roof Tool, and use the Multi-plane construction method and the Complex geometry method.

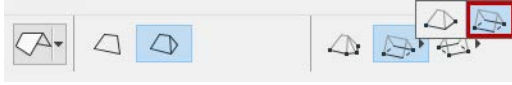


3. In Roof Settings, adjust the main roof parameters (e.g. pitch, eaves overhang)
4. Trace the outline the building; or use the Magic Wand (Space+click) on the edge of the Wall outline to quickly create a Multi-plane roof.
5. The created roof is a single, Multi-plane roof element.

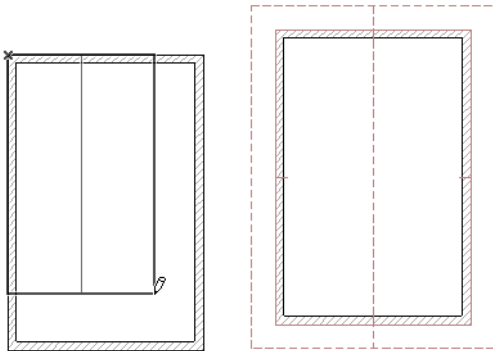


Gable Roof

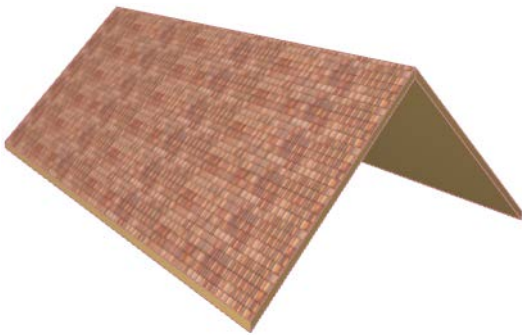
1. Draw the Floor Plan outline of your building (i.e. external walls).
2. Activate the Roof Tool, and use the Multi-plane construction method and the rectangular Gable geometry method (this is found in the pop-up next to the Hip geometry method).



3. In Roof Settings, adjust the main roof parameters (e.g. pitch, eaves overhang)
4. Click twice to place the rectangle-shaped pivot line of the Roof.



5. The created Gable roof is a single, Multi-plane roof element.



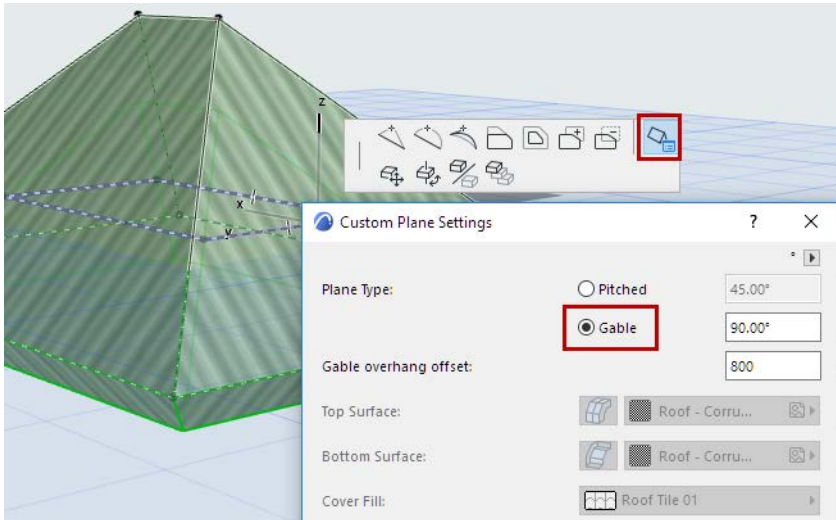
[Watch the Video](#)

Change a Roof Plane into a Gable

In the 3D window:

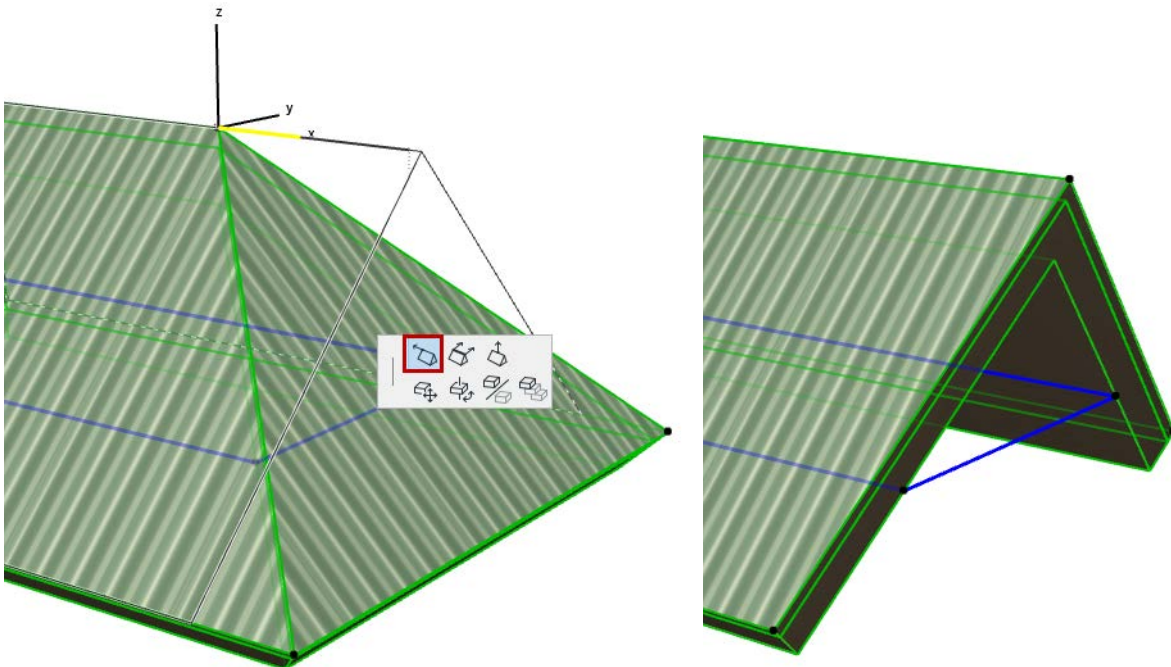
1. Select the Multi-plane roof.
2. Find the pivot line on the roof plane that you want to change into a gable.
3. Click to bring up the Pet Palette, and choose the Custom Plane Settings option.

4. In the appearing dialog box, choose **Gable** to define the selected plane as a gable.



Another method:

1. In the 3D window, select the endpoint of the roof ridge.
2. From the Pet Palette, use the **Stretch Horizontal Ridge** option to drag the ridge endpoint until the plane on that end becomes vertical. (The cursor will snap to the vertical position to make this easy.)

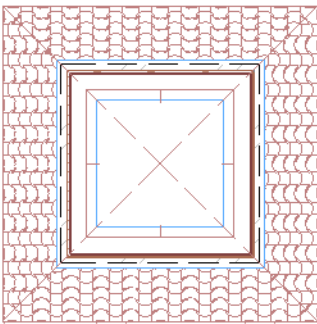
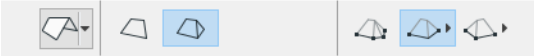


Pyramidal Roof

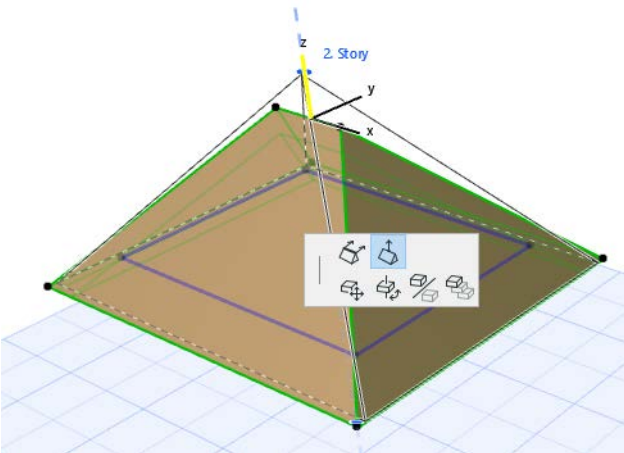


Try two ways of making a pyramidal roof:

- On the Floor Plan, use the Roof Tool - Multi-plane construction method, with the Rectangular Hip geometry method, to draw a square roof outline.



- Another way is to select the ridge of an ordinary hip roof in the 3D Window. Use the “**Stretch Ridge vertically**” command from the pet palette, and move the cursor up until the ridge endpoints meet in the middle.

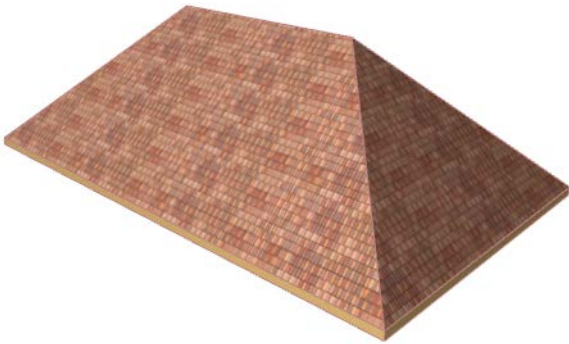


Mansard Roof

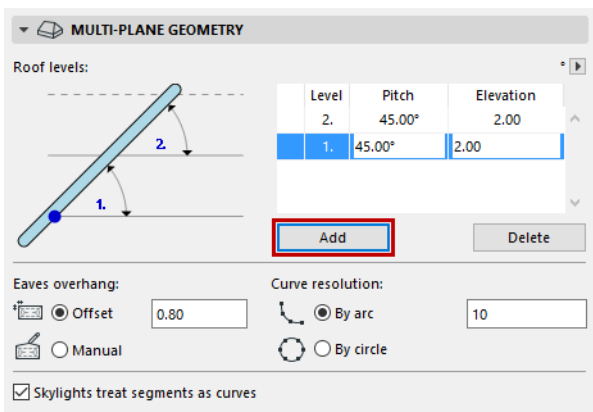


To create a Mansard Roof, you want a Multi-plane Roof that has two different levels to accommodate two different pitches.

1. Begin with a regular Hip Roof.

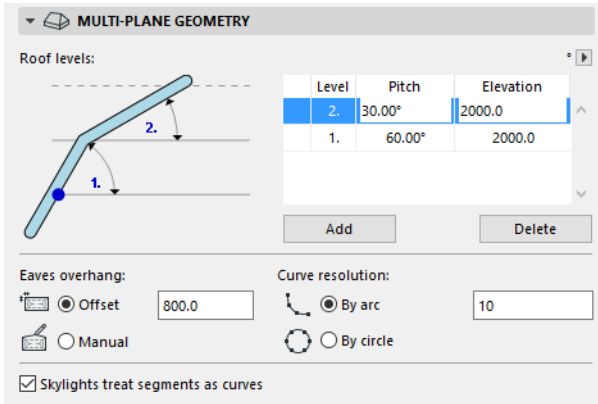


2. Select the Roof.
3. In the Multi-plane Geometry panel of Roof Settings, click **Add** to add a new Roof level.



4. Adjust the pitches of each level. In this example, the first, steeper level is 60 degrees. The top, flatter level is 30 degrees.

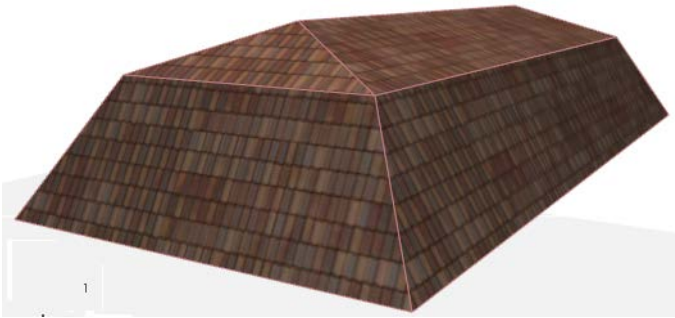
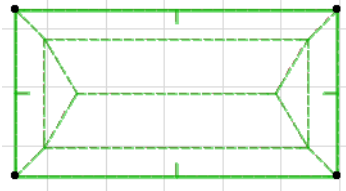
5. Adjust the elevation of Level 1 so that it ends (and Level 2 begins) at 2000 mm.



6. In the Model panel of Roof Settings, adjust the surfaces as desired.

7. Change any other Roof Settings as needed. In our example, we have set the eaves overhang offset to 0.

8. Click OK to close Roof Settings. The Mansard Roof is complete.



As needed, define custom attributes for the individual planes of the mansard roof by changing the parameters in the Custom Plane Settings, available from the pet palette during editing.

[See *Customize Roof Plane*.](#)

For example, you can gable a roof plane at each end of the upper Roof level to create a Gablet roof.

[See *Gablet Roof*.](#)

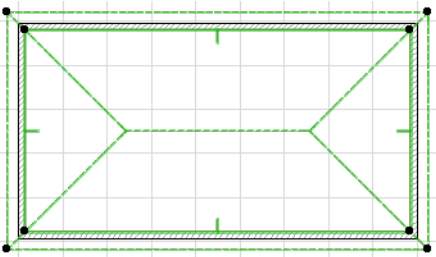
You can further edit the geometry of Roof levels.

[See *Edit Roof Levels*.](#)

Gable Roof



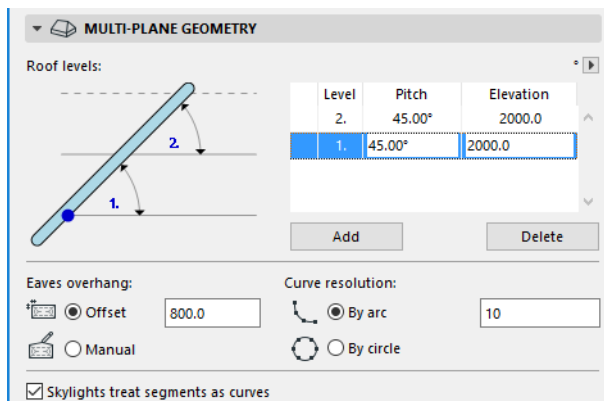
1. Begin with a regular hip roof.



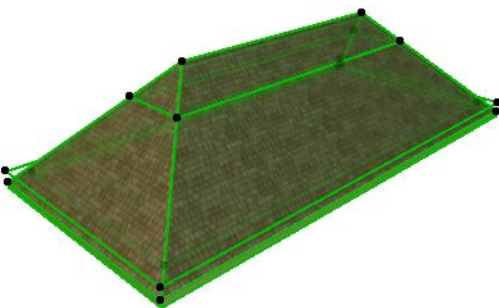
To see how to make a hip roof, see [Hip Roof](#).

2. Make this roof into a two-level roof: open Roof Settings and go to the Multi-plane Geometry tab page.
 - Click **Add** to add a second level.
 - Enter the elevation at which the first level ends (and the second level begins). The pitch can be the same for both levels.

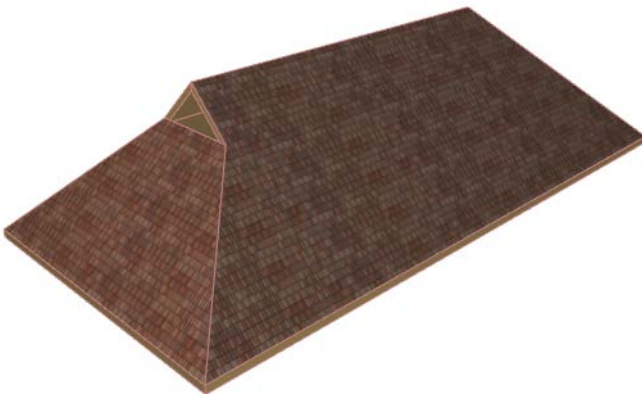
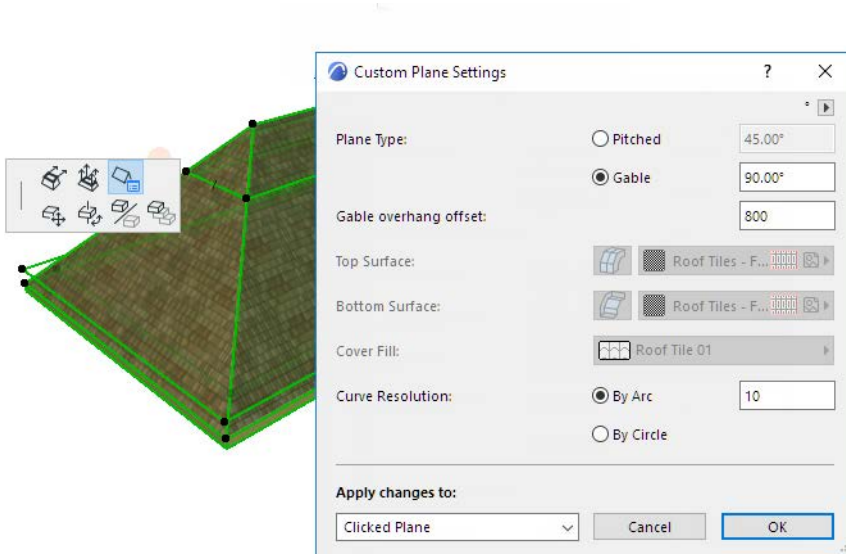
Click OK to close Roof Settings.



3. View the two-level Roof in 3D. Since the pitch is uniform for both levels, you must select it to see where the second level begins.



4. Gable each end of the second level.



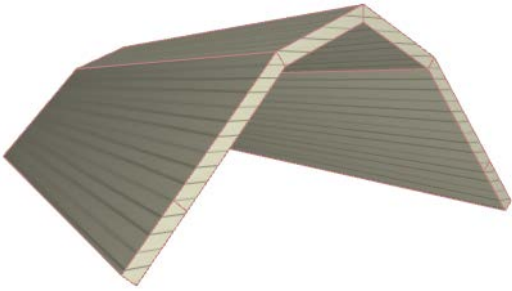
[See *Change a Roof Plane into a Gable*.](#)

5. View the final Roof.

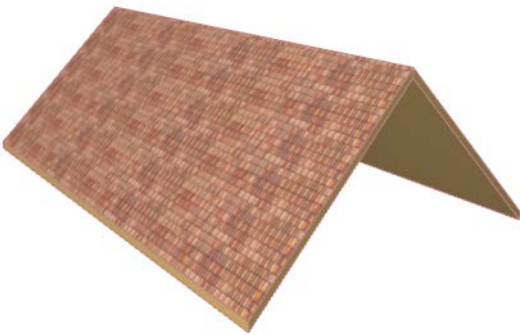


Gambrel Roof

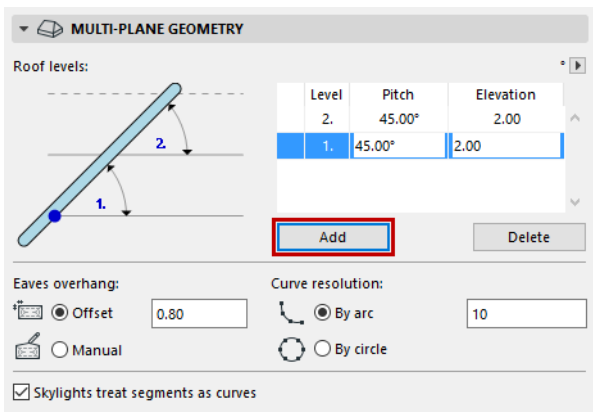
The Gambrel Roof is like the Mansard, except that it is gabled at both ends.



1. Begin with a Gable Roof.

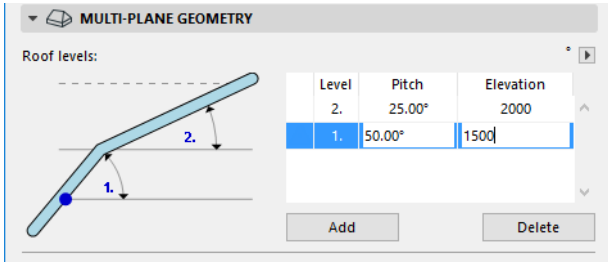


2. Select the Roof.
3. In the Multi-plane Geometry panel of Roof Settings, click **Add** to add a new Roof level.

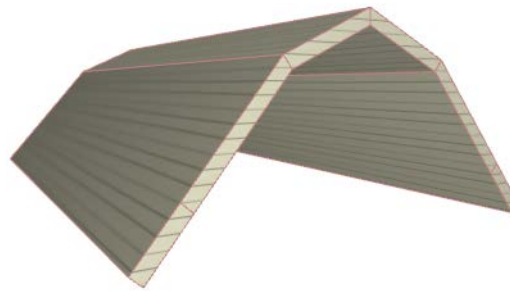


4. Adjust the pitches of each level. The first, steeper level should be 50 degrees. The top, flatter level should be 25 degrees.

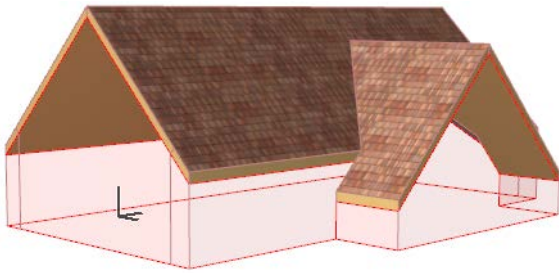
- Adjust the elevation of Level 1 so that it ends (and Level 2 begins) at 1500 mm.



- In the Model panel of Roof Settings, adjust the surfaces as desired.
- Click OK to close Roof Settings. The Gambrel Roof is complete. The additional roof level is indicated by a dotted line on the Floor Plan.

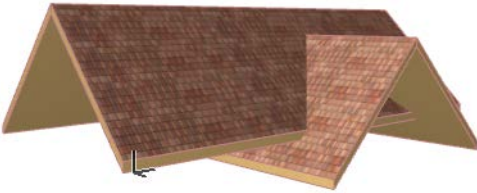


Cross-Gable Roof



Create this by trimming two separate gable roofs to each other.

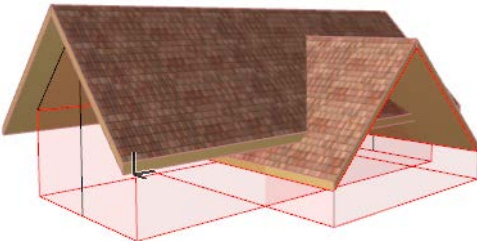
1. Start by creating two roofs gabled at each end, and placed atop each other, like this:



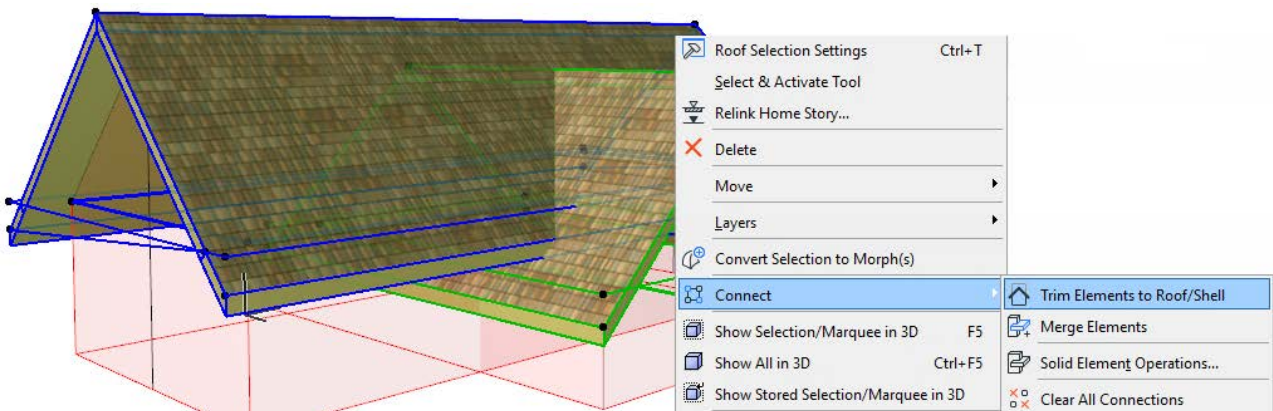
To see how to make a gable roof, see [Gable Roof](#).

You can adjust the pivot line elevations of each roof as needed to achieve the correct geometry.

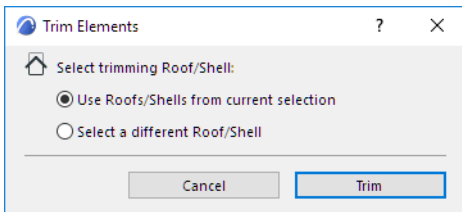
2. Before doing the trim, make the trimming bodies visible at **View > On-Screen View Options > Trimming Bodies**. This shows you how the roofs will cut each other.



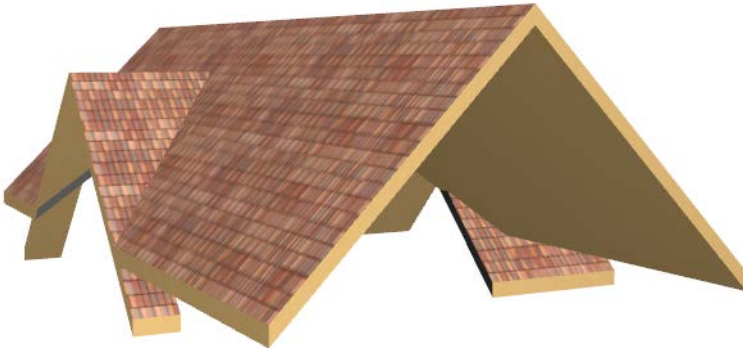
3. Select both roofs. From the context menu, choose **Connect**, then **Trim Elements to Roof/Shell**.



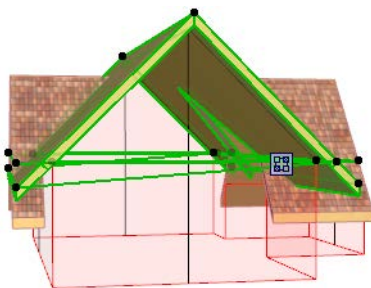
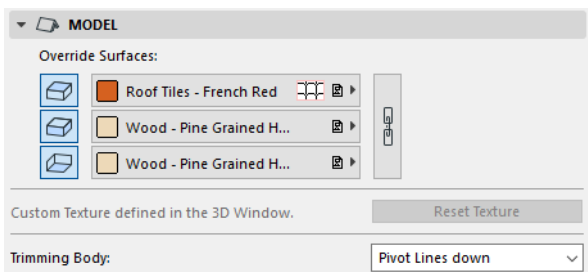
- Since you want both selected Roofs to do the trimming, leave the option as-is in the appearing dialog box and click **Trim**.



- View the result. We see that some bits have been left out of the trim.

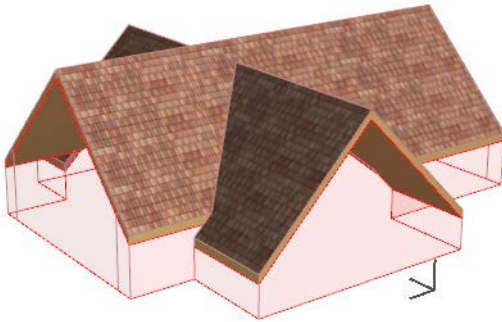
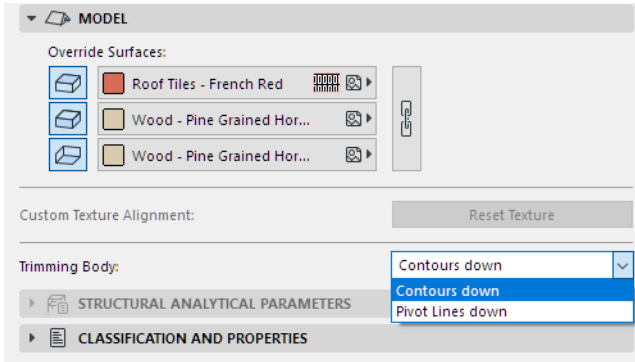


The reason is that the trimming body for both Roofs is the part inside of the pivot line and going downward: this is defined in the Model panel of the Roof Settings dialog box:

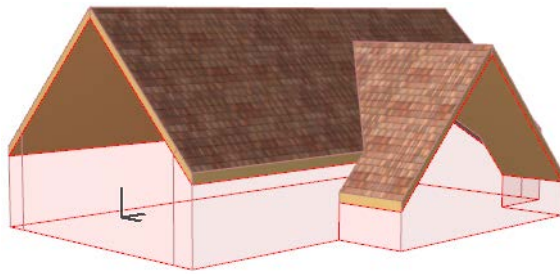


For this reason, any parts of the trimmed Roofs that fall outside of that trimming body are not trimmed.

6. To fix this, just change the trimming body definition of both Roofs to “Contours Down.” See how the trimming geometry changes.



You do not have to re-do the **Trim** command; the trimming relationship is updated automatically.



See also [Trim Multiple Selected Elements: Automatic Trim](#).

Editing Roofs

The following sections describe specific roof-related editing techniques.

[Graphical Editing of Roof Geometry](#)

[Modify the Roof Pitch](#)

[Add Roof Level](#)

[Edit Eaves or Gable Overhang](#)

[Edit Roof Contour](#)

[Edit Roof Ridge](#)

[Customize Roof Plane](#)

[Customize Edge of Roof or Roof Hole](#)

[Create a Hole in a Roof](#)

[Create an Atrium](#)

[Add a Tower to the Roof](#)

[Create Roof Level Lines](#)

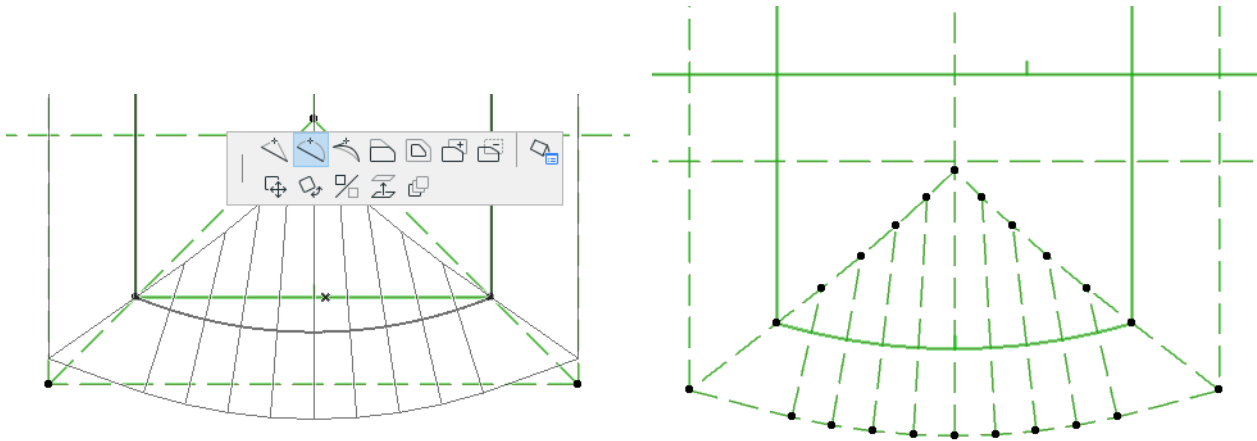
[Intersect Single-plane Roofs](#)

Graphical Editing of Roof Geometry

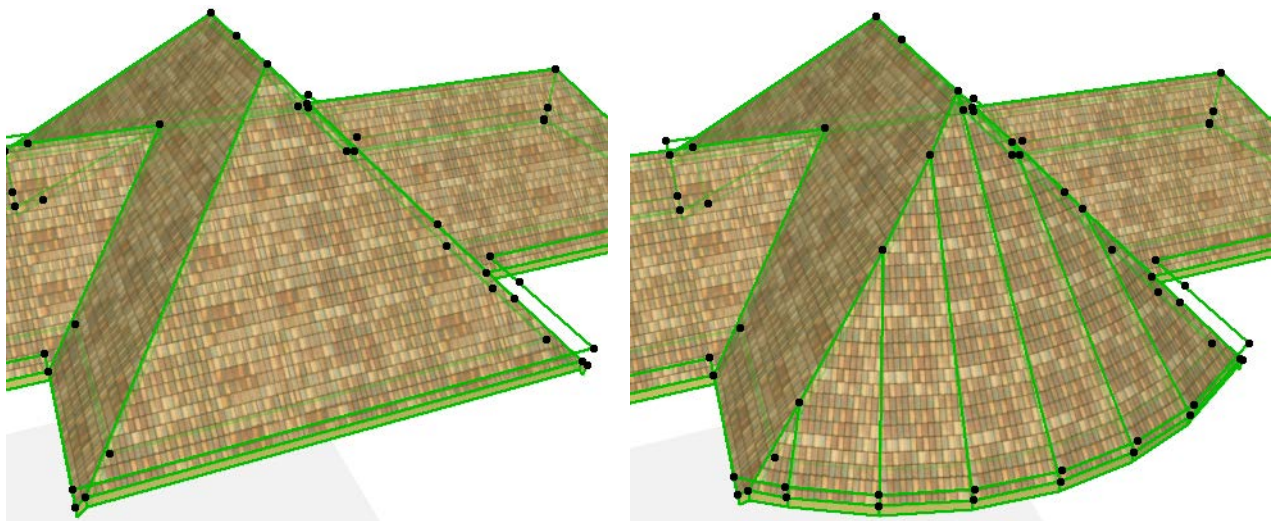
To graphically edit the geometry of a selected Roof, edit any of the following:

- the pivot line
- a ridge line
- the Roof level dividing line, if your Roof has several levels
- the Contour line

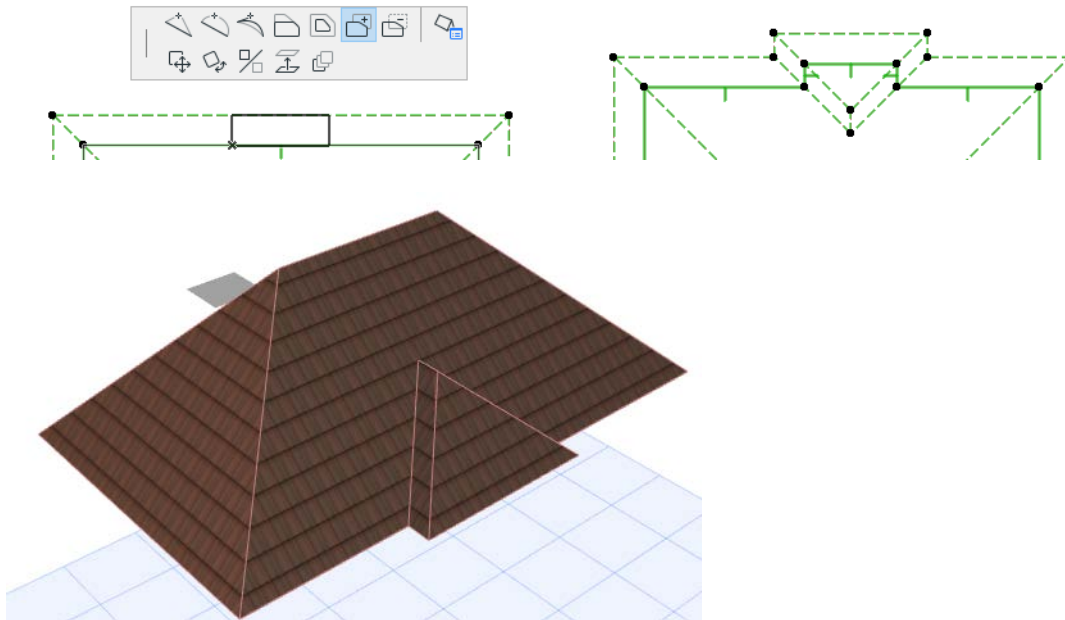
To edit the pivot line, use the familiar polygon editing commands in the pet palette:



As you can see in 3D, editing the pivot line adjusts the geometry of the roof plane and of the adjoining planes as needed:



Below, we have added an extrusion to the roof using the **Add to Polygon** editing command of the pet palette. The resulting roof geometry is modified, with each roof plane adjusted as needed, and the roof remains a coherent single element.



[Watch the Video](#)

Related Topics

[Edit Roof Ridge](#)

[Edit Roof Levels](#)

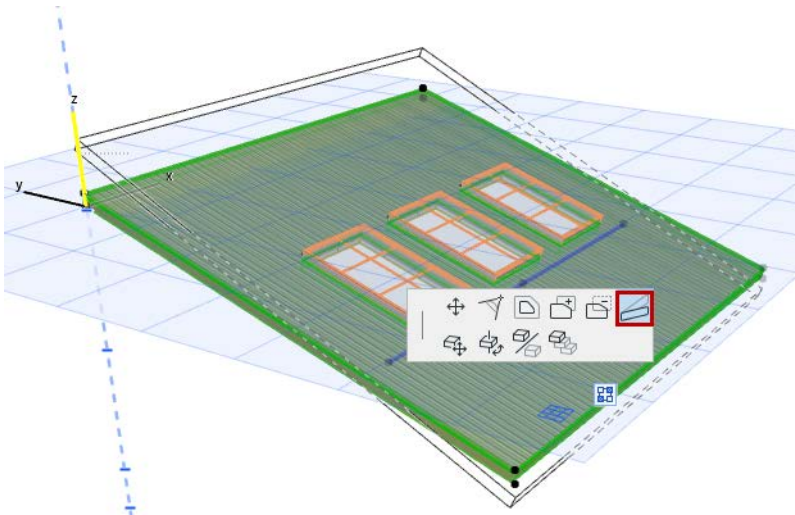
[Skylight Constraint Relative to Roof](#)

Modify the Roof Pitch

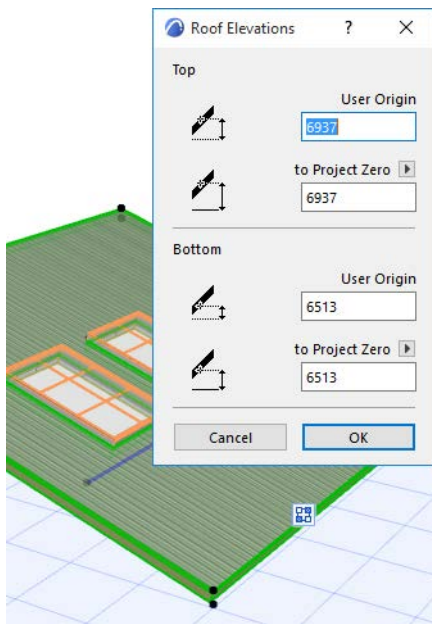
For a Single-plane Roof:

Once you've drawn a roof and selected it, you can modify its pitch in any one of the following ways:

- Change the roof pitch in the Info Box or in Roof Settings.
- In the 3D window, select the roof, click on any point that is not on the pivot line, then select the **Change Roof Pitch** command from the pet palette and graphically adjust the roof angle (or enter a Pitch value into the Tracker.) The roof will rotate around the pivot line.



- In the Floor Plan or 3D Window, select the roof, then **Ctrl+click (Mac: Cmd+Click)** at any point on the roof plane to bring up the **Roof Elevations** dialog box. Enter a numerical value in the top field: this value will be the elevation value of the roof plane at the point at which you clicked it. Altering this elevation value has the same effect as modifying the roof pitch.



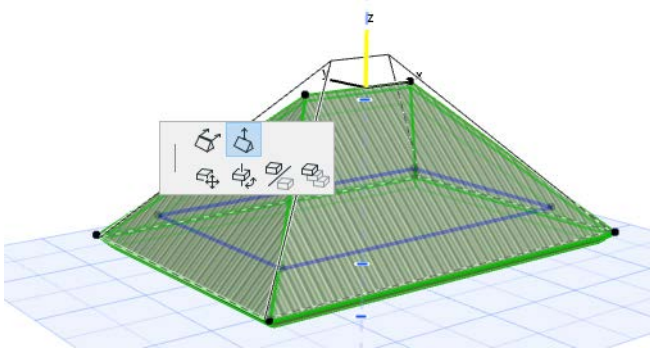
For a Multi-plane Roof:

Modify the roof pitch in one of the following ways:

- Change the roof pitch in the Info Box or in Roof Settings.
- Use the Custom Plane Settings dialog box to set a custom pitch for any selected plane.

See [Customize Roof Plane](#).

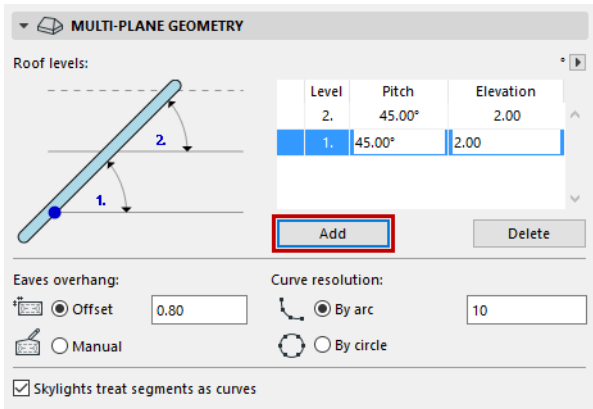
- Use the Pet Palette editing commands (e.g. Elevate Horizontal Ridge) to alter the geometry of the selected plane(s), including the Pitch.



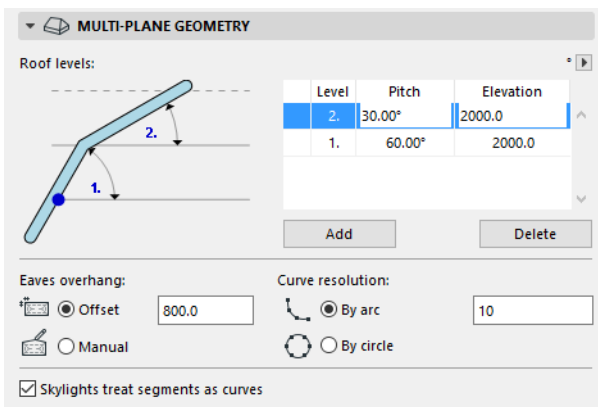
Add Roof Level

You may wish to add additional levels to your Multi-plane Roof: for example, in creating a mansard roof, you want two roof levels, each with a different pitch.

1. Select the Roof.
2. In the Multi-plane Geometry panel of Roof Settings, or from the Info Box, click **Add** to add a new Roof level.



3. Adjust the pitches of each level.
4. Adjust the elevation of any roof level. For example, the elevation value for Level 1 indicates the elevation at which Level 1 ends (and Level 2 begins).

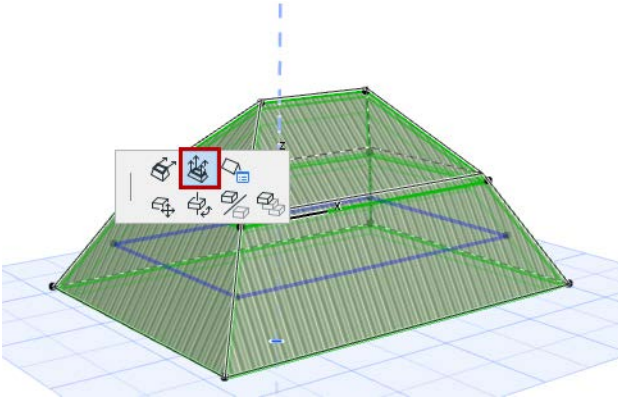


For examples, see [Mansard Roof](#) and [Gambrel Roof](#).

Edit Roof Levels

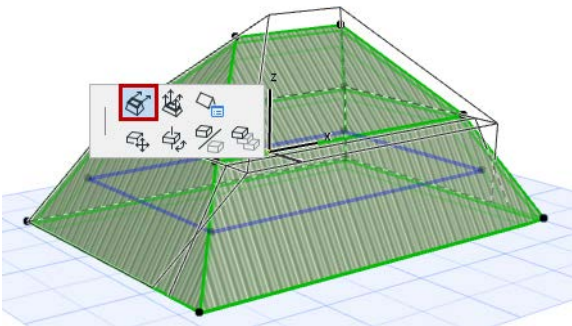
For a multi-level Roof, select the line indicating the Roof level, then use pet palette commands to edit the vertical position or geometry of the Roof level.

Edit Level Height

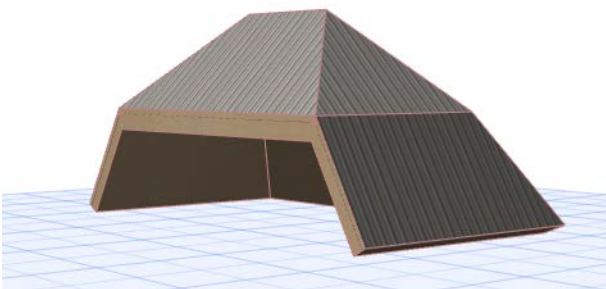


Drag Level Horizontally

Choose the **Drag Level Horizontally** command from the pet palette to move the level horizontally.



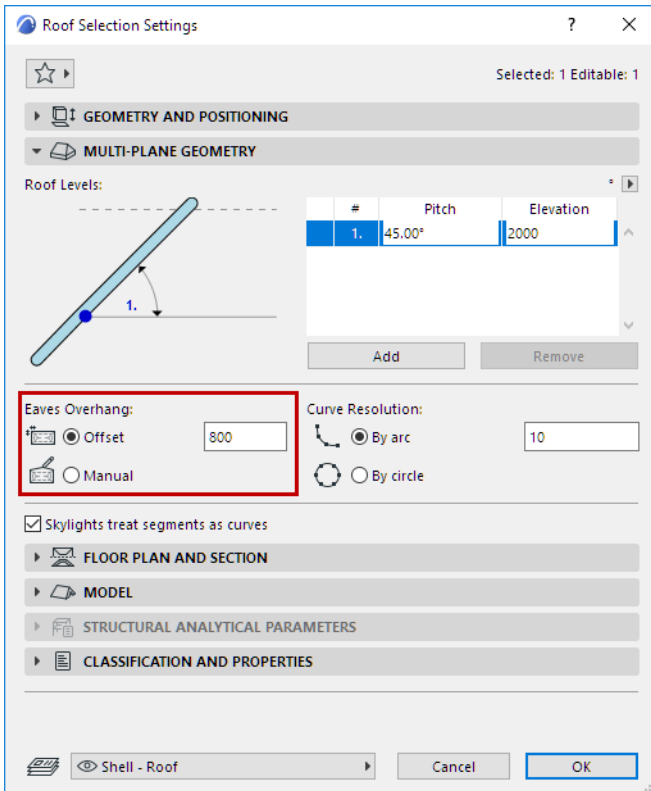
If you drag the level all the way to the edge of the Roof, that edge is gabled:



Edit Eaves or Gable Overhang

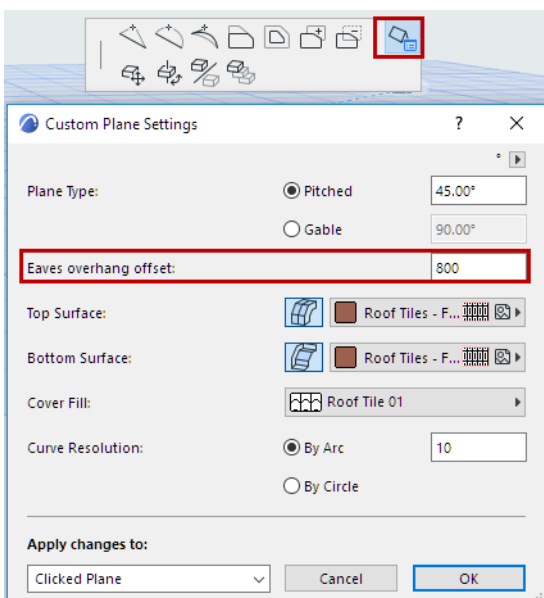
The offset of the eaves (or the gable) from the pivot line can be set:

- For the selected Roof as a whole in Roof Settings:



- For any single panel of a Multi-plane Roof:

Click on the pivot line of the panel whose eaves overhang you wish to customize. Use the Customize Roof Plane dialog box to enter an offset for that roof panel only:

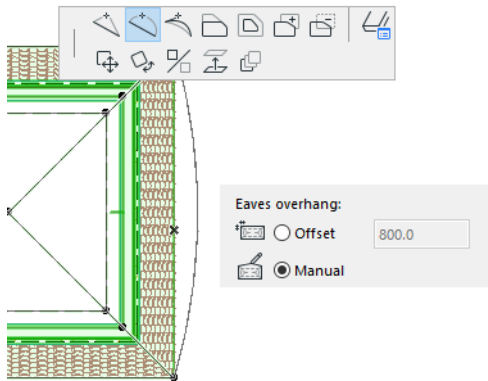


If the selected Roof plane is gabled, the control is called **Gable Overhang Offset**.

Notice that the value in the Eaves Overhang control in Roof Settings has not changed - it still reflects the overhang value set for the roof as a whole. However, the yellow patch indicates that at least one roof panel now has a different, customized overhang value.

If you change the shape of the Roof contour by editing it manually, the Eaves overhang control changes to "Manual".

See [Edit Roof Contour](#).

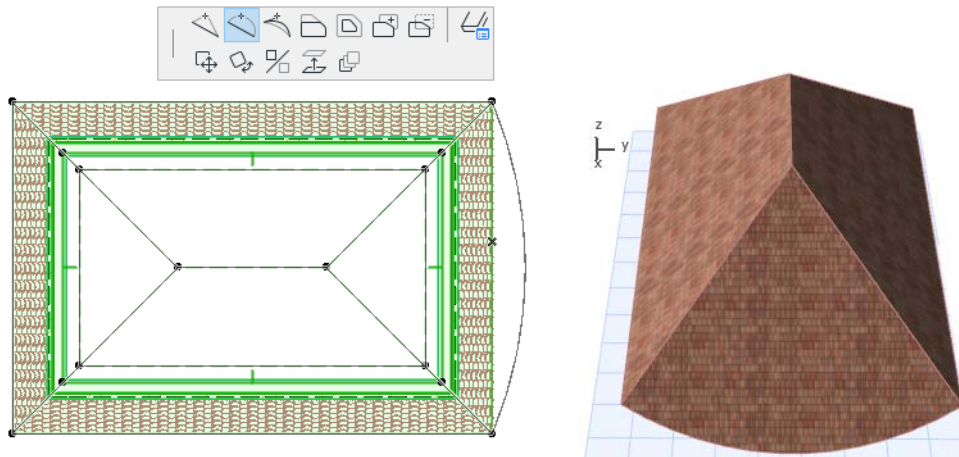


Edit Roof Contour

The Roof contour may be edited graphically. Select the contour line in Floor Plan or 3D and use the available pet palette commands.

Editing the contour (as opposed to editing the pivot line - does not change the overall geometry of the roof, just the shape of the roof's edge.

See [Graphical Editing of Roof Geometry](#).

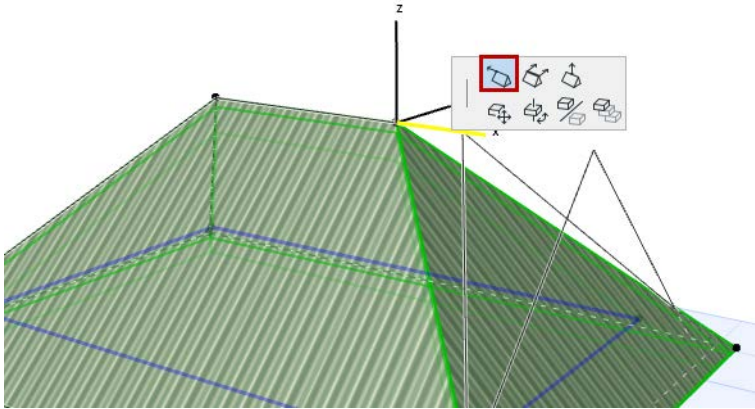


Edit Roof Ridge

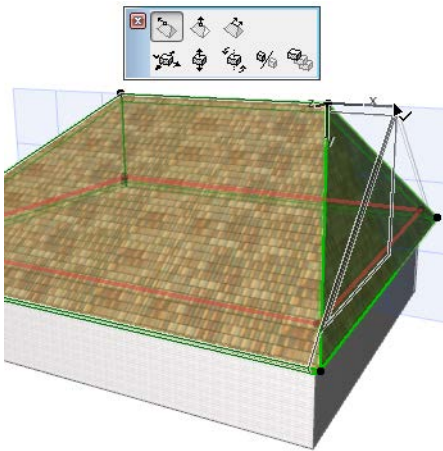
You can modify the roof geometry of a Multi-plane roof by grabbing the ridge line at either endpoint or anywhere along the ridge line.

The pet palette offers three ways to edit the roof:

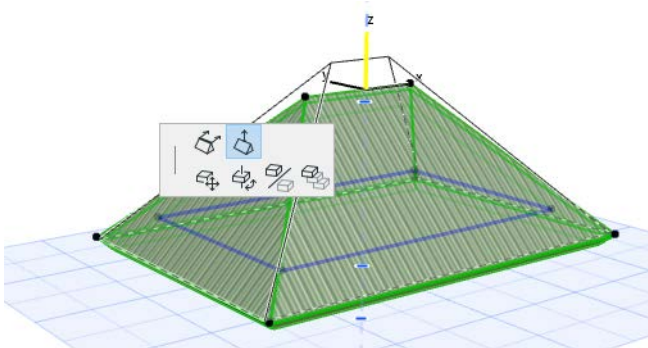
Stretch Horizontal Ridge



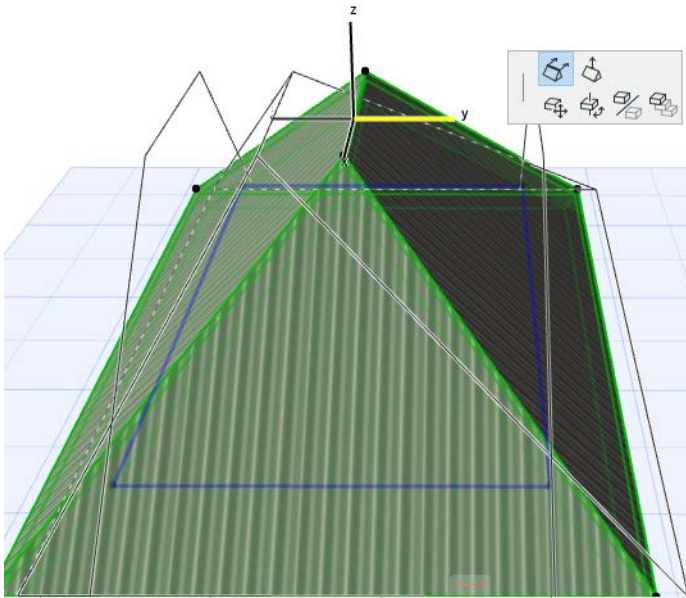
This option has a built-in “snap” when you get to the vertical point. If you click then, you will end up with a gabled roof plane.



Elevate Horizontal Ridge



Drag Horizontal Ridge



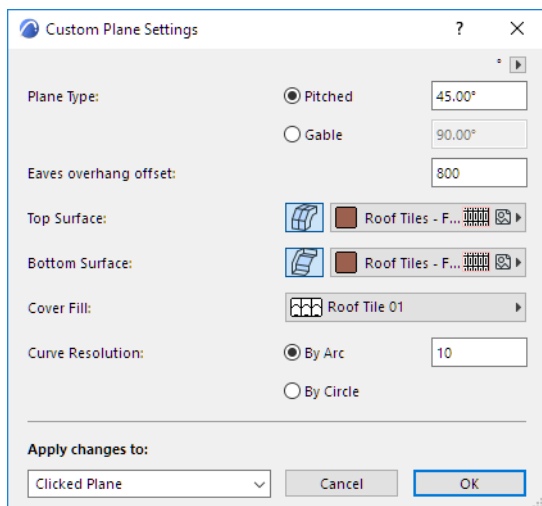
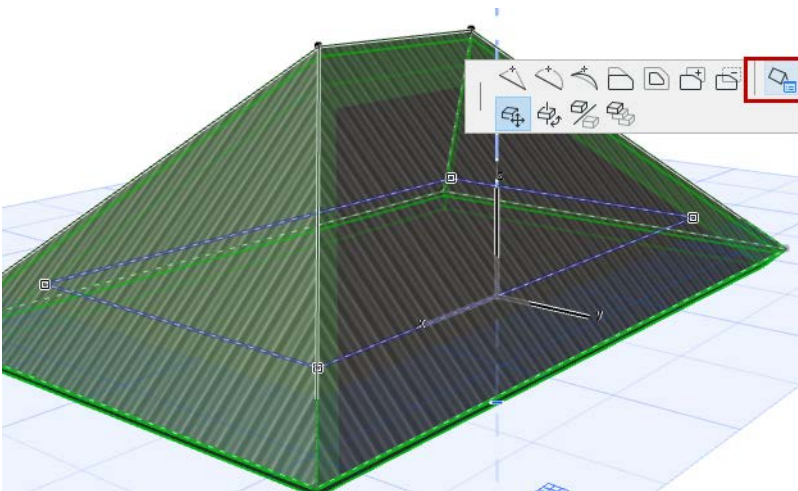
[Watch the Video](#)

Customize Roof Plane

For a Multi-plane Roof, you can customize any of its planes so that its surfaces, eaves overhang, and pitch are different from the Roof Settings in effect for the roof as a whole.

To open the Custom Plane Settings dialog box:

1. Select the Roof.
2. Click the pivot line of the roof plane which you want to customize.
3. From the pet palette, choose Custom Plane Settings:



Here, set one or more of the following custom settings:

- Pitch
- Eaves overhang
- Top or Bottom Surface (available for plane of Pitched Roof only)
 - Note:** To apply a custom Edge Surface, use Custom Edge Settings.
- [See *Customize Edge of Roof or Roof Hole*.](#)
- Cover Fill

Apply Changes

Apply changes to: Choose to apply the custom settings to:

- Clicked Plane (this is the default)
- Clicked Level (all planes of the current Roof Level; this is meaningful only if you have a multi-level roof.)

See [Add Roof Level](#).

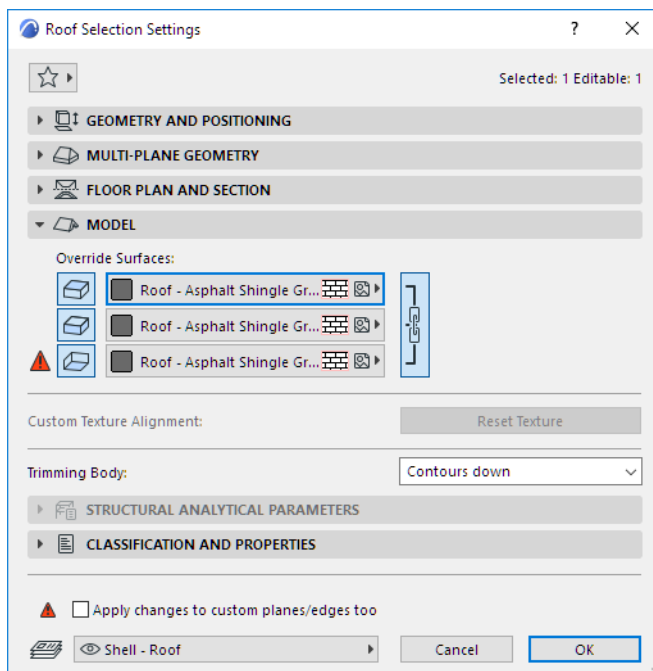
- All Planes

Custom Values in Roof Settings

In **Roof Settings**, the values shown for roof pitch, eaves overhang, surface and/or curve resolution will not change, because they reflect the settings of the roof as a whole.

However, a **yellow patch** appears next to those controls for which at least one roof panel or edge has a different, customized setting.

If you later change the one of these values in Roof Settings while the dialog box is still open, the yellow patch (indicating “custom plane”) gets a red shadow. This means that a Roof plane setting is going to change: do you want to apply this change to your customized plane too? If so, check the box at the bottom of Roof Settings (Apply changes to custom planes/edges too). If you do not check the box, your custom planes remains the way you set them.

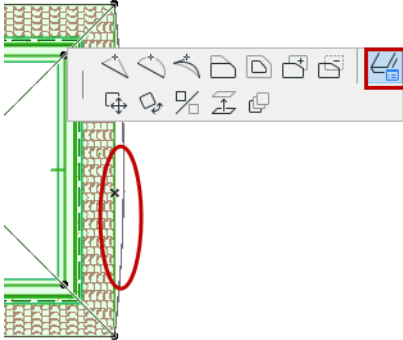


[Watch the Video](#)

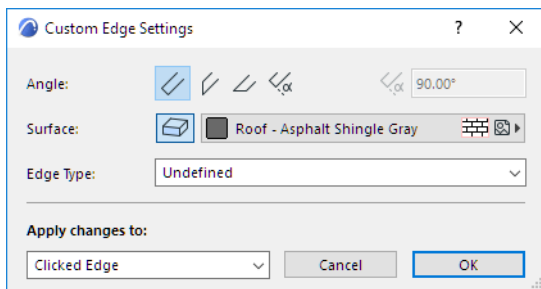
Customize Edge of Roof or Roof Hole

You can customize the surface and/or angle of any edge of a Roof, or the edge of a hole in a Roof. Once you set the customized surface/angle options, you can choose to apply them to the clicked edge only, to all edges of the clicked polygon (hole), or to all roof edges.

1. Select the Roof.
2. Click on the Roof's contour line (not the pivot line!) to bring up a pet palette.



3. Select the **Custom Edge Settings** icon to open the dialog box.



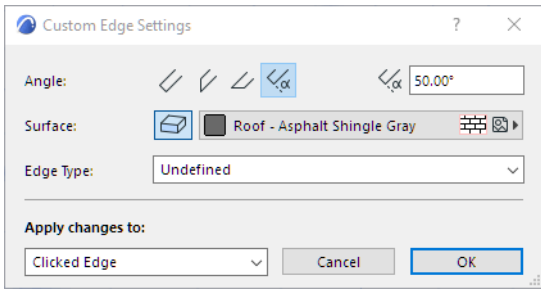
[Watch the Video](#)

Custom Roof Edge Angle

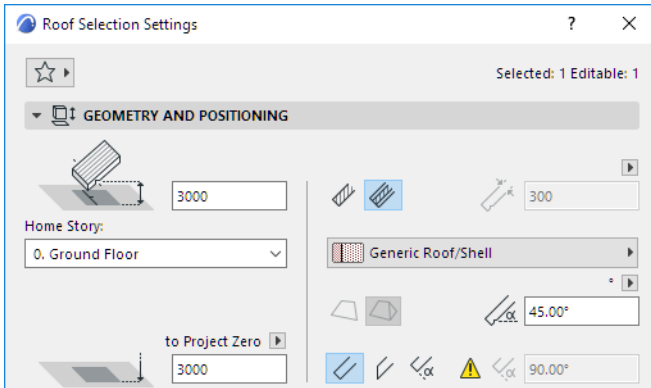
Angle: Choose an angle (vertical, horizontal, perpendicular, custom) for the roof edge. If you choose custom, enter the desired angle value in the field at right.

Note: You can set a horizontal edge angle only if the selected edge is parallel to the pivot line, on a roof that is not flat.

Note: If you have clicked an edge that is the common joint of two selected single-plane roofs, an additional, "Mitered" option is available. Choose this to create a mitered joint along the selected roof edges.



If you apply an edge angle here that is different from the default roof edge angle (set in Roof Settings), the Geometry and Positioning Panel of Roof Settings will indicate that a custom edge angle has been applied.

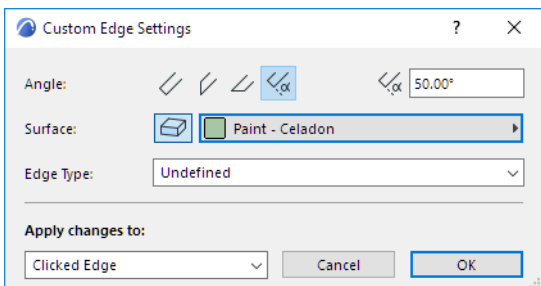


If you later change the Edge angle setting in Roof Settings while the dialog box is still open, the yellow patch (indicating “custom edge”) gets a red shadow. This means that the Roof’s edge angle is going to change: do you want to apply this change to the custom-angle edges too? If so, check the box at the bottom of Roof Settings (Apply changes to custom planes/edges too). If you do not check the box, your custom edge angles remain the way you set them.

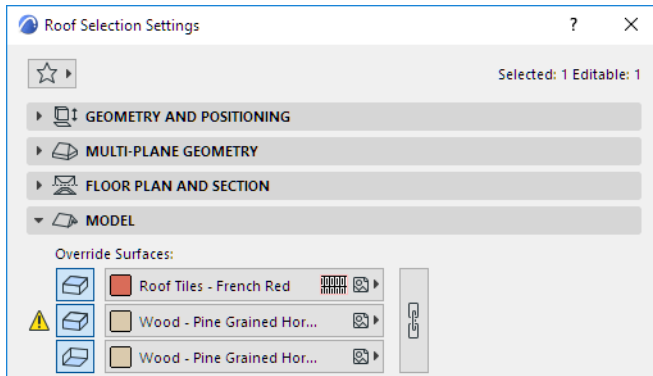
Custom Roof Edge Surface

Note: By default, the Surface material applied to a Roof edge derives from the Building Material assigned to the Roof. You can override *all* edge surfaces of the Roof, using the controls in the Model Panel of Roof Settings. The Custom Edge Settings offers a further way to customize the surface of *any one or more* Roof Edges.

Surface: Click the Surface override button, then choose a custom surface for the roof edge(s).

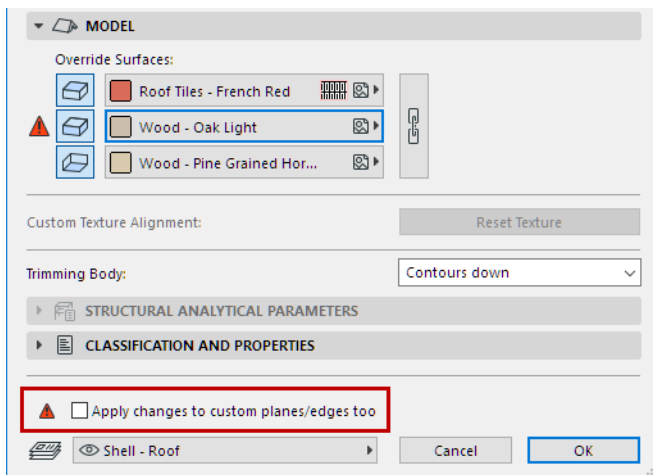


If you apply a surface here to any edge that is different from the edge surface defined in the Model Panel of Roof Settings (or Building Material), the Model Panel will indicate (with a yellow patch) that a custom edge surface has been applied.



If you override the Roof edge surface in Roof Settings while the dialog box is still open, the yellow patch (indicating “custom edge”) gets a red shadow: this means that the Roof’s general edge setting is going to change: do you want to *apply this override to all edges*, including the custom edge(s) too?

If so, check the box at the bottom of Roof Settings (**Apply changes to custom planes/edges too**). If you do not check the box, your custom edge surfaces remain the way you set them.



Custom Roof Edge Type

Edge Type: Optionally, choose a type for the roof edge.

The Roof Edge type you define here can be used as a parameter in all of Archicad’s listing functions. For example, you can add “Eaves Length” and “Gables Length” as parameters of an Interactive Schedule that lists roofs.

For an explanation of edge types, see [Roof Listing Parameters](#).

Apply Changes

Apply Changes to: Once you have set the options as needed, choose the edges to which to apply these changes:

- **Clicked Edge** will apply the options you choose below (angle, surface, edge type) only to the edge you clicked when bringing up the pet palette.
- **Selected Polygon** will apply the chosen options to the edges of the roof only or the edges of the hole only, depending on which one is selected.
- **All Edges** will apply the chosen options to all edges of the roof as well as all edges of its holes.

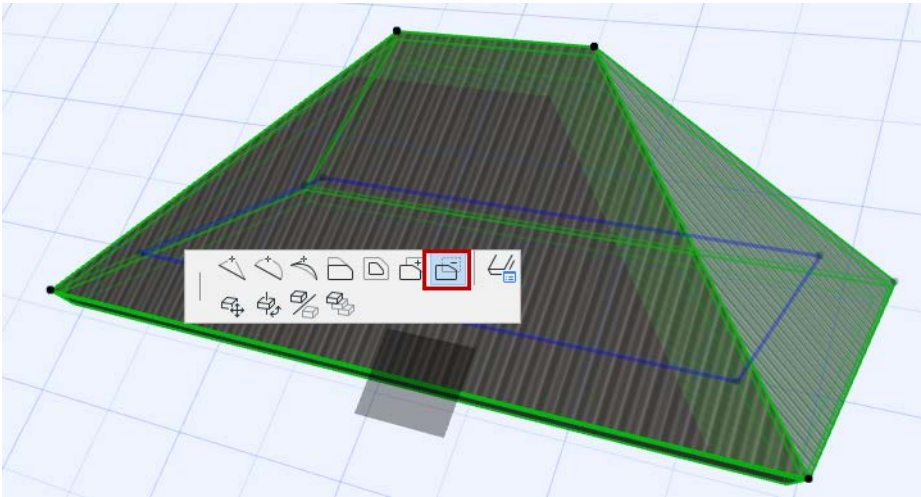
Click **OK** to apply changes.

Create a Hole in a Roof

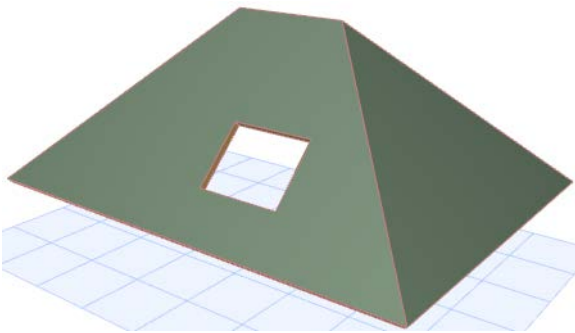
There are two ways to create a hole in a Roof.

Method 1

1. Select the Roof in the Floor Plan or the 3D window.
2. From the contour polygon (not the pivot polygon!), bring up the pet palette and choose the **Subtract from Polygon** command.



3. Draw the hole in the Roof.
4. View the result in 3D.



Method 2

1. Select a Roof.
2. With the Roof tool active, draw a new contour within its boundaries. The new contour will be interpreted as a hole in the Roof.

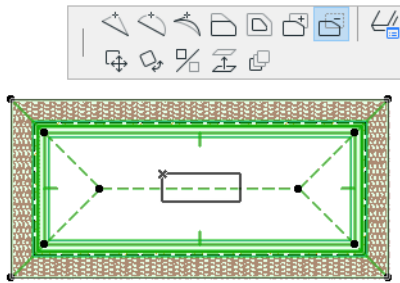
Note: If you create a hole whose outline intersects the host Roof's contour or the outline of other holes in the same polygon, Archicad will display a warning message, but the outline of the hole will still be created.

[Watch the Video](#)

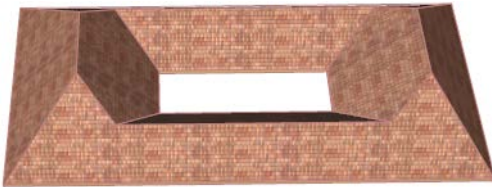
Create an Atrium

To create an atrium in a Multi-plane roof:

1. Select the Roof in the Floor Plan or the 3D window.
2. From the pivot polygon (not the contour polygon!), bring up the pet palette and choose the **Subtract from Polygon** command.



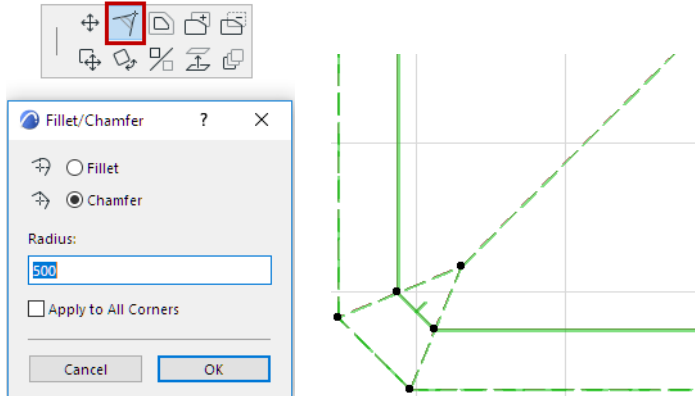
3. Draw the atrium shape in the Roof.
4. View the result in 3D.



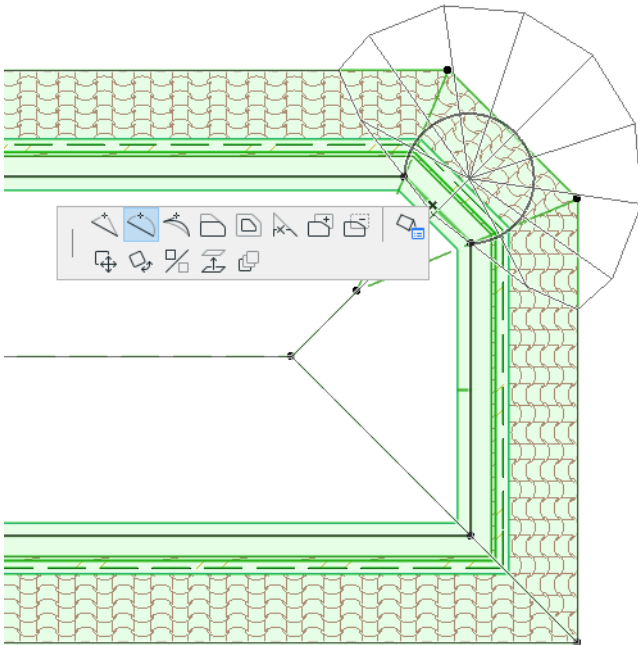
[Watch the Video](#)

Add a Tower to the Roof

1. Begin with a regular hip roof.
2. In the Floor Plan or 3D, select the pivot line on the plane where you want to add the tower.
3. Chamfer the polygon to create an additional roof plane.



4. In this new plane, again select the pivot line and use polygon editing to add a curve to the edge and create a tower shape.



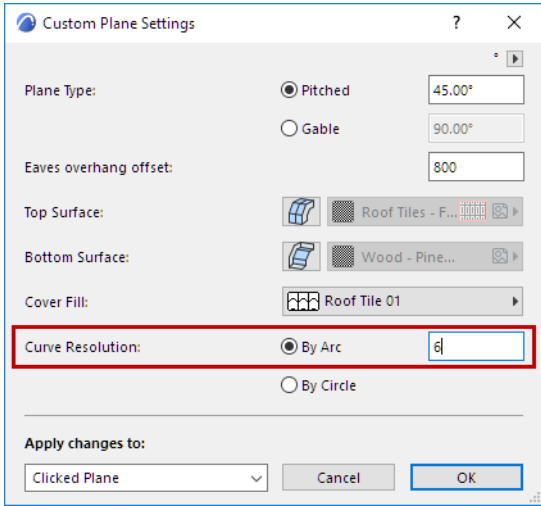
The number of divisions in all the arced planes of your Roof are distributed either by arc or by circle, as set in the curve resolution control of Roof Tool Settings (Multi-plane Roof panel).

[See *Curve resolution*.](#)

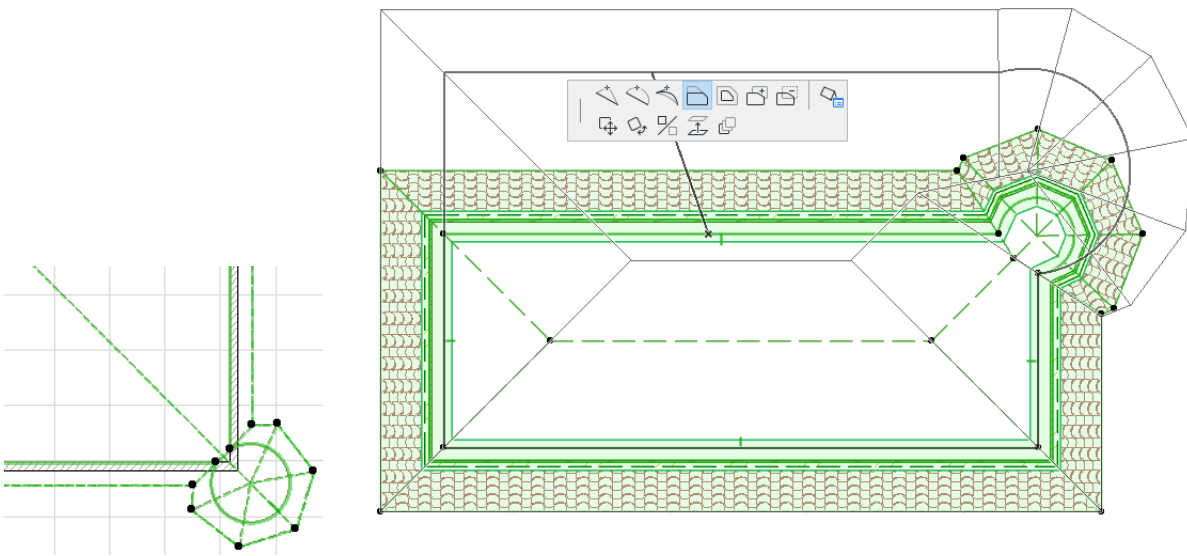
You can customize this arc resolution method for any specific arced plane (such as the tower), using the Customize Plane dialog box.

[See *Customize Roof Plane*.](#)

For example, you can specify that the tower consist of six divisions: choose By arc, then enter 6.



This way, if the geometry of the surrounding roof planes changes, the arced planes of your tower still remain divided into six.



Create Roof Level Lines

Use this command (**Design > Roof Extras > Create Roof Level Lines**) to place a Line on a selected Roof at the height value you specify.

After you place the Roof Level Line, you can place a Level Dimension there to indicate the elevation.

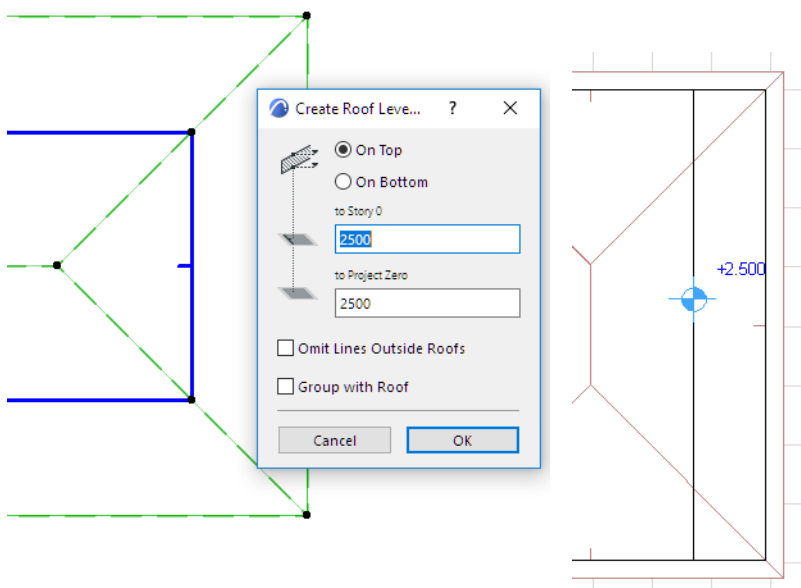
Note: Roof Level Lines can be placed on both single-plane and multi-plane roofs. However, with multi-plane roofs, make sure you are not using legacy intersection mode. (Go to Options > Project Preferences > Legacy, and make sure that the “Use Legacy Intersection methods” checkbox is not active.)

1. On the Floor Plan, select a Roof.
2. Use **Design > Roof Extras > Create Roof Level Lines**,
3. Set preferences in the appearing dialog box, then click OK.
 - **On Top/On Bottom:** Choose whether you want to measure the height value to the top or to the bottom surface of the roof.
 - **Specify Roof Level Lines height level:** Use either input field - the current story level or Project Zero - to define the height of the desired roof level lines.
 - **Omit Lines Outside Roofs:** Check this box to omit level lines that fall outside the floor plan polygon of the roof they belong to.

If the checkbox is disabled, all resulting lines will be drawn. In this case, lines that fall outside the roof polygon will be drawn to be of equal length to the pivot line of the roof they belong to.

- **Group with Roof:** Check this box to group level lines with the roofs they belong to.

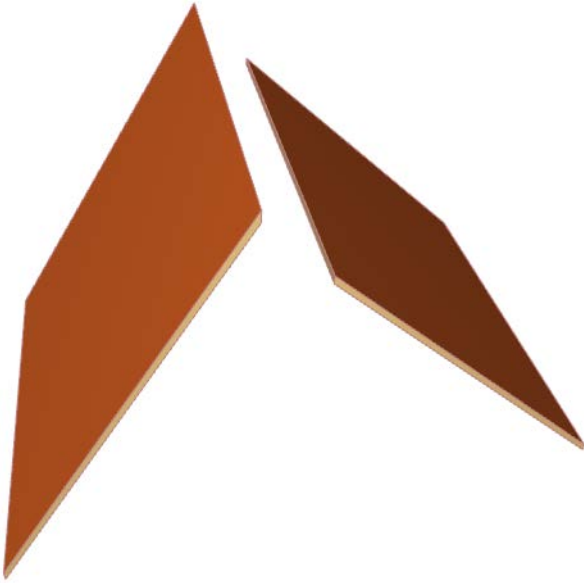
Note: This option is grayed if the **Suspend Groups** toggle is active.



See also [Level Dimensions](#).

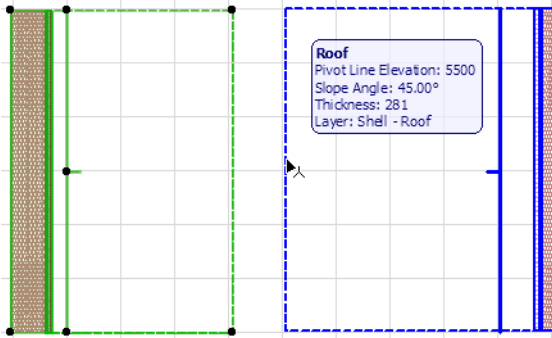
Intersect Single-plane Roofs

You may need to adjust Single-plane roof panes to each other.

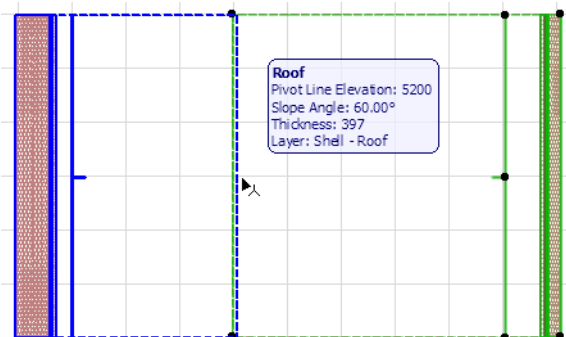


To have these two roof planes intersect:

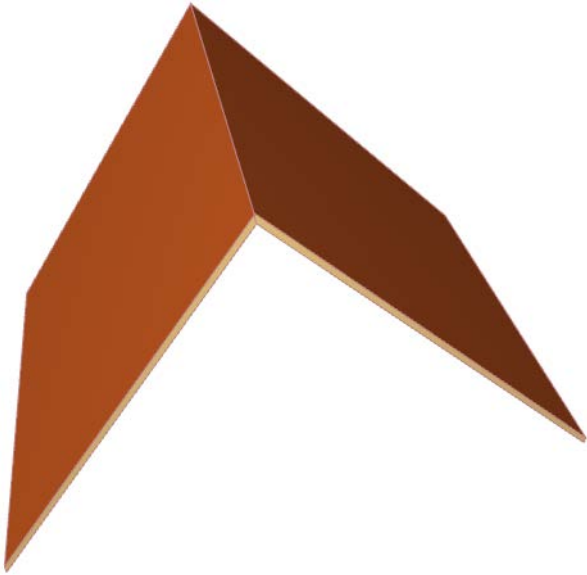
Select one of the Roofs on the Floor Plan, and Ctrl-click (Windows) or Cmd-click (Mac) the ridge line of the other.



Now select the other roof plane and Ctrl-click (Windows) or Cmd-click (Mac) the ridge line of the lower one.



The two faces have been adjusted to each other.



See also [Cropping with a Remote Roof](#).

Crop Elements to Single-plane Roof

To adjust other elements to a single-plane roof, you can use the crop function, described in the following sections:

[Using the Crop to Single-plane Roof Function](#)

[Undo Roof Crop](#)

[Effect of Roof Crop on Wall/Column Height Settings](#)

[Use Roof Crop to Create Complex-Shaped Walls, Beams and Columns](#)

[Cropping Slabs with a Roof](#)

[Cropping with a Remote Roof](#)

Using the Crop to Single-plane Roof Function

To crop an element to a Single-plane roof, use the **Design > Roof Extras > Crop to Single-plane Roof** command.

The **Crop to Single-plane Roof** function is *not* associative. Crop to Roof, available for Single-plane roofs only, creates a changed Wall geometry which is static.

To create associative trims between Roofs and other elements, see [Trim Elements to Roof or Shell](#).

Roofs can crop either the top or the bottom of

- Walls
- Beams
- Columns
- Slabs
- Library Parts (Doors, Windows, Objects)

Note: A similar function is the **Crop Zone** command, which crops Zone volume using surrounding elements, including single-plane Roofs.

[See *Crop Zone to Another Element*.](#)

Elements are cropped only if they intersect a Single-plane roof at any point, and only if you check its element type in the Crop to Single-plane Roof dialog box.

It is usually easiest to use Crop to Roof in the 3D window, where you can see the spatial relationships of roofs to other construction elements.

The **Crop to Single-plane Roof** command is available only if at least one roof or one element of the above types is selected.

To crop elements to a roof, follow these steps:

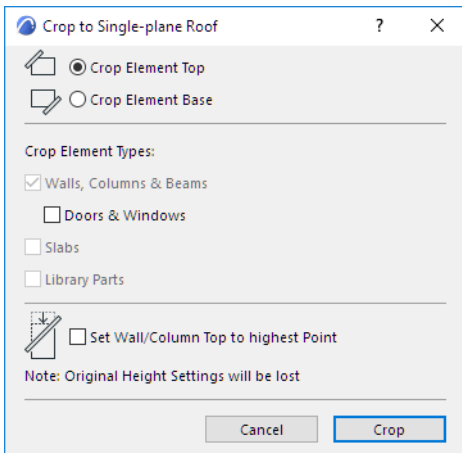
1. Do one of the following:

- Select the elements you wish to crop. (Remember, only elements which intersect a roof at any point can be cropped); or
- Select the roof(s) to which you want to crop elements; or
- Select both elements and roof(s).

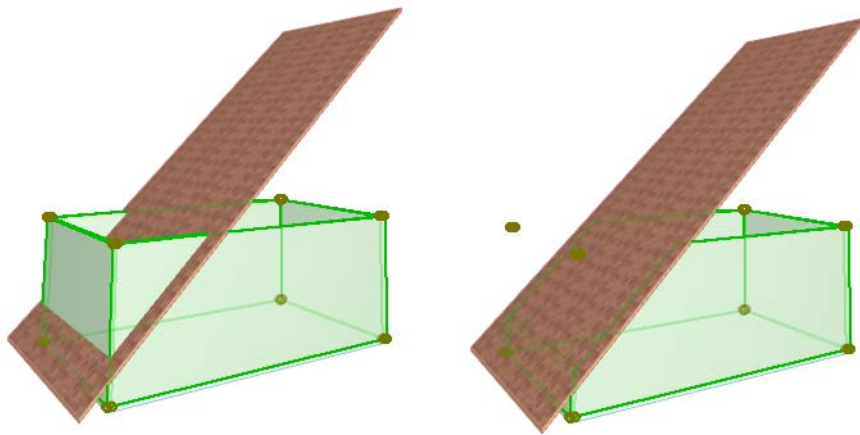
Selection of roofs and/or elements narrows the scope of the crop:

- If you select only a roof, all eligible elements are cropped
- If you select only elements, they will be cropped to all the eligible roofs.

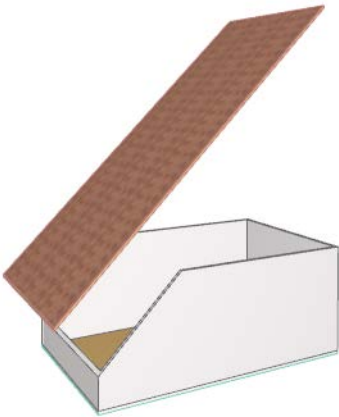
2. Go to **Design > Roof Extras > Crop to Single-plane Roof** to bring up the **Crop to Single-plane Roof** dialog box.



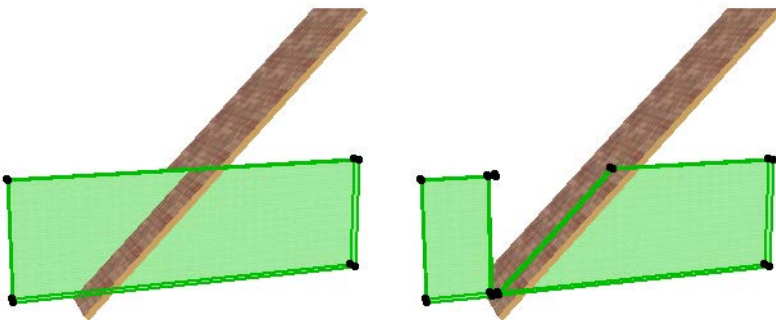
3. Check the element type(s) you want to crop. (If you have already selected certain elements before bringing up the dialog box, these checkboxes are grayed.)
4. Click the preferred cropping option, depending on which part of the eligible element(s) you want to crop:
 - **Crop Element Top:** Crop the part of the element(s) above the roof
 - **Crop Element Base:** Crop the part below the roof.
5. Click **Crop** and view the result.



The crop remains in effect even if you hide the Roof or the cropped elements, move them away from each other or even delete the Roof that is executing the crop.



If you crop a Wall with a roof that crosses the Wall entirely, the original Wall will be cut into two: one Wall that is cut by the cropping Roof, and a second wall that is separate and uncut.

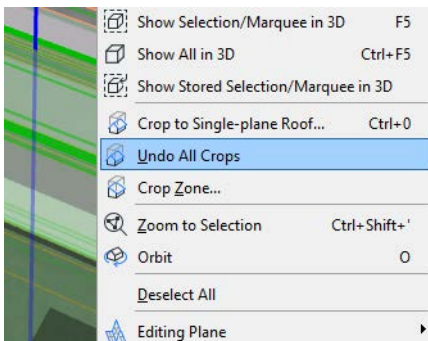


Undo Roof Crop

Cropping an element can either be temporary, for visual display only, or definitive.

You can restore elements after a temporary crop-to-roof. If an element has been cropped to a Roof, the **Undo All Crops** command becomes active:

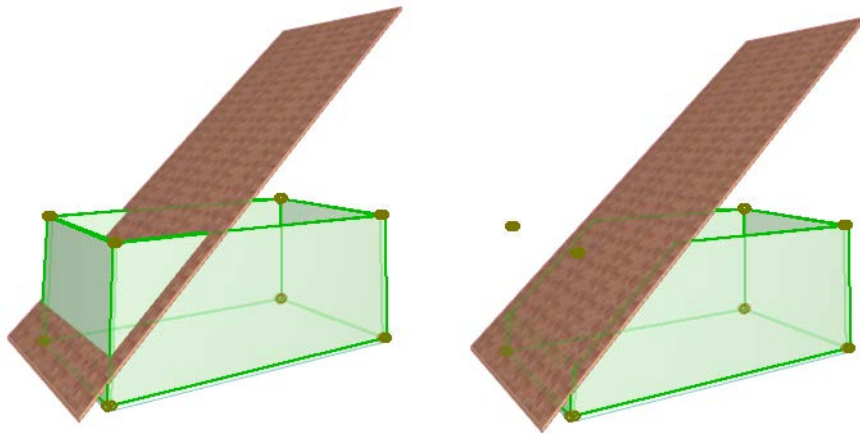
- in the selected element's context menu, and
- at **Design > Roof Extras**



Exception: A crop on a Slab can only be undone through the regular Archicad Undo function (**Edit > Undo**).

Effect of Roof Crop on Wall/Column Height Settings

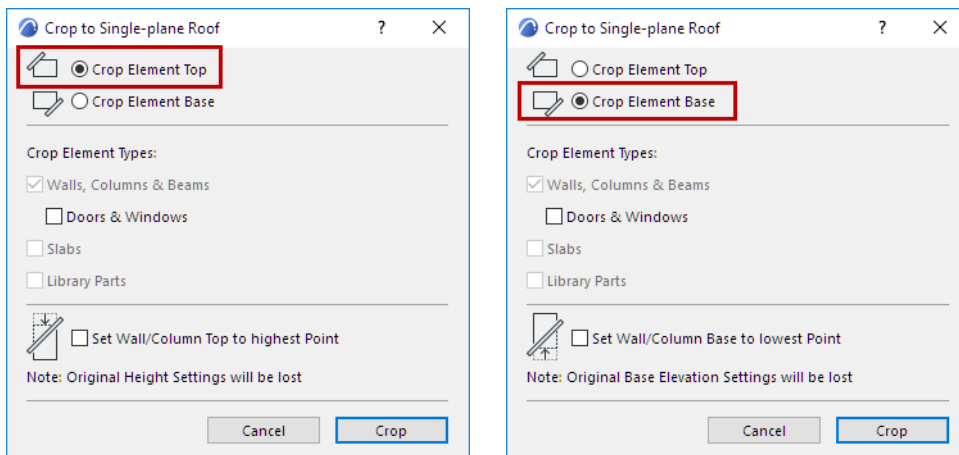
By default, cropping a Wall or Column to a Roof does not alter the cropped element's height. As seen in this image, the cropped Wall's selection dots are still where they used to be before the crop.



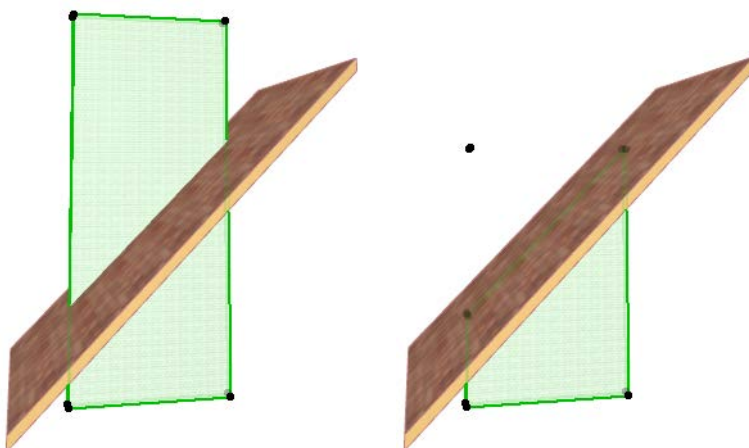
If you want to recalculate the cropped element's height to reflect the cropped geometry, then click the checkbox:

Set Wall/Column Top to highest Point at the bottom of the **Crop to Single-plane Roof** dialog box.

(If you chose **Crop Element Base**, the checkbox refers to the Wall/Column's base elevation (lowest point)).

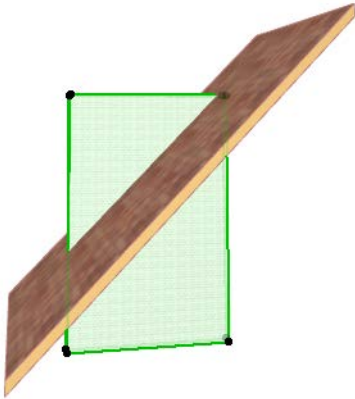


If the box is checked, the height of a cropped wall or column is recalculated to equal the maximum of its cropped height (if you cropped the top) or the minimum cropped base height (if you cropped the bottom). In this crop, the checkbox is checked. This means that the Wall's actual height (as shown by the selection dots) has changed to the highest point of the cropped Wall.



The **Undo All Crops** command will have a different effect depending on the checkbox status:

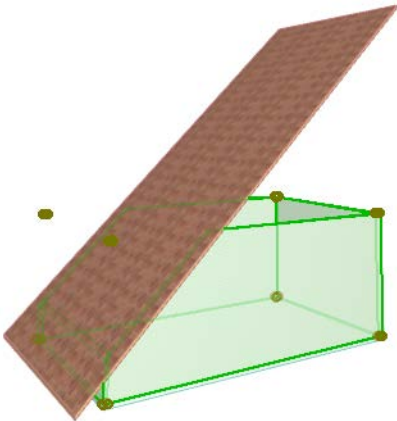
- If you set the Crop operation to recalculate the Wall/Column's height value (if the checkbox was active), then the **Undo All Crops** will only recreate the Wall/Column up to the maximum cropped height, as shown in this image:



- If you did not check the box, then **Undo All Crops** will recreate the original Wall height.

Use Roof Crop to Create Complex-Shaped Walls, Beams and Columns

You can cut complex shapes into walls, beams and columns by cropping them to Roofs. The wall will always follow the shape of the roof(s) above them, no matter how many roof segments you have.



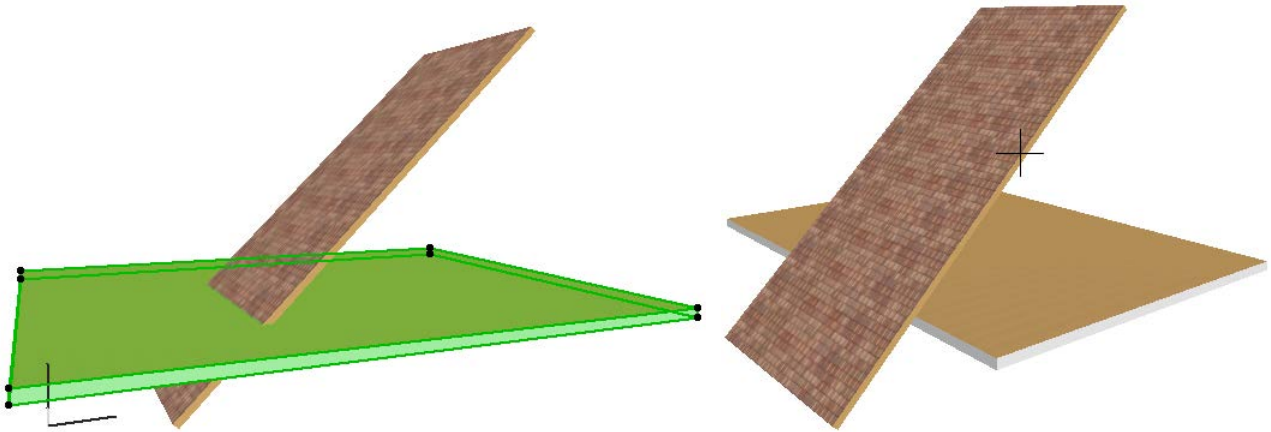
Cropping Slabs with a Roof

Slabs can also be cropped by the lower or upper slope of roofs with the help of the **Design > Roof Extras > Crop to Single-plane Roof** command.

While other elements can be partly cropped, slabs are always cut along the whole section line of the roof and the slab. Moreover, the cropped edge of the slab is always vertical even after being cropped by a slanted roof.

You can, however, manually adjust the slab edge angle.

See [Set Custom Slab Edge Angle and Edge Surface](#).

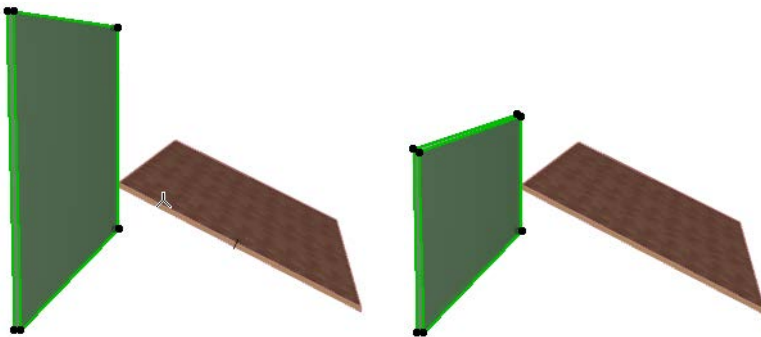


Cropping with a Remote Roof

You can use remote roofs, and even roofs on other stories whose outline is shown on the given story, to crop elements to the plane of that particular roof. In this case, the **Crop to Single-plane roof** command does *not* work (it only works if the elements intersect each other.) However, you can use a keyboard shortcut when cropping with a remote roof.

1. Make sure the Roof tool is selected.
2. Select the elements you intend to cut, then **Ctrl-click (Cmd-click)** an edge or a node of the roof you want to cut them with. The reverse procedure will also work: select the roof you want to cut with and **Ctrl-click (Cmd-click)** the individual elements you want to be cut.

Note: The Wall/Beam/Column is cut whether it is under a Roof or not. Be sure to set a sufficient height for the original rectangular Walls/Beams/Columns so that they are cut correctly.



See also [Intersect Single-plane Roofs](#).

Create Special Roof Objects with RoofMaker

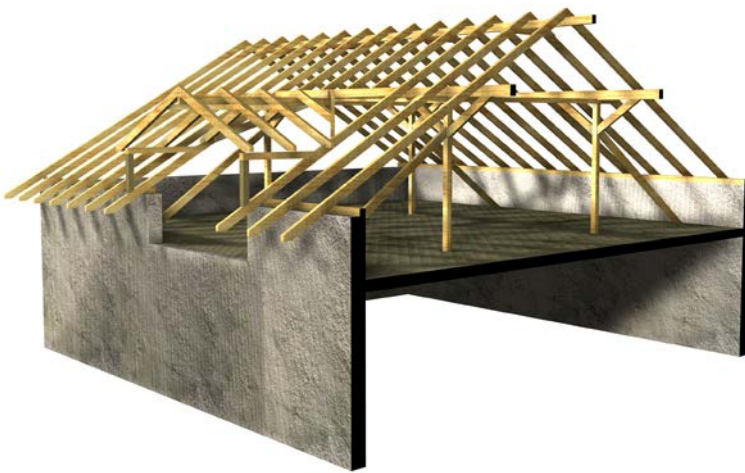
RoofMaker allows you to add GDL Object-based elements to specific parts of your design in order to enhance them in 3D views and Section/Elevation/IE windows. RoofMaker can be accessed from the **Design > Roof Extras** menu. Technically, RoofMaker is an Archicad Add-on and is loaded with Archicad on startup.

After placement, roof construction elements behave like ordinary Objects and can be modified.

The Roof Wizard is a RoofMaker function that allows you to create entire roof constructions quickly and easily.

For more information, see [Using the Roof Wizard](#).

We recommend that you begin modeling the roof construction with the purlins and continue with the hip rafters/valley rafters before placing the rafters. This way you can easily keep track of all the connecting elements. (When placing ridges, you already know the thickness of the purlin; when placing rafters you already know the thickness of purlins, hip and valley rafters, etc.)



To place **rafters**, **ridges** or **purlins** (beams), select a roof surface or Multi-plane roof.

If single-plane roofs are grouped, you must first ungroup them (**Edit > Grouping > Ungroup**), or Suspend Groups (**Edit > Grouping > Suspend Groups**). You will then be able to select roof surfaces individually.

Although the placement of roof construction elements using RoofMaker works using other methods, we strongly recommend setting the reference line of the roof to the upper exterior edge of the wall beneath (as handled by Archicad's Complex Roof Placement Method). The definition of elements, with or without overhang, uses this reference line as a basis.

We recommend setting the roof thickness equal to the general cross-section height of the rafters. This way, you can use the roof to cut elements that are standing upon the rafters (e.g., Posts).

To place a **trimmer** (blocking), a **collar beam** or a **tie beam**, two corresponding rafters must be selected (opposite rafters for a collar beam or a tie beam, and rafters in the same roof surface for a trimmer).

After choosing the command for the placement of an object, a dialog box opens in which certain parameters concerning the object itself can be set. Surfaces and a layer can be assigned to all elements. The current pencolor set for the Object is used by default. If you change the current pencolor, all the subsequently placed elements will have the new color.

The object is then placed with one or more clicks, or is placed automatically, depending on the object type.

To display the RoofMaker commands in a separate floating toolbox, choose **Design > Roof Extras > RoofMaker > Show RoofMaker Toolbox**.



Each command is described in the sections below:

Create a rafter

Create multiple rafters

Create hip or valley rafter

Create a trimmer

Create a purlin

Create an eaves purlin

Create collar beams

Create tie beams

Using the Roof Wizard

Create a rafter

When creating rafters, the bottom plane of the roof serves as a reference plane. Rafters will be automatically placed on top of it.

Select a reference roof surface on the floor plan. Choose **Create a rafter** from the **Design > Roof Extras > RoofMaker** menu (or select the first icon from the RoofMaker toolbox).

Use the appearing Rafter Settings dialog box to set parameters.

[See Rafter Settings Dialog Box.](#)

Clicking OK returns you to the floor plan with the reference roof surface still selected. Click inside the selected roof surface. (If you click outside, an error message appears.)

A rafter is placed with its axis going through the specified point. By definition, rafters are perpendicular to the reference line of the roof.

After the rafter is placed, you can select it and open its Object Settings dialog box. Among others, the parameters include profile and profile thickness. The rafter may have a rectangular (default) profile as well as I-beam, L-beam or C-beam profiles.

Create multiple rafters

Select a reference roof surface on the floor plan. Choose **Create multiple rafters** from the **Design > Roof Extras > RoofMaker** menu (or select the second icon from the RoofMaker toolbox).

Set the desired parameters. You will set the same controls as for a single rafter, but the placement controls for multiple rafters are also active.

[See Multiple Rafters Dialog Box.](#)

Clicking **OK** returns you to the floor plan with the reference roof surface still selected. Click twice inside the selected roof surface to define a placement line. (If you click outside, an error message appears.)

Several rafters are placed along the placement line, with the axis of the first rafter going through the point defined with the first click, and the axis of the last rafter going through the point defined with the second click. By definition, rafters are perpendicular to the reference line of the roof.

Create hip or valley rafter

The hip rafter usually lies between two neighboring roof polygons. Select the roof surface where the hip rafter connects to a purlin instead of another hip rafter; other methods may cause errors.

Choose **Create a hip rafter or valley rafter** from the **Design > Roof Extras > RoofMaker** menu (or click the third icon of the RoofMaker toolbox).

[See *Hip or Valley Rafter Settings*.](#)

After adjusting those settings, click OK to return to the floor plan with the reference roof surface still selected. Click along one of the edges of the selected roof that is NOT parallel or perpendicular to the reference line.

A Hip or a Valley Rafter (or a Valley Rafter without Overhang) is placed along the specified edge of the roof, depending on the position of the edge to the roof's reference line.

Hip and Valley Rafters can also be placed with two roof surfaces selected previously. It is then not necessary to click afterwards to specify an edge; the object will be placed along the common edge of the two surfaces.

The difference between the two methods lies in the shape of the resulting objects; the top cut in the latter method makes the object suitable for steeple type roofs.

Create a trimmer

In order to place a trimmer, you must first select the two rafters the trimmer is going to be placed between. The two rafters have to be within the same roof polygon. (This also means that they have the same pitch angle and they are both perpendicular to the reference line of the roof.)

Choose **Create a Trimmer** from the **Design > Roof Extras > RoofMaker** menu. The Trimmer Settings dialog box appears.

[For more information, see *Create a Trimmer \(or Blocking\)*.](#)

A trimmer is placed at the same height as the rafters. It can be vertical or rotated to a position perpendicular to the pitch angle of the roof. The 2D symbol also shows the current position.

Clicking **OK**, you return to the floor plan with the two rafters still selected. Click between the rafters. (If you click outside them, an error message will appear.)

A trimmer is placed between the two rafters, the axis going through the specified point. Trimmers are always parallel to the reference line of the roof and thus perpendicular to the rafters.

Create a purlin

Select a reference roof surface on the floor plan and choose **Create a Purlin** from the **Design > Roof Extras > RoofMaker** menu (or click the fifth icon from the RoofMaker toolbox). The Purlin Settings dialog box appears in which purlin parameters can be set.

[For more information, see *Create a Purlin \(or Beam\)*.](#)

By default, the purlin is placed under the reference plane defined by the roof, since a purlin generally supports the rafters from below. However, an elevation value can be set for the purlin so that it can be higher or lower than the reference plane. Set the values for the width and the height of the cross-section.

The purlin can also be perpendicular to the rafters and be on the top of them as well. If the purlin is placed on top, the cross section height of the rafters has to be set.

Clicking OK returns you to the floor plan with the reference roof surface still selected. Click either an edge of the selected roof, or inside the roof polygon. (Clicking outside the polygon produces an error message.)

If you click on an edge, a purlin is placed with its axis along the edge. If the edge is not parallel to the reference line of the roof, the endpoints of the axis of the purlin will be at different heights - as indicated in the "Height difference" parameter of the library part - and the purlin will be inclined.

If you click inside the polygon, the axis of the beam will pass through the clicked point parallel to the reference line of the roof.

Create an eaves purlin

Select a reference roof surface on the floor plan and choose **Create an eaves purlin** from the **Design > Roof Extras > RoofMaker** menu (or click the sixth icon in the RoofMaker toolbox). The Eaves Purlin Settings dialog box appears in which eaves purlin parameters can be set.

For more information, see [Create an Eaves Purlin \(or Plate Beam\)](#).

This type of purlin is placed under the reference plane defined by the roof, supporting the rafters from below. However, an elevation value can be set for the eaves purlin so that it can be higher or lower than the reference plane. You can also set the values for the width and the height of the cross section.

Clicking OK returns you to the floor plan, with the reference roof surface still selected. You need to click either one of the edges of the selected roof, or inside the roof polygon. (If you click outside the polygon, an error message appears.)

If you click on an edge, the purlin is placed inside the polygon, with its side along the edge. If the edge is not parallel to the reference line of the roof, the two endpoints of the axis of the purlin will be at different heights, as indicated in the "Height difference" parameter of the library part, and the purlin will be inclined.

If you click inside the polygon, the axis of the purlin will pass through the clicked point, parallel to the reference line of the roof surface.

By default, a background fill is assigned to the 2D symbol of purlins because purlins usually have to cover posts placed underneath. These posts are generally displayed as a larger circle to indicate that there is some kind of supporting structure. You can use the **Bring to Front** and **Send to Back** commands to ensure the accurate positioning of elements.

Create collar beams

In order to place a collar beam you must first select two rafters to place it between. The axes of the two rafters must be along the same line and meet at the top, otherwise you will get an error message.

Choose **Create a collar beam** from the **Design > Roof Extras > RoofMaker** menu (or the seventh icon in the RoofMaker toolbox). The Collar Beam Settings dialog box appears in which different parameters of the collar beam can be set.

For more information, see [Create Collar Beams](#).

A collar beam can be double- or single-sided. Set its cross-section dimensions and elevation, calculated either from Project Zero or the current story elevation.

Clicking **OK** returns you to the floor plan with the two rafters still selected. If a double-sided collar beam has been selected, it is placed automatically in the appropriate position. If a single-sided collar beam is selected, click once more to determine on which side of the rafters the collar beam will be placed.

Create tie beams

In order to place a tie beam, you must first select two rafters to place it between. The axes of the two rafters must be along the same line and meet at the top, otherwise you will get an error message.

Choose **Create a tie beam** from the **Design > Roof Extras > RoofMaker** menu (or eighth icon in RoofMaker Toolbox). The Tie Beam Settings dialog box appears in which different parameters of the tie beam can be set.

For more information, see [Create Tie Beams](#).

In addition to the dimensions of the cross section, an elevation value must be set, calculated either from Project Zero or the current story elevation.

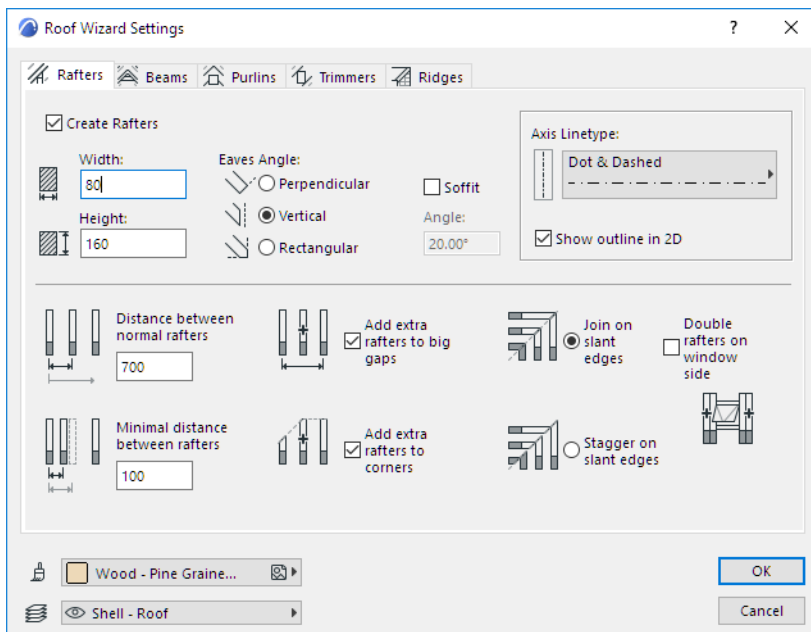
Clicking **OK** returns you to the floor plan with the two rafters still selected. The tie beam is placed automatically in the appropriate position.

Using the Roof Wizard

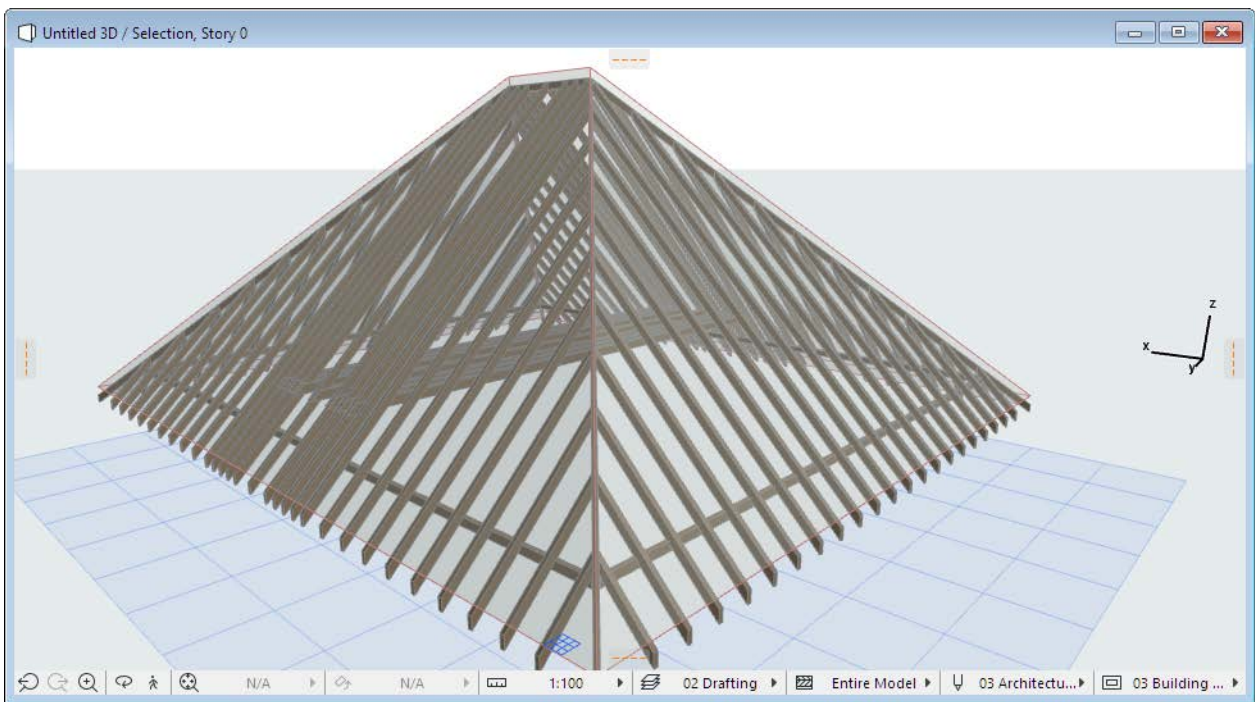
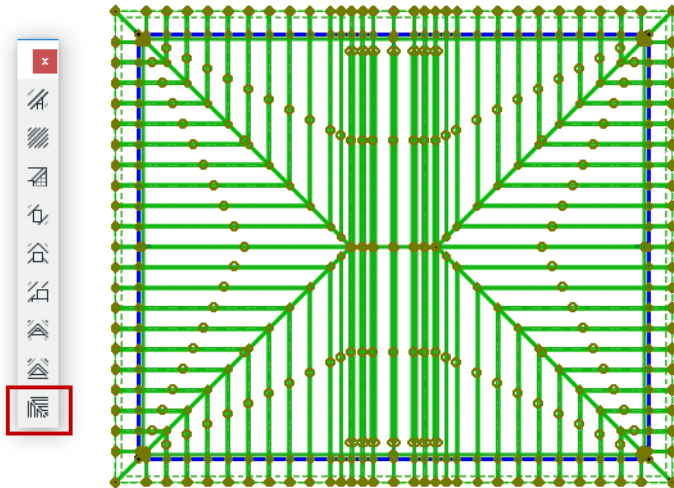
The Roof Wizard allows you to create entire roof constructions quickly and easily. You can place rafters, purlins, trimmers, collar or tie beams in one step.

To begin, select all the roof surfaces in the project that you wish to add structural elements to. Choose the **Roof Wizard** command from the **Design > Roof Extras > RoofMaker** menu (or the last icon in the RoofMaker Toolbox).

See [Roof Wizard Dialog Box](#).



Clicking **OK** returns you to the floor plan. The roof construction elements you have specified are placed automatically in the appropriate position.



The placement of the rafters is optimized to meet the specified requirements. The objects now behave as ordinary Objects and can be modified as such.

Trim Elements to Roof or Shell

The **Trim to Roof/Shell** command allows you to connect Roofs or Shells with each other or with other model elements in any Archicad model window. The result is a more complex structure whose elements are trimmed precisely to each other, although they remain separate elements.

Elements trimmed to each other using this command will also be merged, and will intersect based on their skins intersection priorities.

[See also *Basic Intersection Principles*.](#)

The trimming element is always either a Roof or a Shell. A Roof or Shell can trim any model element (e.g. Wall, Beam, Column).

Roofs and Shells trimmed to each other *act as a single body* when trimming other model elements.

Elements trimmed to each other are *associative*: a change to any element will immediately affect all the connected elements. For example, if you change the Roof pitch, the Wall that it trims will change accordingly.

Note: In contrast, the **Crop to Single-plane Roof** function (known as Trim to Roof in earlier Archicad versions), is *not* associative. Crop to Roof, available for Single-plane roofs only, creates a changed Wall geometry which is static.

[For more information, see *Crop Elements to Single-plane Roof*.](#)

Once you have created a trimming relationship, the elements involved are connected. If you want to change this relationship, you don't have to undo the connection; just do the Trim operation over again.

Related Topics:

[Define Trimming Bodies](#)

[Trim Multiple Selected Elements: Automatic Trim](#)

[Trim Elements to Roof: Manual Trim](#)

[Merge Trimming Elements](#)

[Managing Element Connections](#)

[Trim Elements Using Scissors](#)

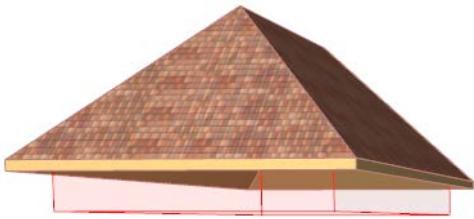
Define Trimming Bodies

The trimming element is always either a Roof or a Shell. The Roof/Shell has a “trimming body” attached, which represents the direction and shape in which it will execute the trim on the other element(s).

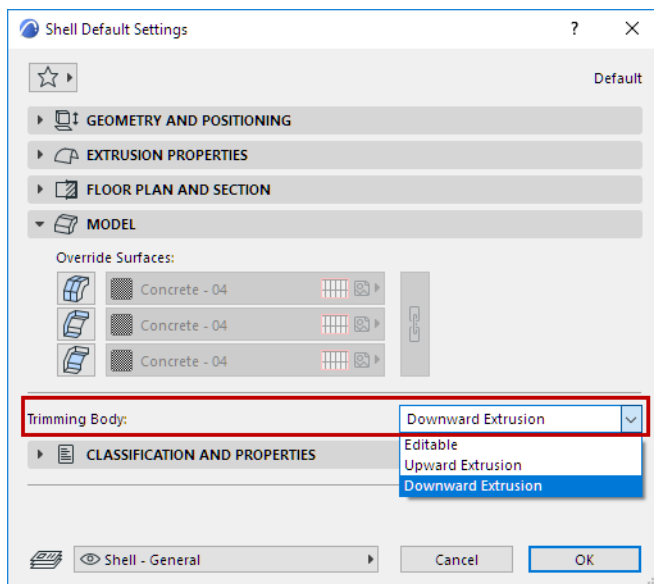
Since a Roof/Shell shape is often quite complex in itself, it may be difficult to visualize exactly what will happen when you embark on a Trim operation. To see these trimming bodies while you work (in the 3D window only), turn on **Trimming Bodies** in the **View > On-Screen View Options** menu.

Note that all the trimming bodies (except the “editable” one for Shells), in fact represent bodies that are extended infinitely upward or downward (this is not shown in the feedback).

By default, the Roof’s trimming body is “Pivot lines down”, that is, the pivot line is the boundary of the trimming body, and the body extends downward from there.

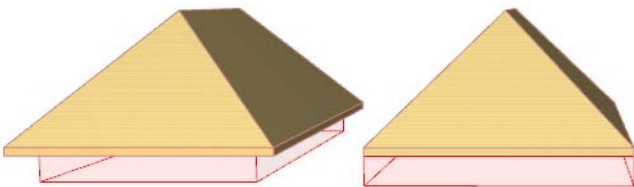


For a Shell, the default body is defined as “downward extrusion” from the Shell membrane.

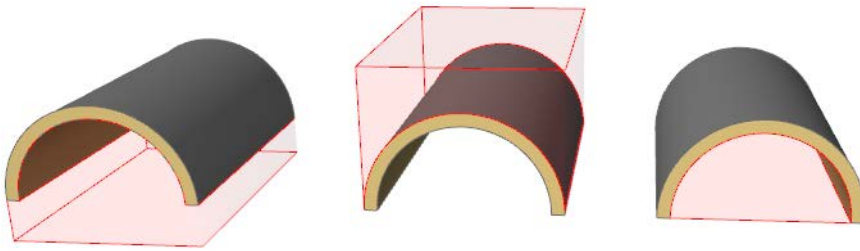


However, you can change the geometry of this trimming body in Roof or Shell Settings: use the “Trimming bodies” control in the Model panel of Shell (Roof) Settings.

The trimming body options are illustrated here for both Roofs and Shells:



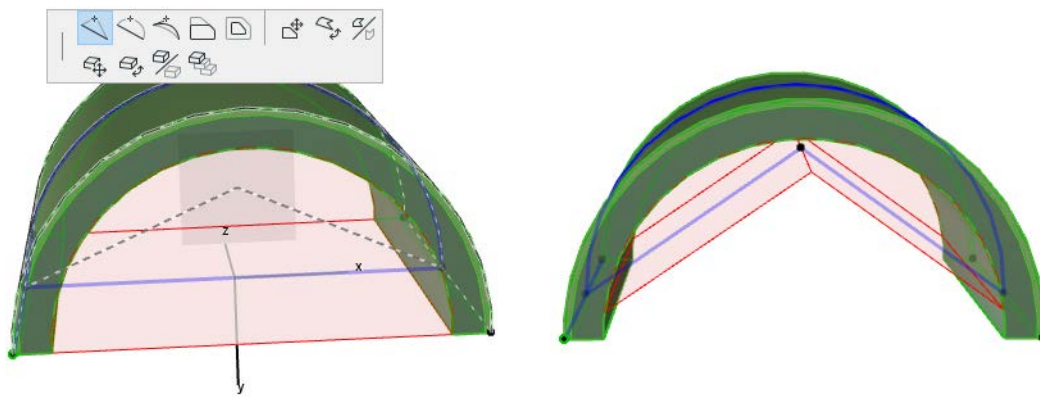
Pivot Lines down - Contours down



Downward extrusion - Upward extrusion - Editable

Editable Trimming Body (Shell)

For a Shell whose trimming body is set to “Editable”, the selected Shell will show a line connecting the two ends of the Shell profile. (This line represents the limit of the trimming body.) This line is editable, allowing you to change the shape of the trimming body.



Note: If you created a profile that is a closed polygon, you cannot edit this bottom line.

Trim Multiple Selected Elements: Automatic Trim

Use the automatic Trim workflow if you want to trim selected multiple elements to each other using the default trimming logic, without any special requests for which pieces should remain and which to eliminate, and without specifying which Roof or Shell should do the trimming job.

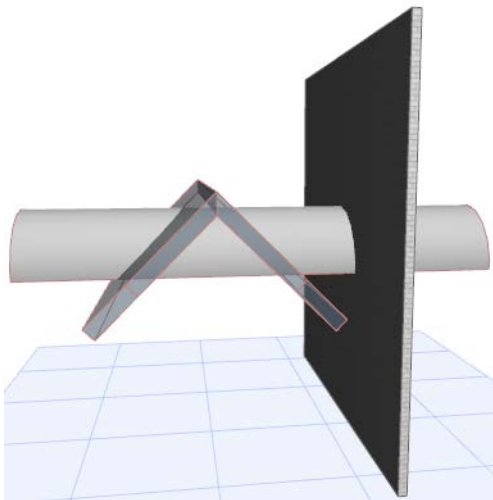
For a more specific workflow in special situations, see: [Trim Elements to Roof: Manual Trim](#).

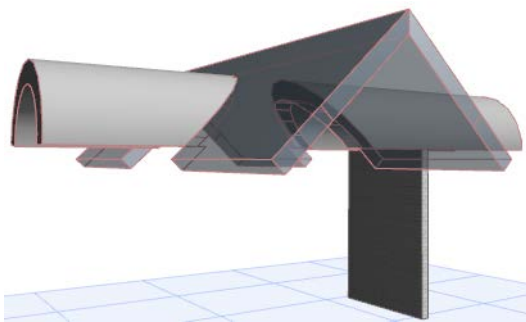
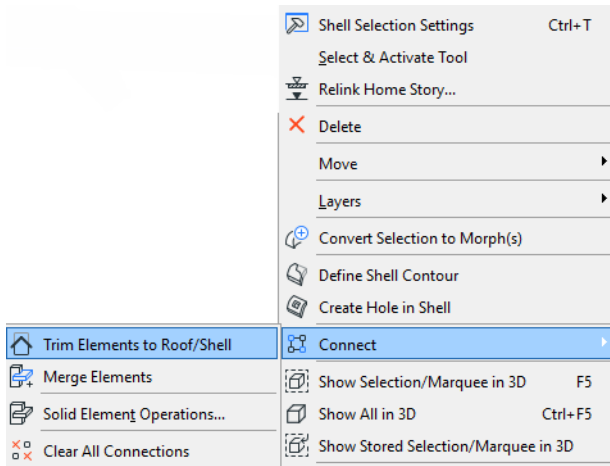
For the regular element trimming command, see: [Trim Elements Using Scissors](#).

The default trimming logic works as follows:

1. You select multiple elements which include one or more Roofs or Shells.
2. Select the **Design > Connect > Trim Elements to Roof/Shell** command (or use context menu).
3. From the Trim Elements dialog box, choose the default **Use Roofs/Shells from current selection** and click **Trim**.
4. Archicad will automatically use all Roofs and Shells in the selection to trim each other, and other selected model elements, as follows:
 - Roofs and Shells trim each other so that any element parts that fall inside of the other's trimming body are eliminated. That is, the outside parts remain; what falls *inside* is cut out (The intersection priority among trimmed Roofs and Shells is determined by their respective Building Materials.)
 - The trimming bodies of the involved Roofs and Shells are merged into a single trimming body.
 - This single trimming body proceeds to cut the non-Roof and non-Shell model elements (e.g. Walls) such that what falls outside the Roof/Shell structure is eliminated.

This process is carried out automatically and immediately without any further input from you.





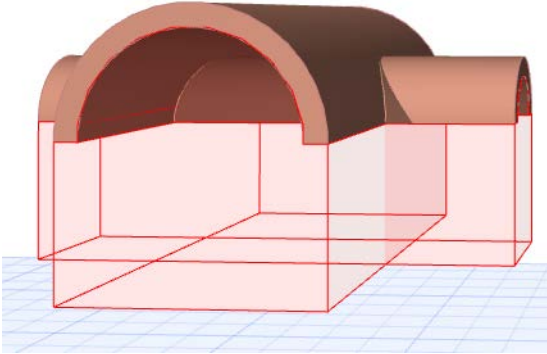
See the following descriptions:

Automatic Trim: Example 1

Automatic Trim: Example 2

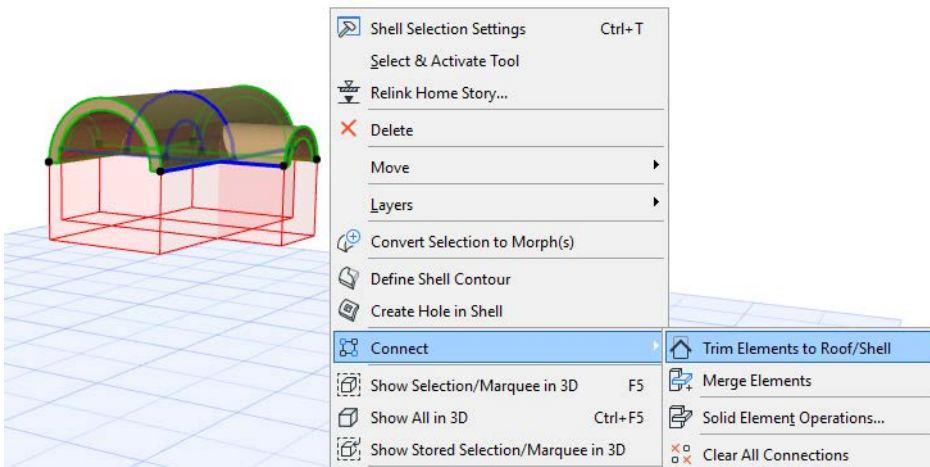
Automatic Trim: Example 1

Suppose we have the two intersecting Shells shown here and wish to trim them to each other.

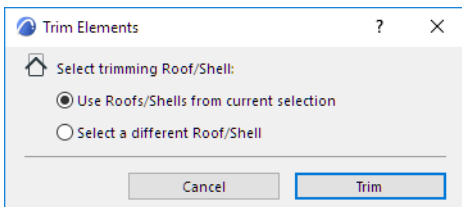


Before you start, you can turn on the trimming bodies if you want to see them while you work. (**View > On-Screen View Options > Trimming Bodies**)

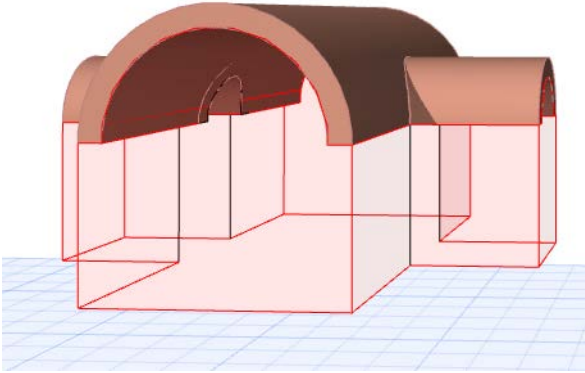
1. Select both Shells.
2. From the context menu, go to **Connect** and view the available commands.



3. Choose **Trim Elements to Roof/Shell**.
4. In the appearing dialog box, click **Trim**.

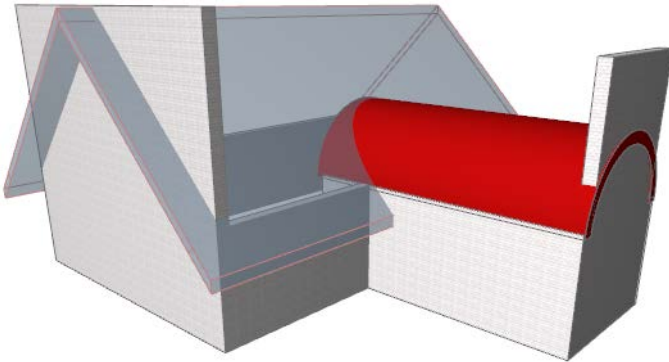


5. View the result in 3D. The internal parts of both elements that coincided with each other's trimming body have been eliminated. Note that the trimming bodies of the two Shells have been merged into a single trimming body.

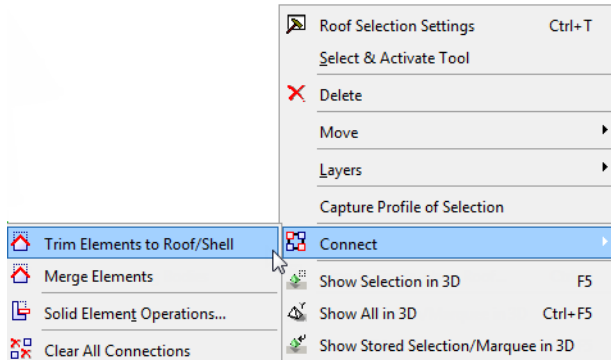


Automatic Trim: Example 2

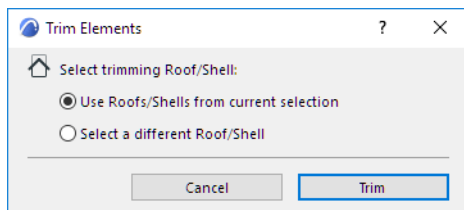
Suppose you have the following Shells and Walls:



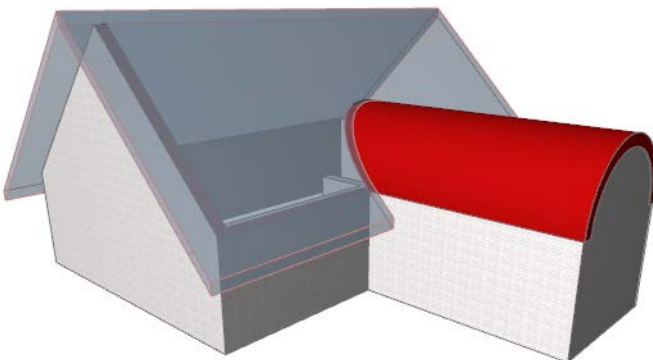
1. Select all the elements shown (Roof, Shell, Walls).
2. From the context menu, go to Connect and choose **Trim Elements to Roof/Shell**.



3. In the appearing dialog box, click **Trim**.



4. View the result in 3D:



For another example of an automatic trim, see [Cross-Gable Roof](#).

Trim Elements to Roof: Manual Trim

Use the manual Trim workflow if you want to trim one or more model elements to a specific Roof or Shell while deciding which parts (e.g. inside or outside) of the trimmed elements should remain.

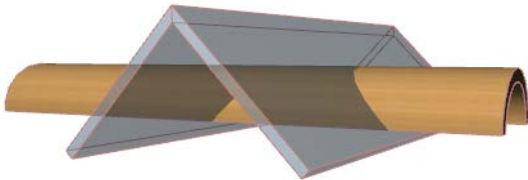
For the automatic trim logic, to trim several selected elements automatically in one step, see: [Trim Multiple Selected Elements: Automatic Trim](#).

For the regular element trimming command, see: [Trim Elements Using Scissors](#).

Manual Trim: Example

Suppose you have two intersecting Shells as shown. (A V-shaped Shell and a tunnel-shaped Shell running through it.) You wish to trim them so that

- *only* that part of the tunnel-shaped Shell that is *inside* the V-shell should remain; and
- the outside parts of the tunnel should be chopped off; and
- the V-shaped Shell should be cut to make holes for the tunnel



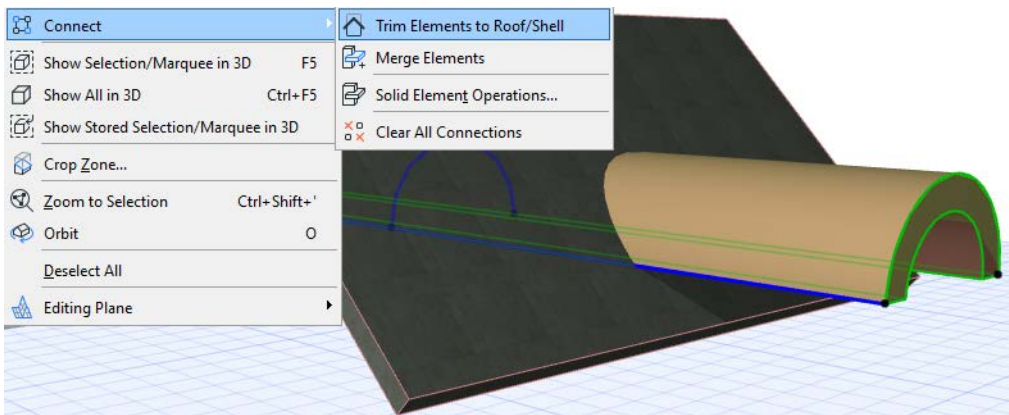
The automatic trimming logic will not suffice; you would end up with the inside part missing, which is not what you want:



See [Trim Multiple Selected Elements: Automatic Trim](#).

Instead, use the following trimming workflow:

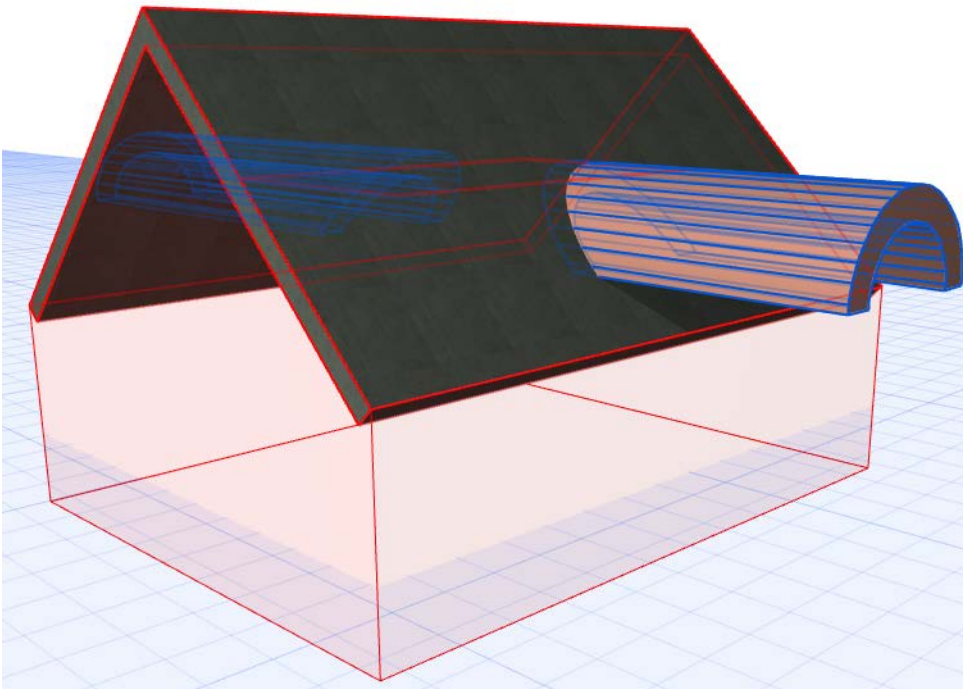
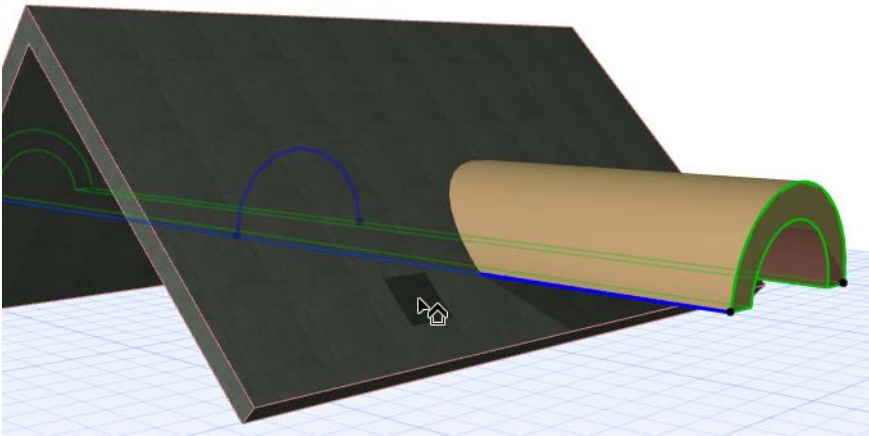
1. Select the Shell you want to trim (the tunnel).
2. From the context menu, choose **Connect > Trim Elements to Roof/Shell**.



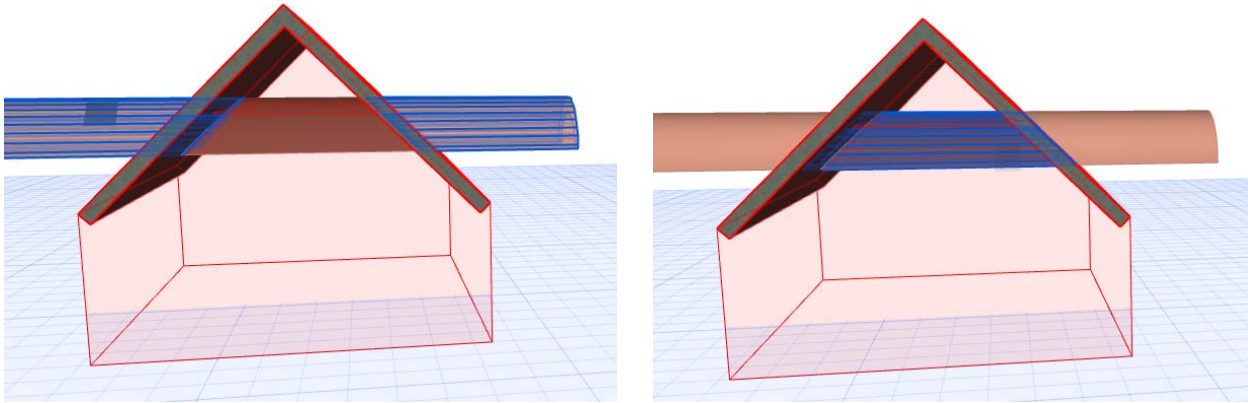
3. Now you must select the trimming element.

Note that the cursor has changed to a little roof. Use it to click the element to use as the trimming element (here, the V-shaped shell, with red feedback).

Note: If Trimming Bodies are not on, they will appear here automatically for the duration of the trimming operation.

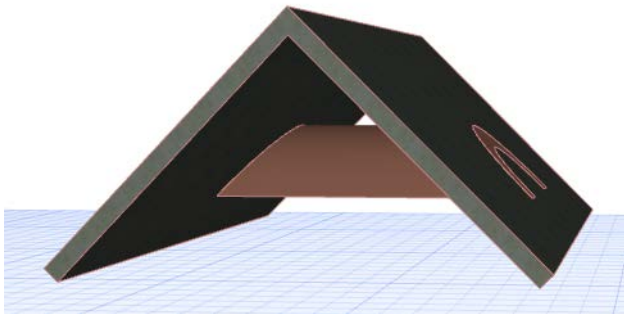


- Now you must select the part of the tunnel that you want to *retain*. Move the cursor over the parts of the tunnel (inside and outside of the V-shell) and see how the blue feedback identifies them: this indicates the options for which parts of the tunnel can be retained after the trim:



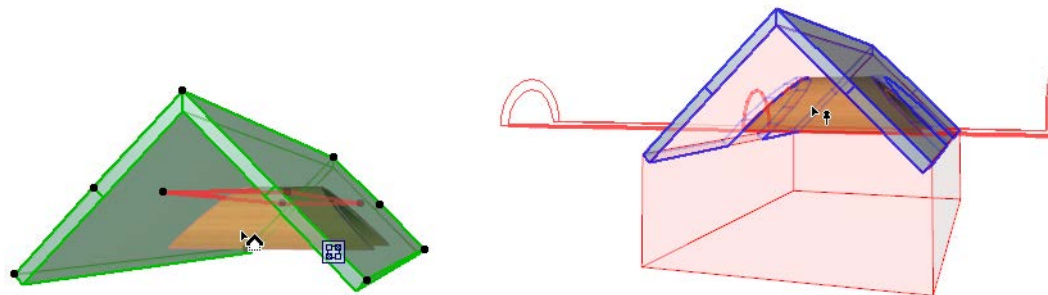
- You want to **retain** the part that is inside, so click on that part. The trim is executed.

Note: Alternatively, Ctrl-click one of the parts to eliminate it. The feedback will always show in blue the part that will be retained.



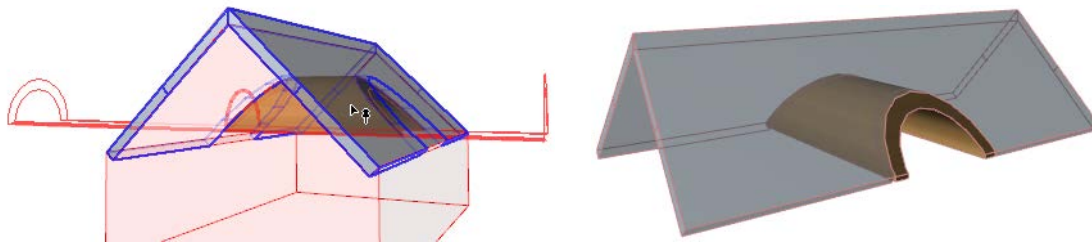
Now you must trim the other element, the V-shaped Shell, to cut holes for the tunnel. Repeat the trimming process:

- Select the Shell you want to trim (the V-shell).
- From the context menu, choose **Connect > Trim Elements to Roof/Shell**.
- Now, with the little roof cursor, you must choose the trimming element: in this case, the tunnel. It will now be shown with red feedback.



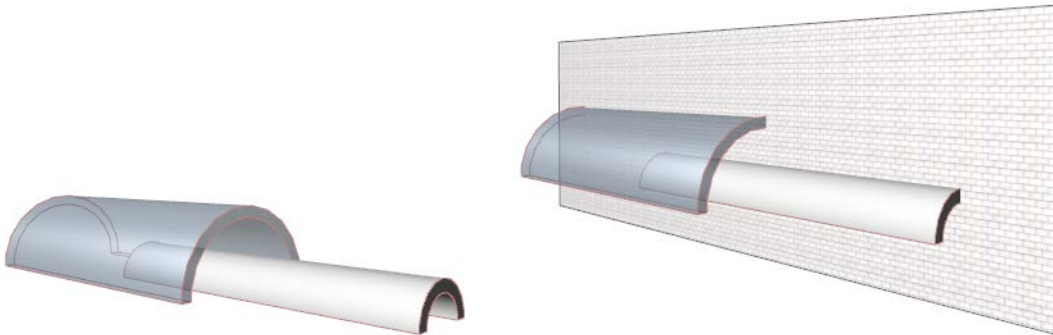
- Now select the part of the V-shell that you want to retain. Move the cursor and see how the blue feedback identifies the options for which parts can be retained after the trim:

5. Click on the part of the V-Shell to be retained. The trim is executed.



Merge Trimming Elements

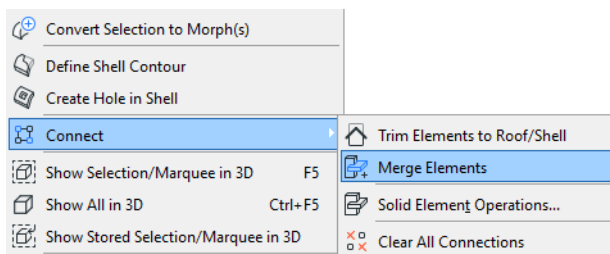
Suppose you start with the following two-Shell design, and then run a Wall through it.



You want to trim the Wall in such a way that it does not extend above either Shell. Yet the Shells themselves should remain intact.

The solution is to combine the trimming bodies of the two Shells, without trimming the Shells themselves. Then you will use the combined trimming bodies to trim the Wall in one step.

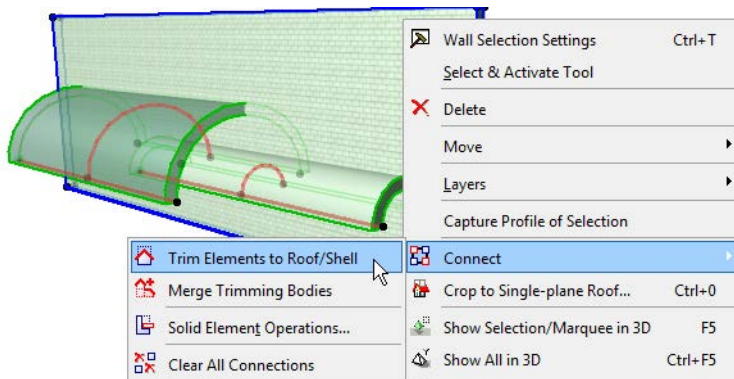
1. Select both Shells.
2. From the context menu, choose **Connect > Merge Elements** (or **Design > Connect > Merge Elements**).



You have merged the Shells' trimming bodies, without trimming off any parts of the Shells themselves.

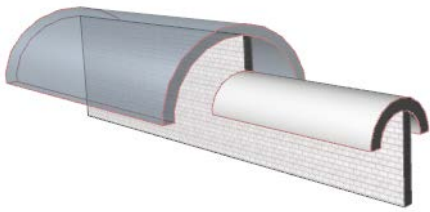
Now, using this merged but intact Shell complex as the trimming element, you can do the automatic trimming procedure on the Wall:

1. Select the Wall and the Shells together.
2. From the context menu, choose **Connect > Trim Elements to Roof/Shell**.



3. In the appearing dialog box, click **Trim**.

4. View the result in 3D.



Merge Elements: Roofs, Shells, Morphs

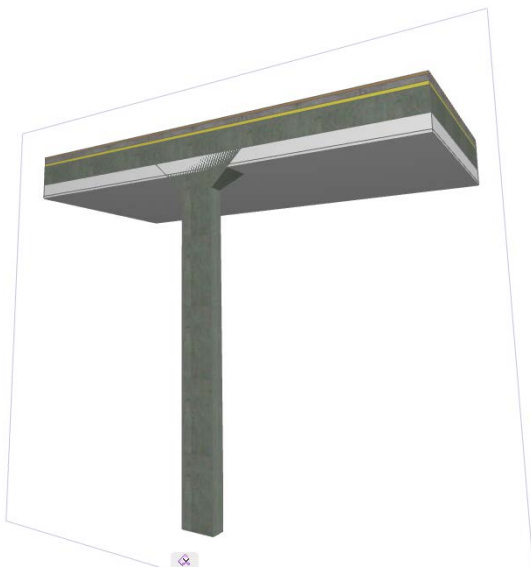
Use the **Connect > Merge Elements** command to enable priority-based intersections among any number of Roof/Shell/Morphs and other elements. (At least one of the merged elements must be a Roof, Morph or Shell.)

Note: Alternatively, you can use Roofs and Shells to trim other elements (not just merge them): this is another way to enable priority-based connections for Roofs/Shells, as well as physically trimming the target element.

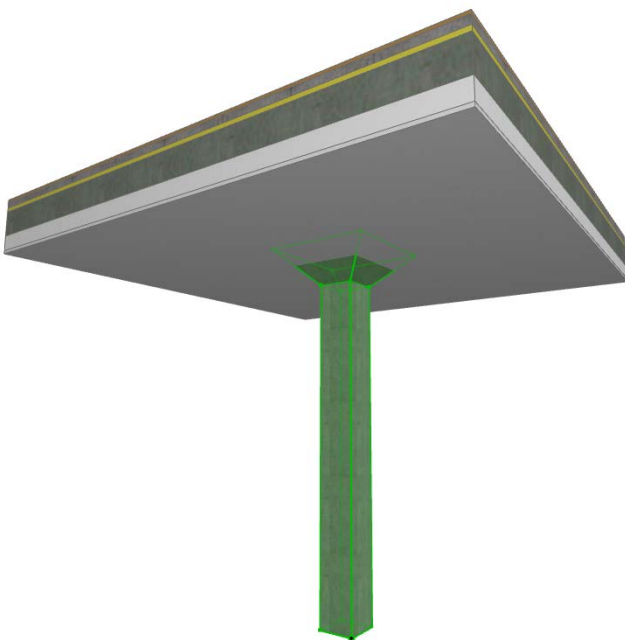
[See *Trim Elements to Roof or Shell*.](#)

Merge Elements: Example

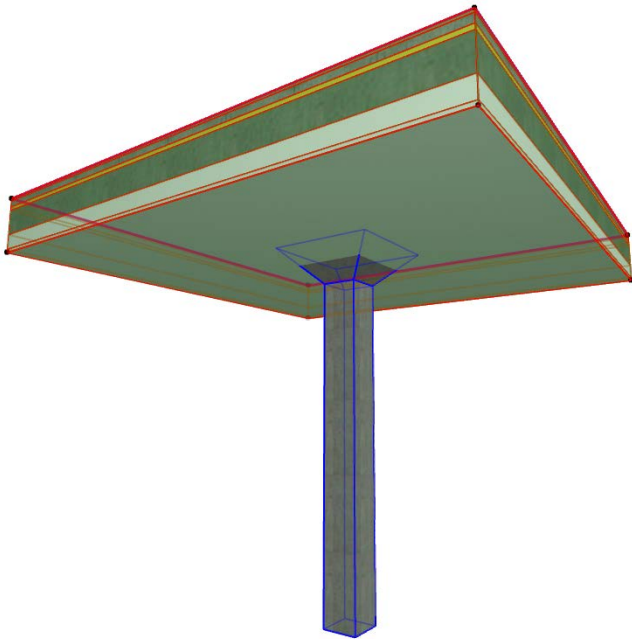
1. Suppose you have two colliding construction elements: a Morph and a composite Slab.



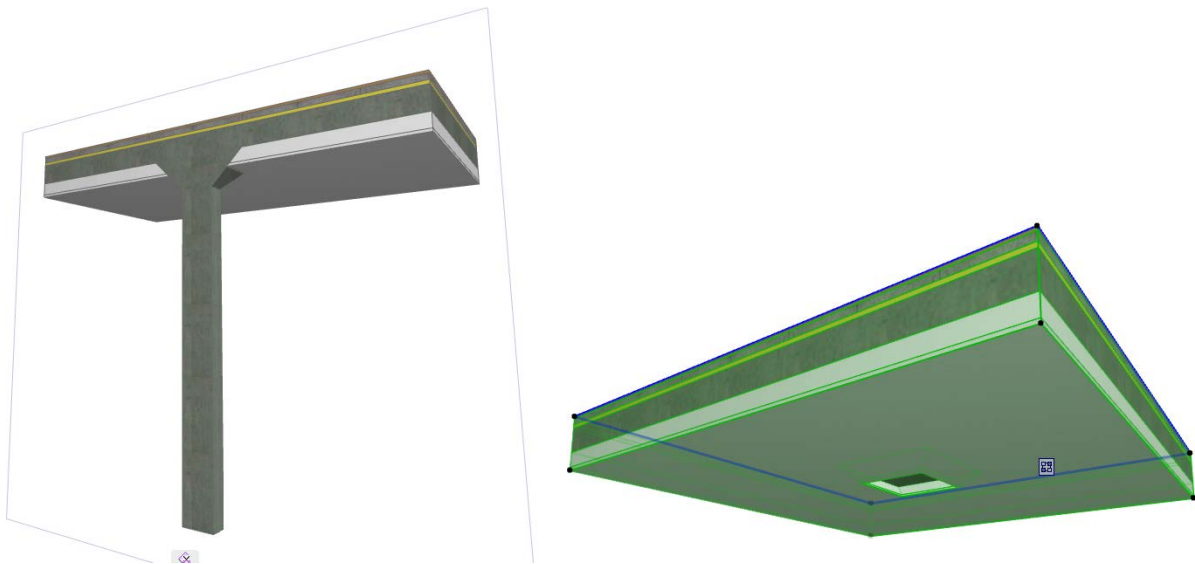
2. Select the Morph.



3. Use **Design > Connect > Merge Elements**. You are prompted to choose the element you want to merge it with.



4. Click on the target element. The two elements are now merged, with their priority-based skin intersections correct.



Note: You can also select both elements before using **Design > Connect > Merge Elements**. In this case, you don't have to specify any more elements, and the merge occurs in one step.

For general information on how intersections (junctions) work among construction elements:

[See *Element Intersections*](#).

For information on managing merged elements:

[See *Managing Element Connections*](#).

Managing Element Connections

Connected elements can involve the following operations (located by default in the Design menu):

- Trim to Roof/Shell
- Merge Elements
- Join Wall to Curtain Wall
- Solid Element Operations

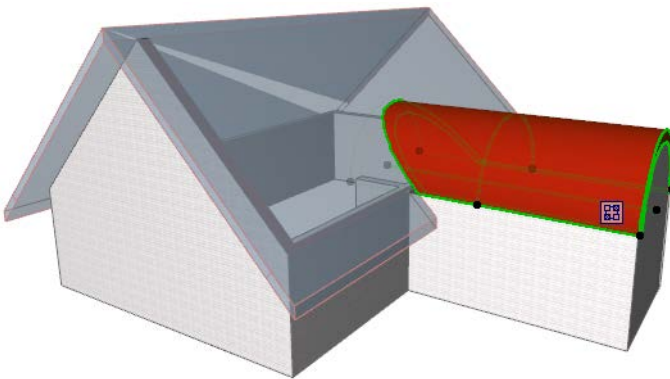
Elements connected to each other are associative: a change to any element will immediately affect all the connected elements. Thus, it can be important to keep track of which elements are connected to which.

The **Connections icon** and the **Connections List** provide in-place interface to help you manage these relationships.

The following examples involve Trimming relationships, but the feedback works the same way for Solid Element Operations, Merged Elements and joined Curtain Walls.

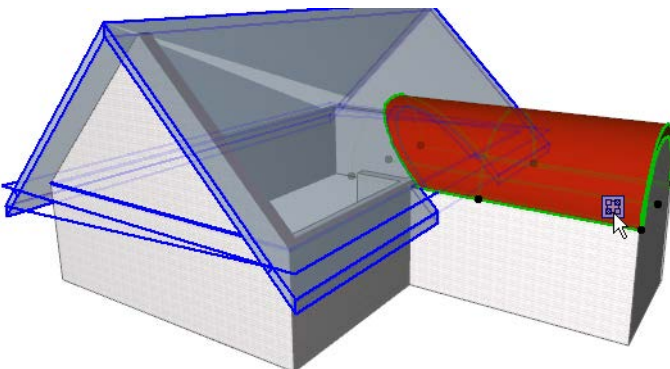
Managing Trims

If you select a single model element that is part of a Trim relationship, you will see a Connections icon appear:



Note: The Connections icon will not appear if you have selected multiple elements.

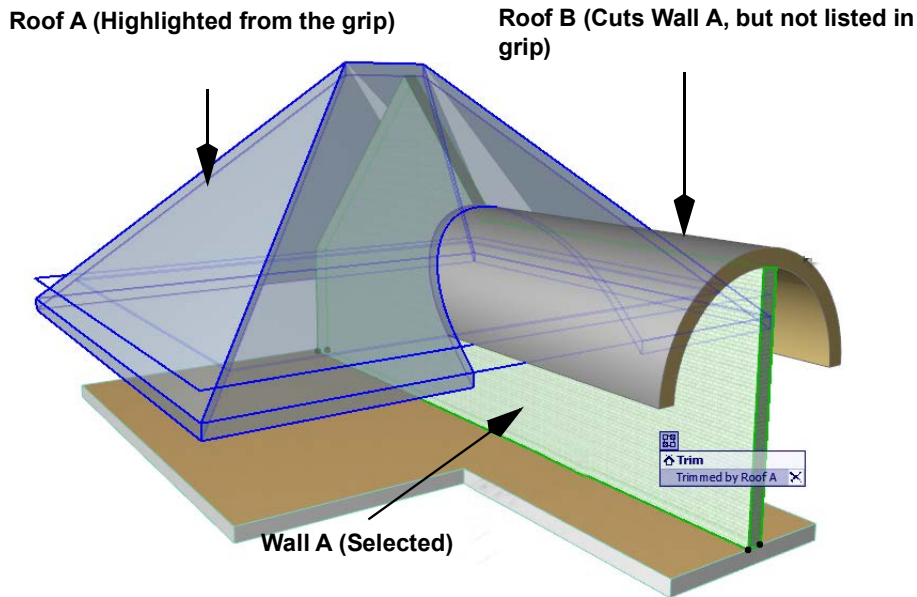
If you move your cursor on top of this icon, any connected element will be highlighted in blue.



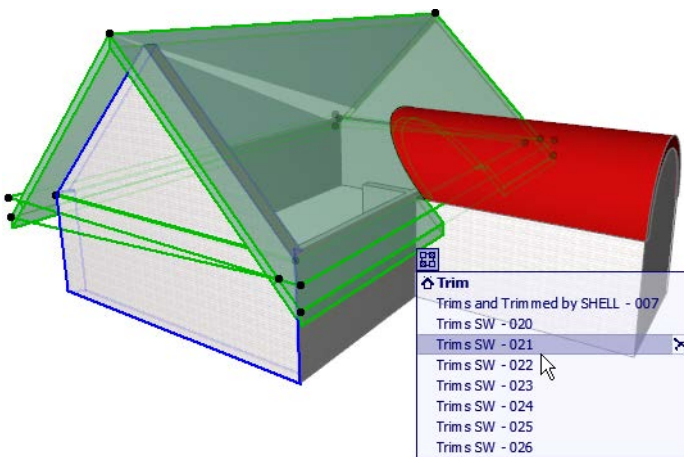
If the selected element (such as the Roof, below) is involved in multiple Trim relationships, click the icon to view each connection relationship in a list.

If Roof A is connected to Roof B, and Wall A is trimmed by Roof A, then Wall A will be trimmed by Roof B as well.

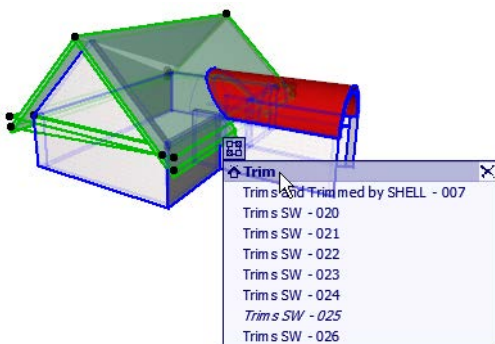
Wall A's connection grip will list only Roof A, since only they have direct connections.



Move your cursor through the list: as you hover over any listed item, that connected element is shown with a highlight in the project (e.g. the gable Wall in the image below).

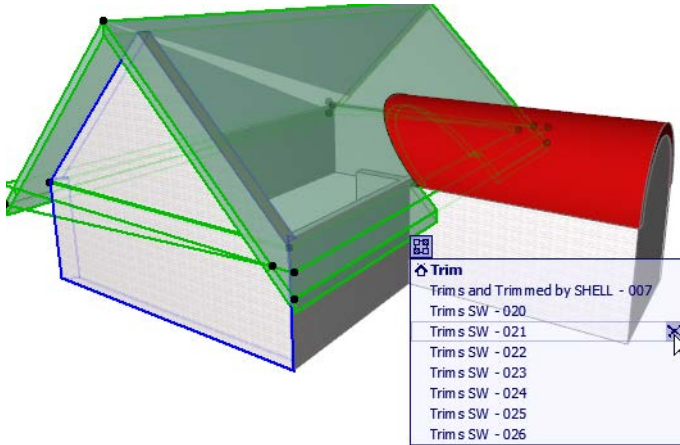


To highlight *all* of the elements connected to the selected element, hover over the header (e.g. **Trim**) in the connection list:

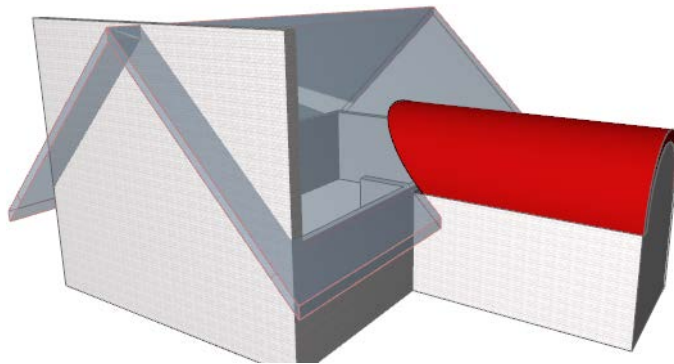


If the list includes connections to elements not visible in the current view (e.g. it is on a hidden layer, or on a different story), that connection item is listed in italic type.

To clear a connection (that is, to sever the associative relationship and undo the Trim), click the X icon next to that connection in the list. The connection is cleared. To clear a whole group of Connections, click the X next to the header in the list (e.g. **Trim**).

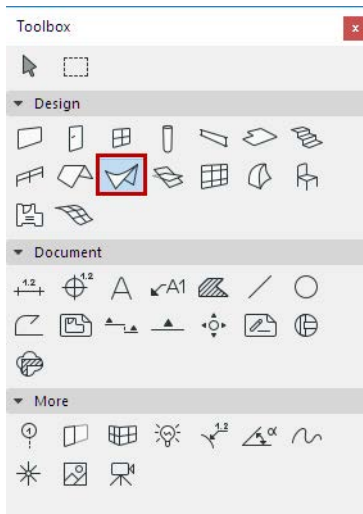


To clear multiple connections all at once, select one or more model elements. From the context menu, choose **Connect > Clear All Connections**. All connections of the selected element(s) are cleared.



Shells

The Shell in Archicad is a real building element with a wide range of creative uses. Use the Shell tool to model elements ranging from a building's entire exterior to a single custom object.



About Shells

Extruded Shells

Revolved Shells

Ruled Shells

Sample Workflows for Shell Creation

Graphical Editing of Shells

Create Hole in Shell

Edit Shell Hole

Define Shell Contour

Customize Edge of Shell or Edge of Shell Hole

Trim Elements to Roof or Shell

Composite Structures

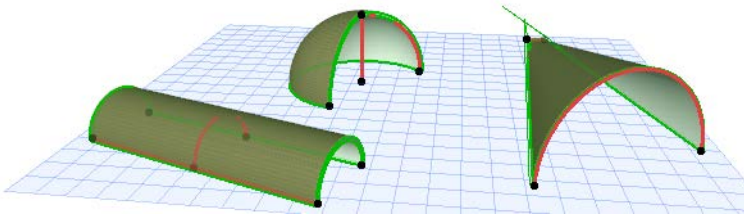
Shell Tool Settings

About Shells

Basic Shell Creation

Shells can be created in either the Floor Plan or 3D Window. Use one of three basic shapes:

- Extruded
- Revolved, or
- Ruled



Creating a basic Shell of any of these types is easy. Then you can develop the Shell further by graphically editing it, freely rotating it, adding openings or cutting its contour into any shape.

See [Graphical Editing of Shells](#).

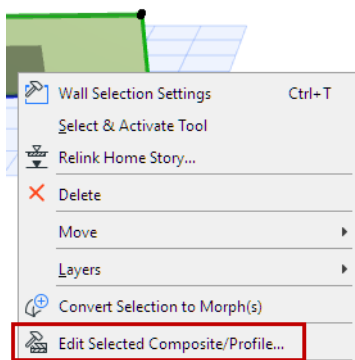
Shell Structure

Like other construction elements, a Shell's essential structure derives from its Building Material, assigned in Shell Tool Settings.

A Shell (like a Wall, Slab or Roof) can have a composite structure. If the structure of the Shell is a composite, then the Shell's thickness is defined in **Options > Element Attributes > Composites**, and equals the sum of the skins' thicknesses.

For a composite Shell, the "Reference Side" surface is the first listed skin of the composite, in Composite Settings.

Note: You can edit a selected composite by accessing its settings from its context menu:

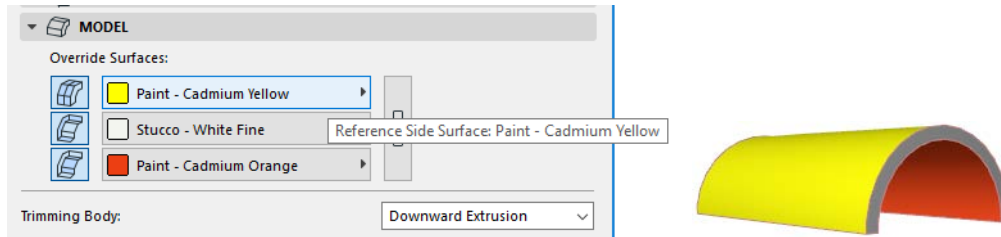


The Shell is constructed out of a membrane (the Reference side) plus a Shell body attached to one side of the membrane. The Shell body is "grown downwards," with the Reference side on top.

The Reference side of the Shell is significant when:

- calculating surface area of the Shell or its skins

- overriding surfaces as needed, in the Model Panel of Shell Settings.



Use **Flip** (from Shell Settings or the Info Box) if you want to move the Shell body to the other side of Shell membrane.



Shell Display on the Floor Plan

- Shell display on the Floor Plan is based on its real 3D geometry. Thus, you will see the effects of Solid Element Operations on Shells on the Floor Plan.
- Shells can display a cover fill, which - as with Roofs - can be set to reflect the vectorial hatching of the Shell's surface, either as defined in the Building Material or (if the override is on) its top surface override. (Enable the **Use Fill from Surface** checkbox under the Cover Fill settings, in Shell Settings.)

Shell Connections

For information on how intersections (junctions) work among construction elements, including Shells:

[See *Element Intersections and Merge Elements: Roofs, Shells, Morphs*](#).

Like Roofs, Shells are trimming elements and participate in associative model connections. This means you can trim Shells to other elements to achieve precise geometries and complex structures.

[See *Trim Elements to Roof or Shell*](#).

Shell Openings

Holes and Skylights can be placed into Shells.

[See *Create Hole in Shell and Skylights*](#).

Gravitate to Shell

Archicad's Gravity feature works on Shells. That is, when placing a new Wall, Column, Beam or Object-type element, the **Gravity** function lets you place it directly on top of an existing Shell, thus taking on the elevation of the element it is placed on.

[See also *Gravity*](#).

Extruded Shells

Use the extruded method to create a Shell by drawing its profile on a plane (either freehand polygon or a default shape), then extruding the profile. For example, you can extrude a Shell over the walls of a building.

See the following sections for basic creation of Extruded Shells:

[Simple Extruded Vault: Parallel to the Floor Plan](#)

[Simple Extruded Vault: Perpendicular to the Floor Plan](#)

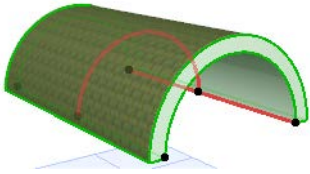
[Extruded Shell with Freely Defined Profile](#)

For more examples, see [Sample Workflows for Shell Creation](#).

[Watch the Video](#)

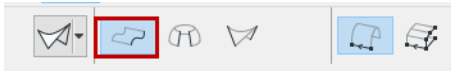
Simple Extruded Vault: Parallel to the Floor Plan

Use this method to place a vault-shaped Shell profile, then extrude it graphically.



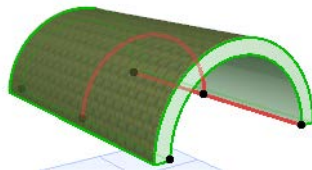
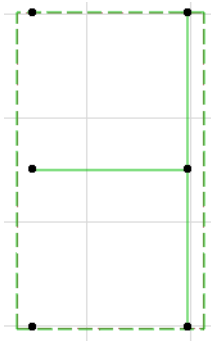
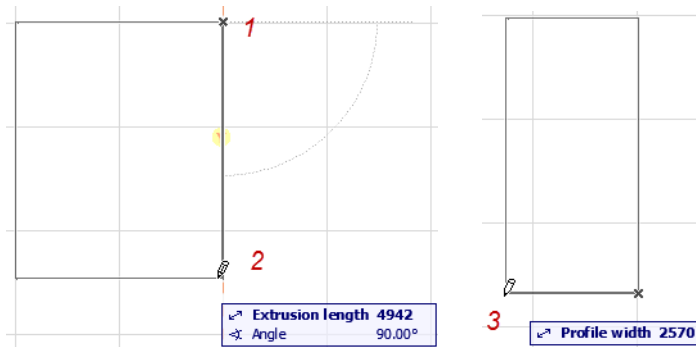
The Shell's profile here is always a half-circle (vault). (You can edit the profile later.)

1. Activate the Shell Tool and choose the extruded geometry method with simple input method.



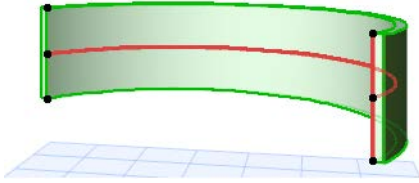
On the Floor Plan or 3D Window:

2. Click twice to define the Shell's extrusion length.
3. Click a third time to define the width of the Shell Profile.
4. The Shell is complete.



Simple Extruded Vault: Perpendicular to the Floor Plan

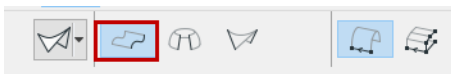
Use this method to place a vault-shaped Shell profile, then extrude it up or down (perpendicular to the Floor Plan).



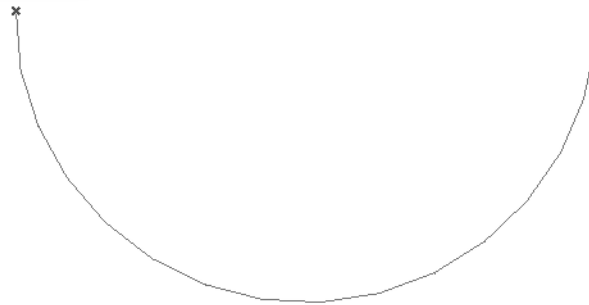
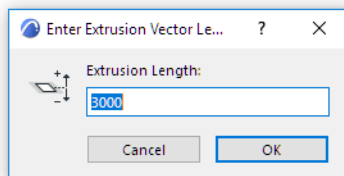
The Shell's profile here is always a half-circle (vault). (You can edit the profile later.)

On the Floor Plan:

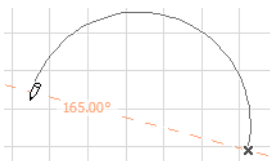
1. Activate the Shell Tool and choose the extruded geometry method with simple input method.



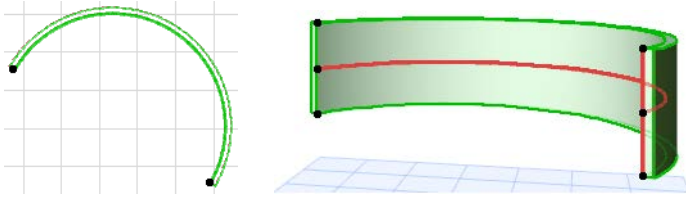
2. Double-click at the place you want to place the Shell. (A temporary profile appears.) Its base elevation depends on the value set in Shell Settings.
3. In the appearing dialog box, enter the length of the Shell's extrusion. A positive number will extrude the Shell upward (toward you). A negative number will extend the vector away from you. Click OK to close the dialog box.



4. Move the cursor to define the Shell position, then click to define the width of its profile.

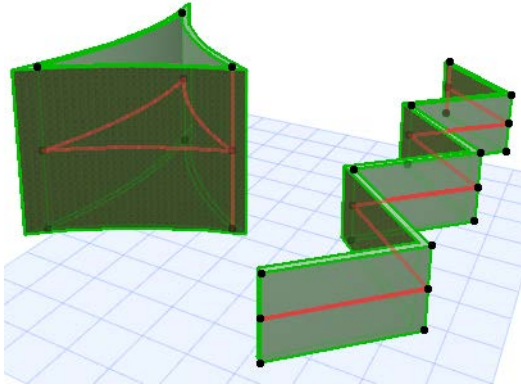


5. The Shell is complete.



Extruded Shell with Freely Defined Profile

Use this method to define the Shell profile with a polyline, then extrude it perpendicularly to the editing plane. Two examples are shown here.

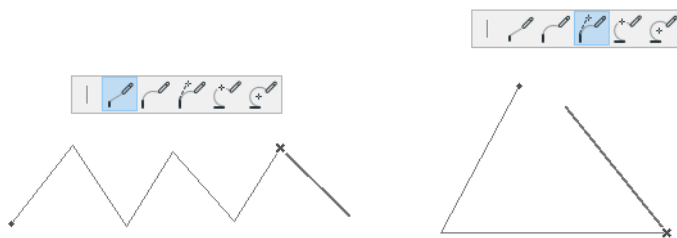


1. Activate the Shell Tool and choose the extruded geometry method with detailed input method.

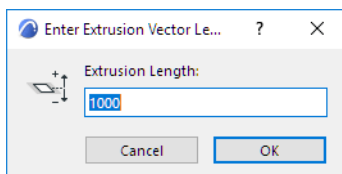


On the Floor Plan:

2. Draw the Shell profile - a polyline or a closed polygon - with the usual polygon input method.

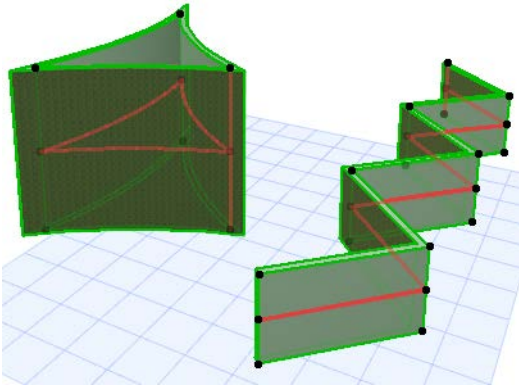


3. Double-click to complete the polyline or polygon.
4. In the appearing dialog box, enter the length of the Shell's extrusion. A positive number will extrude the Shell upward (toward you). A negative number will extend the vector away from you. The extrusion is always perpendicular to the Floor Plan. Click OK.



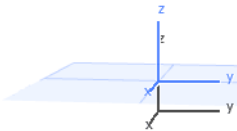
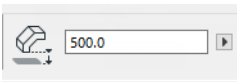
5. The Shell is complete. View it in 3D. (In the 3D window, the profile line of a selected Shell is displayed in a separate color.)

Note: You can change this color using the "Reference Lines in 3D" setting at **Options > Work Environment > Selection and Element Information**.

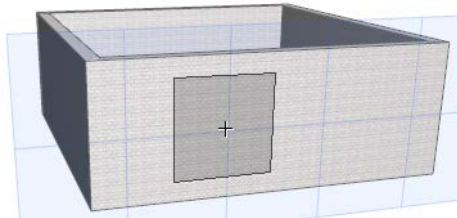
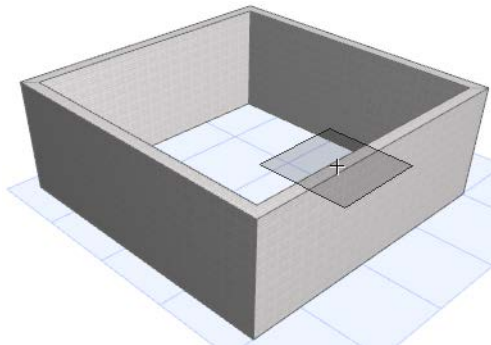


In the 3D Window:

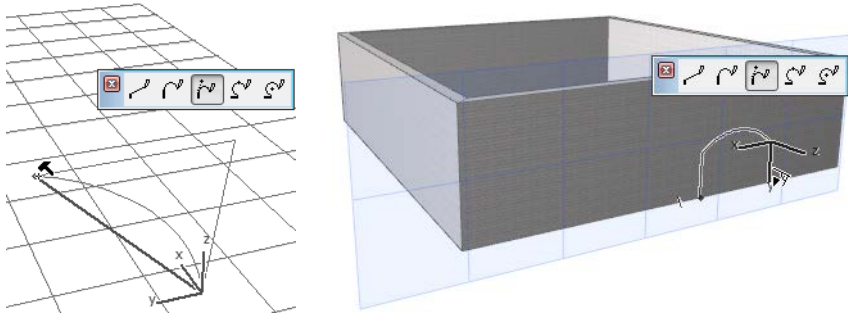
1. Choose the input plane. Either click to find the default plane, or choose another input plane based on existing elements.
 - If you click in “empty space”, the default input plane is at the base elevation defined in Shell Default Settings (here, 500).



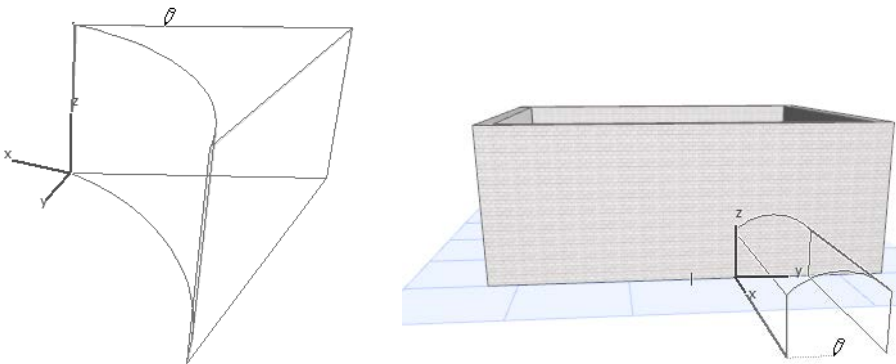
- To place an input plane defined by points or surfaces of an existing element, choose it by clicking on any points and/or edges or surfaces: the automatic feedback (a gray square) will show you where the potential input planes are. When you get the plane you want, click to make it appear.



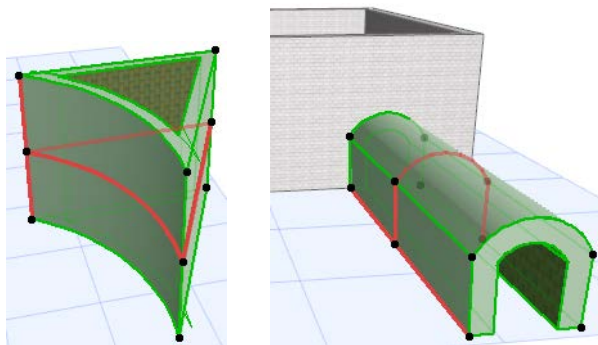
2. Draw the Shell profile - a polyline or a closed polygon - with the usual polygon input methods. (Here, two different examples are shown.)



3. Complete the polygon.
4. Extrude the Shell by dragging the cursor. It will be extruded perpendicular to the input plane.



5. Click to complete.



Revolved Shells



Use the revolved method to create a Shell by defining its profile (either a default shape or a freehand profile), then revolving the profile around an axis.

Related Topics:

[Revolved Shell: Simple Input](#)

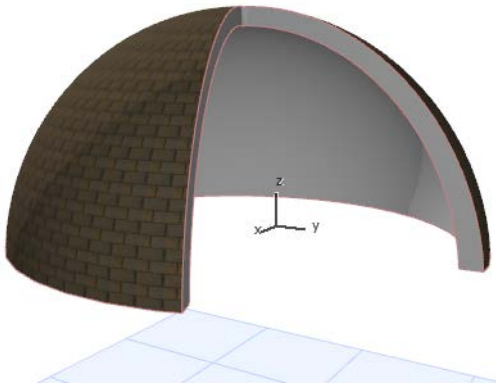
[Revolved Shell with Freely Defined Profile](#)

[Sample Workflows for Shell Creation](#)

[Segment Surface Along Rotation](#)

[Watch the Video](#)

Revolved Shell: Simple Input



In the Floor Plan or 3D window:

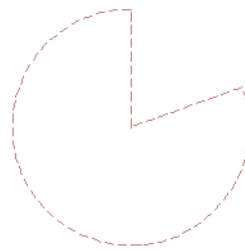
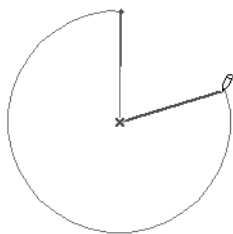
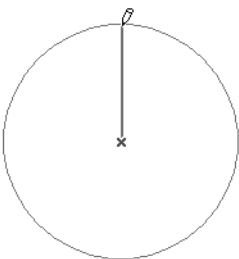
1. Activate the Shell Tool and choose the revolved geometry method with simple input method.



2. Click to define the Shell axis at the center of the dome. (The initially placed axis is always vertical, though you can edit the axis slant later.)
3. Drag the cursor to define the size (width) of the Shell. Click where you want to begin revolving it around the axis.



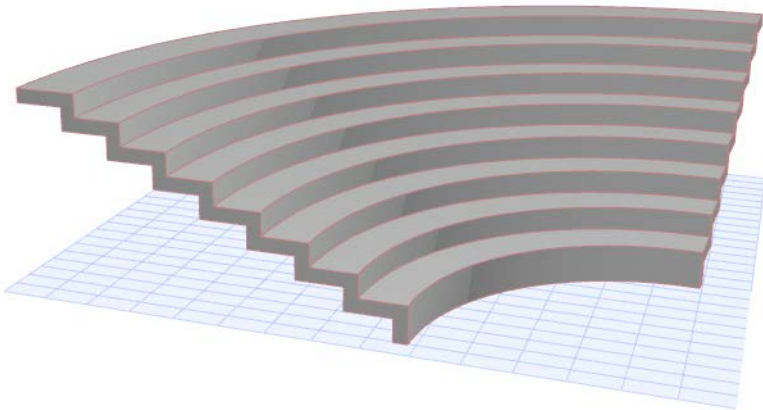
4. Drag the cursor and click to complete the angle of revolution.



5. View the result in 3D.



Revolved Shell with Freely Defined Profile

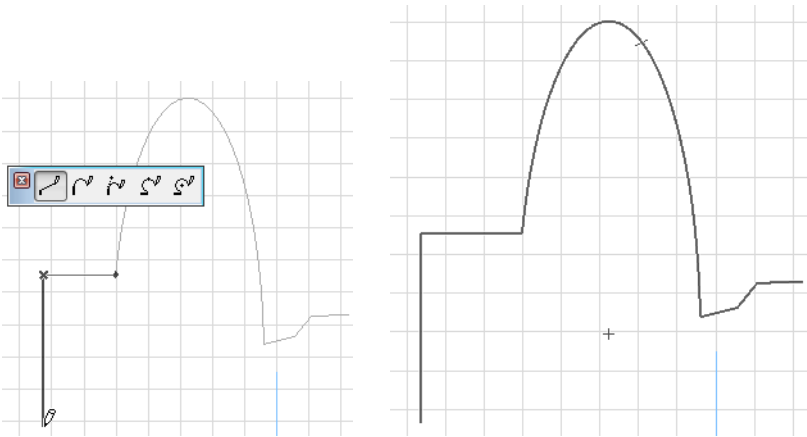


1. Activate the Shell Tool and choose the revolved geometry method with detailed input method.

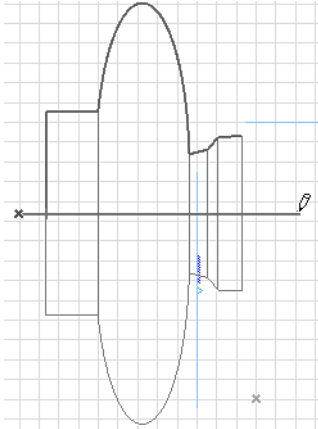


On the Floor Plan:

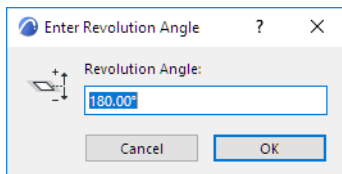
2. Draw the Shell profile - a polyline or a closed polygon - with the usual polygon input method.
3. Double-click to complete the polyline or polygon.



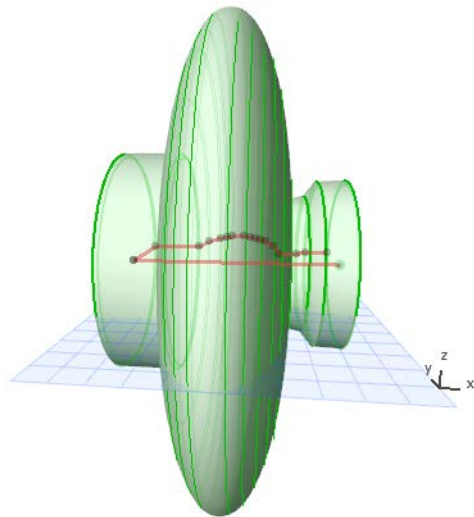
- Click twice to draw the revolution axis. The initially placed revolution axis is always parallel to the Floor Plan, though you can edit it later.



- In the appearing dialog box, enter the Shell's revolution angle. A positive number will revolve the Shell toward you. A negative number will revolve the Shell away from you.

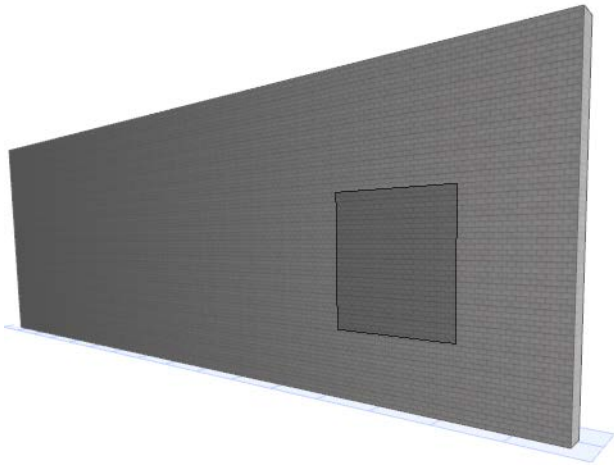


- Click OK to complete the Shell. View it in 3D.

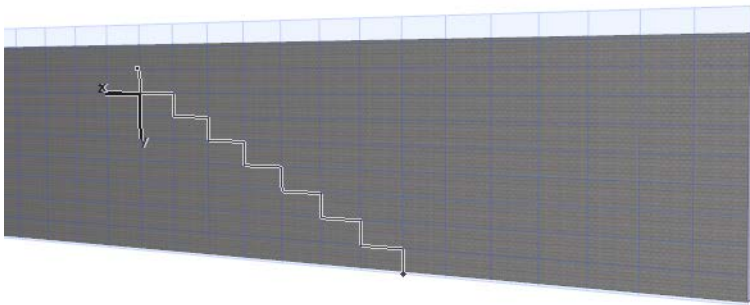


In the 3D Window:

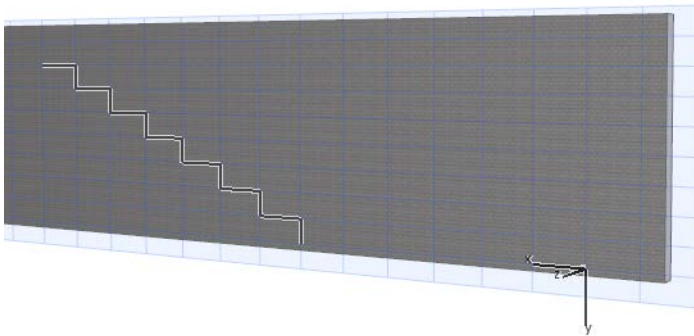
- Choose the input plane. Either click to find the default plane, or choose another input plane based on existing elements. This is the plane on which you will draw the profile and then its revolution axis.
 - If you click in “empty space”, the default input plane is at the base elevation defined in Shell Default Settings.
 - To place an input plane defined by points or surfaces of an existing element, choose it by clicking on any points and/or edges or surfaces: the automatic feedback (a gray square) will show you where the potential input planes are. When you get the plane you want, click to make it appear.

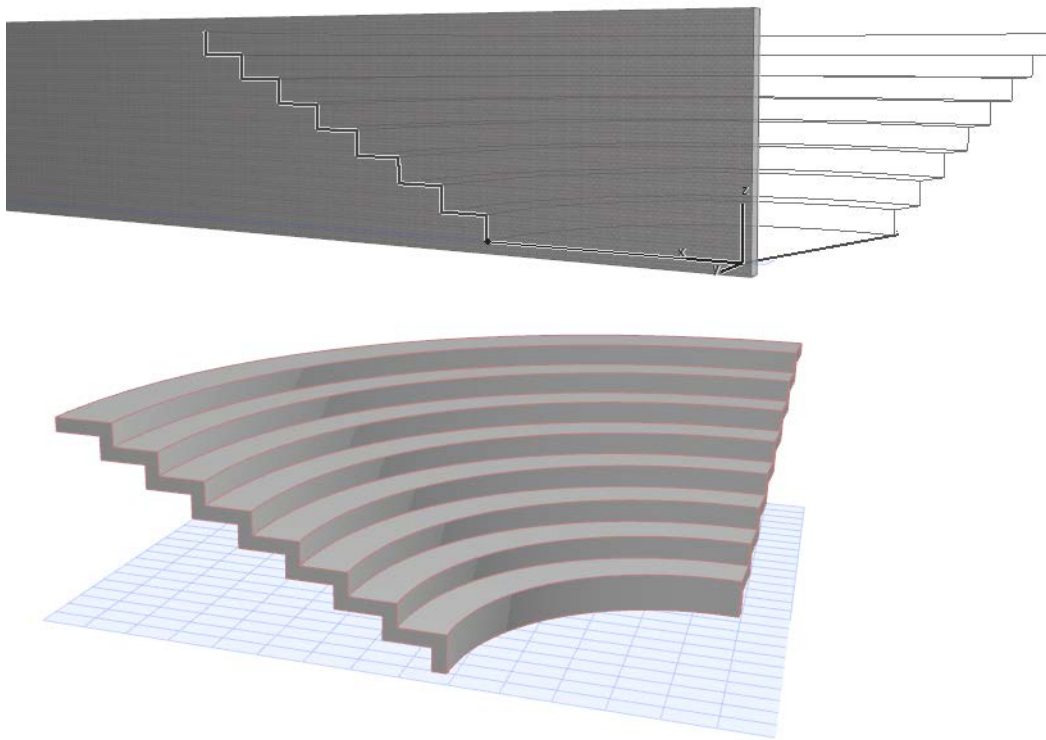


2. Draw the Shell profile - a polyline or a closed polygon - with the usual polygon input methods.
3. Complete the polygon.



4. Click to define the center of the revolution axis.
5. Move the cursor to complete the revolution axis. (Navigate in the 3D view to find the best angle for seeing the feedback.)





6. The Shell is complete.

Ruled Shells

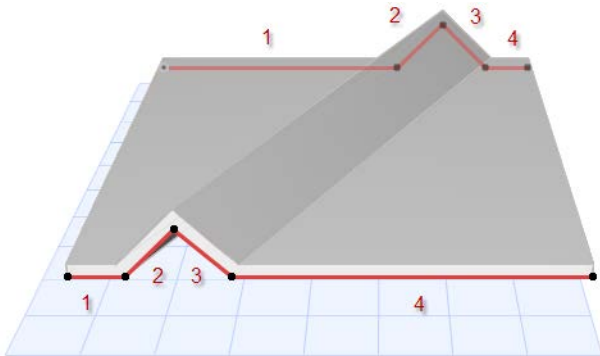
A Ruled Shell requires two separate profiles in any two planes.

Each of the two profiles, like line-type elements, consists of segments and intermittent nodes. To create a ruled Shell, Archicad will draw connecting lines between the respective segments and nodes of each of the two profiles. The Shell is then “grown” onto the resulting geometry.

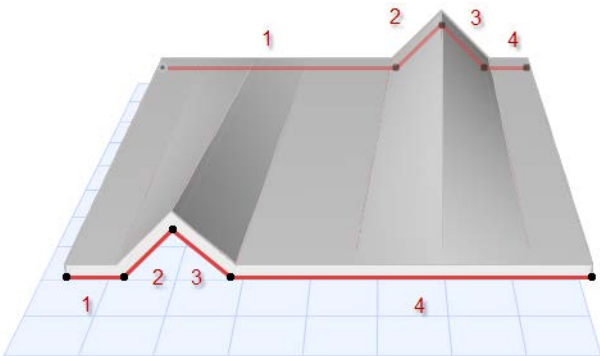
Ruled Shell Morphing

The logic (called “Morphing rules”) by which the two profiles are connected can take two forms. Choose the desired icon from the Info Box or the Geometry and Positioning Panel of Shell Settings, with the Ruled Construction method selected.

- **Paired:** Profiles are connected by mapping each segment/node on one profile to a single segment/node on the other profile. If the segments/nodes cannot be matched up one-to-one, then they are paired off proportionately.

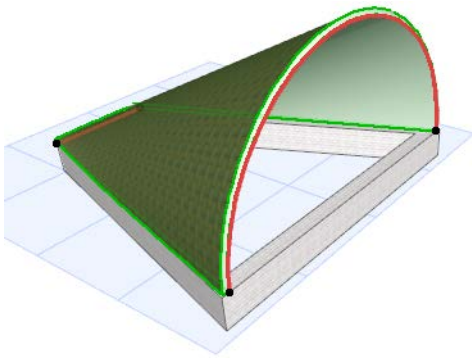


- **Smooth:** Profile polygons are connected by joining the respective segments that are of proportionate lengths, without calculating the position of nodes. This may be advantageous if you have drawn the profiles based on a spline, which creates a large number of seemingly random nodes.



Related Topics:

[Ruled Vaulted Shell](#)



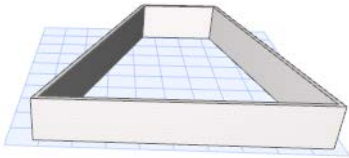
For more examples, see [Sample Workflows for Shell Creation](#).

Ruled Vaulted Shell

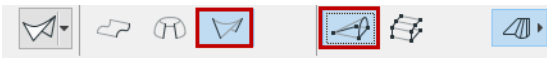
To create a Ruled Shell with the simple input method, you will define two profiles on parallel planes. With this input method:

- The first profile is always a vault shape.
- The second profile is always a straight line.

Here we will place a vaulted ruled Shell atop this wall complex.

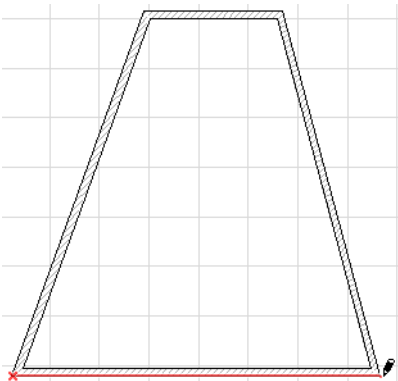


1. Choose the Shell tool's Ruled geometry method and the Simple input method.

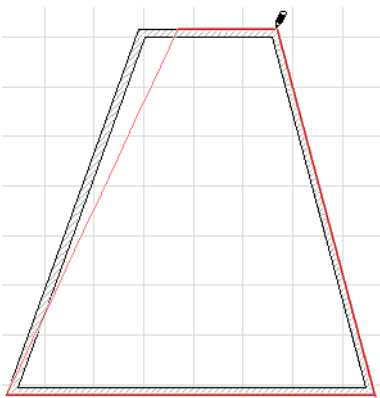


On the Floor Plan:

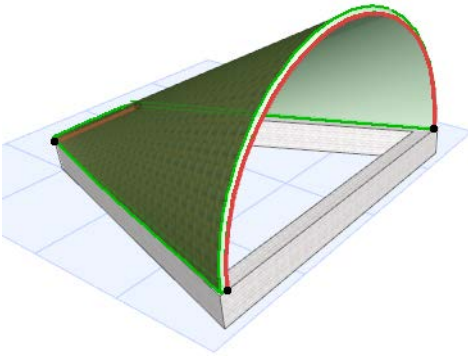
2. Click on each endpoint of the lower wall segment to define the width of the first, vault-shaped profile.



3. Drag the cursor to the opposite wall segment, then click on each endpoint of that wall to define the width of the second, straight-line profile.

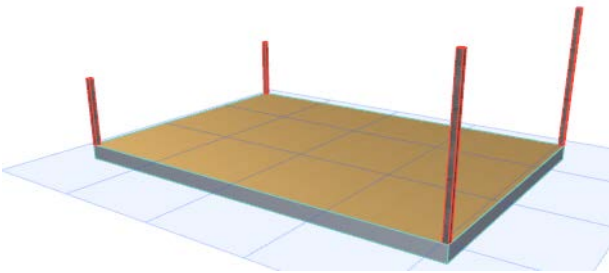


4. The Ruled vault-shaped Shell is complete. View in 3D.



In the 3D Window:

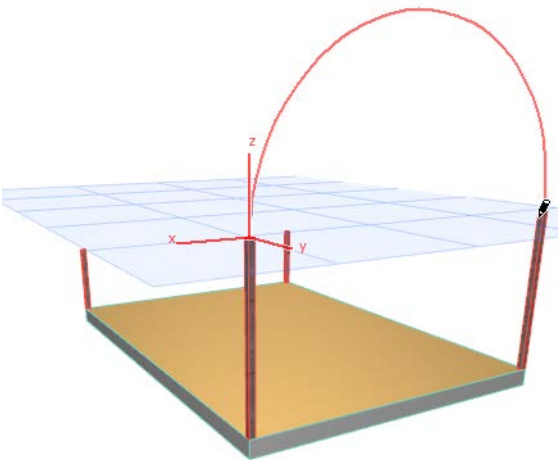
Here we will place a vaulted ruled Shell atop two pairs of columns of different heights.



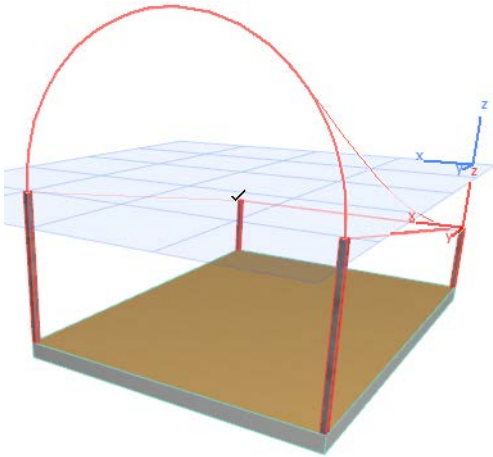
1. Choose the Shell tool's Ruled geometry method and the Simple input method.



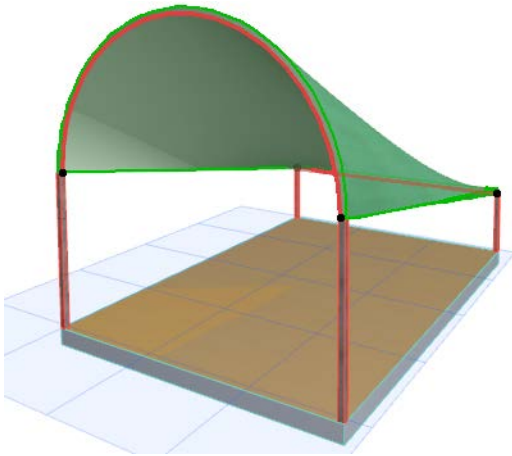
2. Click on each of the two taller columns to define the width of the first, vault-shaped profile.



3. Drag the cursor to the opposite, shorter columns and click on each to define the width of the second, straight-line profile.



4. The Ruled vault-shaped Shell is complete. View in 3D.



[Watch the Video](#)

Sample Workflows for Shell Creation

The following sections show you how to create the following Shell types, step by step:

Extruded Shell: Wavy Canopy Roof

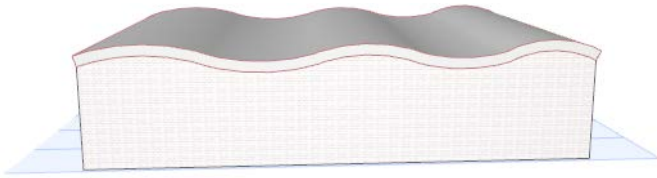
Revolved Shell: Arced Triangular Roof Complex

Revolved Shell: Define Contours in Section

Ruled Shell: Tennis Court Shell

Ruled Shell: Twisted Tower

Extruded Shell: Wavy Canopy Roof



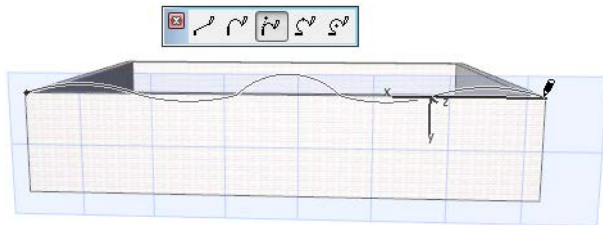
1. Activate the Shell Tool and choose the extruded geometry method with detailed input method.



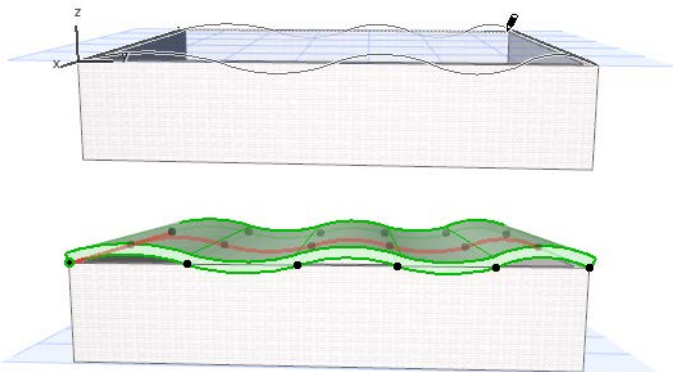
2. In the 3D window, click on a surface, or click three points, to define an input plane onto which to draw the profile.



3. Draw the desired profile, in this case a wavy line, with the usual pen palette options. Click to complete.



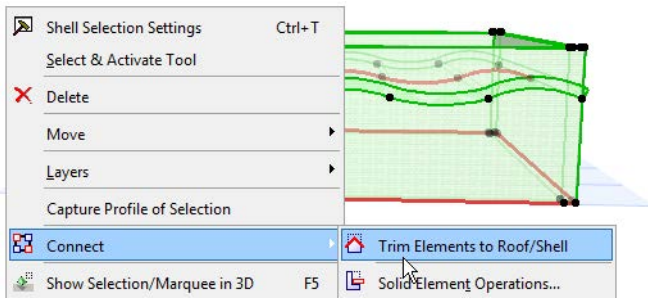
4. Next, extrude the profile: extend it back to the far wall and click to place it. View the result.



5. Raise the Wall height so that it intersects the Shell, so that you can trim the Wall to the Shell.

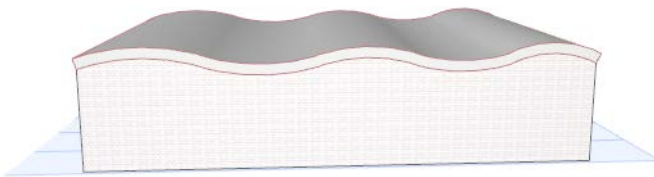


6. Select the Wall and Shell which you want to trim. From the context menu, choose **Connect > Trim Elements to Roof/Shell**.

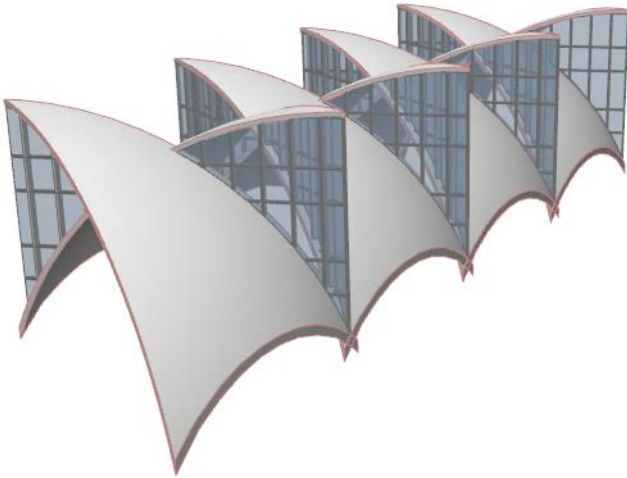


For more information on Shell connections, see [Trim Elements to Roof or Shell](#).

View the result.



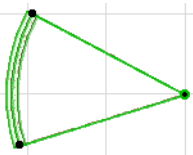
Revolved Shell: Arced Triangular Roof Complex



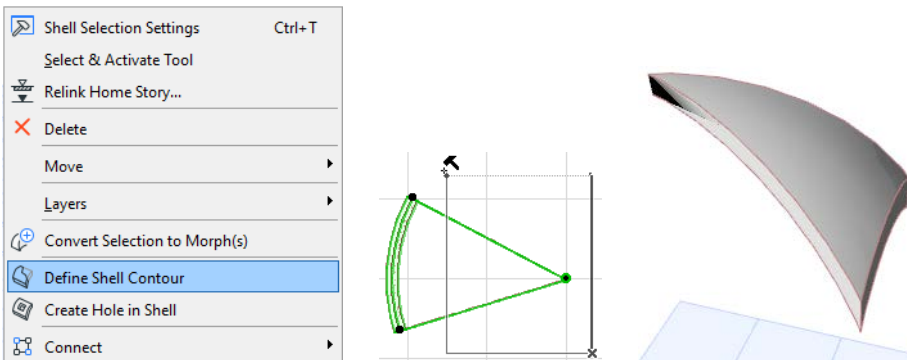
1. On the Floor Plan, activate the Shell Tool and choose the Revolved geometry method with simple input method.



2. Create a Revolved Shell with a 45-degree angle of revolution.

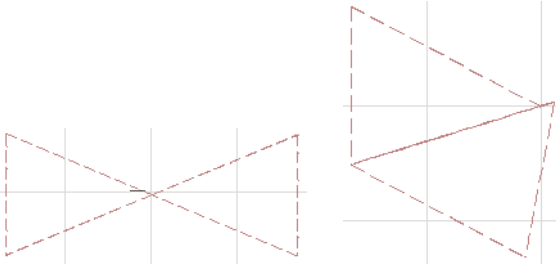


3. With the Shell selected, choose **Define Shell Contour** from the context menu.
4. Define a square contour with which to cut the edge of the Shell, as shown. View the result in 3D.

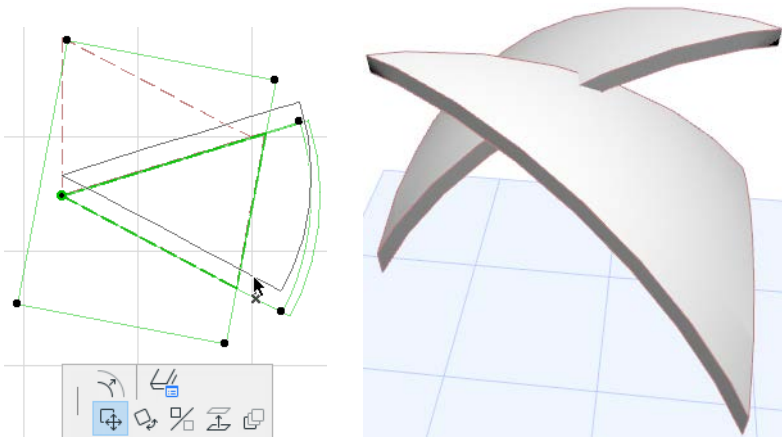


5. Back on the Floor Plan, mirror a copy of this Shell.

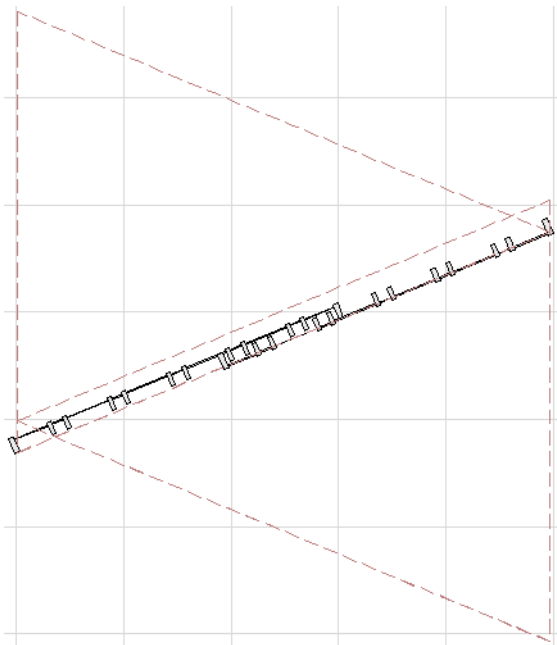
6. Move the second Shell so that it is alongside the first one.



7. Move the second Shell slightly upward on the Y-axis so that it slightly overlaps the first Shell (this will make it easier to trim the elements to each other.) View it in 3D.

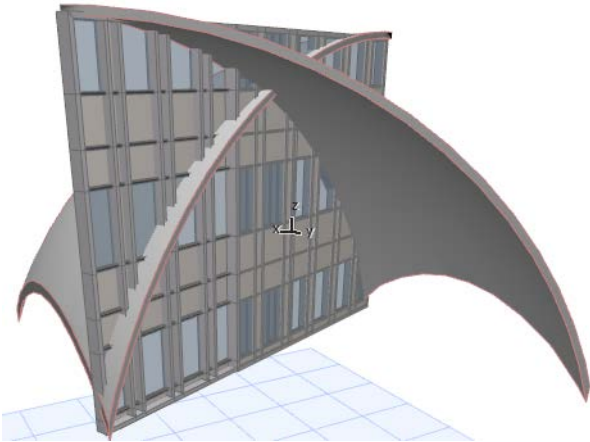


8. Return to the 2D window. Create two Curtain Walls, each facing in the opposite direction, running between the two Shells as shown.



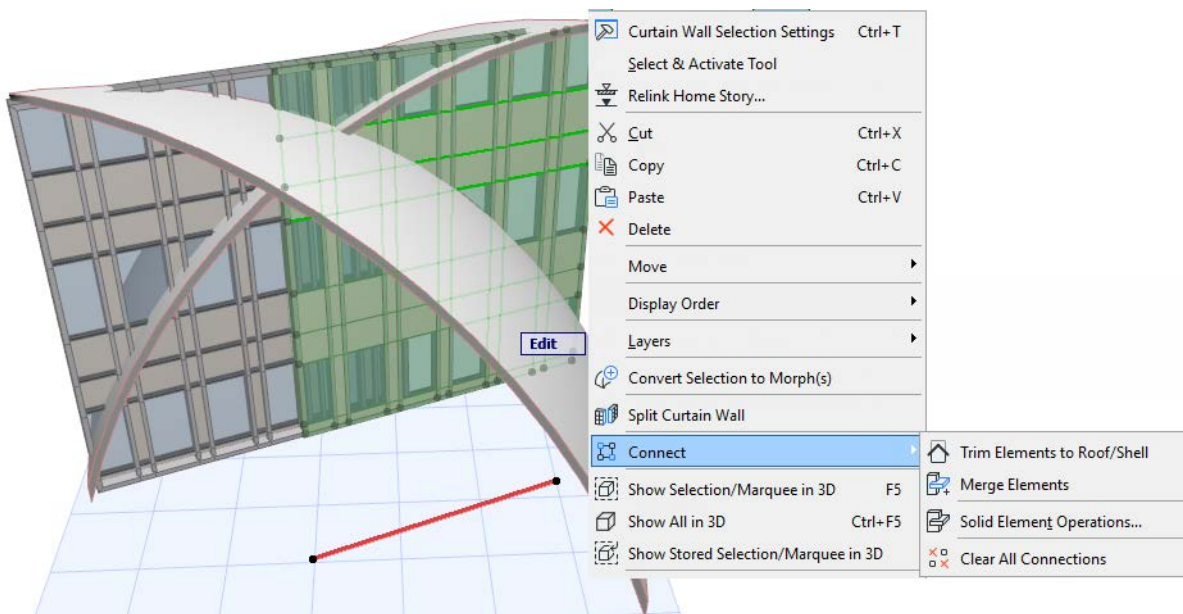
9. View in 3D.

10. Edit the position of the Curtain Walls into the right place: elevate the Curtain Wall, then stretch its top corner to the right height.

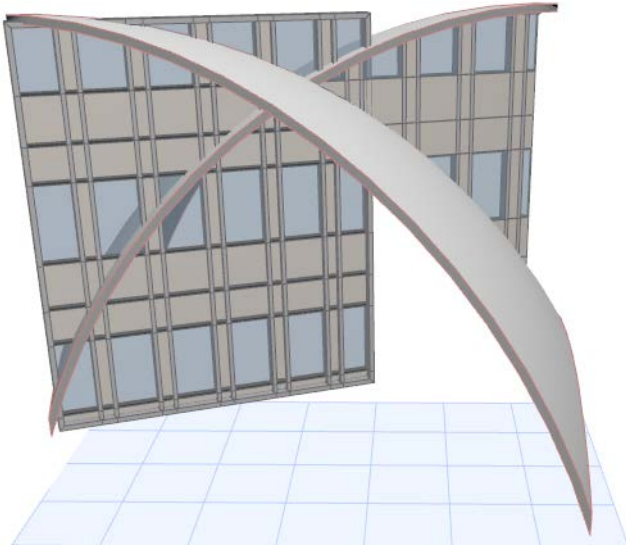


11. Now select *each* Curtain Wall in turn and trim it to *each* of the Shells to achieve the desired geometry, by following these steps.

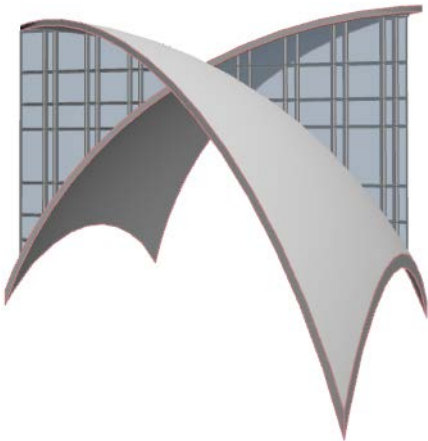
- Select one of the Curtain Walls and use the **Connect > Trim Elements to Roof/Shell** command.



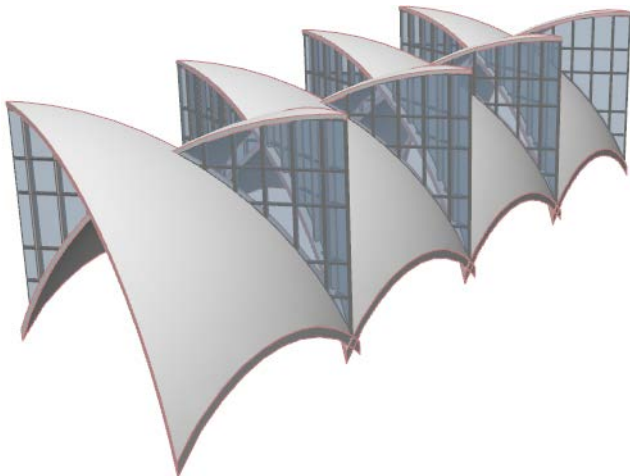
- Trim the selected Curtain Wall to *each* of the two Shells.



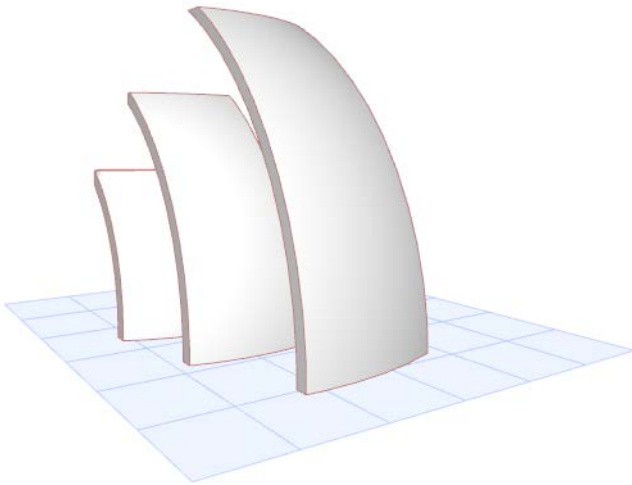
- Now select the other Curtain Wall, and trim it to *each* of the two Shells.



- Multiply the resulting structure as needed.



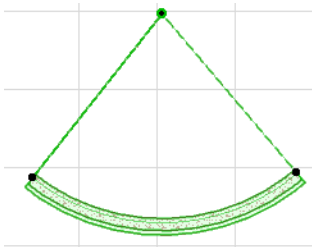
Revolved Shell: Define Contours in Section



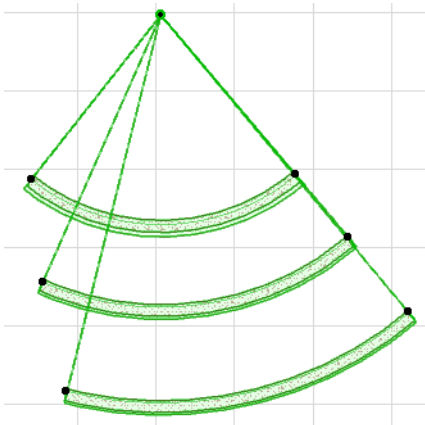
Here, you draw several revolved Shells having a common endpoint for their profiles, but slightly offset from each other.

Then, in the Section window, select each Shell separately and draw its contour in the shape of a rectangle - use progressively smaller rectangles. The result is several overlapping Shell segments, each of a slightly different size.

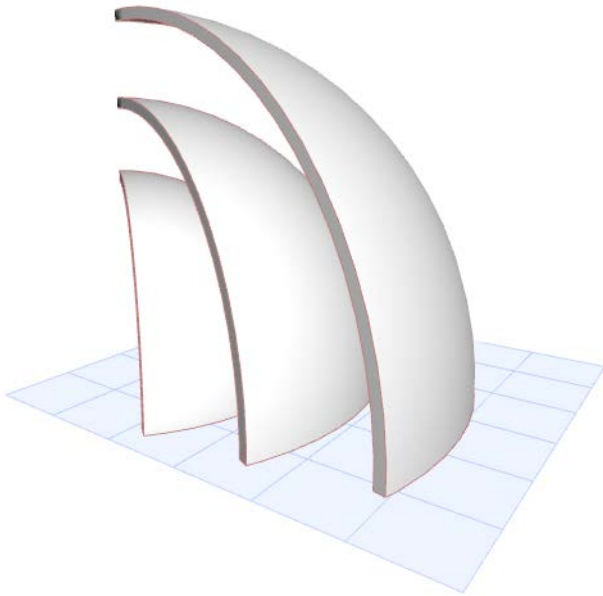
1. Ensure that the default Shell elevation is 0.
2. Draw the first revolved Shell on the Floor Plan as shown.



3. Draw two more revolved Shells using the same profile starting point, but each slightly larger and offset from the previous one.

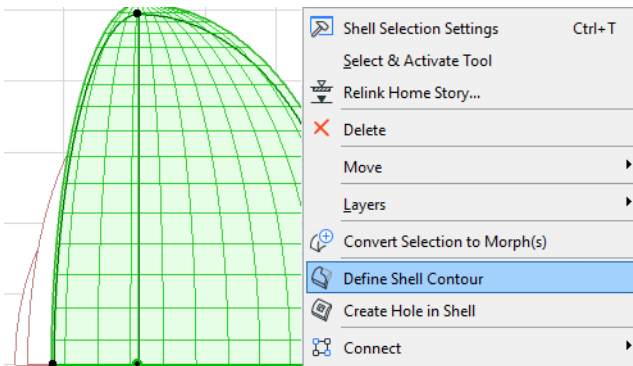


4. View the result in 3D.

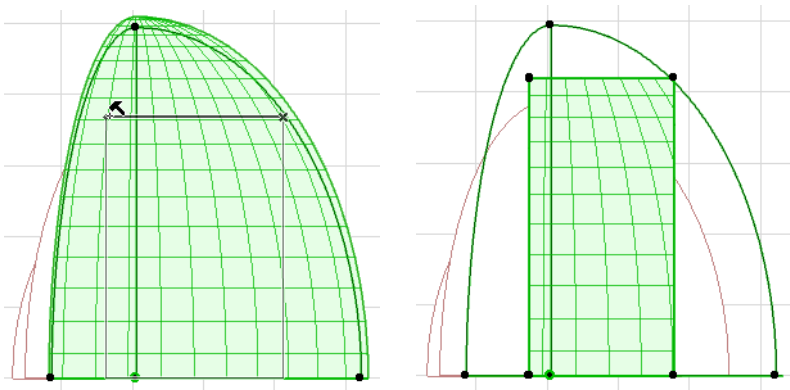


5. Now create a Section for these Shells and open the Section window.

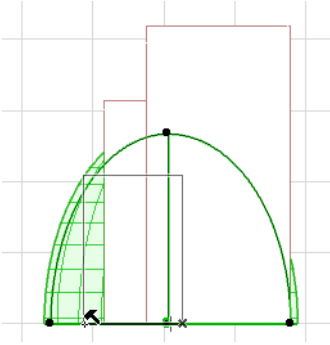
6. Select the Shell closest to you. From the context menu, choose Define Shell Contour.



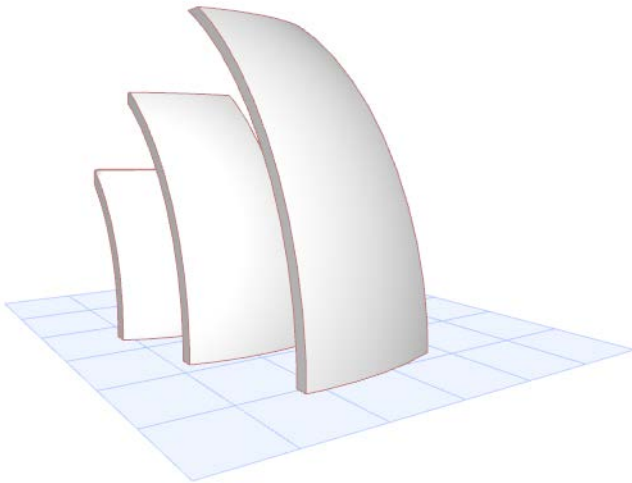
7. Draw the contour as a rectangle.



8. Select the other two Shells and define rectangular contours for them.



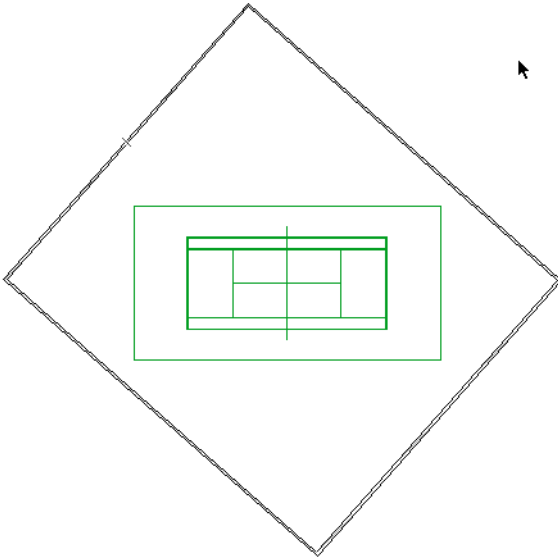
9. View the result in 3D and fine-tune as needed.



Ruled Shell: Tennis Court Shell

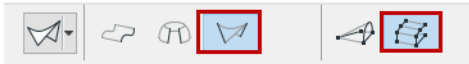
Create a Shell atop a tennis court stadium.

1. Place the Tennis Court object.
2. Draw a rectangle of walls around it at 45 degrees to the stadium; this wall will serve as the reference for the Ruled Shell's profiles (it will be deleted later).

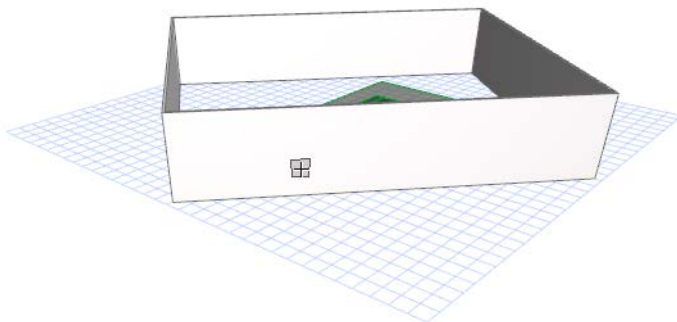


Place a Ruled Shell on top:

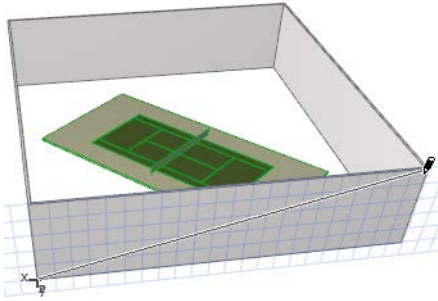
1. Go to the 3D window and activate the Shell tool, with the Ruled geometry method and Detailed input method.



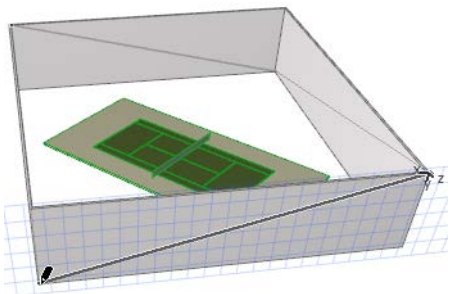
2. Define the first input plane on one side of the wall rectangle, by clicking on the outside surface of the wall.



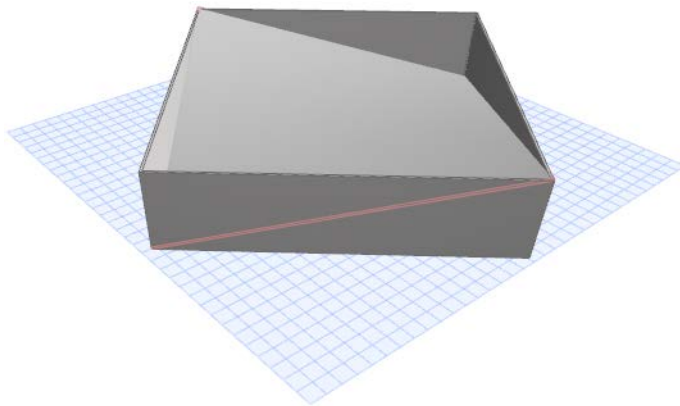
3. Draw a line from one top corner to the other and double-click to complete the first profile: a straight diagonal line as shown.



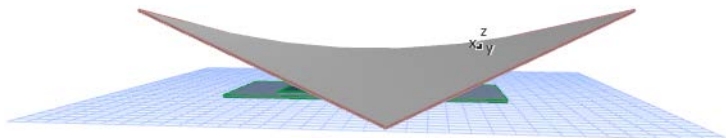
4. In the 3D window, turn the view so that the opposite wall is now closest. Define the second input plane on the outside of this opposite wall, by clicking on the wall's outside surface.
5. On this plane, draw the second profile, relying on the feedback to see how the Ruled Shell will be formed. In this case, you will draw a diagonal, straight line. Double-click to complete.



The initial Shell is completed.

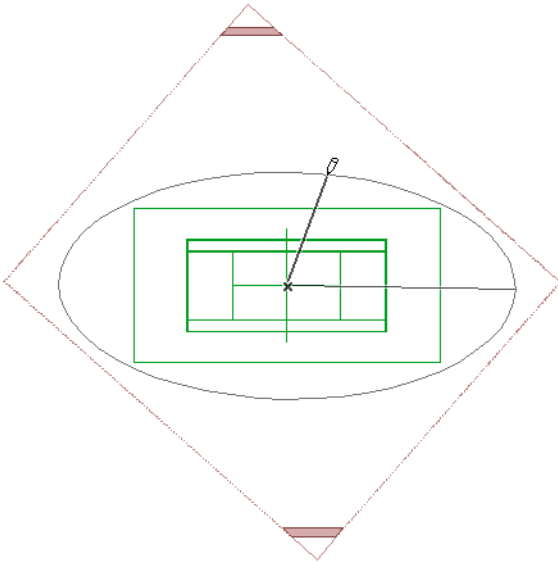


You can delete the wall rectangle and view the result.

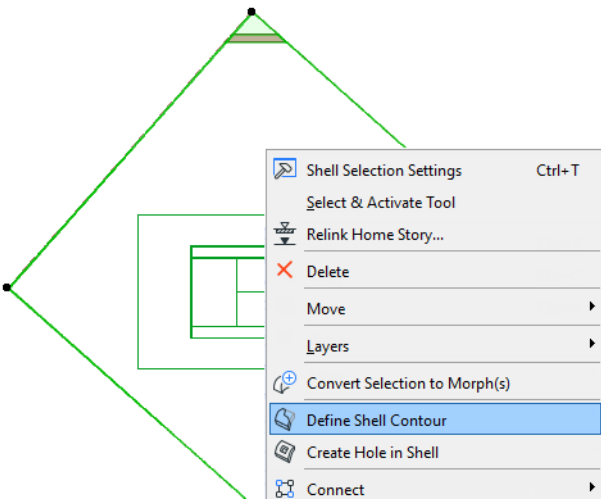


Next, define the Shell contour:

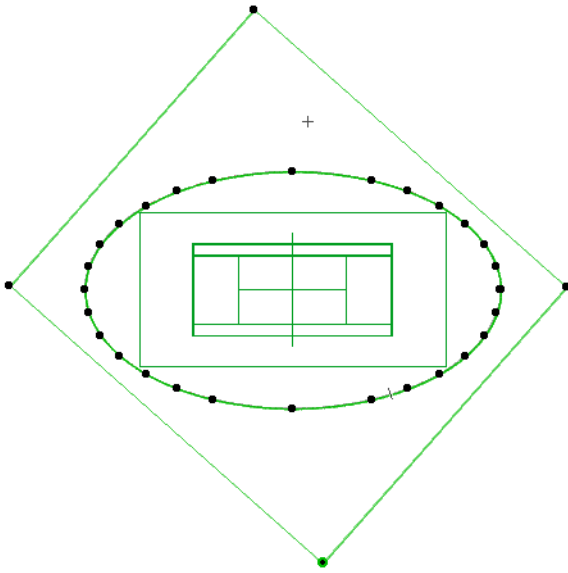
1. Go to the Floor Plan.
2. Use the circle tool to draw an ellipse in the desired shape of the stadium roof.



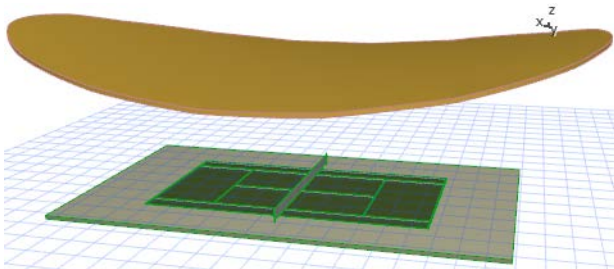
3. Select the Shell. From the context menu, choose Define Shell Contour.



4. With the Magic Wand (press the spacebar), click on the ellipse to define the Shell's contour in the ellipse shape. (You can then delete the ellipse element.)



5. View in 3D.
6. Place surrounding walls to make a stadium as needed, and trim them to the Shell.

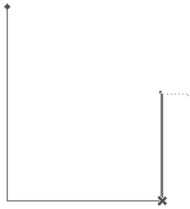


Ruled Shell: Twisted Tower

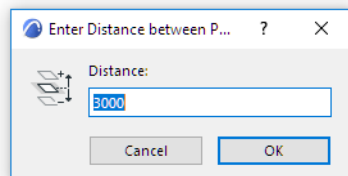
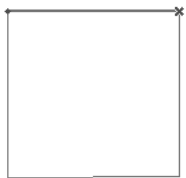
1. Choose the Shell tool's Ruled geometry method and the Detailed input method.



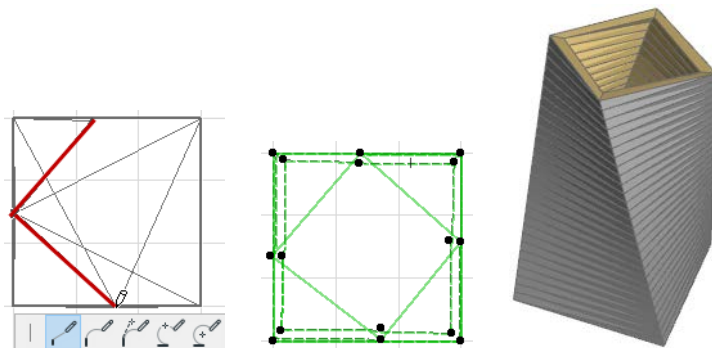
2. On the Floor Plan, draw the shape of your first profile polygon (its plane is flat on the Floor Plan). In this case, we draw a square. Double-click to complete the first profile polygon.



3. In the appearing dialog box, enter the distance between the two planes of your Ruled Shell profiles (between i.e. the first profile, already placed, and the second one, which you are about to draw.)



4. Now draw the polygon of the second profile, whose plane is parallel to the first. We draw a skewed square (here, shown in red). The feedback shows the ruled Shell taking shape.
5. Double-click to complete the polygon and the Shell.
6. View the result in 3D.



Graphical Editing of Shells

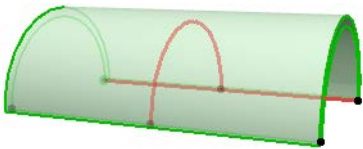
In Floor Plan, Section and 3D view, a wide range of editing possibilities are available for the Shell. Select your Shell and try out the pet palette commands: twist, turn, and rotate your Shell into the shape and position that you need.

In the 3D window, the **editable profiles**, **axes** and **extrusion vectors** are shown in a different color, so it is easy to find them.

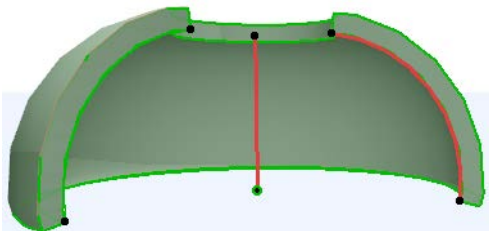
Note: You can change this color using the “Reference Lines in 3D” setting at **Options > Work Environment > Selection and Element Information**.

See images for each type of Shell when selected in 3D:

- An Extruded Shell has a **profile** and an **extrusion vector**; both are editable.



- A Revolved Shell has a **profile** and an **axis**.



- A Ruled Shell has **two profiles**, on any **two planes**, connected by the Shell body. The profiles can be edited and rotated. You can also rotate either or both of the profile planes.



As you edit the Shell, any cutting bodies it may contain will remain in place in the global coordinate system. This means that after the editing is done, the holes may be located in a different part of the Shell.

See [Edit Shell Hole](#).

Options for Editing Extruded Shells

Options for Editing Revolved Shells

Options for Editing Ruled Shells

Options for Editing Extruded Shells

About the Extrusion Vector

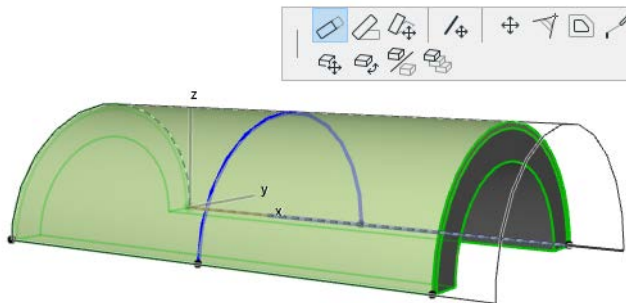
- One end of the extrusion vector (its initially placed point) is distinguished by a larger hotspot; this represents the Shell's base height.
- The extrusion vector can be located apart from the Shell body itself.



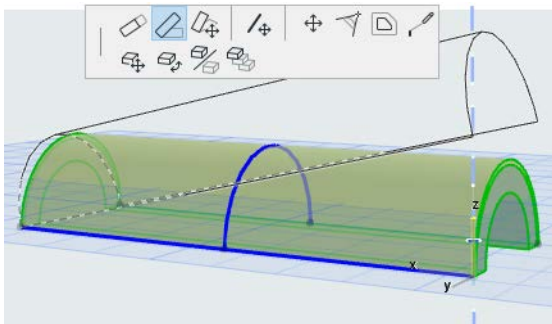
Edit Extrusion Vector

Click on either end of extrusion vector to bring up these commands on the pet palette.

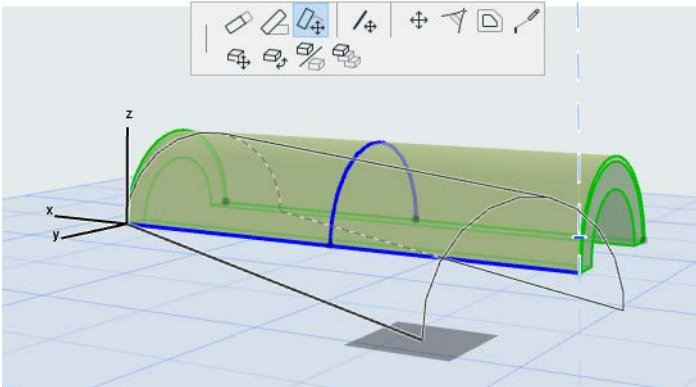
- **Edit Extrusion Length:** Edit the length of the extrusion vector. Its slant and its other geometric parameters remain unchanged.



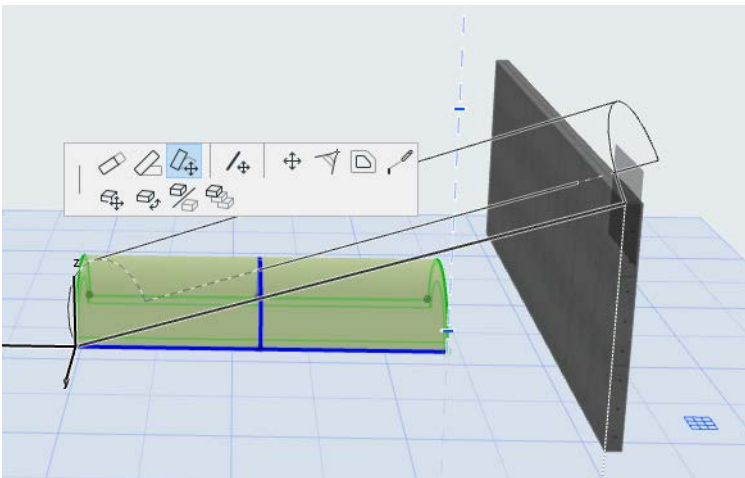
- **Edit Extrusion Slant:** Edit the slant of the extrusion vector. The extrusion length, as well as the Shell starting/ending angles and the distortion angle, remains unchanged.



- **Stretch:** Edit the length of the extrusion vector and its rotation. You can also rotate its slant, if you join the vector endpoint to an existing element. The starting/ending angles and the distortion angle remain unchanged.



If you stretch the Shell to reach a point on another existing element, the Shell slant changes accordingly:

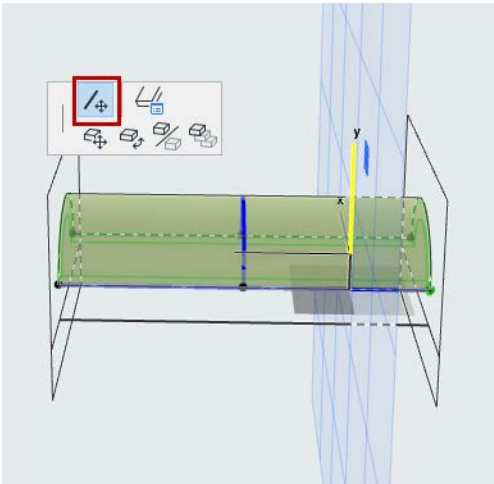


- **Drag Extrusion Vector:** Drag the Shell's extrusion vector to another location. Click anywhere on the Extrusion vector to bring up this command from the pet palette.

If you drag the extrusion vector, two things happen:

- The Shell's base elevation changes to the new location of the vector

- The Shell's start and end angles are now editable based on this new vector location. (The vector serves as the axis of rotation for the Shell start/end angles.)

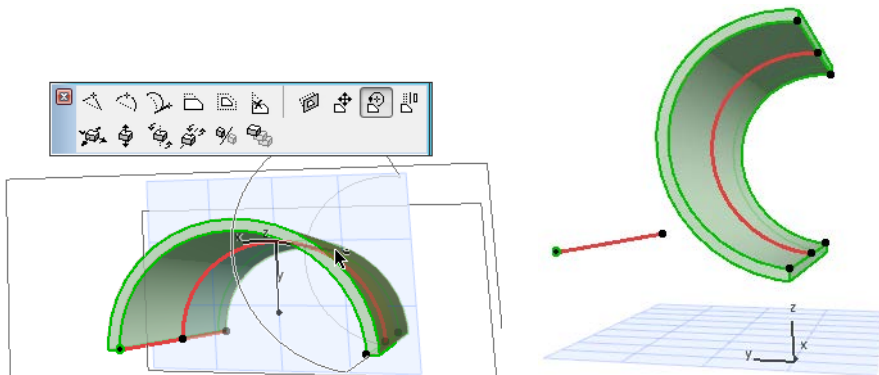
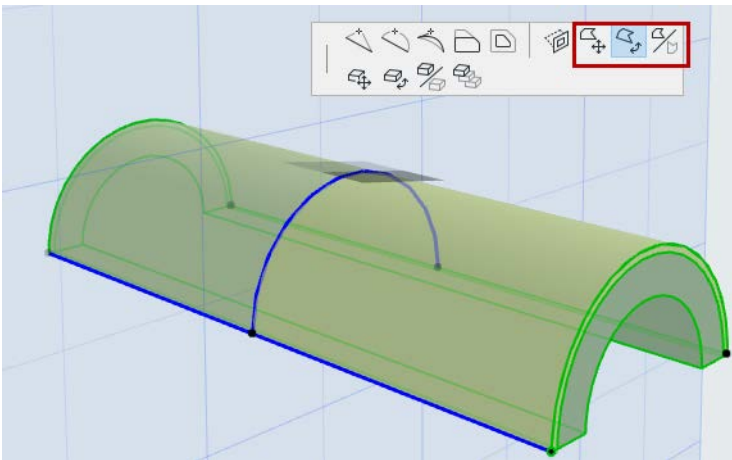


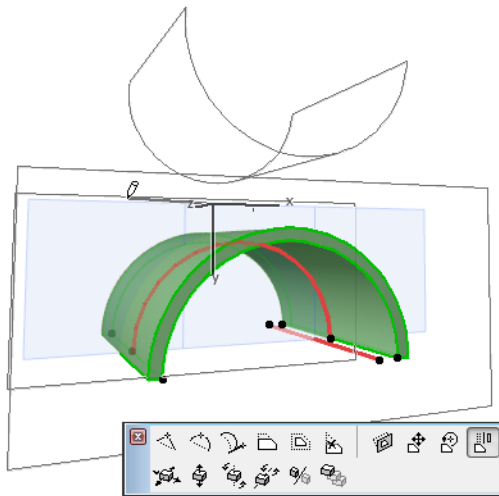
Edit Profile of Extruded Shell

Select the Shell and use the familiar pet palette commands to edit the profile. Note that even if you move the profile, the extrusion vector remains in place.

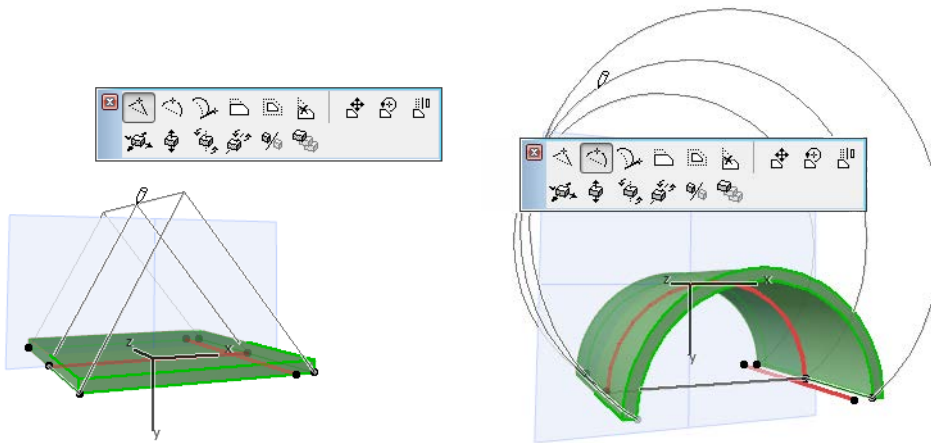
- Drag, Rotate, Mirror Profile

These three commands operate on the Shell profile on its own plane (the plane is displayed in the 3D window while you edit):



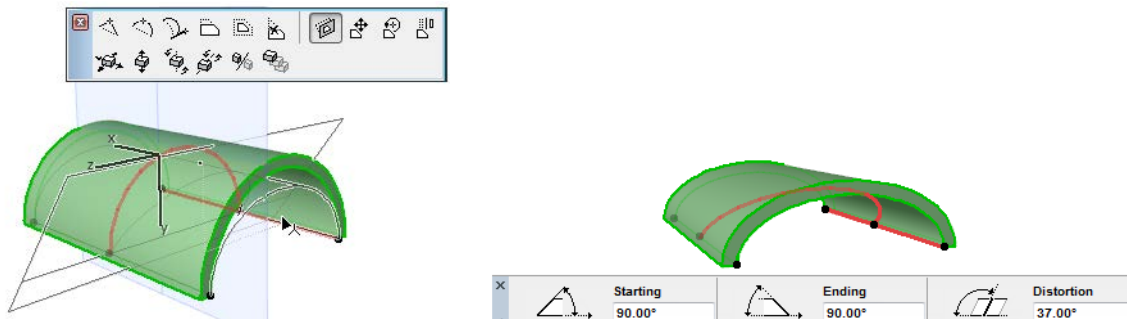


- Insert New Node
- Curve Edge



- Set Distortion Angle

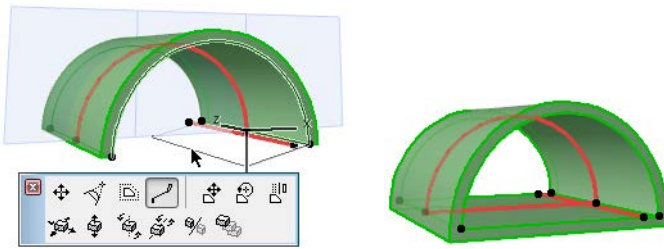
Select the Shell profile and use the pet palette command to set the distortion angle graphically. This can be useful, for example, to “flatten” your vault-shaped Shell into more elliptical form.



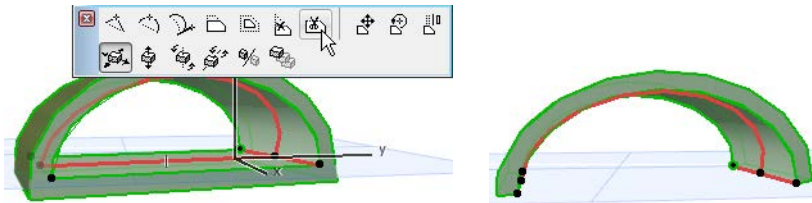
View the resulting distortion angle in the Tracker, the Info Box or Shell Settings.

- Continue Profile

If you select the endpoint of the Shell Profile, the **Continue Profile** command is available in the pet palette. Use this add one or more segments to the profile, as if you were extending a polyline (double-click to complete the polyline, or click on the other profile endpoint to create a closed profile.)



Remove profile Edge: For a closed Shell Profile, the **Remove edge** command is available in the pet palette from any selected profile edge. Click to remove that segment.

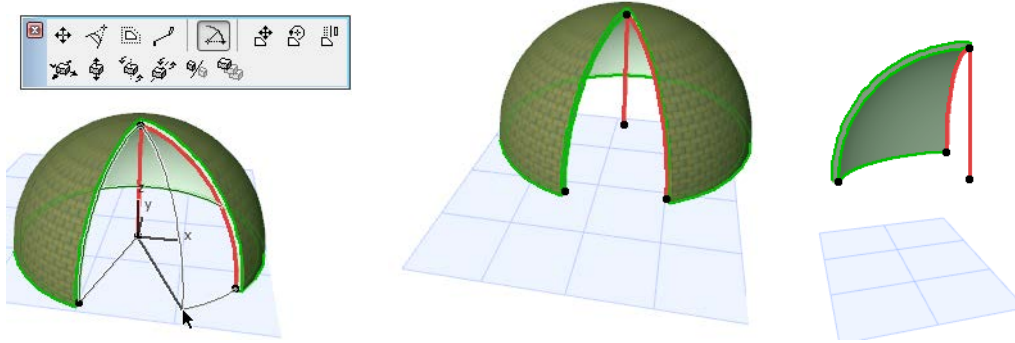


Options for Editing Revolved Shells

Select the axis or the profile of the Revolved Shell. One end of the axis is distinguished by a larger hotspot; this represents the Shell's base height.

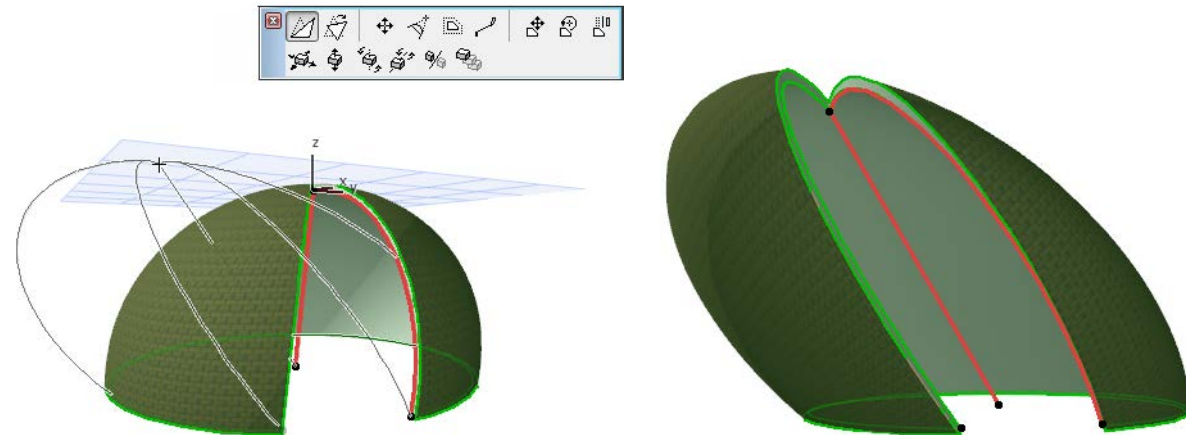
- **Edit Revolution Angle**

Click anywhere on the profile of the selected Shell to bring up this command in the pet palette. Choose **Edit Revolution Angle**.

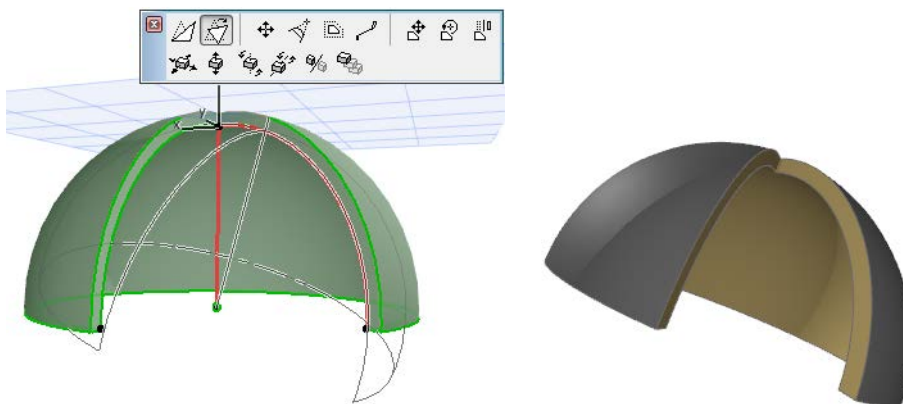


To bring up the next two commands in the pet palette, click on the endpoint of the Revolution Axis.

- **Distort Axis**

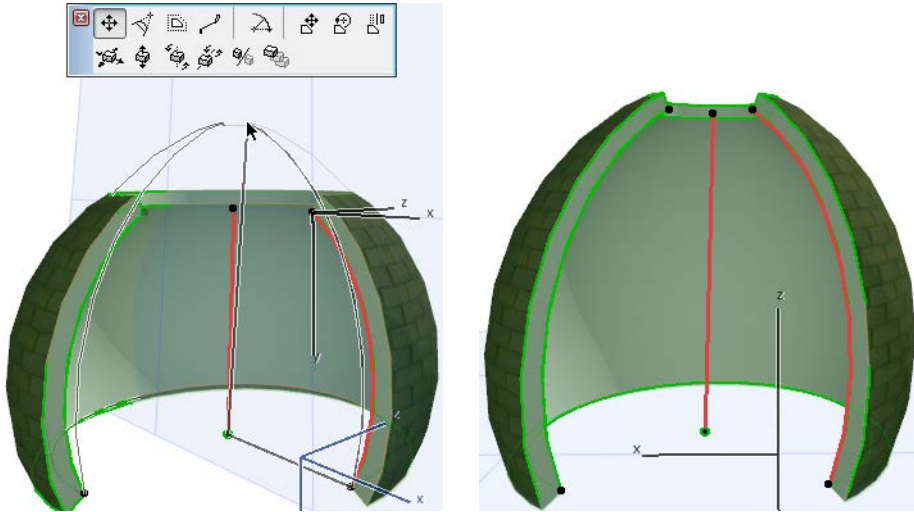


- **Slant Axis**

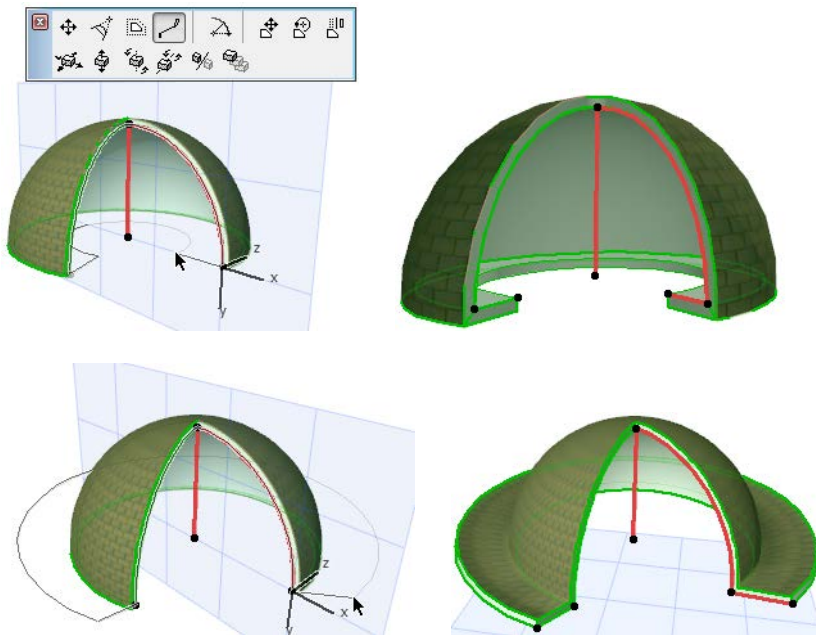


[Watch the Video](#)

- **Move Profile Node**

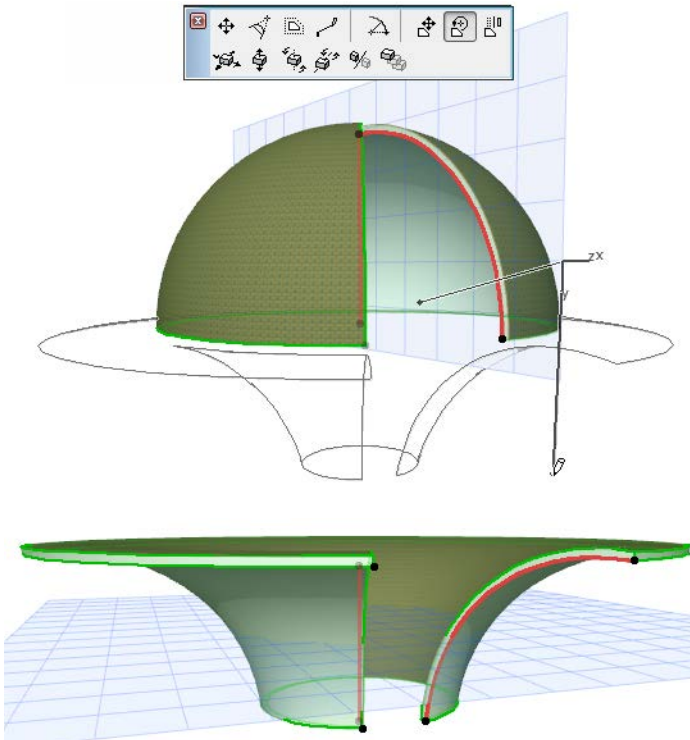


- **Continue Profile**



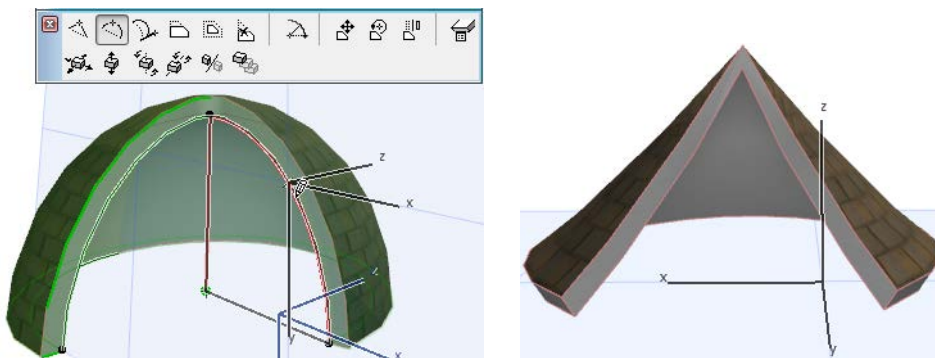
- **Rotate Profile**

Select the **Rotate profile** command from the Pet Palette. Then (as when rotating other elements) click on a point, then draw a rotation axis. Click to complete the rotation.



- **Curve Edge**

Click anywhere along the Shell profile to bring up this command on the pet palette.



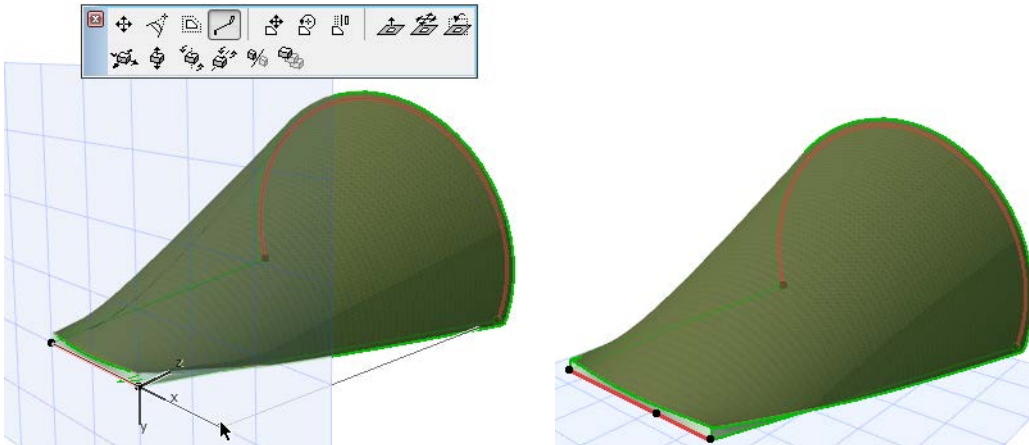
Options for Editing Ruled Shells

The Ruled Shell has two profiles. One end of the first profile is distinguished by a larger hotspot; this represents the Shell's base height.

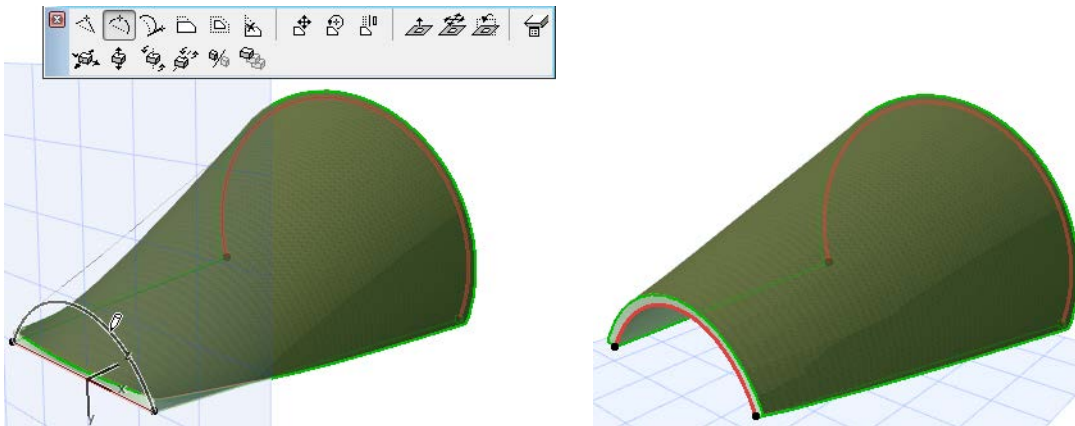
Here are some editing possibilities for either profile:

- **Continue Profile**

For example, extend the straight-line profile to make that end of the Shell wider.

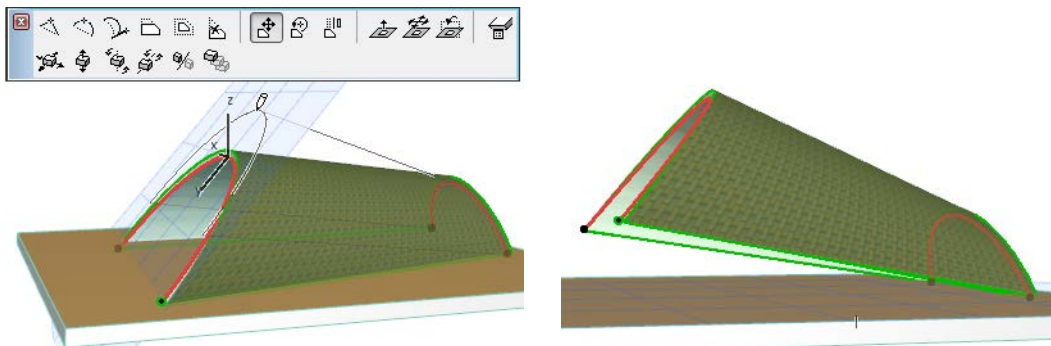


- **Curve Edge**



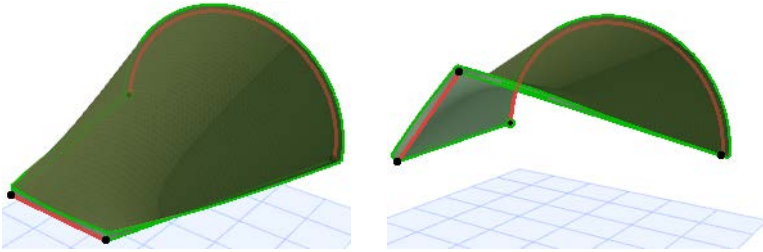
- **Drag Profile**

Use this command to drag the profile within its own plane, in horizontal and/or vertical directions, to a new location.



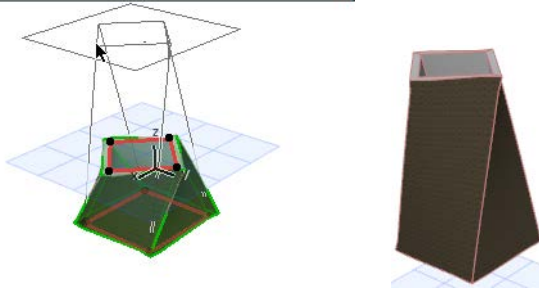
- **Rotate Profile**

Use these commands to rotate the selected profile in space.



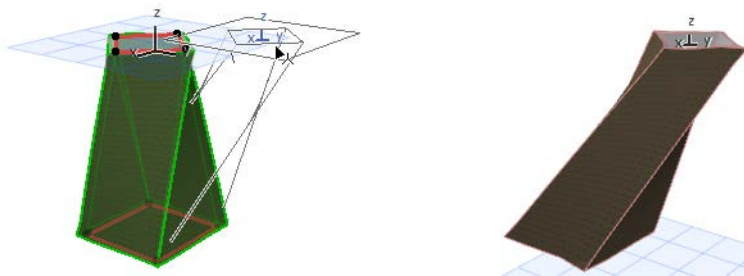
- **Elevate Profile Plane**

Here you move the entire plane of the profile in the vertical direction.

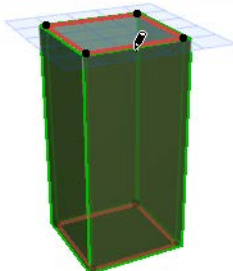


- **Drag Profile Horizontally**

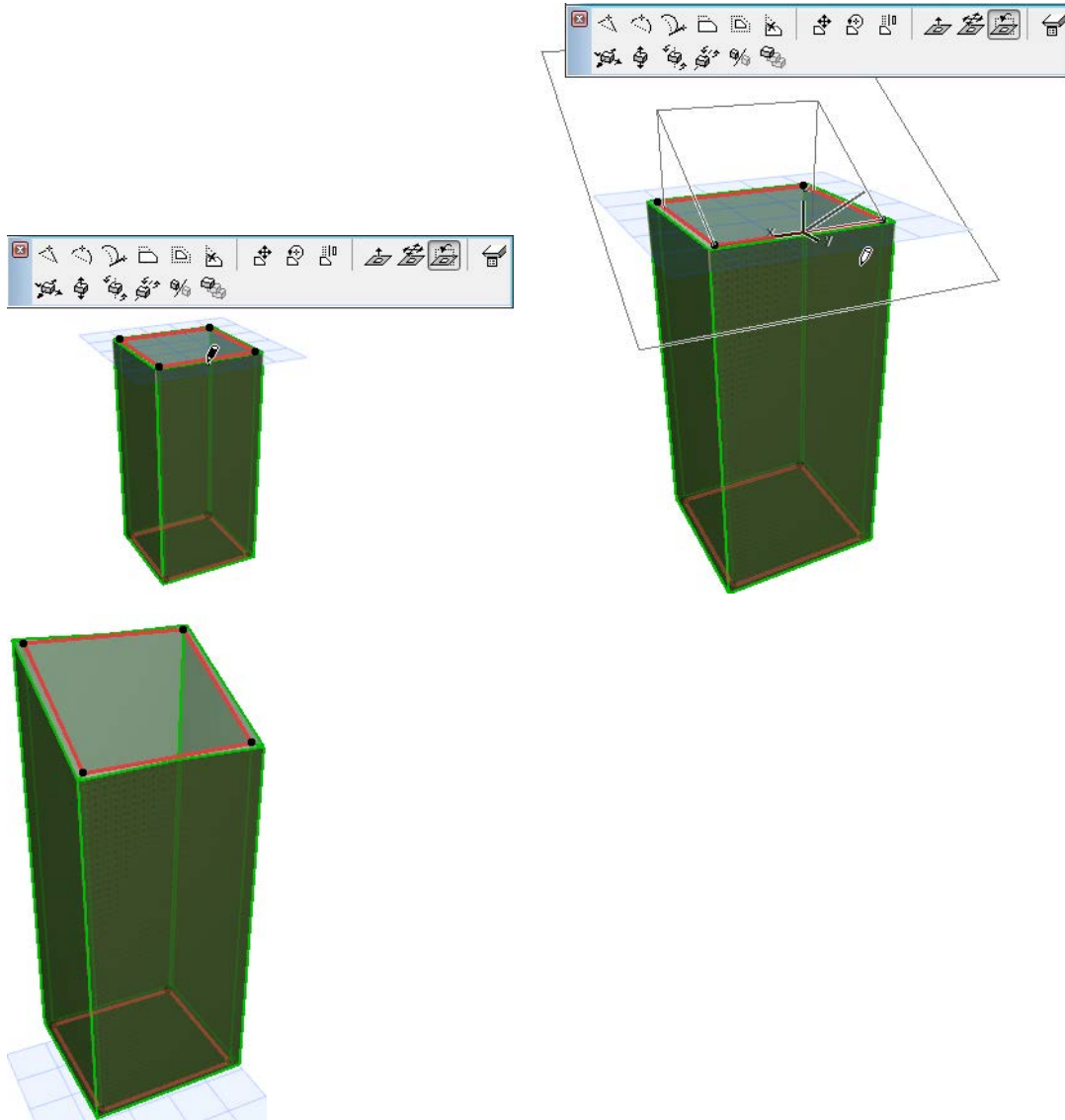
Here you take the profile plane and drag it horizontally.



- **Turn Profile Plane**



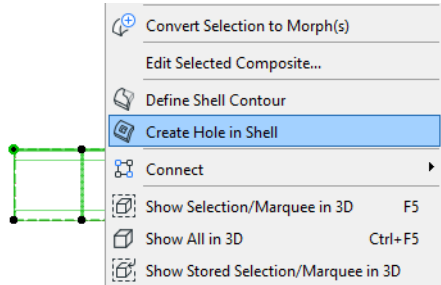
Click to enter the turning point. Then move your cursor to define the turn direction, and click to complete.



Create Hole in Shell

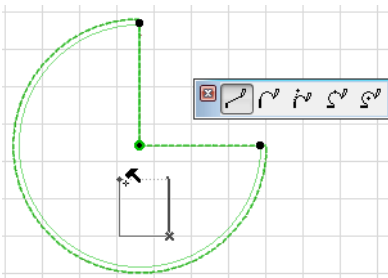
On the Floor Plan or Section window:

1. Select any Shell on the Floor Plan or Section.
2. From the context menu, choose **Create Hole in Shell**.

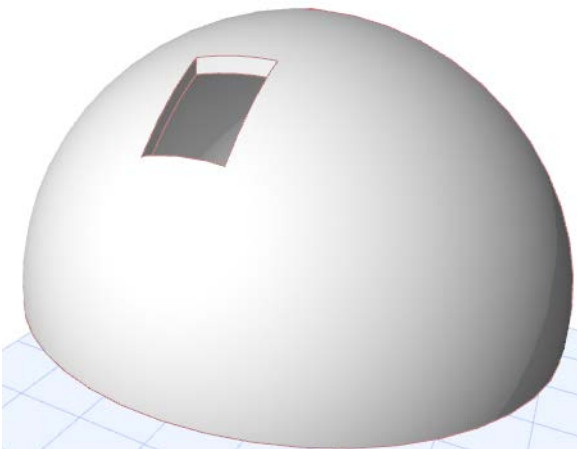


3. Draw the desired polygon onto the Shell. (The opening's plane is the Floor Plan, but you can edit this later.)

See [Edit Shell Hole](#).



4. View the result in 3D:



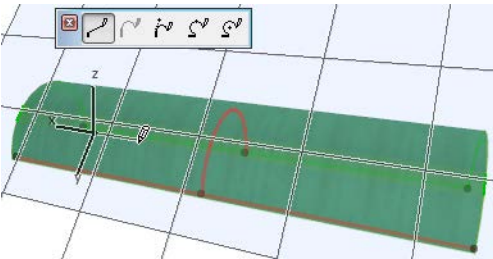
In the 3D Window:

1. Select the Shell.
2. From the context menu, choose **Create Hole in Shell**.
3. Choose an input plane on which you want to draw the hole: use the feedback to choose the desired plane location. (Choose a plane anywhere in the model; it does not have to be on top of the Shell.)

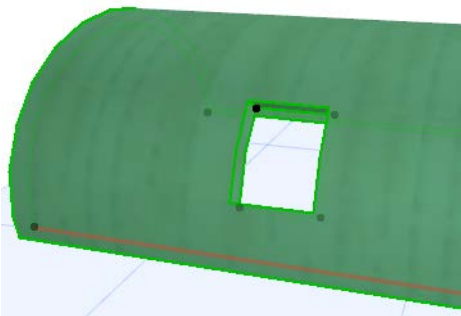


4. Draw the desired polygon onto the input plane. You can edit this later.

See [Edit Shell Hole](#).



5. View the result.



[Watch the Video](#)

Listing Shell Holes in the Interactive Schedule

Properties of Shell Holes (e.g. Holes Surface and Perimeter) can be listed in an Interactive Schedule. As with other elements, you can set a preference (“Calculation Rules”) for the size of holes which should be taken into account when calculating Shell volume or surface.

For details, see [Calculation Rules Preferences](#).

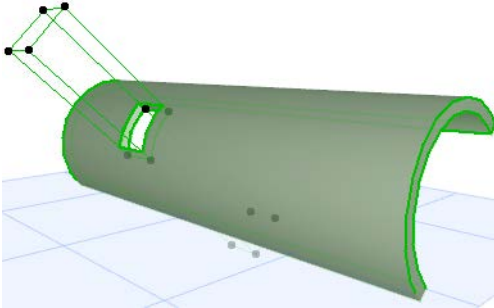
Edit Shell Hole

For information on customizing Shell edge surface, see [Customize Edge of Shell or Edge of Shell Hole](#).

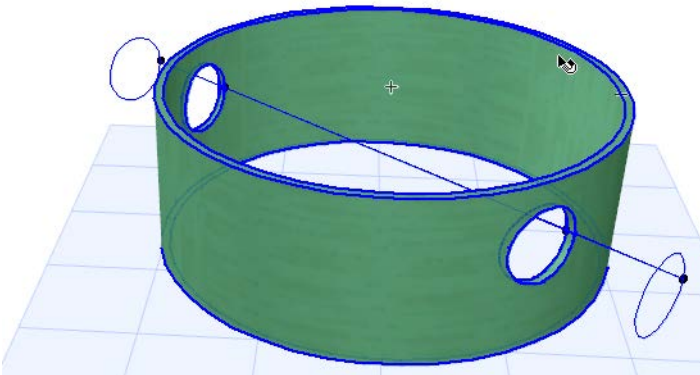
The Shell hole is created by an editable cutting body. The cutting body is visible when you select the hole itself, not the Shell.

To select the hole, zoom in on the hole and click its edge.

Note: When selecting the hole, you must click the hole on the side of the Shell that coincides with the profile line. Hole edge selection is not available on the side of the Shell opposite to the profile line.



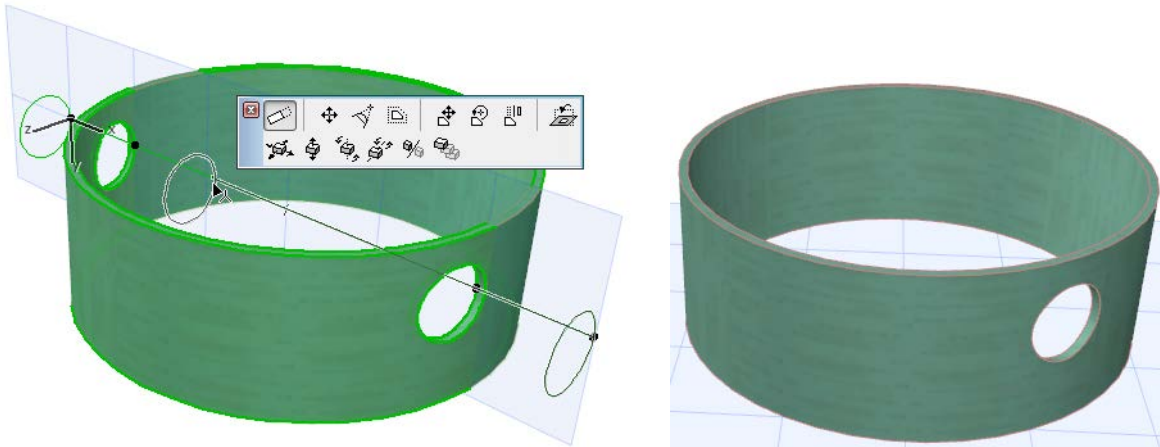
Each cutting body is specific to its own Shell and cannot cut any other elements. However, a Shell that bends back upon itself is automatically cut twice (or more) by the single cutting body.



When you move the Shell as a whole from one place to another, the cutting body moves with it. However, editing the Shell itself (e.g. its profile or drawing its contour) will not affect the hole geometry.

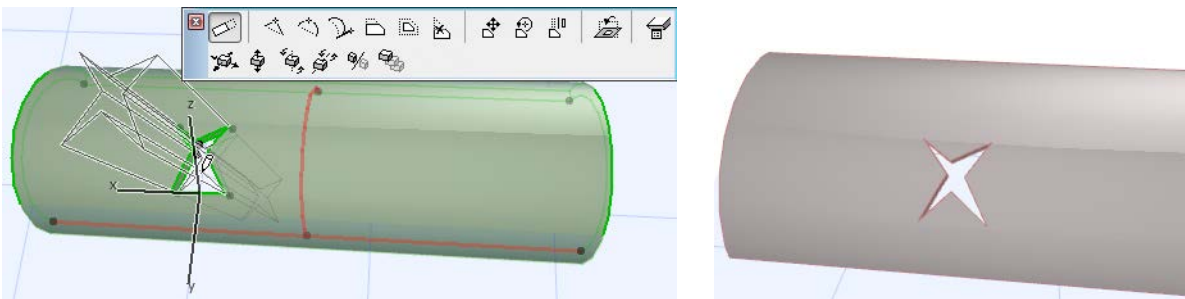
Edit Extrusion Length of Cutting Body

To limit the extent of the hole's cutting body, select the hole and use the **Edit Extrusion Length** command from the pet palette to change the length of the cutting body. This way, for example, you can eliminate a hole created by a cutting body that runs through both sides of your Shell.



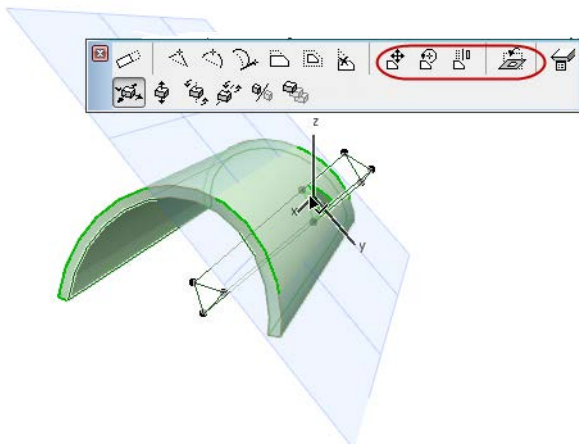
Edit Hole Contour

Select the hole so that you see its cutting body. Edit either the hole edge itself, or the contour of the cutting body (the result is the same). Use the familiar polygon editing commands from the pet palette.



Edit Hole on its Own Plane

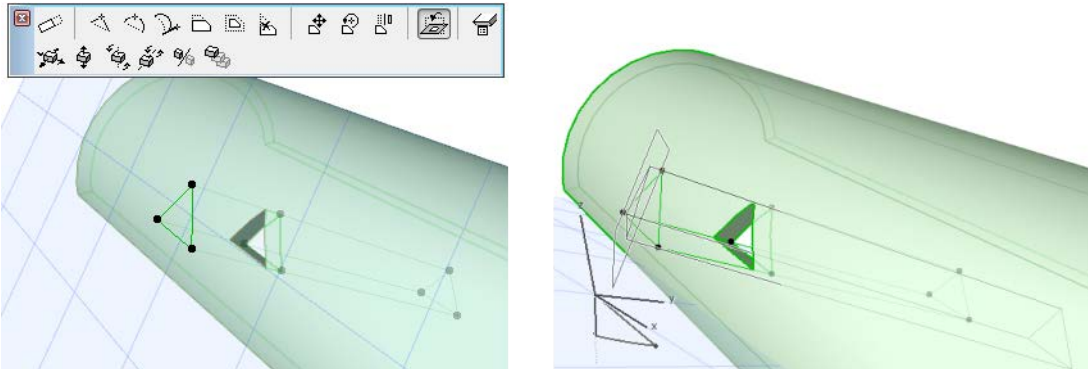
The cutting body that creates the hole has its own plane. When you select the hole, you can edit the hole on this plane using the pet palette commands marked here. Then the hole plane is also displayed.



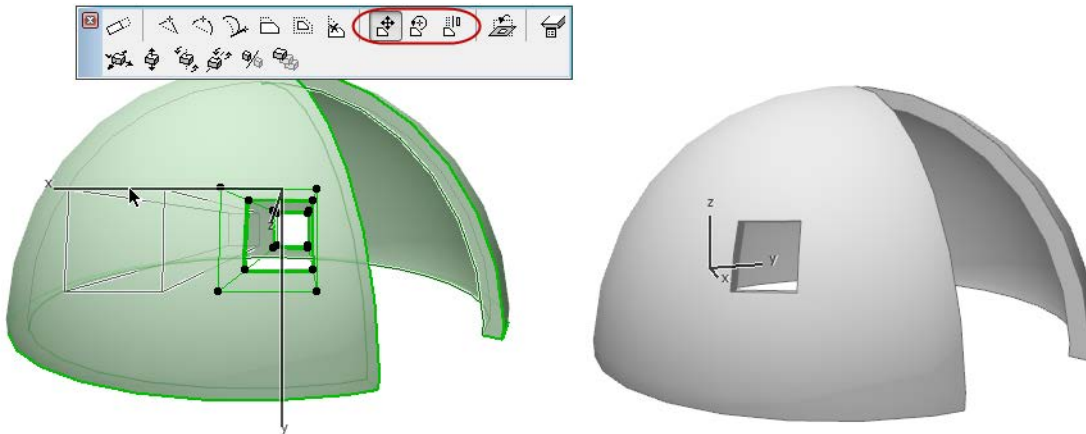
- **Turn Hole Plane:** This command rotates the hole plane, and thereby changes the hole geometry within the Shell. You can also rotate the hole itself.

See [Rotate cutting body to change hole geometry](#).

Select the **Turn Hole Plane** command. Define the rotation arc, then move the cursor to rotate the plane.



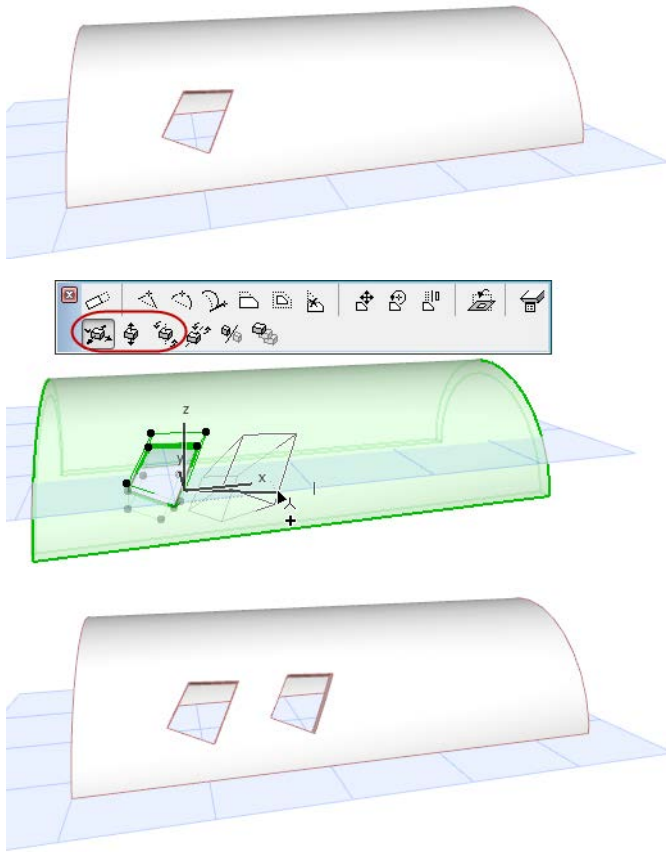
- **Drag, Rotate, Mirror Hole:** These edits move the hole on its own plane.



Edit Hole on Horizontal/Vertical Project Planes

- Drag or Drag a Copy Horizontally/Vertically

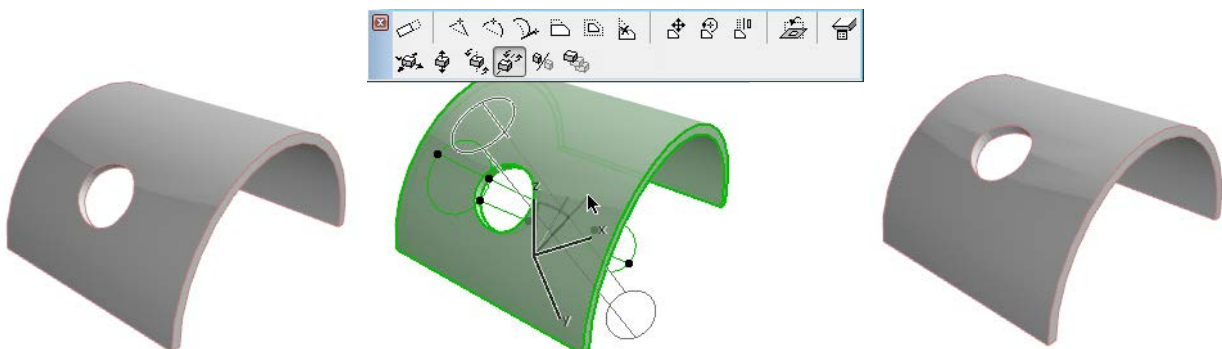
These commands drag the hole itself (or a copy) on the project's horizontal/vertical planes.



Rotate cutting body to change hole geometry

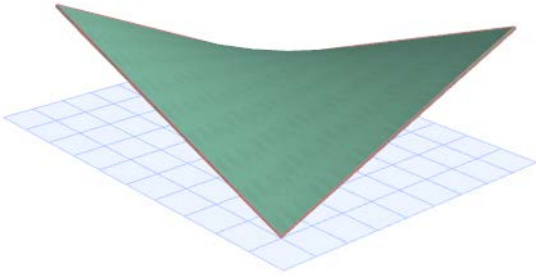
One way to change the hole geometry is to free-rotate its cutting body.

1. Select the hole.
2. Use the Free Rotate command from the pet palette.
3. Define a rotation axis with two clicks.
4. Define the starting point of the rotation arc with a third click, then free rotate the cutting body to its desired location.
5. Click to complete.



[Watch the Video](#)

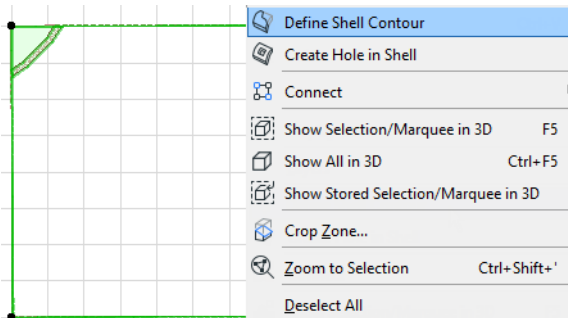
Define Shell Contour



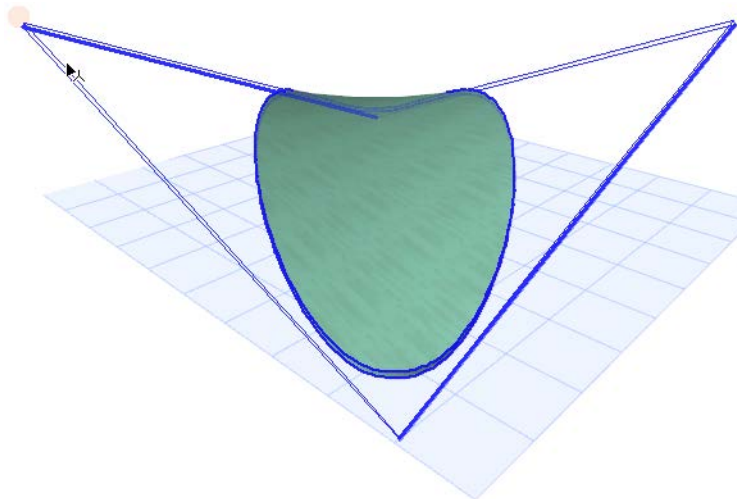
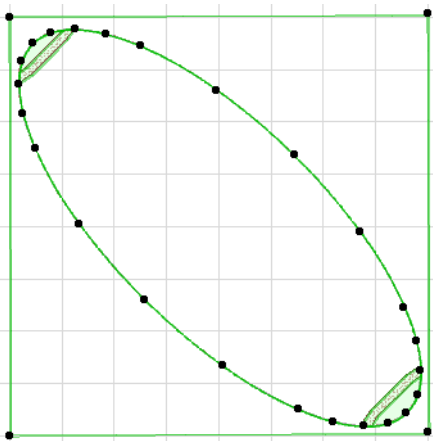
Suppose we have a ruled Shell like the one above.

You can trim it, or any Shell geometry, to the desired shape by defining its contour.

1. Go to the Floor Plan or Section window.
2. Select the Shell.
3. From the context menu, choose **Define Shell Contour**.



4. Draw the desired polygon: in this case, an ellipse. (Or use the Magic Wand on an existing polygon.)
5. Click to complete the polygon. The Shell is trimmed.



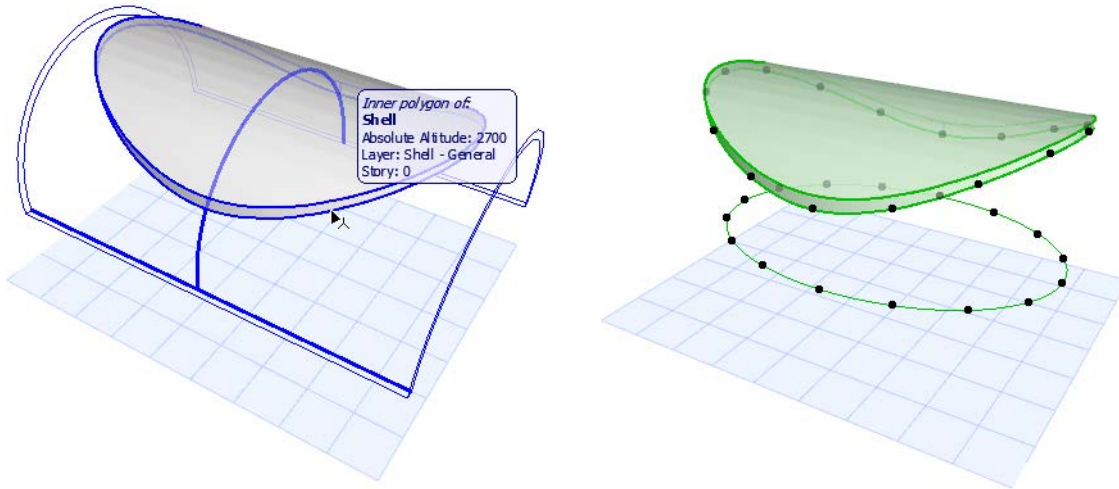
See also [Edit Shell Contour Polygon](#).

Edit Shell Contour Polygon

The Shell's contour polygon, and the plane of this contour polygon, are editable.

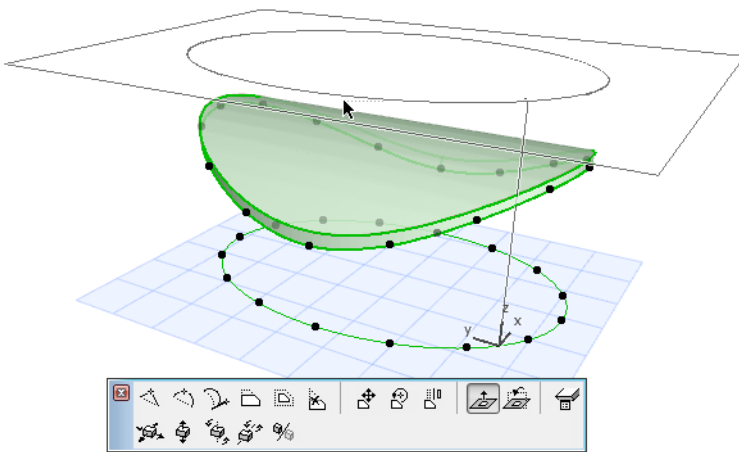
Select Contour Polygon

To edit the contour, you must select the contour polygon itself, not the Shell. Move the cursor to the edge of the contour and make sure that the Info Tag indicates that the Shell's Inner Polygon is going to be selected.



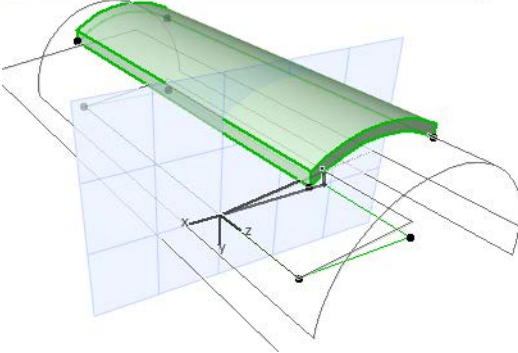
Offset Contour Polygon

Before you edit the Shell contour, it may be helpful to offset the contour polygon, so that it is easier to do your editing. Select the Shell contour, then choose Offset Contour Polygon from the pet palette. Offset to the desired location.

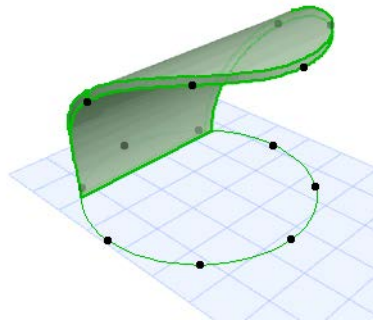
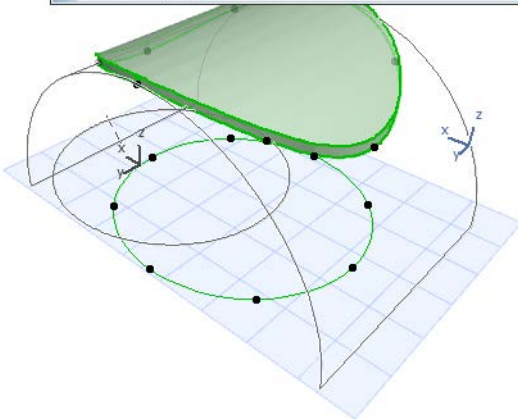


Turn Contour Plane

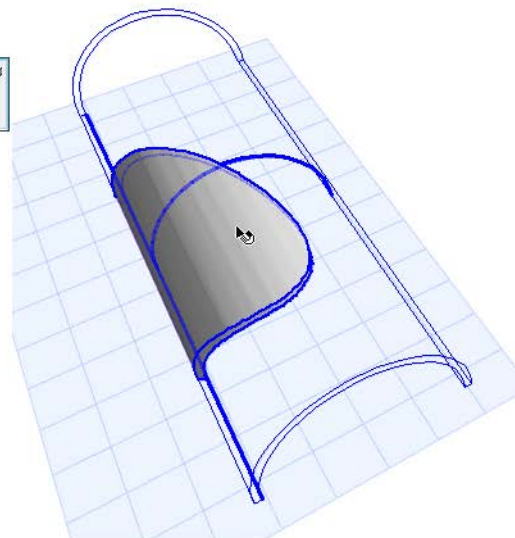
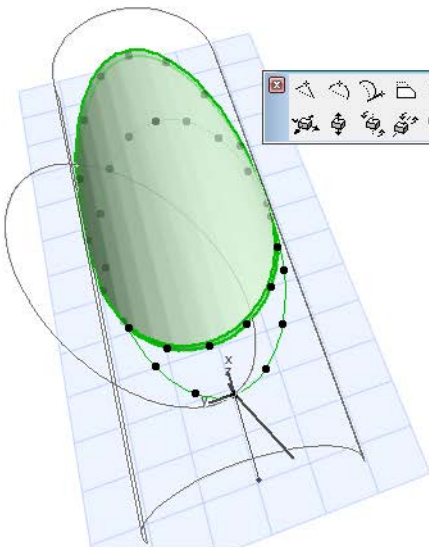
By default, the plane of the contour polygon is parallel to the Floor Plan. Use the **Turn Contour Plane** palette command to change this polygon plane. This may also affect the geometry of the Shell contour.

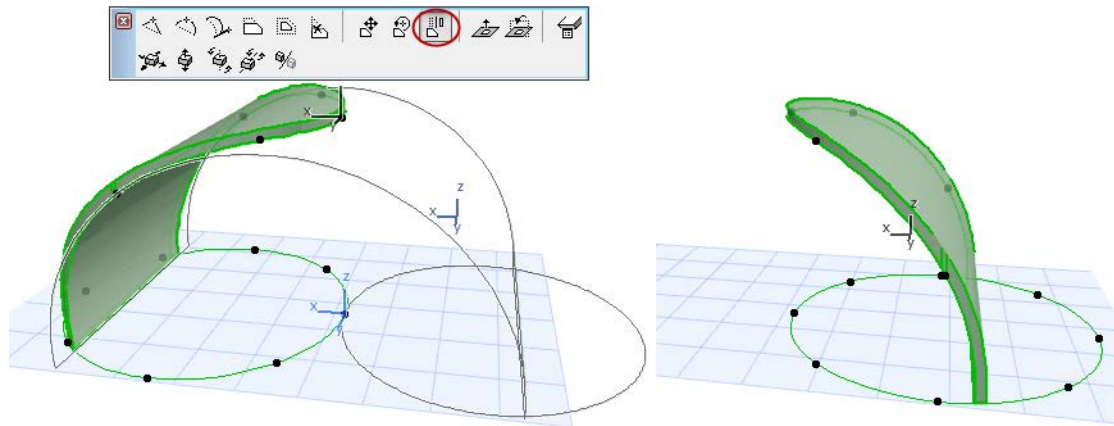


Drag Contour Polygon



Rotate, Mirror Contour Polygon





[Watch the Video](#)

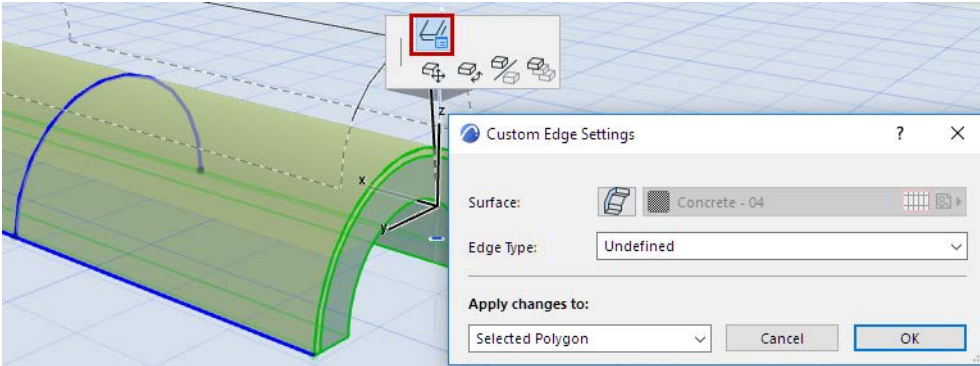
[Watch the Video](#)

Customize Edge of Shell or Edge of Shell Hole

You can customize the surface and/or type of any edge of a Shell, or the edge of a hole in a Shell.

Once you set the customized surface/type options, you can choose to apply them to the clicked edge only, to all edges of the clicked polygon (hole), or to all Shell edges.

1. Select the Shell.
2. Click on an edge to bring up the pet palette.
3. Select the **Customize Edge settings** icon to open the **Custom Edge Settings** dialog box.

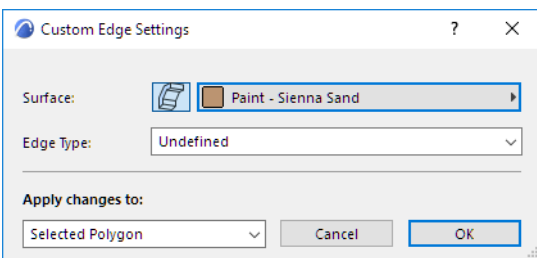


4. Use the controls to adjust the Surface and Edge Type as needed.

Custom Shell Edge Surface

Note: By default, the Surface material applied to a Shell edge derives from the Building Material assigned to the Shell. You can override *all* edge surfaces of the Shell, using the controls in the Model Panel of Shell Settings. The Custom Edge Settings offers a further way to customize the surface of *any one or more* Shell Edges.

Surface: Click the Surface override button, then choose a custom surface for the Shell edge(s) or end surfaces.

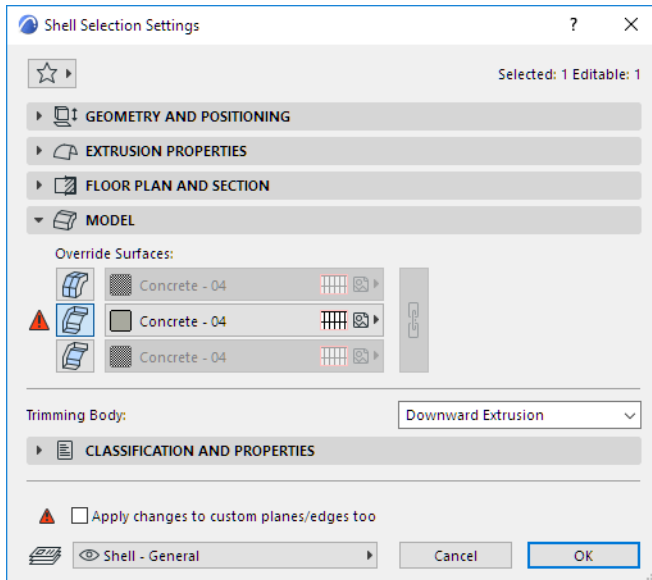


If you apply a surface here to any edge that is different from the edge surface defined in the Model Panel of Shell Settings (or in the Building Material), the Model Panel will indicate (with a yellow patch) that a custom edge surface has been applied.



If you override the Shell edge surface in Shell Settings while the dialog box is still open, the yellow patch (indicating “custom edge”) gets a red shadow: this means that the Shell’s general edge setting is going to change: do you want to *apply this override to all edges*, including the custom edge(s) too?

If so, check the box at the bottom of Shell Settings (**Apply changes to custom planes/edges too**). If you do not check the box, your custom edge surfaces remain the way you set them.



Custom Shell Edge Type

Edge Type: Optionally, choose a type for the Shell edge.

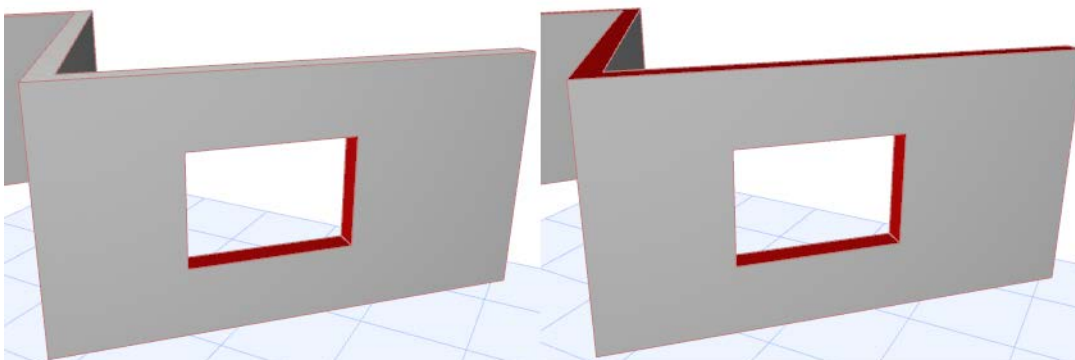
The Shell Edge type you define here can be used as a parameter in all of Archicad’s listing functions.

For an explanation of edge types, see [Shell Listing Parameters](#).

Apply Changes

Apply Changes to: Once you have set the options as needed, choose the edges to which to apply these changes:

- **Clicked Edge** will apply the options you choose below (surface, edge type) only to the edge you clicked when bringing up the pet palette.
- **Selected Polygon** will apply the chosen options to the extruded ends of the Shell only or the edges of the hole only, depending on which one is selected.
- **All Edges** will apply the chosen options to all edges of the Shell as well as all edges of its holes.



5. Click **OK** to apply changes.

[Watch the Video](#)

Slabs

Slabs are the basic horizontal building blocks in Archicad. They are typically used for modeling floors, split levels or ceilings.

Attributes for each part of the Slab's representation are set in the Slab Settings dialog box.

Like other construction elements, a Slab's essential structure derives from its Building Material.

Note: The Floor Plan outlines of Meshes and Slabs on remote stories (if they are shown on stories other than their home stories) are displayed using the line type set at **Project Preferences > Legacy**.

While a Slab's surface is defined as part of its Building Material, you can override each of the three Slab surfaces (top, bottom and side) surfaces in the Model Panel of Slab Settings.

It is also possible to assign a separate surface and a separate edge angle to each edge of the Slab.

To aid in precise element placement, each Slab has a Reference Plane that helps you control connections to other elements.

The Gravity function also works for Slabs: turn on the **Gravitate to Slab** command as an aid in placing elements onto the surface of the Slab.

Related Topics:

[Slab Reference Plane](#)

[Creating Slabs](#)

[Composite Structures](#)

[Placing Holes in Slabs](#)

[Set Custom Slab Edge Angle and Edge Surface](#)

[Slab Tool Settings](#)

[Adjust Elements to Slabs](#)

[Gravity](#)

Slabs can be included in a Structural Analytical Model generated in Archicad.

For details, see [Structural Analytical Model](#).

Slab Reference Plane

Each Slab has a reference plane which plays a role in intersections with other elements. The ability to flexibly assign a Slab reference plane helps you to align intersections easier and to control the changes that result if you alter the Slab structure.

If you redefine an existing Slab’s reference plane, the entire Slab will change its position accordingly. This way, the connections previously made to the slab will remain intact.

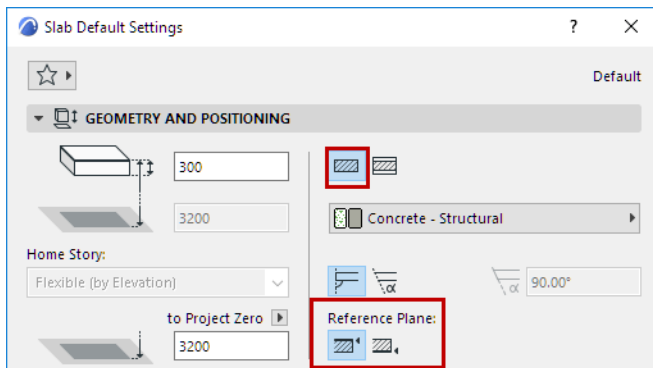
Note: To modify the Slab reference plane while keeping the Slab position *unchanged*, use the Modify Slab Reference Plane command.

See [Modify Slab Reference Plane](#), below.

Define Slab Reference Plane

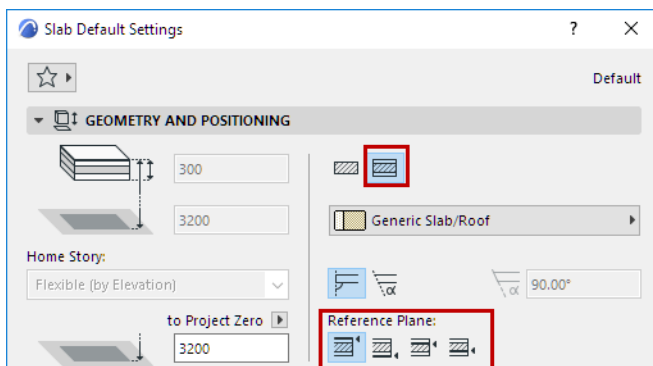
Use the Info Box or Geometry and Positioning Panel of Slab Settings to choose a reference plane. Click the “C” shortcut to cycle through the reference plane options in the Slab Info Box.

A **basic Slab** has two possible reference planes: either the top or the bottom plane.



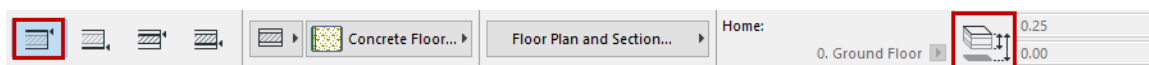
A **composite Slab** has four possible reference planes: the top or the bottom of the Slab core, or the top or bottom of the Slab.

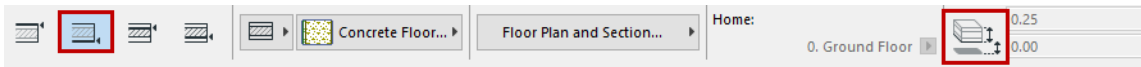
Note: If the chosen composite structure has no core, then the core-related reference plane choices are grayed out.



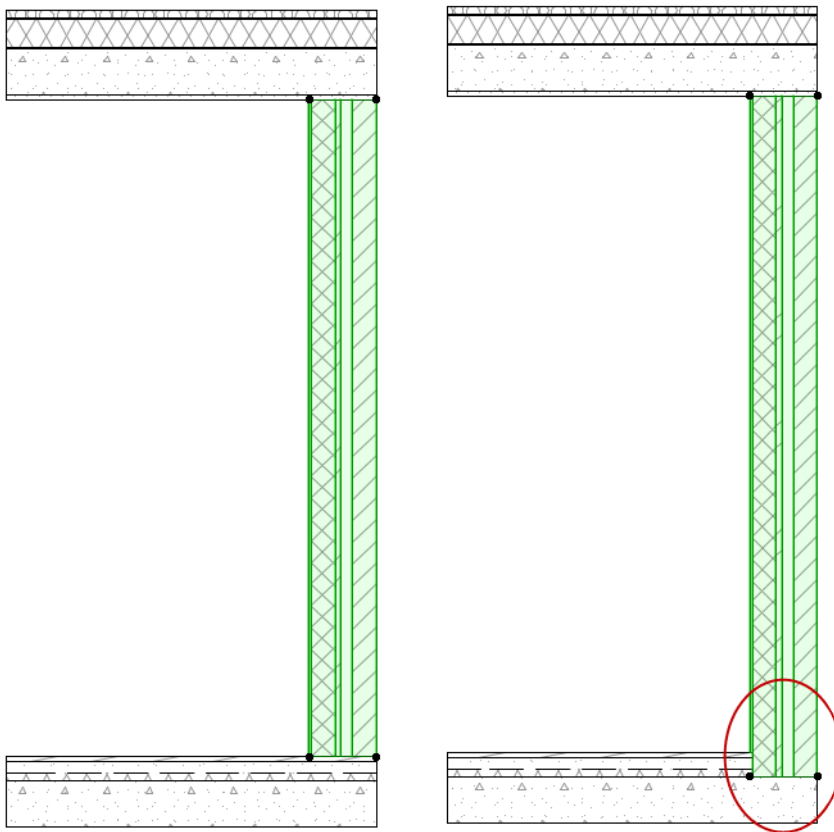
For a composite Slab, the “top” surface is the first listed skin of the composite, in Composite Settings.

Notice that the Info Box elevation icons and fields vary depending on the location of the Slab’s reference plane.



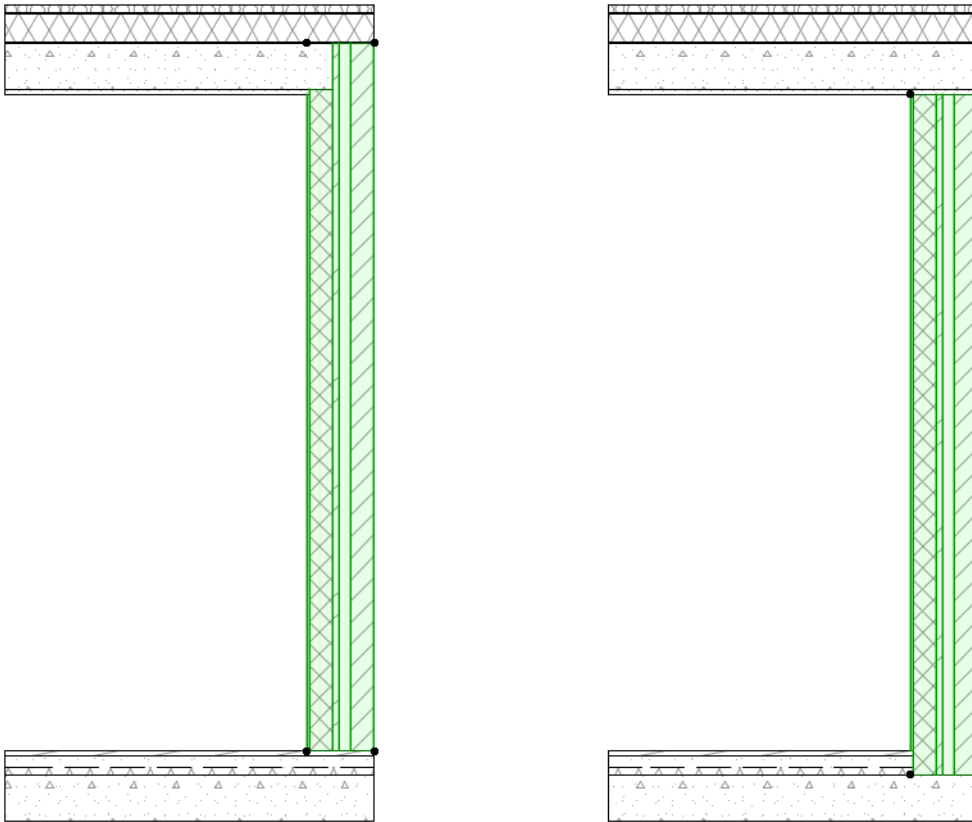


Thus, Walls can join Slabs by connecting to the top of the slab core, rather than the top of the Slab finish.



Use Cases

- When a Slab is used to model a ceiling: the reference plane can be placed at the Slab base, rather than the top.
- If you redefine an existing Slab's Reference Plane from Core Top to Core Bottom, the entire Slab will change its position accordingly. This way, the connections previously made to the slab will remain intact. Here, the Walls are connected to the Slab's Core Top and Bottom, respectively.

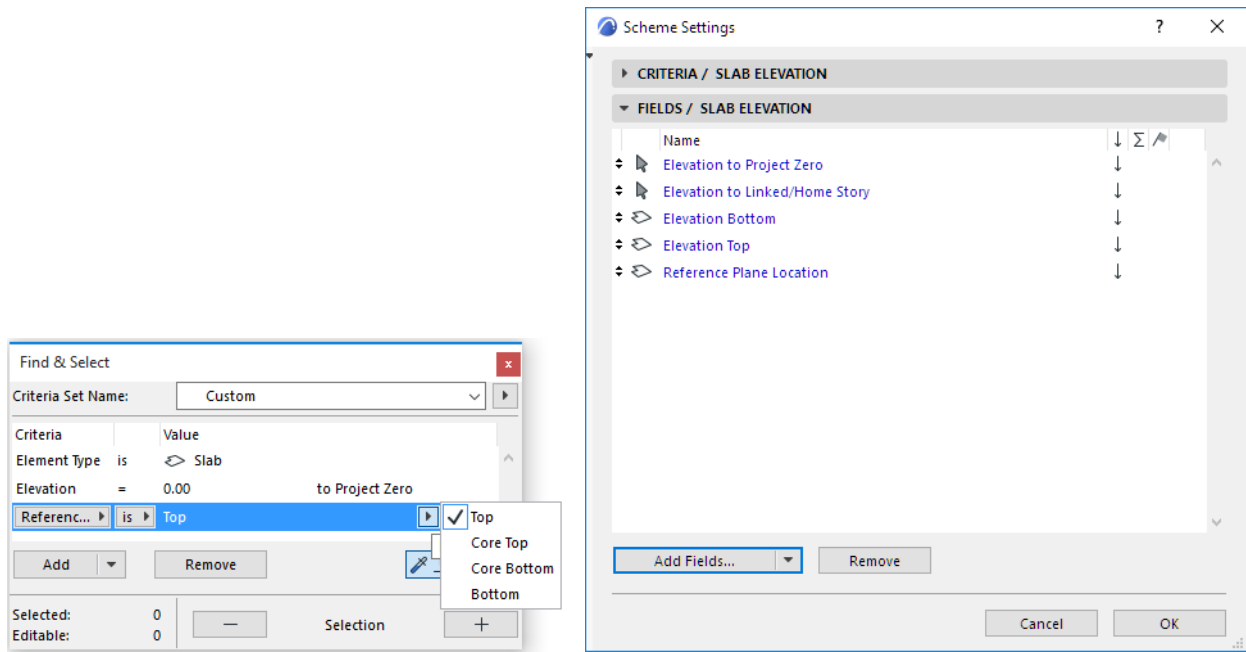


[Watch the Video](#)

Reference Plane in Interactive Schedule/Find & Select

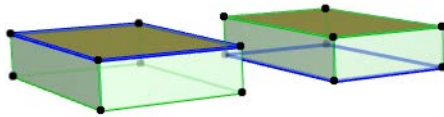
In the Interactive Schedule, you can list Slab elements using separate fields for the element's top and bottom elevation; but the "Elevation to Project 0" and "Elevation to Story" refer to the elevation of the Slab reference plane (not the elevation of the Slab top).

It is also possible to Find & Select slabs in your project, or list them in the Interactive Schedule, based on the position of their reference planes:



Editing Slabs by Reference Plane

In 3D view, the Slab Reference Plane is shown as a thick blue line. Its shape is a projection of the top of the Slab; to edit the Slab, you can click either on the Reference Plane line or the Slab’s top outline, or any of the nodes.



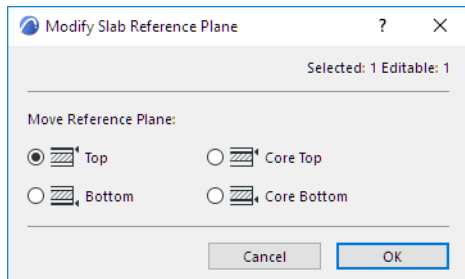
Adjust Elements to Slabs

Use this function (**Edit > Reshape > Adjust Elements to Slabs**) to easily adjust elements (Walls, Columns or horizontal Beams) to specific levels of one or more Slabs that are located above and/or below them.

[See *Adjust Elements to Slabs*.](#)

Modify Slab Reference Plane

This command (at **Edit > Reference Line and Plane > Modify Slab Reference Plane**) enables you to modify the Slab reference plane of one or more selected Slabs, while keeping the Slab position *unchanged*.



Note: If the selection includes one or more basic Slabs, then choosing the “Core Top” option here will result in a “Top” reference plane for these basic Slabs.

This function can be helpful if you use basic structures to model during the sketching phase, then later convert basic Slabs to composites. In such a case, you can adjust the reference plane to the core top rather than the top, as needed.

When migrating older-version files, you may find it useful to use this function to change the Slab reference plane from the top to other locations.

[Watch the Video](#)

Creating Slabs

Set Slab parameters in Slab Settings.

[See Slab Tool Settings.](#)

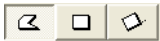
New Slabs can be created in either the Floor Plan or the 3D Window.

Depending on the structure you choose for the Slab (either a basic or a composite slab), you have varying construction method options for the Slab Reference Plane:

[See Slab Reference Plane.](#)

Slab Geometry

Choose one of the three Slab Geometry Methods from the Info Box.



- With the first icon on the left, you can create a **Polygonal** Slab. Just like for Walls, the pet palette appears and allows you to draw straight and curved segments for the Slab's outline.
- The second and third icon allow you to create a **Rectangular** or a **Rotated Rectangular** Slab. The rectangle is defined by placing two opposing corner nodes. A rectangle Slab is always aligned orthogonally with the normal grid. The Rotated Rectangular method requires that you first define a rotation vector, then drag the cursor in a perpendicular direction to complete the slab.

Create a Composite Slab

You can apply or customize a composite structure for a Slab.

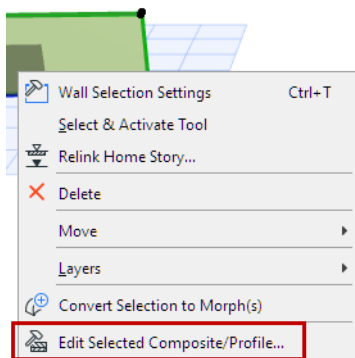
[For more information, see Composite Structures.](#)

(**Remember** that the model's Partial Structure Display settings will affect the display of composite Slabs.)

[See Partial Structure Display.](#)

If the structure of the slab is a composite, then the slab's thickness is defined in **Options > Element Attributes > Composites**, and equals the sum of the skins' thicknesses.

Note: You can edit a selected composite by accessing its settings from its context menu:



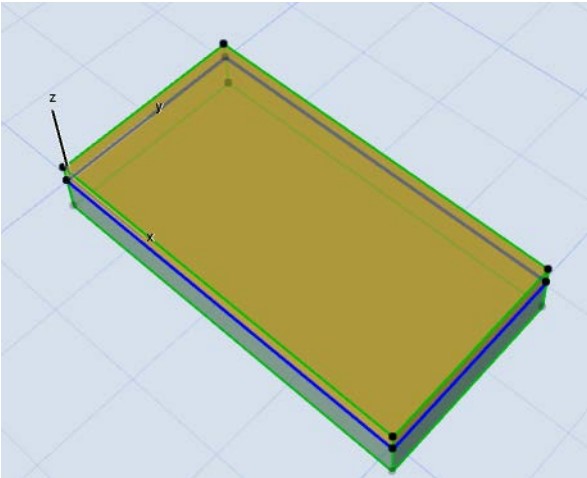
You have four options for the reference plane of a composite Slab: top of slab, bottom of slab; top of core, bottom of core.

[See also Slab Reference Plane.](#)

Placing Holes in Slabs

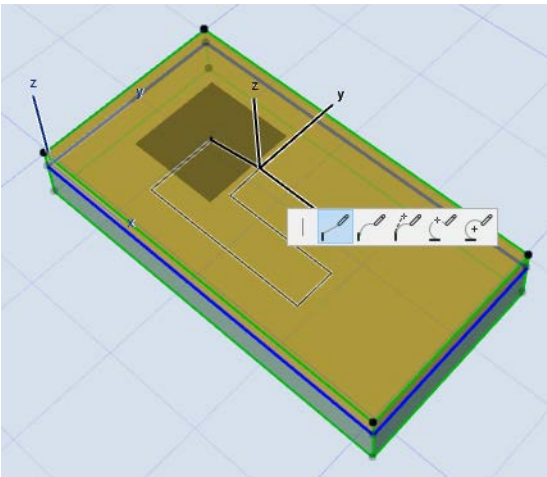
You may need to make holes in Slabs to accommodate objects such as staircases, or for an open atrium space.

1. Select the existing Slab.

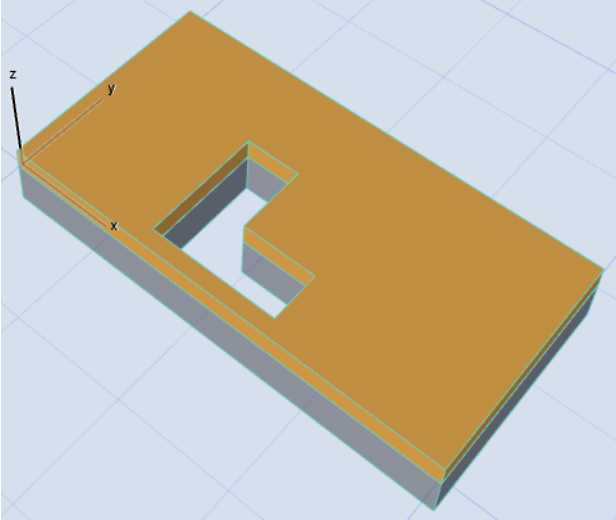


First Select the Existing Slab

2. With the Slab tool active, draw a new contour within the Slab's boundaries.



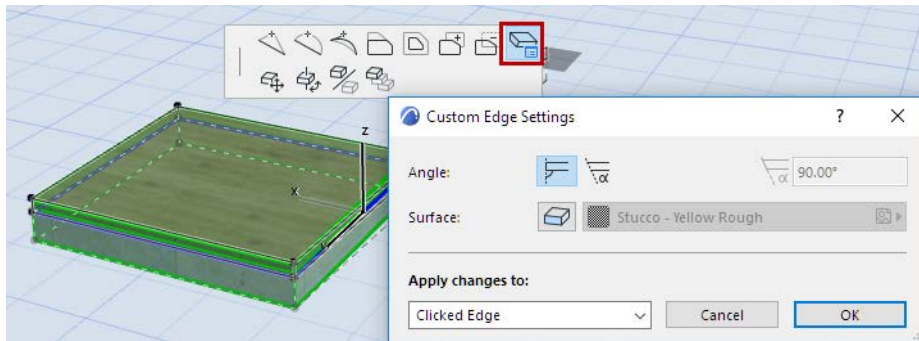
3. This new contour will be interpreted as a hole in the Slab.



See also [Selecting Slab Hole](#).

Set Custom Slab Edge Angle and Edge Surface

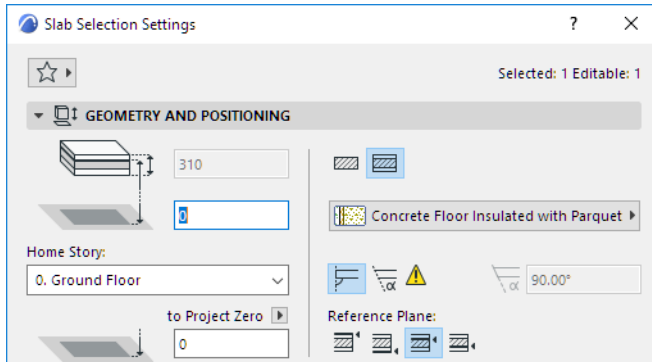
To customize a slab edge angle, and to apply a custom surface to the edge(s) of a slab and/or its holes, select the slab (or just its hole) and click on an edge with the Mercedes cursor to bring up a pet palette. Select the **Custom Edge settings** icon to open the dialog box.



Customize Slab Edge Angle

Angle: Choose an angle (Vertical or Custom) for the slab edge. If you choose Custom, enter the desired angle value in the field at right.

If you apply an edge angle here that is different from the default edge angle (set in Slab Settings), the Geometry and Positioning Panel of Slab Settings will indicate that a custom edge angle has been applied.

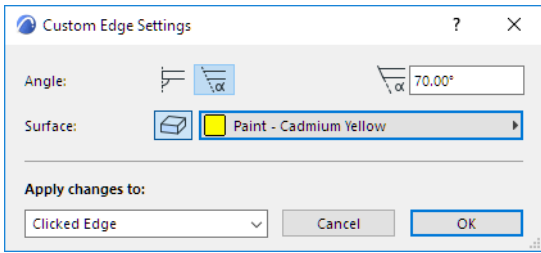


If you later change the Edge angle setting in Slab Settings while the dialog box is still open, the yellow patch (indicating “custom edge”) gets a red shadow. This means that the Slab’s edge angle is going to change: do you want to apply this change to the custom-angle edges too? If so, check the box at the bottom of Slab Settings (Apply changes to custom planes/edges too). If you do not check the box, your custom edge angles remain the way you set them.

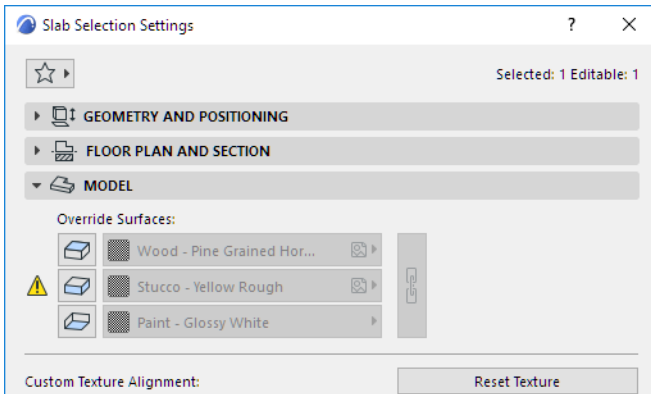
Custom Slab Edge Surface

Note: By default, the Surface material applied to a Slab edge derives from the Building Material assigned to the Slab. You can override *all* edge surfaces of the Slab, using the control in the Model Panel of Roof Settings. The Custom Edge Settings offers a further way to customize the surface of *any one or more* Slab Edges.

Surface: Click the Surface override button, then choose a custom surface for the slab edge(s).

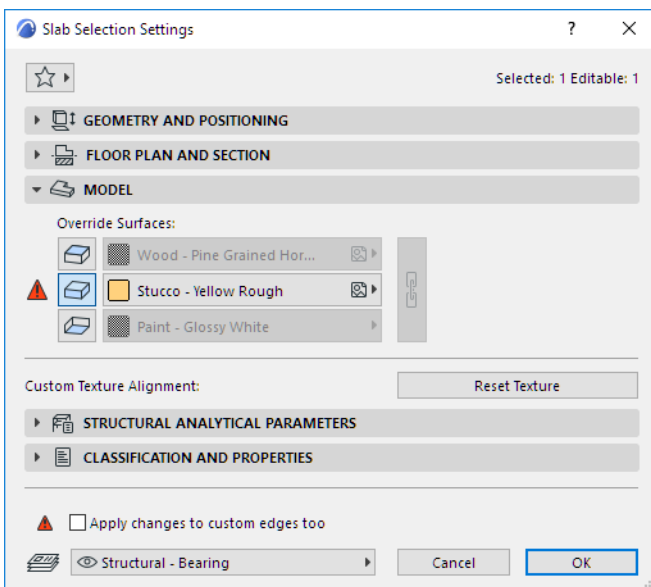


If you apply a surface here to any edge that is different from the edge surface defined in the Model Panel of Slab Settings (or Building Material), the Model Panel will indicate (with a yellow patch) that a custom edge surface has been applied.



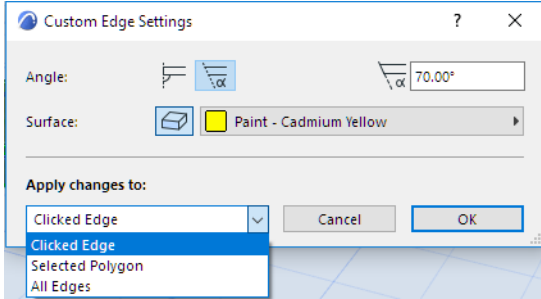
If you override the Slab edge surface in Slab Settings while the dialog box is still open, the yellow patch (indicating “custom edge”) gets a red shadow: this means that the Slab’s general edge setting is going to change: do you want to *apply this override to all edges*, including the custom edge(s) too?

If so, check the box at the bottom of Slab Settings (**Apply changes to custom planes/edges too**). If you do not check the box, your custom edge surfaces remain the way you set them.



Apply Changes

Apply changes to:



- **Clicked Edge** will apply the options you choose below (angle, surface) only to the edge you clicked when bringing up the pet palette.
- **Selected Polygon** will apply the chosen options to the edges of the slab only or the edges of the hole only, depending on which one is selected.

See [Selecting Slab Hole below](#).

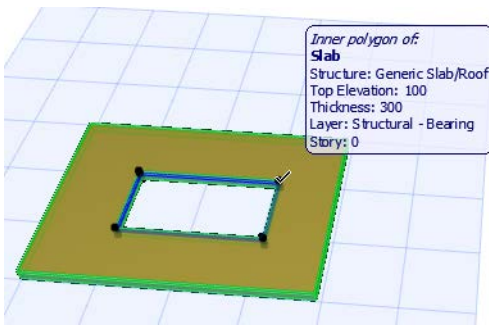
- **All Edges** will apply the chosen options to all edges of the slab as well as all edges of its holes. (If the selected slab has no holes, then Selected Polygon and All Edges have the same effect.)

Selecting Slab Hole

To customize the Slab Hole edge or angle, you must select the hole, rather than the entire slab.

To ensure that you have selected the hole, make sure the Info Tag says “Inner polygon of” the selected Slab:

To apply the customized values to the hole only, then choose **Selected Polygon** from the Apply On pop-up.



Meshes

Meshes are surfaces of any form created by defining the elevation of their characteristic points and interpolating between them.

A Mesh is created on the foundation of the Mesh Reference Plane and the ridges of the Mesh.

Like other construction elements, a Mesh's essential structure derives from its Building Material.

To aid in precise element placement, you can turn on the “**Gravitate to Mesh**” command to place additional elements onto the surface of the Mesh.

For information on converting Surveyors' data into an Archicad Mesh, see [Surveyors' data \(.xyz\)](#).

Related Topics:

[Display of Meshes](#)

[Create a Mesh](#)

[Edit Elevation of a Mesh Point](#)

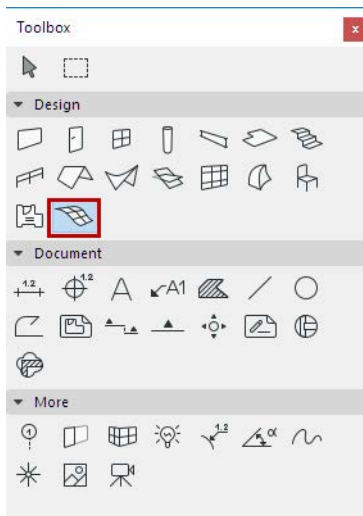
[Add New Points to the Mesh](#)

[Create a Hole in the Mesh](#)

[Mesh Tool Settings](#)

Create a Mesh

New Meshes can be created in either the Floor Plan or the 3D Window.



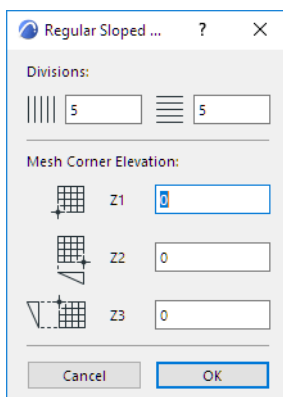
See also [Surveyors' data \(.xyz\)](#).

The Mesh has four geometry methods in the Info Box. You can create **Polygonal**, **Rectangular** and **Rotated Rectangular** shapes or a **Regular Sloped Mesh**.



In all cases, you will draw a Mesh polygon at the elevation of the Mesh reference plane defined in the Settings dialog box.

- The **Polygonal** and **Rectangular/Rotated Rectangular** Mesh geometry methods work the same as similar controls for the Slab tool.
- The **Regular Sloped Mesh** method helps you quickly create regular Meshes with plane surfaces. After defining the rectangle contour of the Mesh, the **Regular Sloped Mesh** dialog box opens automatically.



- **Divisions:** In the first part of the dialog box, set the number of divisions in either coordinate direction.
- **Mesh Corner Elevation:** Set the elevation values of three corner points of the Mesh; the fourth one is generated automatically.

If needed, select the Mesh and edit its points, add new points, or create a hole.

Related topics:

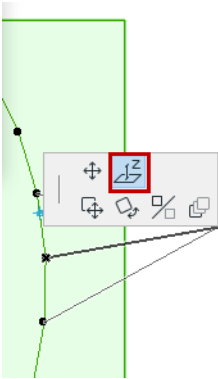
[Edit Elevation of a Mesh Point](#)

[Add New Points to the Mesh](#)

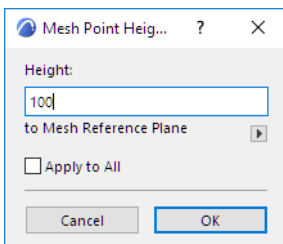
[Create a Hole in the Mesh](#)

Edit Elevation of a Mesh Point

1. Select the Mesh.
2. With the Mesh tool active, click a Mesh Point to bring up the pet palette.



3. Choose the **Elevate Mesh Point** command from the pet palette.
If you are in the Floor Plan, this opens the **Mesh Point Height** dialog box.

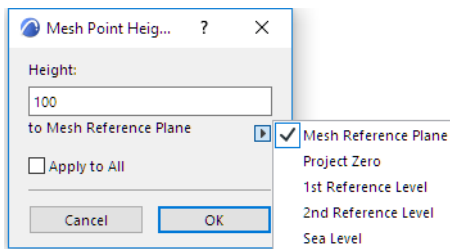


4. Enter a new height value in the text field.

Checking the **Apply to All** box will set all Mesh points to this new height.

Changing the elevation of a single point will not affect the elevation of the neighboring points.

The arrow below the text field opens a pop-up menu in which you can choose a reference level for the modified height.



Add New Points to the Mesh

To add points to an existing Mesh, draw a line or polygon onto the existing Mesh using either the Mesh tool or the Line/Polyline tools. The workflow is a little different for each:

Use Mesh Tool

1. Select the Mesh.
2. Activate the Mesh tool.
3. Draw a closed polygon or a straight segment within the Mesh polygon.

(If you draw outside of the contour of the first polygon, only points within the contour will constitute the Mesh.)

The **New Mesh Points** dialog box opens.

Continue by choosing an option for the new points (see below).

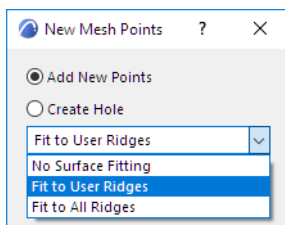
Use Line or Polyline Tool

1. Activate the Line or Polyline Tool.
2. Draw a line or polygon to create a set of points on the existing Mesh.
3. Select the Mesh.
4. Activate the Mesh tool.
5. With the Magic Wand method (Spacebar + click), click on the newly drawn line/polyline.

The **New Mesh Points** dialog box opens.

Continue by choosing an option for the new points (see below).

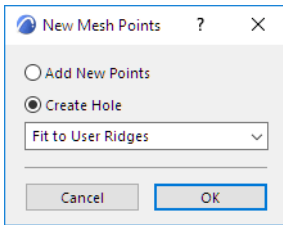
New Mesh Points Options



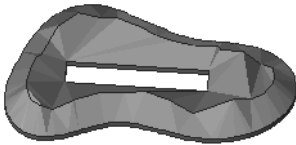
1. Choose the **Add New Points** radio button. (If you drew a two-point line rather than a polygon, this is the default and only choice.)
2. Choose an option from the drop-down menu to define the relationship between the newly created points and existing ones.
 - If you choose **No Surface Fitting**, the new points will be added with the height value currently set in the **Mesh Settings** dialog box. Intersected user-defined ridges will keep their height.
 - If you choose **Fit to User Ridges**, the new points will be placed on the current surface of the Mesh, and only user-defined ridges will keep their height. (The internal triangulation of the current Mesh will be altered.)
 - If you choose **Fit to All Ridges**, the new points will be placed on the current surface of the Mesh, and all ridges will keep their height. (The internal triangulation of the current Mesh will be unaffected.) This method is the most precise one, but it may generate a rather large number of ridges.

Create a Hole in the Mesh

1. Select the existing Mesh.
2. With the Mesh tool, draw a closed polygon that falls entirely within the Mesh polygon.
3. The New Mesh Points dialog box opens.



4. Choose the “Create Hole” radio button.
5. Choose an option from the drop-down menu to define the relationship between the newly created points and existing ones.
 - **No Surface Fitting:** the slopes of the Mesh will be connected to the bottom elevation of the Mesh.



- **Fit to User Ridges** or **Fit to All Ridges:** With these options, no slopes will be connected to the bottom of the Mesh, allowing you to place your building exactly into the excavated site.



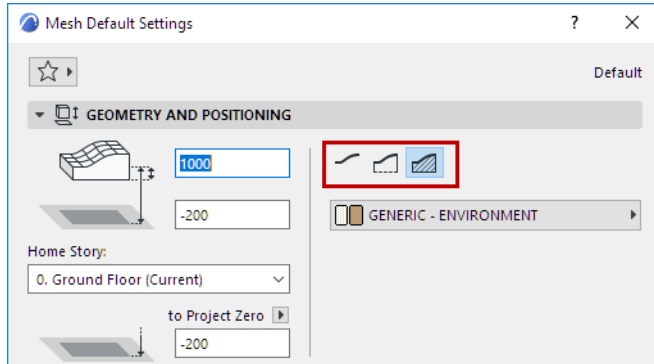
Display of Meshes

Mesh Structure

In the Floor Plan, only the outline and the ridges of the Mesh are shown.

Note: The Floor Plan outlines of Meshes and Slabs on remote stories (if they are shown on stories other than their home stories) are displayed using the line type set in **Project Preferences > Legacy**.

In 3D, depending on the construction method chosen in the **Info Box** or Mesh Settings, the Mesh can be displayed in three ways:

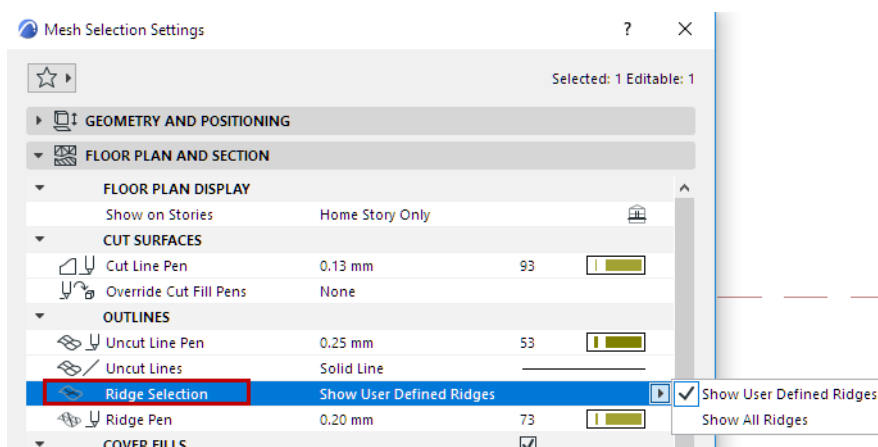


- Surface Only
- Surface with Skirt
- Solid Body

Display of Ridges

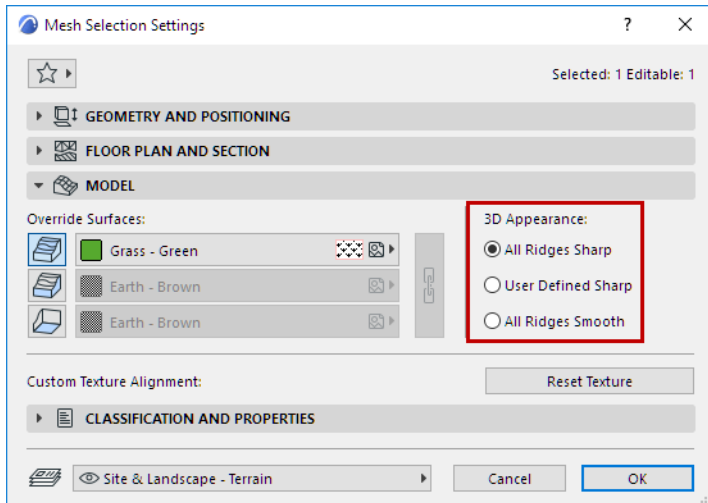
There are two types of ridges in a Mesh: **user-defined** and **generated ridges**. Each generated ridge connects two Mesh points at different heights that are not already connected by a user-defined ridge.

- User-defined ridges are always displayed.
- If the **Show All Ridges** option is selected, Archicad will also show the ridges generated through the connecting Mesh nodes. (This control is located in Mesh Tool Settings). Generated ridges are shown only if they connect points that differ in elevation.



3D Appearance

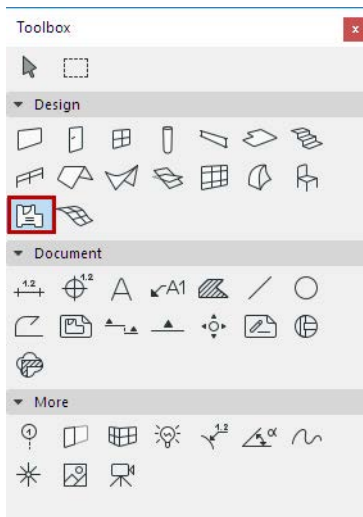
The controls of the Model panel of Mesh Settings defines whether the Mesh ridges are displayed as smooth or sharp.



- **All Ridges Sharp:** Show the Mesh in the 3D window with sharply connecting triangles.
- **User Defined Sharp:** Show the Mesh surfaces with sharply connecting user defined ridges, and with smoothly connecting surfaces everywhere else.
- **All Ridges Smooth:** Show the Mesh with smoothly connecting surfaces.
If you choose **All Ridges Smooth**, only user-defined ridges will appear in 3D view.

Zones

Zones are spatial units in your project, created with the Zone tool.



Zones usually represent rooms; wings of a building; blocks of a housing estate; or functional areas of a building. Zones in 3D can also be used for simple mass modeling; and they form the basis of the Model Review process in the Energy Evaluation function.

[Creating Zones](#)

[Edit Zone Polygon](#)

[Zone Categories](#)

[Display of Zones in Floor Plan](#)

[Display of Zones in Section](#)

[Zone Stamps](#)

[3D Display of Zone Spaces](#)

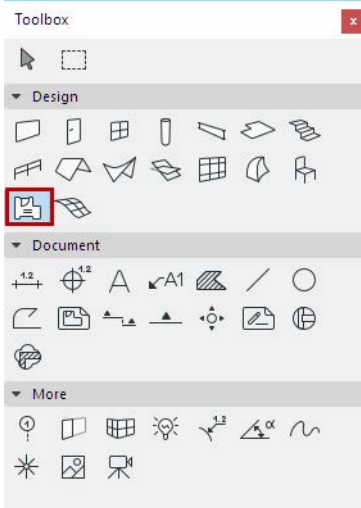
[Crop Zone to Another Element](#)

[Calculating Zone Area and Zone Volume](#)

[Updating Zones](#)

[Zone Tool Settings](#)

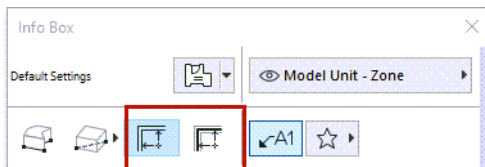
Creating Zones



Create an Automatic Zone

Most Zones are surrounded by Walls and the only openings in them are Doors and Windows. In this situation, the easiest way to create a Zone is using one of the automatic methods: **Inner Edge** or **Reference Line**.

1. Activate the Zone tool.
2. In the Info Box, choose the **Inner Edge** or **Reference Line** method (see below for details)



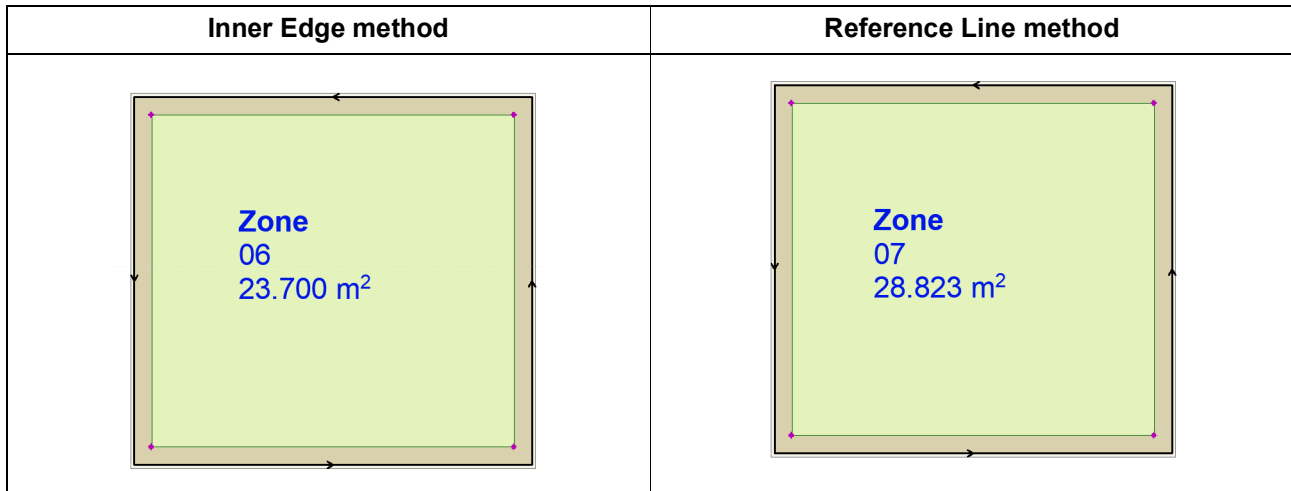
3. In the Floor Plan or 3D window, click within a closed polygon that consists of Walls and/or Curtain Walls.
4. Click with the Hammer cursor to place the Zone Label (either inside or outside the Zone polygon)

[See *Place Zone Label Automatically*.](#)

Zone Construction Methods: Inner Edge or Reference Line

- **Inner Edge** method: Zone area extends to inner edges of Walls.
- **Reference Line** method: Zone area extends to Wall reference lines.

You can see the difference in the Measured Area of the Zone:

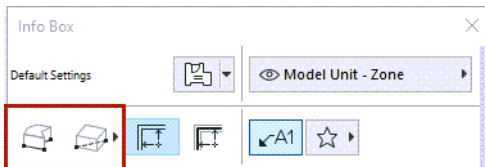


Note: For a Zone created with the Reference Line method, you can set its Floor Plan display as either “Gross” or “Net,” using the radio buttons in Zone Settings. [See Zone Polygon.](#)

Manual Zone Construction

Use a manual method if the Zone’s boundary is not clearly surrounded by elements (e.g. when doing space planning).

1. Activate the Zone tool.
2. In the Info Box, choose a Manual method: Polyline, Rectangular, or Rotated Rectangular



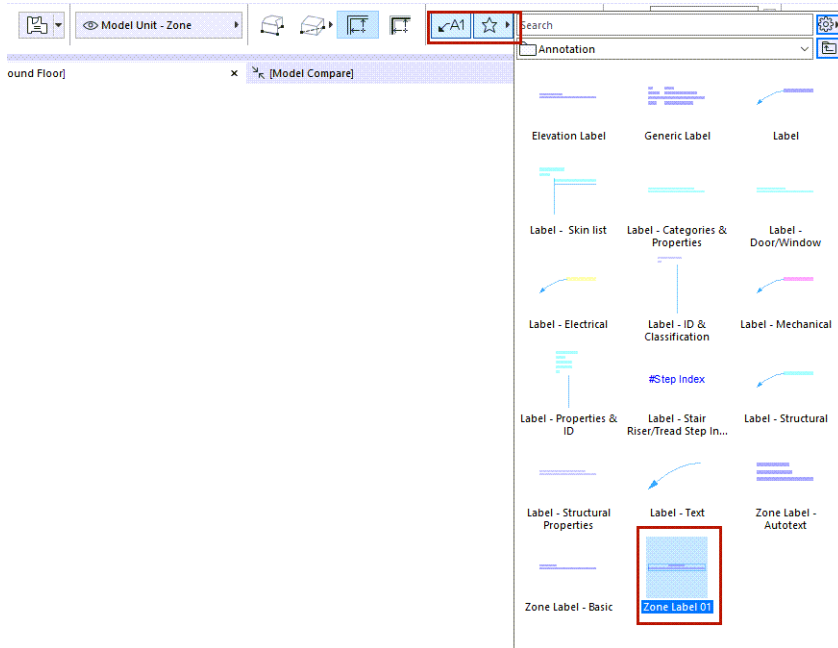
3. In the Floor Plan or 3D window, draw the Zone contour.
4. Click with the Hammer cursor to place the Zone Label (either inside or outside the Zone polygon.)

Place Zone Label Automatically

By default, Zone Labels are created automatically when you click to place the Zone: the **Place Label** toggle is on in the Zone Info box.



Choose a Zone Label from the Favorites pop-up.



If you don't want to place a Label when creating a Zone, turn the **Place Label** toggle off in the Zone Info Box.

Exclude a Wall as a Zone Boundary

By default, the Relation to Zones setting in the Model Panel of Wall Settings is set to “Zone Boundary.”

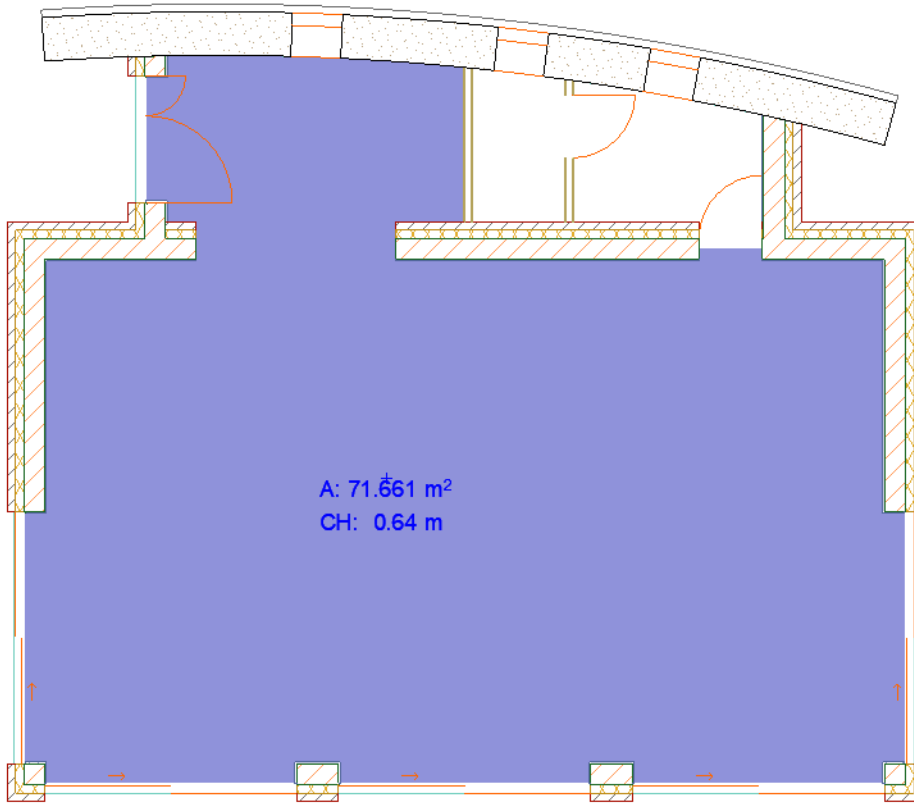
This means that all walls enclosing a closed space are considered Zone boundaries by default.

To exclude any wall as a Zone boundary, just change the Relation to Zones setting to **No Effect on Zones**.

See [Relation to Zones](#).

Create a Manual Zone Boundary

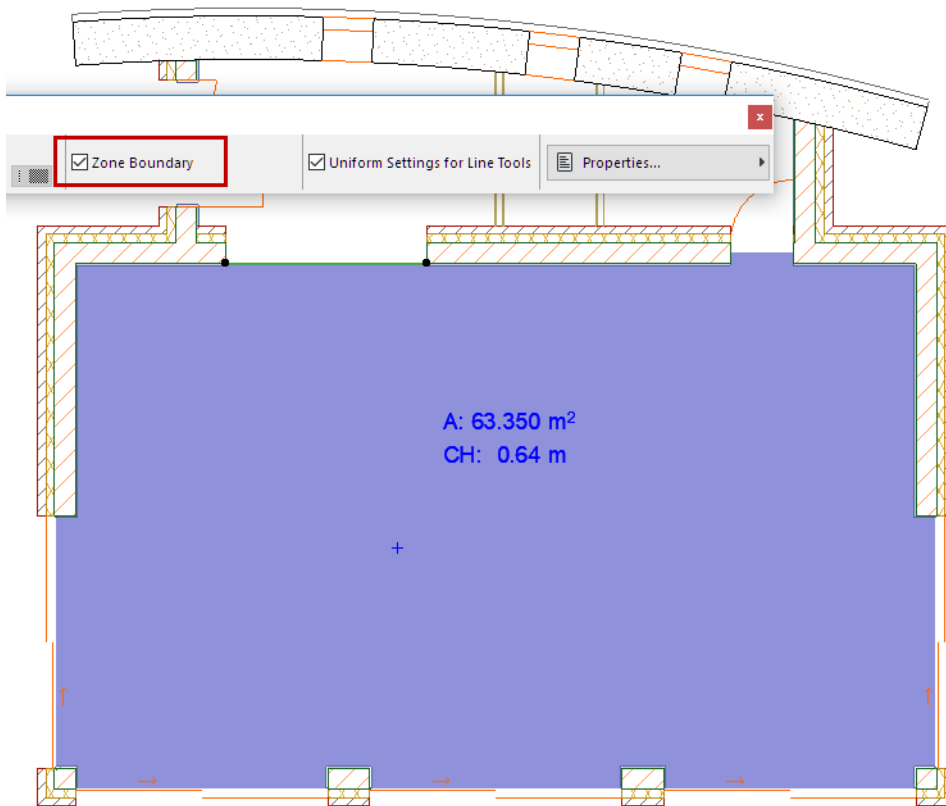
If the Zone has openings that are not Doors or Windows, Archicad will continue searching for boundaries outside the spatial unit that you wished to identify as a Zone. In this case, the result may not be what you expect.



To resolve this, do one of the following:

- Use the Manual construction method
- Place an Empty Opening Door object into the Wall which is the same height as the Wall - the Wall is not shown in 2D, but it does serve as a Zone Boundary

- Draw a Line, Arc or Spline to serve as a Zone boundary: check the **Zone Boundary** box in its Settings.



If your desired Zone boundaries also encompass freestanding shapes or walls, you may get a warning message when clicking to place the Zone Stamp. In this case, you must click closer to the bounding elements. Then the Zone will be created correctly.

Related Topics:

[Updating Zones](#)

[Relation to Zones](#)

[Calculating Zone Area and Zone Volume](#)

Edit Zone Polygon

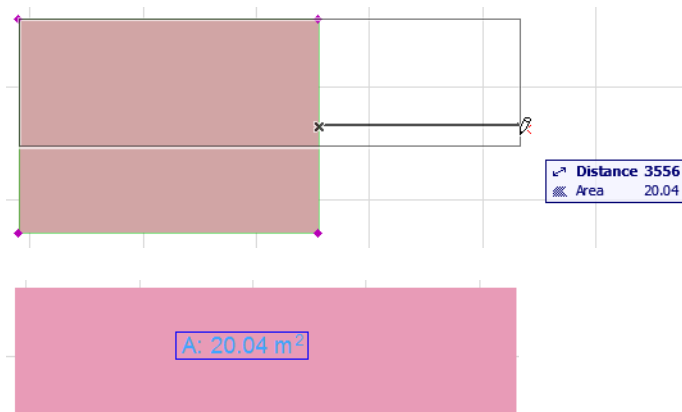
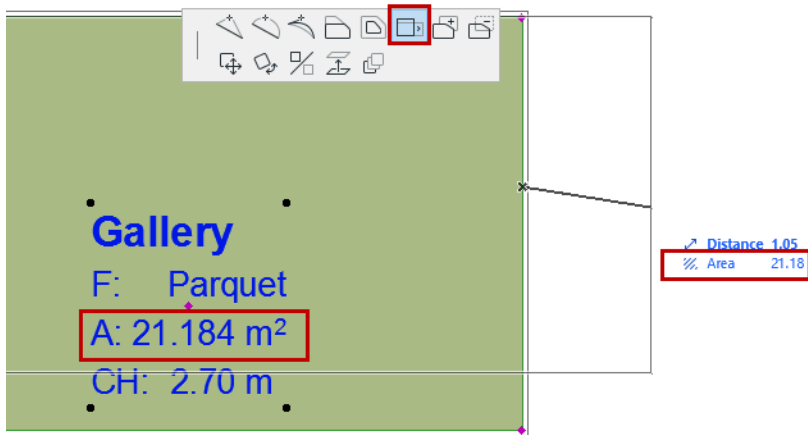
Use the familiar pet palette commands to edit the Zone polygon.

See [Reshaping Polygons and Chained Elements](#), and [Offset All Edges](#).

Offset Edge - Fixed Area

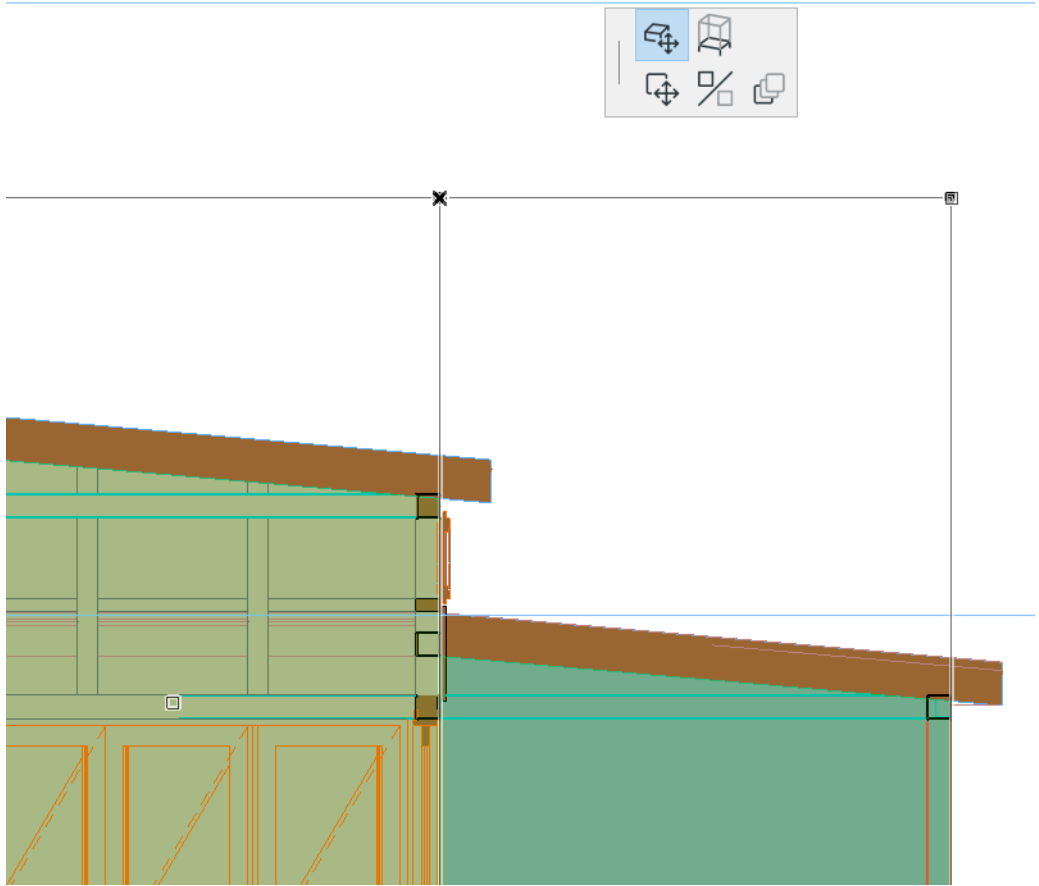
Use this pet palette command to offset the clicked edge while keeping the Zone area constant. This is useful if you are doing space planning based on defined areas.

In this example, you offset the right side of the Zone - and the shape is automatically constrained so that the area stays constant (as seen in the tracker):

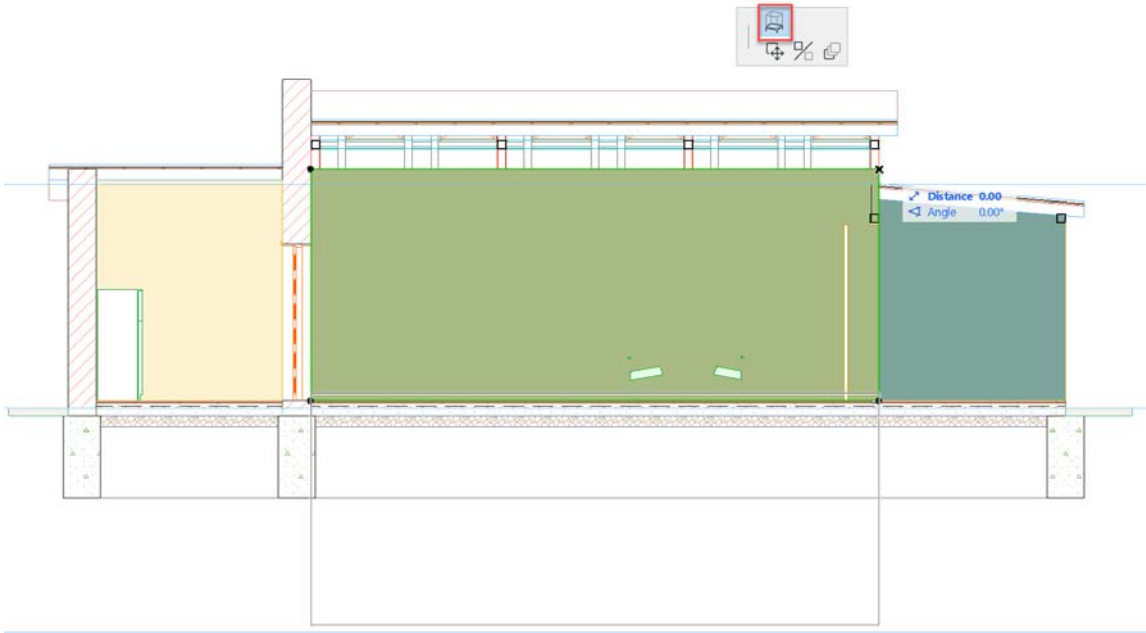


Zone Interactions on Section-Elevation

Use pet palette commands from selected Zone nodes, even if the Zone is cropped: Drag, Rotate, Mirror, Stretch Height.



If you use Stretch Height to modify the bottom of the Zone, its sub-floor thickness value remains unchanged.



Stretch Height of a Zone in Section

Zone Categories

Each Zone you create in your project is assigned a **Zone Category** in the Zone's Settings dialog box.

Zone Categories are Archicad attributes.

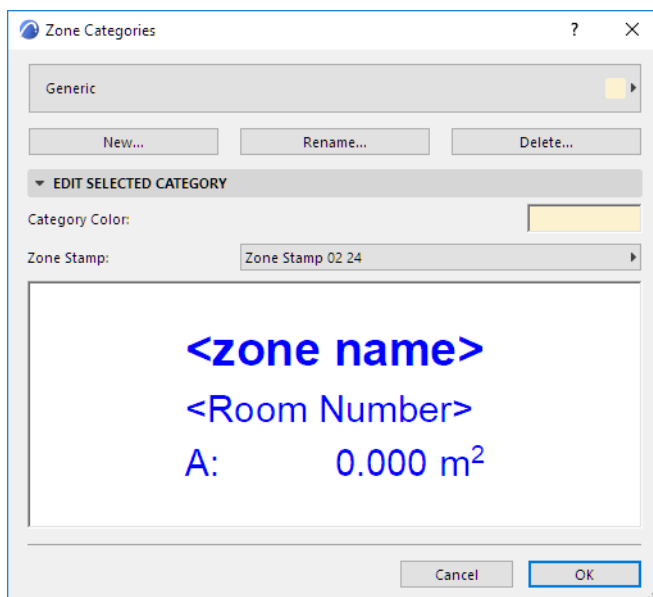
Zone Category defines:

- the default color of the Zone background (for all Floor Plan Zones, and for cut Zones in Section-type views and 3D Documents), and
- the Zone Stamp object used for the Zone. ([see Zone Stamps](#))

Note: Zone Stamps are turned off by default. (As of Archicad 25, Zones are placed with Zone Labels instead.)

Zone Categories Dialog Box

To define or redefine Zone Categories, go to **Options > Element Attributes > Zone Categories**.



Use the pop-up at top to choose a Zone Category from list of predefined Zone categories. Either edit its properties directly, or choose one of the following commands:

- **New:** Create a copy of the selected Zone Category under a new name.
- **Rename:** Rename the currently selected Zone Category.
- **Delete:** Remove the selected Zone Category from the project.

[See Delete attributes.](#)

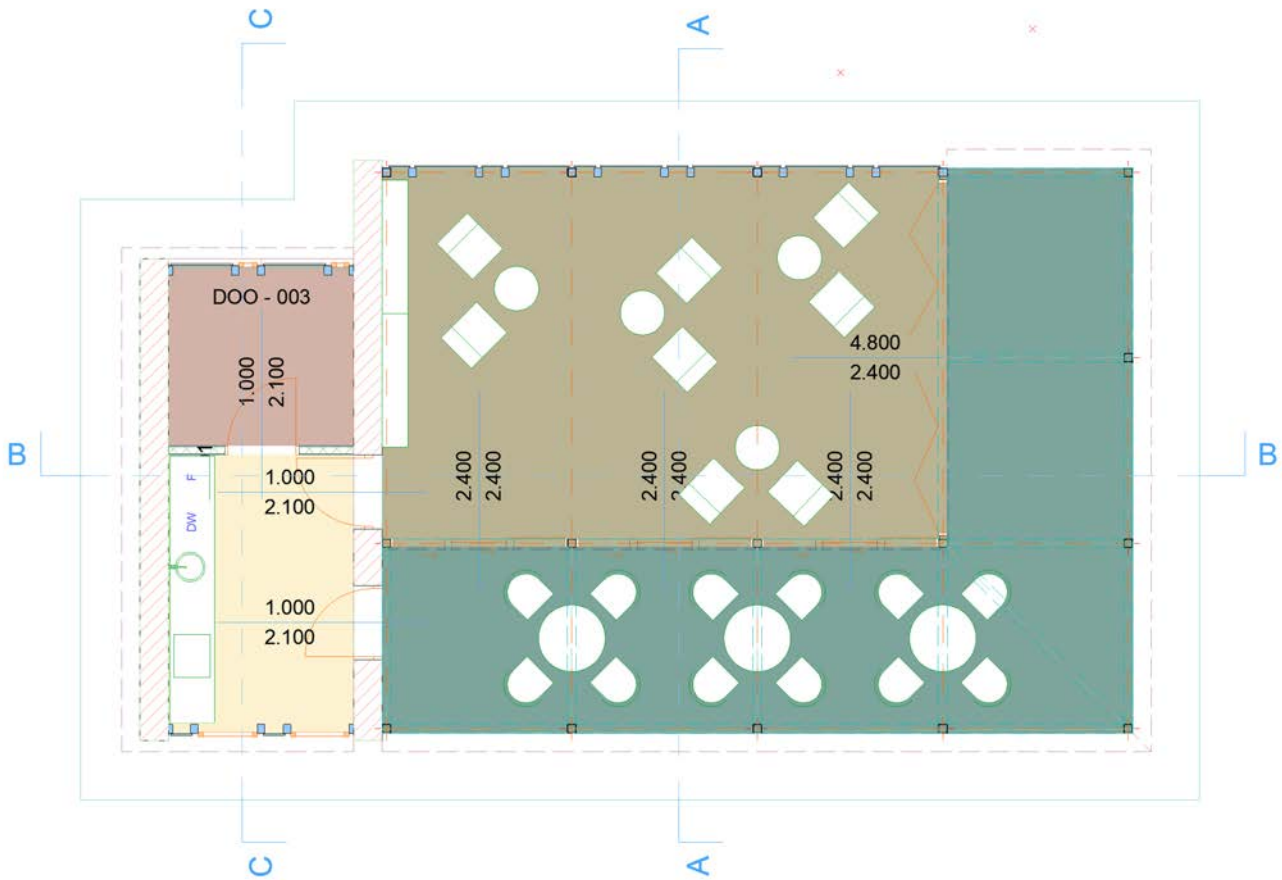
Zone Stamp pop-up: Choose a Zone Stamp from your loaded library.

Color: Choose a color for the selected Zone Category. ([see Zone Stamps](#))

A preview window helps you visualize the Zone Stamp parameters.

Display of Zones in Floor Plan

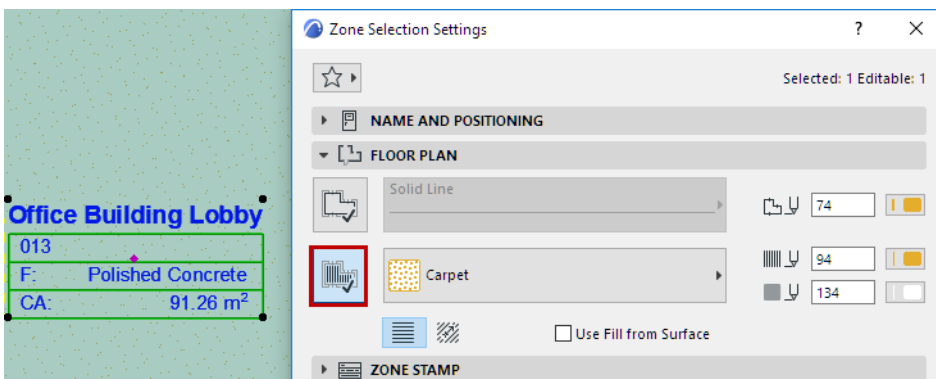
By default, Zone backgrounds are displayed with Zone Category color.



See [Zone Categories](#).

You can assign a different Zone fill by using any cover fill.

1. Go to the Floor Plan panel of Zone Settings.
2. Activate the Cover Fill icon at the left.



3. Choose a fill pattern from the fill pop-up menu.

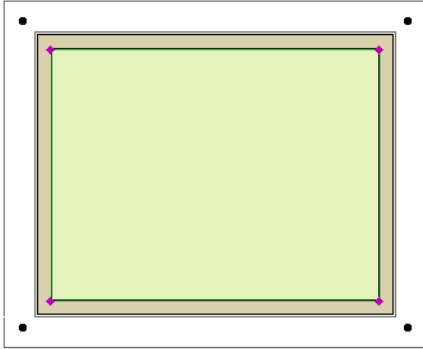
If you have assigned a Zone fill, it will be displayed in Floor Plan by default.

Note: Zone backgrounds (and other fills) can use Graphic Overrides.

See [Graphic Override Rules](#).

Display of Zones with Slanted Walls

Zones bounded by one or more slanted walls will accurately consider the slanted geometry. Such Zones are displayed on the Floor Plan with two separate Zone polygons:



- The polygon with **editable selection dots** delimits the visible part of the Zone on the Floor Plan (while you can edit this polygon, such editing has no effect on the true 3D Zone), and
- The polygon with **regular selection dots** indicates the true extent of the base of the 3D Zone (you cannot edit the Zone shape by editing this polygon.)

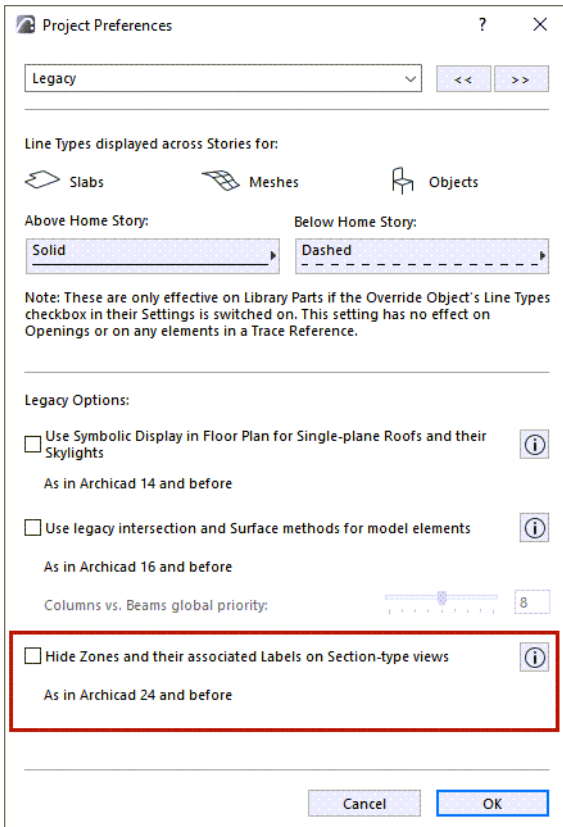
Note: For a Zone created with the Reference Line method, you can set its Floor Plan display as either “Gross” or “Net,” using the radio buttons in Zone Settings. See [Zone Polygon](#).

Related Topic

[Edit Zone Polygon](#)

Display of Zones in Section

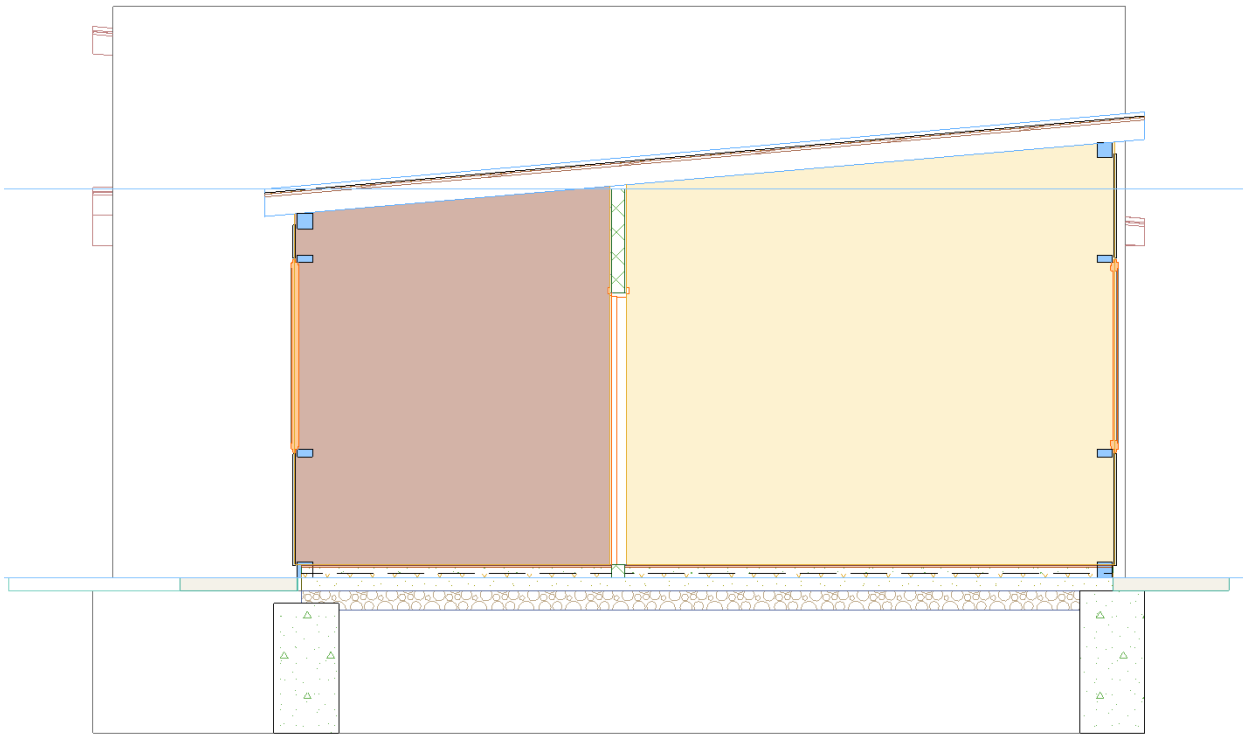
Display of Zones is new as of Archicad 25 Update 2. Make sure the checkbox at Options > Project Preferences > Legacy is off:



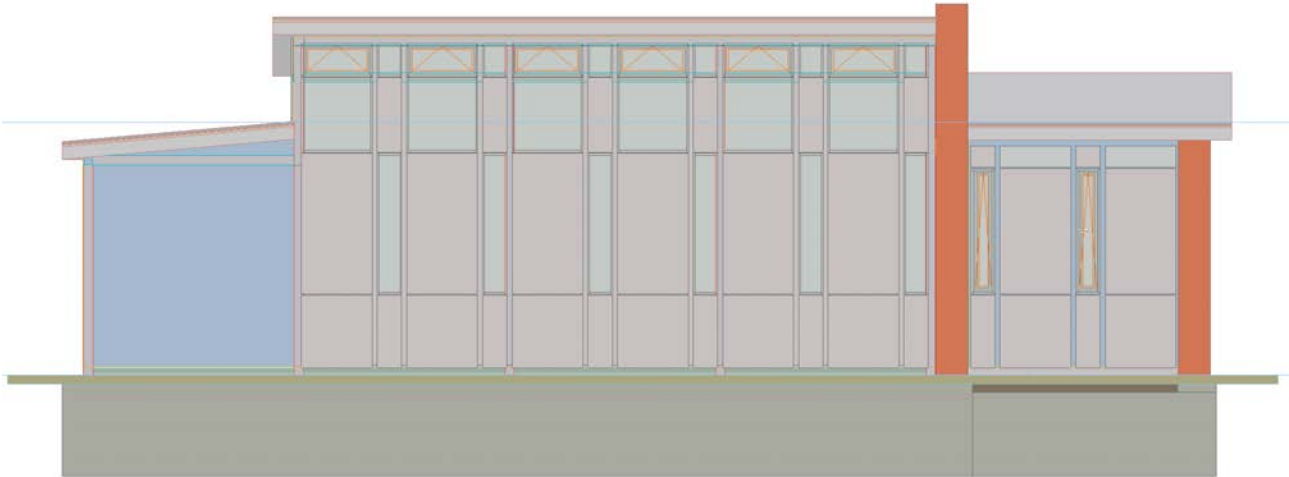
When opening projects started in Archicad 25 or before, this checkbox is switched on by default.

Default Display of Zones

	Cut section display	Uncut section display
Floor plan (Symbolic)	Category Color	Category Color
Section/Elevation/Interior Elevation	Category Color	Surface
3D Document	Category Color	Surface
3D window + cutaway	Surface	Surface



Section - cut default (Category color)



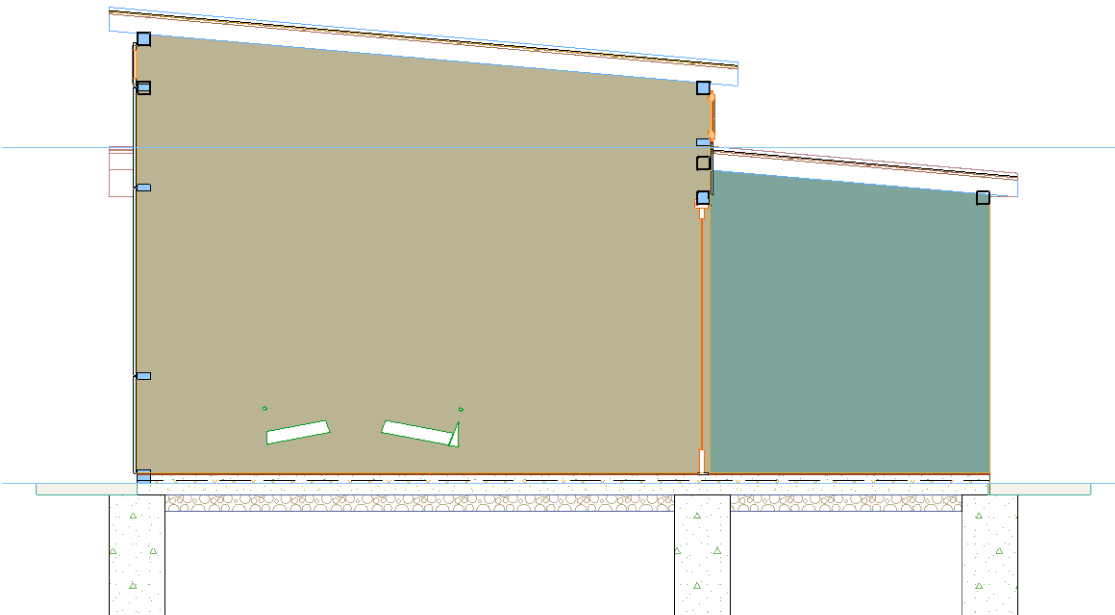
Elevation - uncut default (Surface)

Set Transparency of Cut Zones in Section

If you have a cut Zone in a Section, you can make it fully transparent by setting its surface transmittance parameter to a value of at least 50.

If the transmittance parameter is 49 or below, the Zone is opaque.

In a transparent Zone, the contours of objects will be displayed, but their colors are not affected.



Opaque Zone (transmittance value below 50)

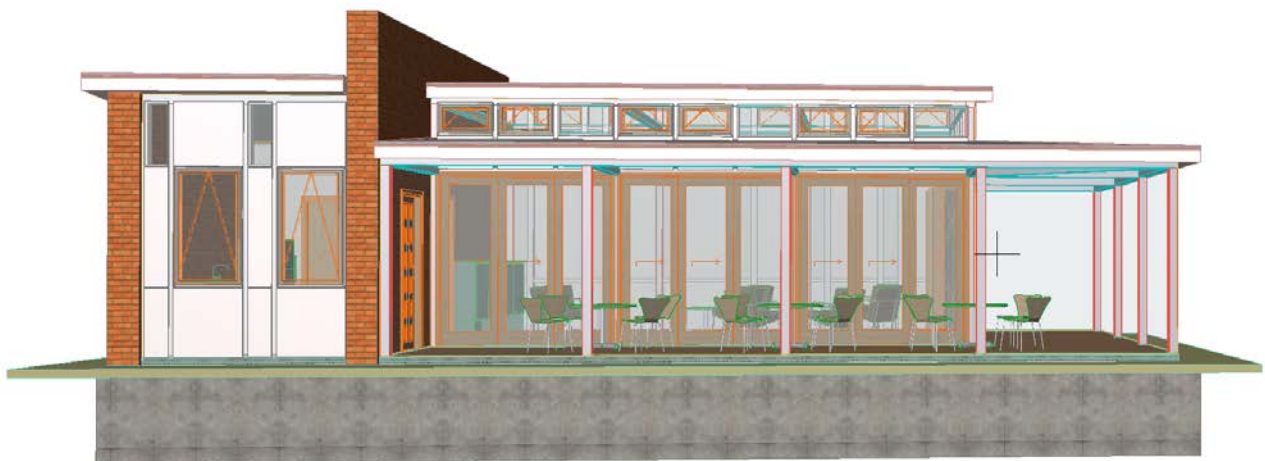


Transparent Zone (transmittance value above 50)

Set Transparency of Uncut Zones

If you have an uncut Zone (e.g. in an Elevation or 3D view), you can adjust its transparency using the surface transmittance parameter.

Any value over 0 makes the Zone transparent, but the transparency level varies depending on the transmittance value (a higher transmittance value makes the Zone increasingly transparent).



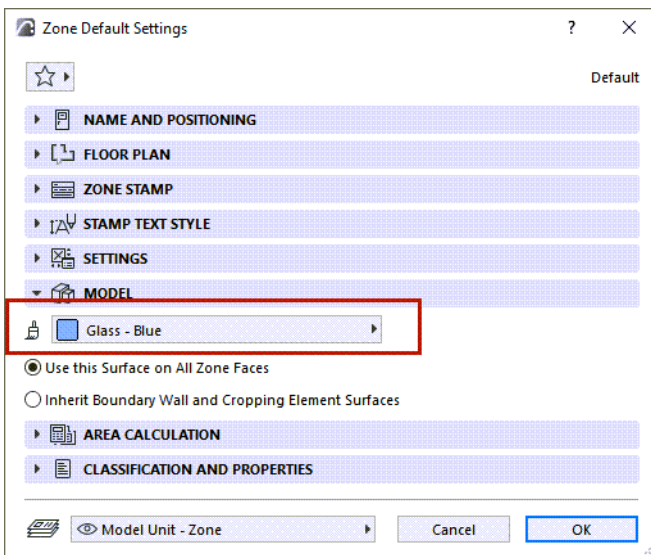
Uncut Zones - High Surface transmittance value



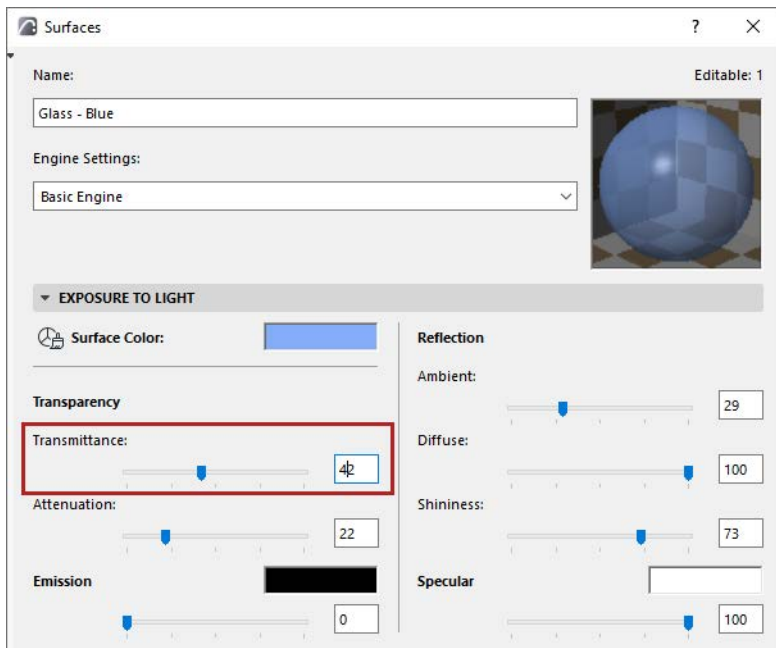
Uncut Zones - Low Surface transmittance value

Set Surface Transmittance for a Zone

1. Select the Zone and open Zone Settings
2. In the Model panel, check the name of the Surface.



3. Go to Options > Element Attributes > Surfaces to adjust the Transmittance level:



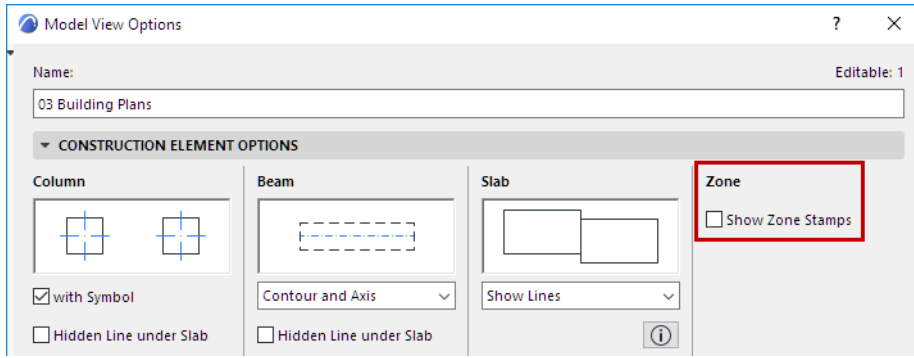
Related Topic

[Edit Zone Polygon](#)

Zone Stamps

Note: Zone Stamps are turned off by default. (As of Archicad 25, Zones are placed with Zone Labels instead.)

To display Zone Stamps in the project, check **Show Zone Stamp** in **Document > Model View > Model View Options** (Construction Element Options panel).



Zone Stamps are intelligent parametric GDL Objects whose look, contents and behavior can be fitted to local architectural practice. They reside in the Archicad Library.

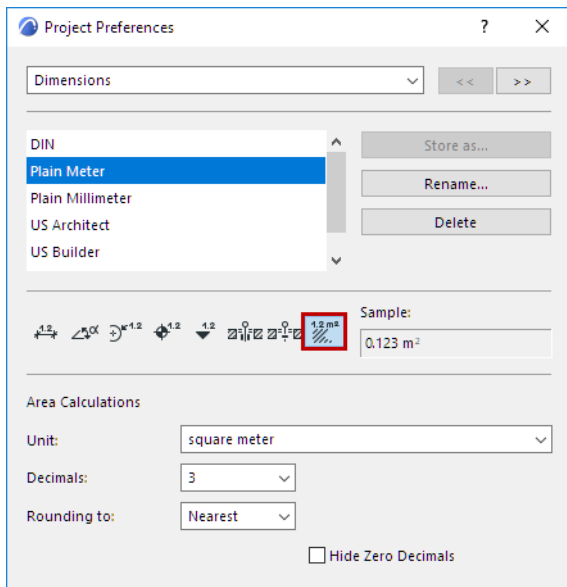
The Zone Stamp object assigned to your Zone depends on the Zone Category you have chosen.

[See Zone Categories.](#)

Each Zone Stamp contains textual information about the Zone, including its name, number, area, and other optional parameters; the exact content of your Zone Stamp depends on the parameters you set in the Zone Stamp Panel of Zone Settings.

[See Zone Tool Settings.](#)

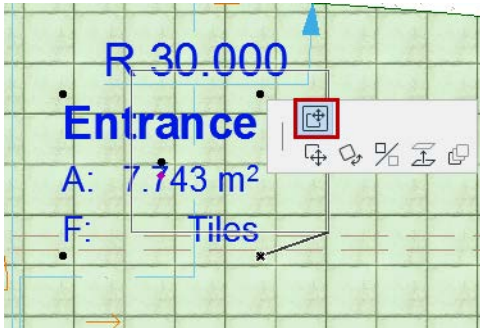
Note: To set the units for the Zone Area display on this Zone Stamp, use **Options > Project Preferences > Dimensions**, and select Area Calculation units.



Moving Zone Stamps

Zone Stamps can be moved independently of the Zone:

- Click on a node of the Zone Stamp.
- Choose the **Move Sub-Element** icon from the pet palette.



- Drag the Zone Stamp.

Zone Reference Point (for Zone Stamp only)

If you place the Zone Stamp with one of the automatic recognition methods, a cross appears inside the Zone at the location that you first clicked to define the Zone's area. This cross indicates the reference point of the Zone.



This point is used as the starting point of boundary recognition for subsequent updating actions.

3D Display of Zone Spaces

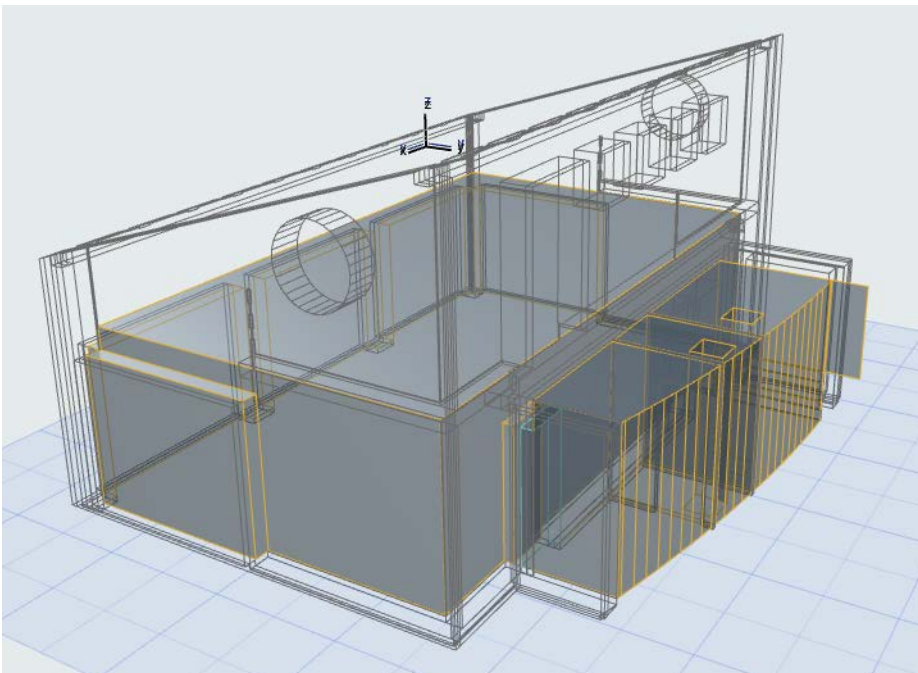
To view Zone spaces in 3D:

Go to **View > Elements in 3D View > Filter and Cut Elements in 3D** and make sure the “Zone” checkbox is selected.

Use the Model Panel of Zone Settings to choose a Surface for the Zone in 3D.

See [Zone Model Panel](#).

Note: You may wish to display 3D Zones as solid bodies in shading mode, and all other elements types in wireframe mode. To achieve this effect, apply the predefined “Show 3D Zones as Solid” Layer Combination.



Crop Zone to Another Element

Zone spaces can be cropped to single-plane roofs, slabs and beams with the **Design > Roof Extras > Crop Zone** command.

You can either select an element (Roof, Slab, Wall, Column, Beam, Curtain Wall or Library Part) and crop a Zone to it if the element protrudes into the Zone, or select a Zone and then choose which element types to crop it to.

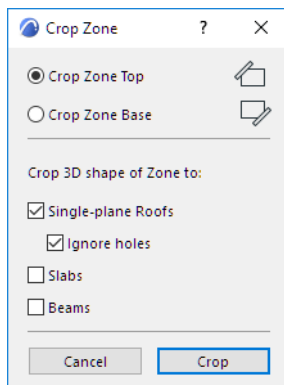
Note: Zone space will be cropped by any multi-story element with which it intersects, on any story.

Note: The **Crop Zone** command can crop to single-plane roofs only. If you have a multiplane roof, you can use the associative trim function instead.

See [Trim Elements to Roof or Shell](#).

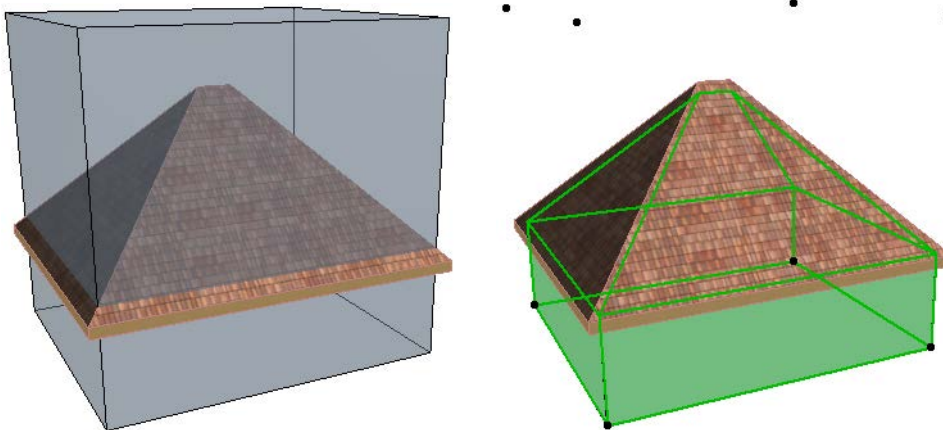
When cropping a Zone to a single-plane roof, a checkbox gives you the option of ignoring holes (that is, to include any hole in the roof as part of the roof for the purposes of a Zone boundary.)

You can choose between cropping the base or the top of Zone spaces. Check the corresponding boxes to define which elements you wish to crop to the Zone.



The crop will be executed by clicking the **Crop** button in the dialog box.

If you select only one type of cropping element in the project (e.g. Roof), all the checkboxes are grayed.



Zone space crops can be undone by selecting the cropped Zone space and pressing the **Undo All Crops** command from the selected Zone's context menu, or at **Design > Roof Extras**.

Note: You can also opt to undo the Crop when updating modified Zones.

For more information, see [Updating Zones](#).

Calculating Zone Area and Zone Volume

Zone area and volume calculation depends on several different settings:

- **Relation to Zones** options chosen for Walls, Columns, and Curtain Walls, which can affect the Zone's area and/or volume
- **Zones Preferences**, at **Options > Project Preferences > Zones**, containing global controls for calculating Door/Window recesses in Walls, and for refining the area calculation of Walls or Columns in some cases
- The **Area Calculation Panel** of Zone Settings, which allows you to check the measured area of the Zone and refine the calculated values.

Note: To see the effect of any changes in Zone calculation settings, you must execute the **Design > Update Zones** command.

See [Updating Zones](#).

Related Topics

[Relation to Zones](#)

[Zone Area Calculation Panel](#)

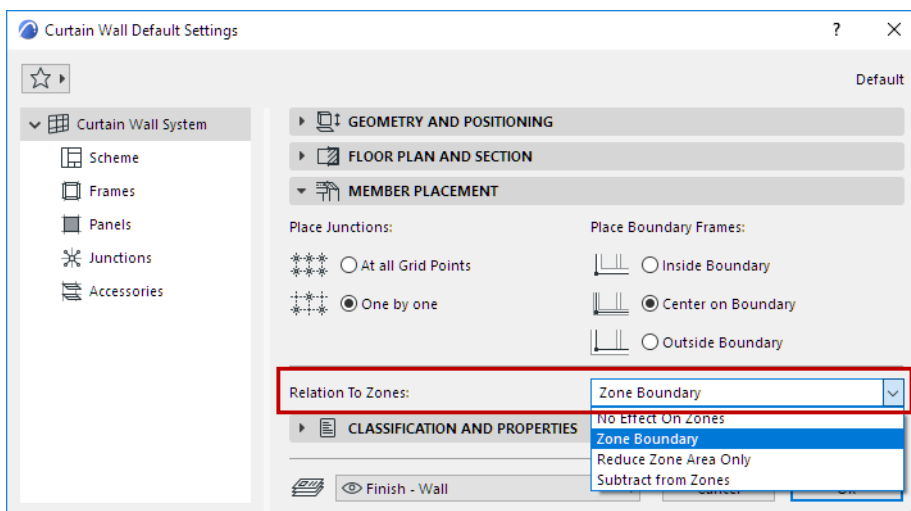
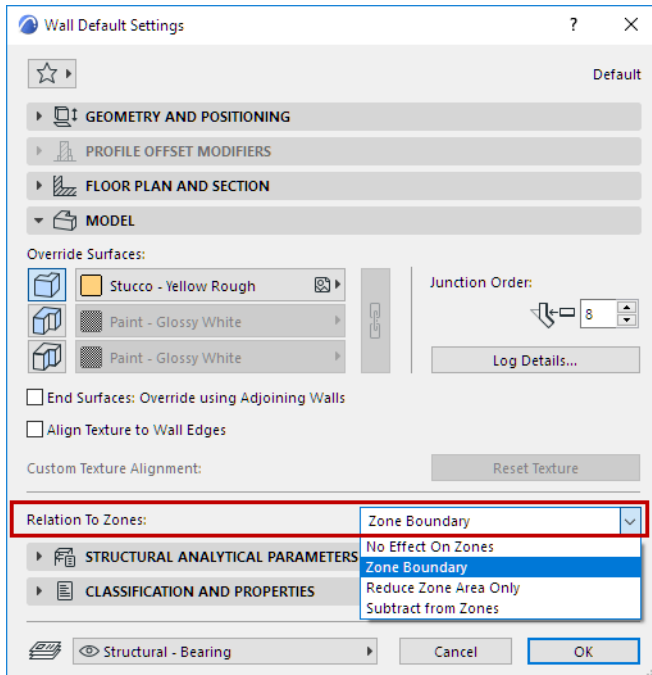
[Zones Preferences](#)

Relation to Zones

This setting is available for Walls, Curtain Walls, and Columns. It is relevant if the element is located within a Zone polygon: Zone area calculation depends on the element's Relation to Zones setting. That is: should the Zone area include or exclude the area of this element?

For each Wall, Curtain Wall and Column, define its Relation to Zones using the pop-up in Element Settings.

- for Walls: use the **Model Panel**
- for Columns: use the **Positioning Panel**
- for Curtain Walls: use the **Member Placement Panel** of Curtain Wall System Settings



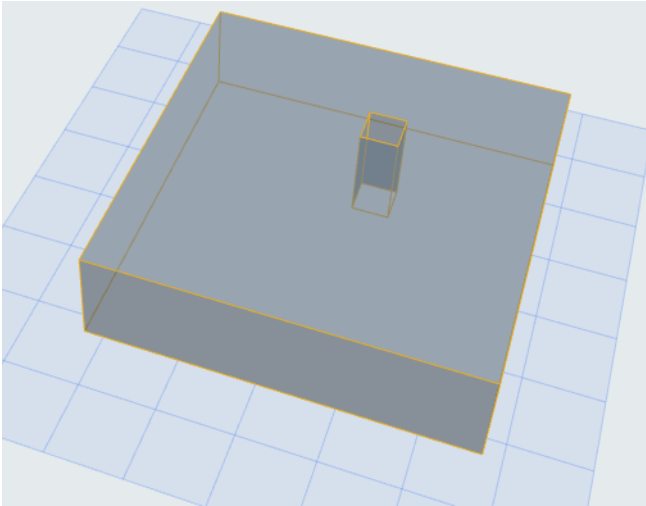
Note: To see the effect of any changes in Zone calculation settings, you must execute the **Design > Update Zones** command.

See [Updating Zones](#).

Zone Boundary

A Wall or Curtain Wall inside the Zone, if set to “Zone Boundary”, delimits the Zone. Thus, the Zone boundary extends up until the Wall/Curtain Wall. If the Wall/Curtain Wall is slanted, the Zone’s geometry is slanted accordingly.

- The Wall/Curtain Wall, if located inside the Zone area, is *not* included in the Measured Area (Zone parameter).
- A vertical Column set to “Zone Boundary” is *not* included in the Measured Area (Zone parameter).
- A slanted Column cannot be set to “Zone Boundary”. For a slanted Column, use the Subtract from Zone setting.



Vertical Column Set to “Zone Boundary”: Excluded from Measured Area (Zone Listing Parameter)

Note: Any line-type element can also be set as “Zone Boundary” in its Settings dialog box. Such a line acts as a 2D delimiter of the Zone.

Reduce Zone Area Only

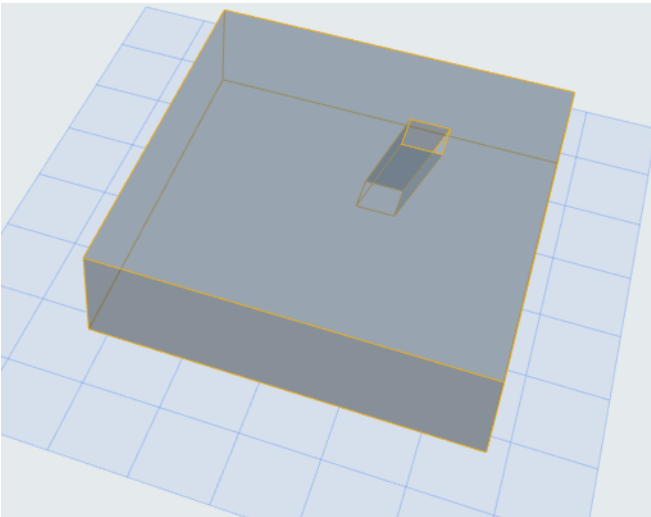
- Area of the element is *excluded* from the Measured Area
- Volume of the element is *included* in the Zone volume

Note: If you have set a Wall or Column to “Reduce Zone Area”, you may prefer to ignore this setting for very small-sized walls or columns. To set the minimum limit for excluding such walls and columns, go to **Options > Project Preferences > Zones**.

To see how much of a Zone’s area has been reduced due to Wall/Column reduction, select the Zone and view the data in the Area Calculation Panel of Zone Settings.

Subtract from Zones

- Area of the element is *subtracted* from the Measured Zone Area and the Net Area. Unlike the Zone Boundary setting, this setting is available for slanted Columns.
- Volume of the element is *subtracted* from the Zone volume



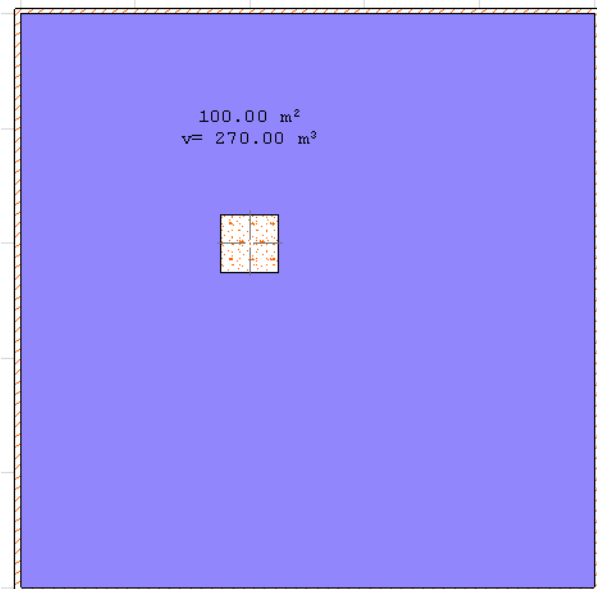
Slanted Column Set to "Subtract from Zones": Excluded from Measured Area (Zone Listing Parameter)

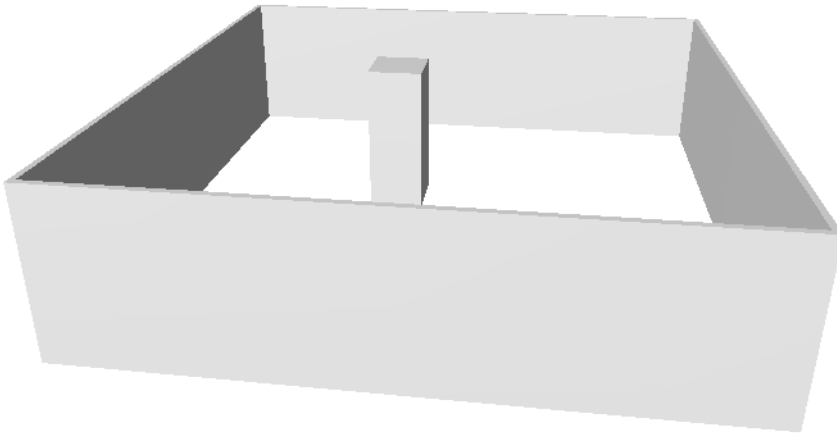
No Effect On Zones

The element has no effect on Zone calculation: the Zone area and volume will include the area and volume occupied by the element.

Example

In this example, the Zone area is 100 square meters, with a height of 2.70 meters. The Zone includes a square 1-meter Column.





Relation to Zones	Measured Area	Volume
Zone Boundary	99 m ²	267.3 m ³
Reduce Zone Area Only	99 m ²	270 m ³
Subtract from Zones	99 m ²	267.3 m ³
No Effect on Zones	100 m ²	270 m ³

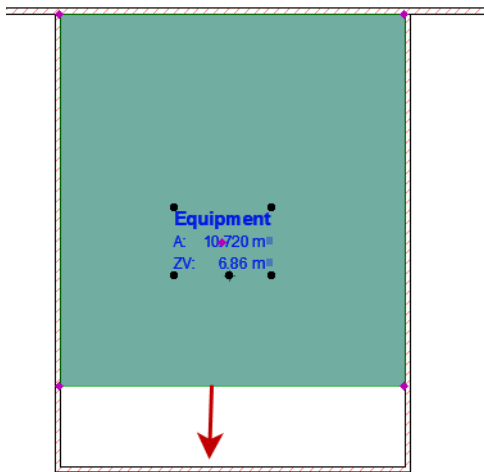
Related Topics

[Zone Area Calculation Panel](#)

[Zones Preferences](#)

Updating Zones

When you modify your design, automatically created Zones do not automatically adjust themselves to the changes, so they may require you to use the Update Zones command.

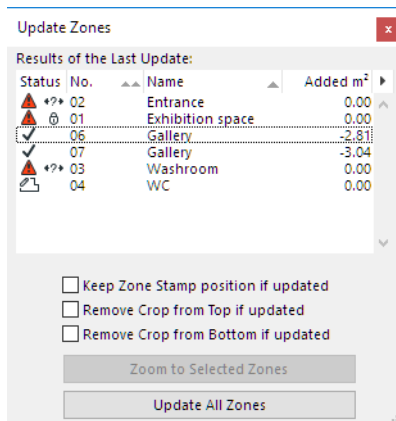


To adjust Zones to your updated design and recalculate the Zone area:

Go to **Design > Update Zones**. This dialog lists the Zones in your project. Selecting a Zone in the list selects it in the project, and vice versa.

- If any Zone is selected, click **Update Selected Zones**.
- If none is selected, click **Update All Zones**.

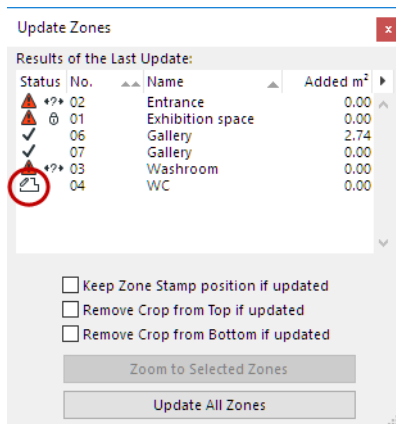
Successfully updated Zones are shown with a checkmark, with the added (or subtracted) Zone area resulting from the update.



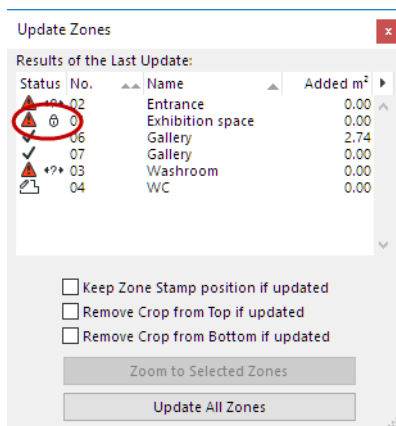
Possible Status Problems when Updating Zones

Icons in the Update Zones dialog box alert you to any problems:

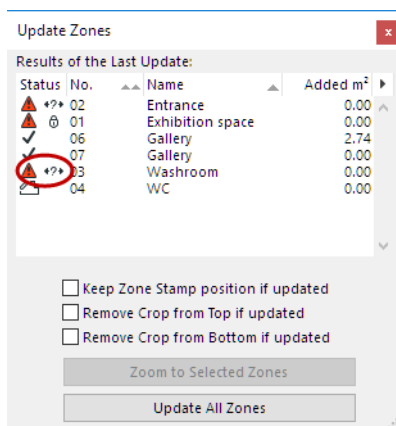
- The Zone was created manually and cannot be automatically updated.



- The layer on which the Zone resides is locked. Unlock the layer in the **Layer Settings** dialog box and repeat the update process.



- As a result of the changed model (e.g. moved Wall), the automatically created Zone is no longer surrounded by a closed polygon, and can't be updated.



Check the Zone by selecting it in the list, and clicking the **Zoom to Selected Zones** button. Fix the problem and repeat the update process.

Curtain Wall

Create a Curtain Wall in the Floor Plan, Section/Elevation/IE, or 3D window.

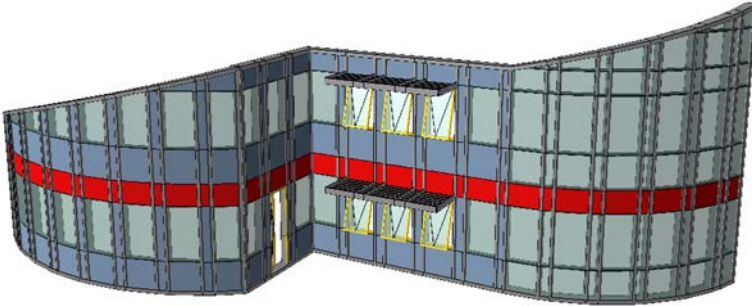
Related Topics:

- [Working with Curtain Walls: Overview](#)
- [Curtain Wall Components](#)
- [Curtain Wall Frames](#)
- [Curtain Wall Frame Settings](#)
- [Curtain Wall Panels](#)
- [Place Door- or Window-Type Panel in Curtain Wall](#)
- [Curtain Wall Panel Settings](#)
- [Curtain Wall Scheme Settings](#)
- [Set up Grid Pattern in Scheme Settings](#)
- [Create Grid Pattern Graphically](#)
- [Curtain Wall Input Methods](#)
- [Model View Options for Curtain Walls](#)
- [Curtain Wall Edit Mode](#)
- [Edit Grid and Grid Lines in Edit Mode](#)
- [Graphical Editing of Curtain Wall](#)
- [Add Additional Curtain Wall Boundary](#)
- [Curtain Wall Junctions](#)
- [Curtain Wall Accessories](#)
- [Curtain Wall System Settings](#)
- [Curtain Walls and Other Elements](#)

Working with Curtain Walls: Overview

The Curtain Wall is a single element consisting of Frame, Panel, Accessory and Junction components: these are the Members of the Curtain Wall, arranged on a Base Surface according to a pre-defined and editable Scheme.

The Base Surface can be planar or spherical; other curves are not supported.



To control the whole hierarchy at once, use Curtain Wall System Settings to define the overall scheme and configure the Curtain Wall members.

Creating a Curtain Wall involves a simple graphical input. Activate the Curtain Wall tool, choose a geometry method, and then draw a reference line, polyline, contour or arc.

If you use the **Boundary method**, then your input defines the entire geometry of the Curtain Wall.

If you **draw a Reference Line/Arc/Polyline**, then the Curtain Wall will be extruded from this line/polyline/arc.

[For details on Curtain Wall geometry methods in different windows, see *Curtain Wall Input Methods*.](#)

Curtain Walls consist of a set of Members (such as Frames, Panels and Junctions). Each Member's parameters are defined on its respective page in Curtain Wall Settings.

[See *Curtain Wall Components*.](#)

You can edit the Curtain Wall at two levels:

System-Level Editing

Select the Curtain Wall; open Curtain Wall Settings; and edit any of its System-level settings again.

Click **OK** to redefine the Curtain Wall according to the modified settings.

[See *Curtain Wall System Settings*.](#)

Edit Individual Curtain Wall Members

Select the Curtain Wall; go into **Edit** mode.

[See *Curtain Wall Edit Mode*.](#)

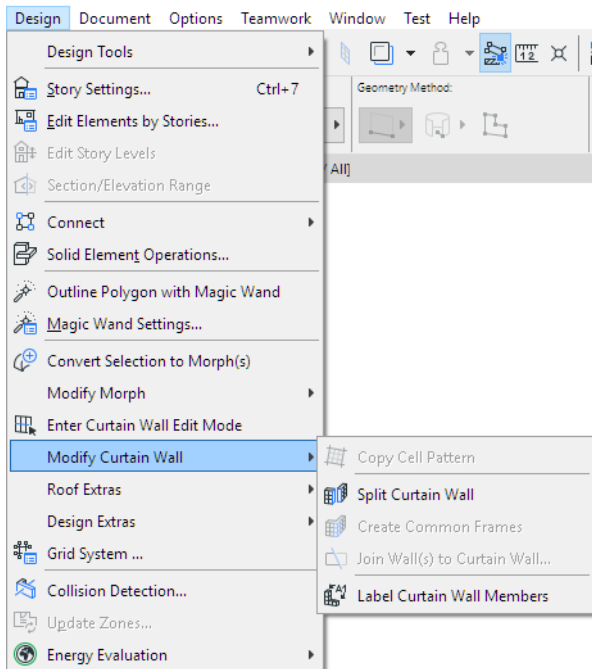
In Edit mode, you cannot place additional Curtain Walls, but you can edit any members of the selected Curtain Wall or add additional members to it. In Edit mode, each member (Scheme, Frame, Panel, Accessory, Junction) has its own Dialog box and its own Tool, enabling you to change settings and place new Members in this Curtain Wall.

To customize the settings of any member of the Curtain Wall, select it and use the dialog box to change its settings. For example, select a panel and change its surfaces. Changing the settings of a selected member will transform that member into a "Custom" member. You can have any number of different customized members. Once a member has Custom settings, the System-level settings no longer apply to it.

In Edit mode, you can create additional members for your Curtain Wall, using the Scheme, Frame, Panel, Junction and Accessory Tools. When using any of these tools to create a new member, you have the choice of using the System-level default settings, or else defining custom settings.

Additional Curtain Wall Commands

The **Design > Modify Curtain Wall** menu contains a collection of commands which are specific to Curtain Wall functions. These functions are described elsewhere in this documentation.



Curtain Wall Components

Reference Line

The **Reference Line** is the original input line/polyline/arc you draw in the window. If you use a Boundary method to create a Curtain Wall, the first drawn segment of the Boundary is the Reference Line.

You can Flip the Curtain Wall orientation relative to the Reference Line, using the **Flip** command in the Info Box.

The Reference Line is part of the Curtain Wall Scheme (together with the Scheme Grid pattern, the Boundary, and the Base Surface.)

Later, you can edit the Reference Line of a placed Curtain Wall.

[See *Edit Curtain Wall Reference Line.*](#)

Reference Surface

The Reference Surface is an imaginary plane or arced surface automatically created by Archicad by extruding the input line (polyline, arc) you have drawn. Often, you will place your input line along an existing Archicad element (e.g. slab); the Reference Surface is extruded from this input line.

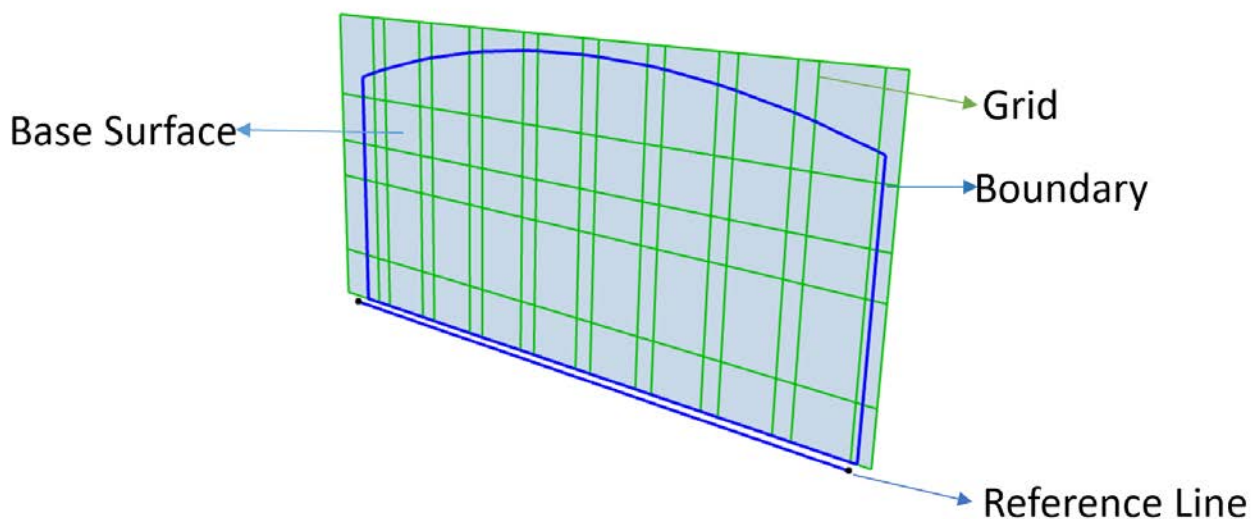
Base Surface

The **Base Surface** defines the shape and orientation of the Curtain Wall. It is the surface in which the physical members of the Curtain Wall (Frame, Panels) lie. The Base Surface may coincide with the Reference Surface; more likely, you will offset it. When you move or rotate the Curtain Wall as a whole, it is the Base Surface which you are moving; the Reference Line and Reference Surface move along with the Base Surface.

The Base Surface of Curtain Walls is an infinite plane - except if it is based on a curved shape. (The Base Surface of a curved Curtain Wall is cylindrical, and is infinite only in the direction of the cylinder's axis.)

A Curtain Wall's infinite Base Surface gives you great freedom in connecting the Curtain Wall to other elements, and in adding multiple, bounded Curtain Wall portions to any Curtain Wall.

[See *Add Additional Curtain Wall Boundary.*](#)



Boundary

The Boundary is a simple polygon on the Base Surface, which represents the physical limits of the Curtain Wall.

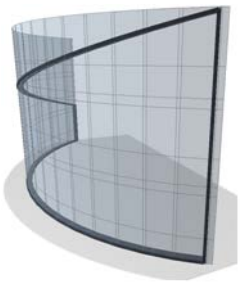
The Boundary is part of the Curtain Wall Scheme (together with the Scheme Grid, the Base Surface and the Reference Line).

Depending on the geometry method used to create the Curtain Wall, the Boundary is either drawn by you or created by Archicad in accordance with input parameters.

You can draw additional boundaries on an existing Curtain Wall - this gives you freedom to separate the Curtain Wall into parts, while still keeping it as a single element: all parts of the Curtain Wall use the identical Scheme.

See [Add Additional Curtain Wall Boundary](#).

The Curtain Wall's Boundary Frame always coincides with this Boundary polygon.



You can edit the Boundary of the Curtain Wall like any other polygon.

Scheme

The Scheme contains the essential geometry definitions (Base Surface, Scheme Grid, Reference Line) of the Curtain Wall.

Use Scheme Settings to arrange the grid line pattern in two directions ("row" and "column"). The Scheme also defines the classes to which frames and panels within this basic pattern belong.

See [Curtain Wall Scheme Settings](#).

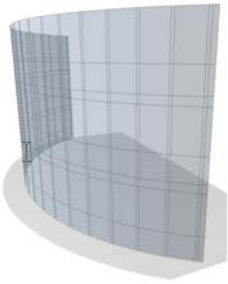
All other members of the Curtain Wall (Frames, Panels, Accessories, Junctions) are attached to the Scheme.

Scheme Grid

The Base Surface is crisscrossed by a **Scheme Grid** (grid lines in rows and columns, defined as part of the Curtain Wall Scheme.)

The Scheme Grid defines the default location and arrangement of Frames and Panels in the Curtain Wall. You can edit the Scheme Grid (delete, add grid lines, move or rotate it). Changes to Scheme Grid geometry will change the geometry of its Frames and Panels accordingly.

See [Edit Grid and Grid Lines in Edit Mode](#).



Frame

Each Frame class (e.g. Boundary, Mullion, Transom, or any other user-defined class) has unique settings. Frames are attached to the Scheme Grid when you create the Curtain Wall, so if you move or delete a Grid Line, you move or delete the Frame. (In Curtain Wall Edit mode, you can place additional Frames independent of the Scheme Grid.)

[See *Curtain Wall Frames*.](#)



Panel

Panels are the flat surfaces - generally glazed - of the Curtain Wall. Curtain Walls have two predefined classes: Main and Distinct, plus any user-defined Panel classes. Panels are always placed between Frames.

[See *Curtain Wall Panels*.](#)



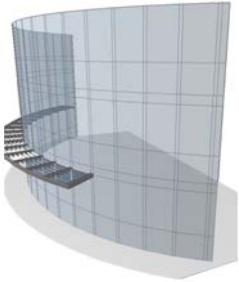
Junction

A Curtain Wall Junction is an optional structure that serves to connect panels to the Frame. In Archicad, a Junction is a GDL object, and can be placed either one by one, or automatically at every intersection.

[See *Curtain Wall Junctions*.](#)

Accessories

Accessories are optional, non-load-bearing members - such as a sun shade - attached to the Curtain Wall. These are GDL objects placed at any Frame of an existing Curtain Wall.



[See *Curtain Wall Accessories*.](#)

Curtain Wall Frames

Frames are placed along the Scheme Grid when you create the Curtain Wall.

The Curtain Wall's pattern of Frames and Panels can be defined in Scheme Settings. ([See Set up Grid Pattern in Scheme Settings.](#))

- A Frame must be joined to another Frame at both ends (it cannot “float” independently).
- A Frame with adjoining Panels always divides two separate Panels (it cannot bisect a single Panel).
- In a multi-segment (Chained) Curtain Wall, a Frame is always placed where two adjacent Curtain Wall segments meet. Such a Frame cannot be deleted, although you can make it invisible.
- A Boundary Frame cannot be renamed or removed. The edges of a Curtain Wall always use a Boundary Frame.

Frame Classes

Edit Curtain Wall Frames

Add New Frame to Curtain Wall

Copy Cell Pattern in Curtain Wall

Curtain Wall Frame Intersections

Curtain Wall Corner Frames

Corner Frames: Complex Profiles

Create Common Frame for Adjoining Curtain Walls

Curtain Wall Frame Settings

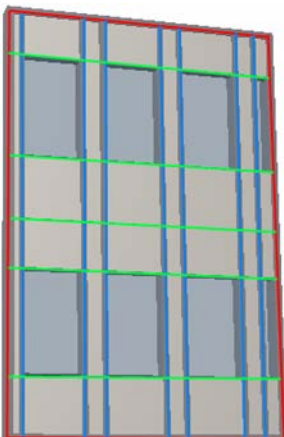
Frame Classes

By default, a Curtain Wall has four classes of Frames: Corner, Boundary, Mullion and Transom.

You can also create other Frame classes: [see Curtain Wall Frame Settings.](#)

- **Boundary Frames** are placed along the Curtain Wall Boundary.
- **Mullion and Transom Frames** are placed along either the Columns or Rows of the grid lines
- **Division Frames** are invisible, but they serve to divide panels (they can replace a deleted Frame). [See “Division” Frame Class.](#)

In this image, the Frames in the Boundary class are red; the Frames in the Mullion class (coinciding with the grid columns) are blue; and the Frames in the Transom class (coinciding with the grid rows) are green.



Edit Curtain Wall Frames

1. Select the Curtain Wall.
2. Enter Edit mode.
3. Select a Frame.
4. Do any of the following:
 - Open Frame Selection Settings and change them
 - Stretch, Drag, Mirror, Rotate or Multiply the Frame
 - Delete the Frame

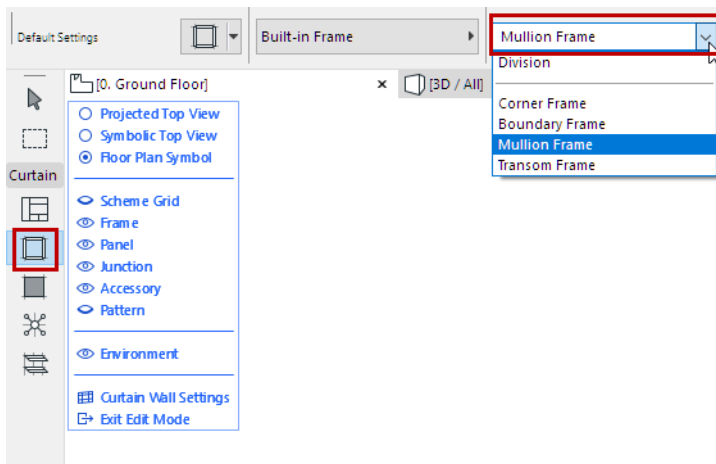
Notes:

- Deleting a Frame is only possible if the two panels on either side of the Frame are both in the same plane. On a non-planar Curtain Wall surface, you must first delete the grid line which runs between them. The Frame is deleted along with the grid line.
- If you move or delete a grid line, you move or delete the Frame.

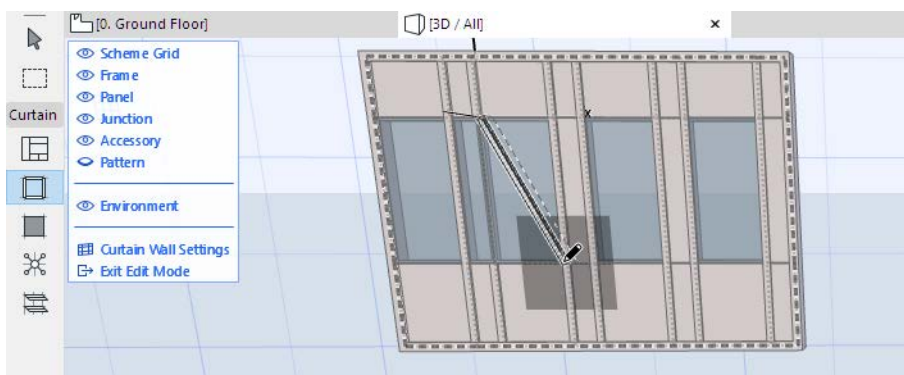
See also [Multi-selection of Frames and Panels](#).

Add New Frame to Curtain Wall

1. To add an additional Frame, enter Curtain Wall Edit mode.
2. Activate the Frame Tool.
3. Use the Info Box to define the Frame type and the Frame class whose parameters it should use (e.g. Mullion).



4. Draw a new Frame on the Curtain Wall in Edit mode.

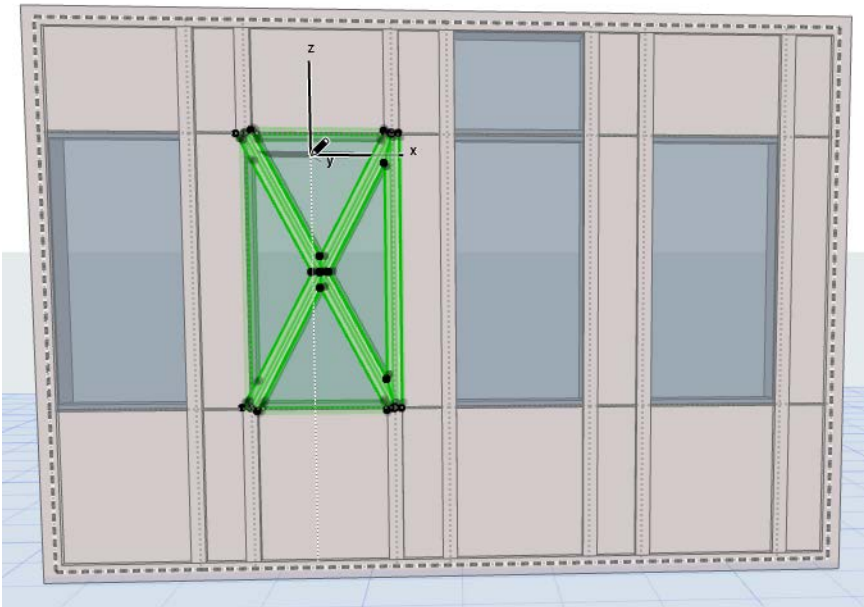


- If you draw the Frame on a Grid line, then the Frame will be attached to the grid line. Moving the Scheme Grid will move the Frame.
- If you draw a new Frame that crosses an existing grid line, the Frame will be cut into multiple Frames automatically where it crosses the grid line.

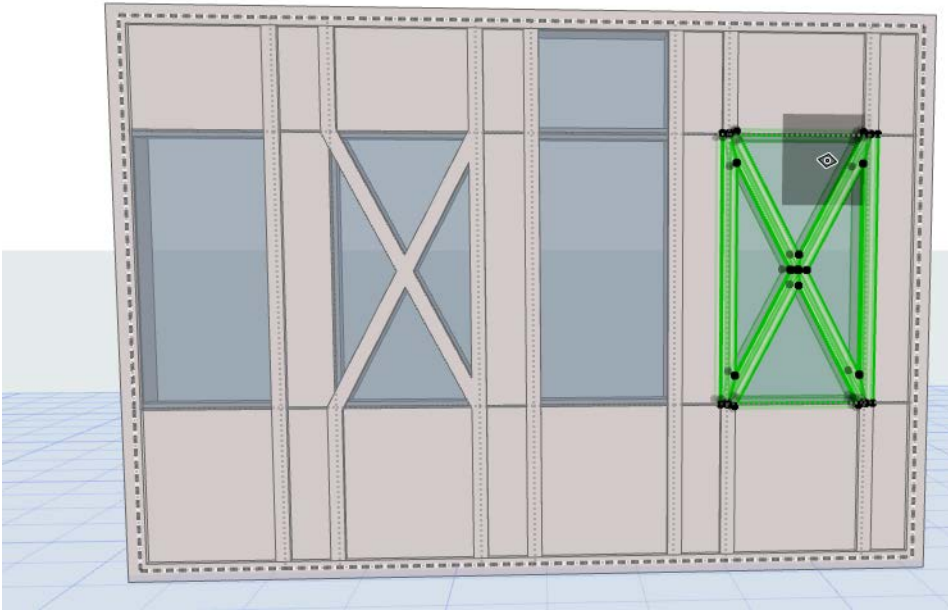
Copy Cell Pattern in Curtain Wall

To copy the Frame pattern of any cell in the grid to another cell:

1. Make sure you are in Curtain Wall Edit mode, with the Frames visible.
2. Use **Design > Modify Curtain Wall > Copy Cell Pattern**.
3. Click in the cell whose Frame pattern you wish to copy.



- Click in the cell to which to copy the Frame pattern. The Frame pattern is duplicated.



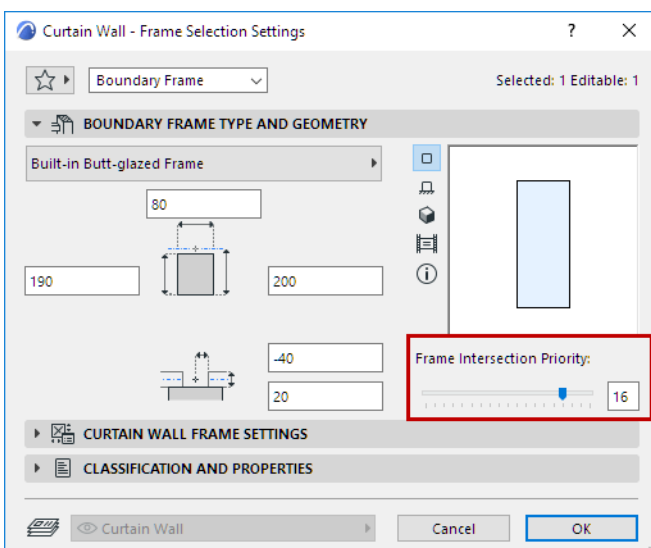
Curtain Wall Frame Intersections

When two or more Frames meet in a Curtain Wall, the Frame with the higher intersection priority will cut the other one.

Note: While a Curtain Wall Frame does have a Building Material, the Building Material’s intersection priority is ignored in this case.

Frame Priorities are defined for each class on the Frames page.

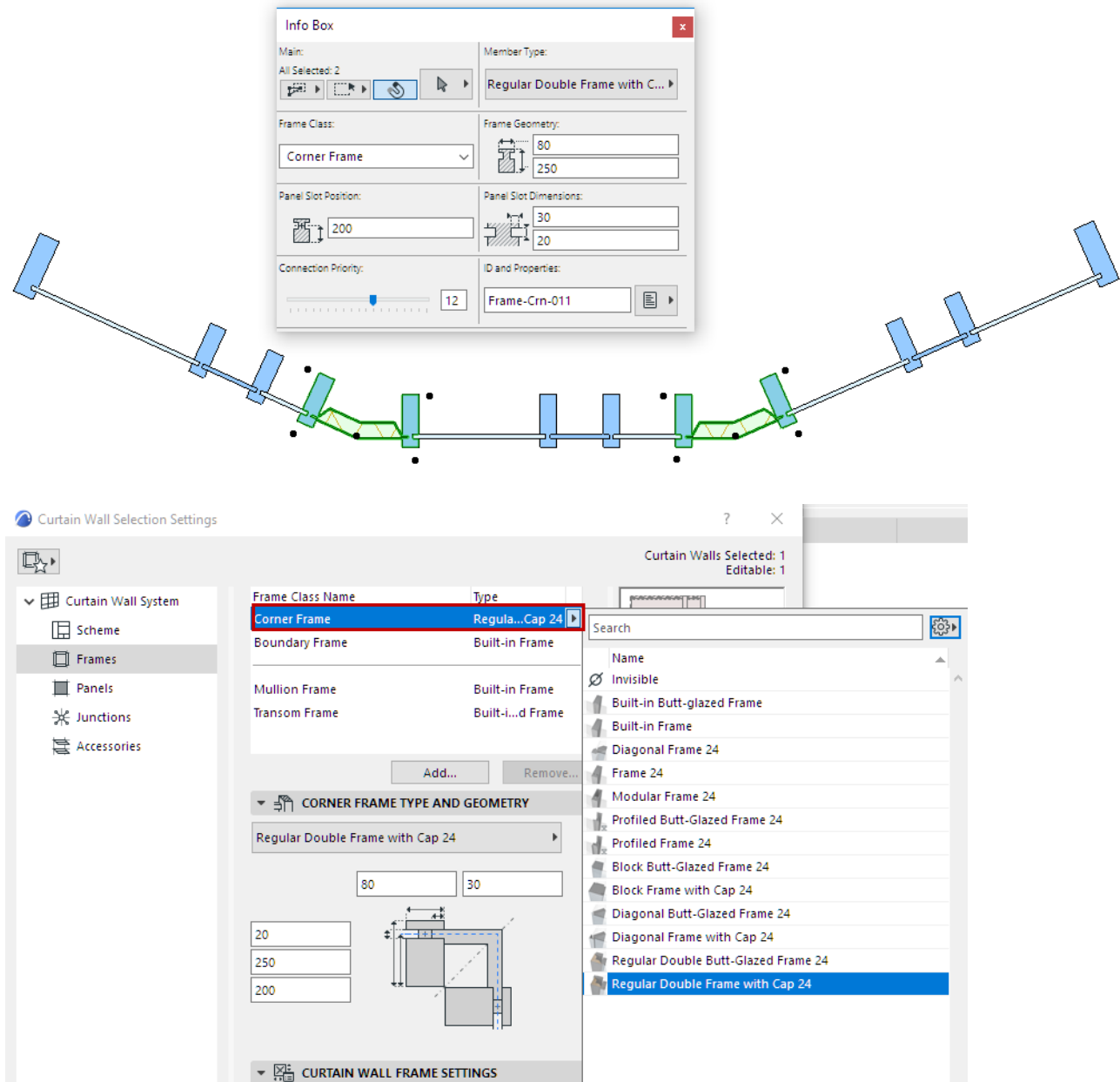
However, you can change the intersection priority of any single selected Frame. In Curtain Wall Edit mode, select the Frame whose intersection priority you want to change. Go to Frame Selection Settings and adjust the slider to the desired level. Changing the Frame’s intersection level - as when changing any other Frame parameter - will result in a Custom Frame.



Curtain Wall Corner Frames

Corner Frames are a separate Frame Class. You cannot rename or remove the Corner Frame class.

When you place a Curtain Wall with multiple segments, the Frame used by the Corner Frame class is placed at the place where each segment joins the next one. (Corner Frames are not placed at curved segments.)

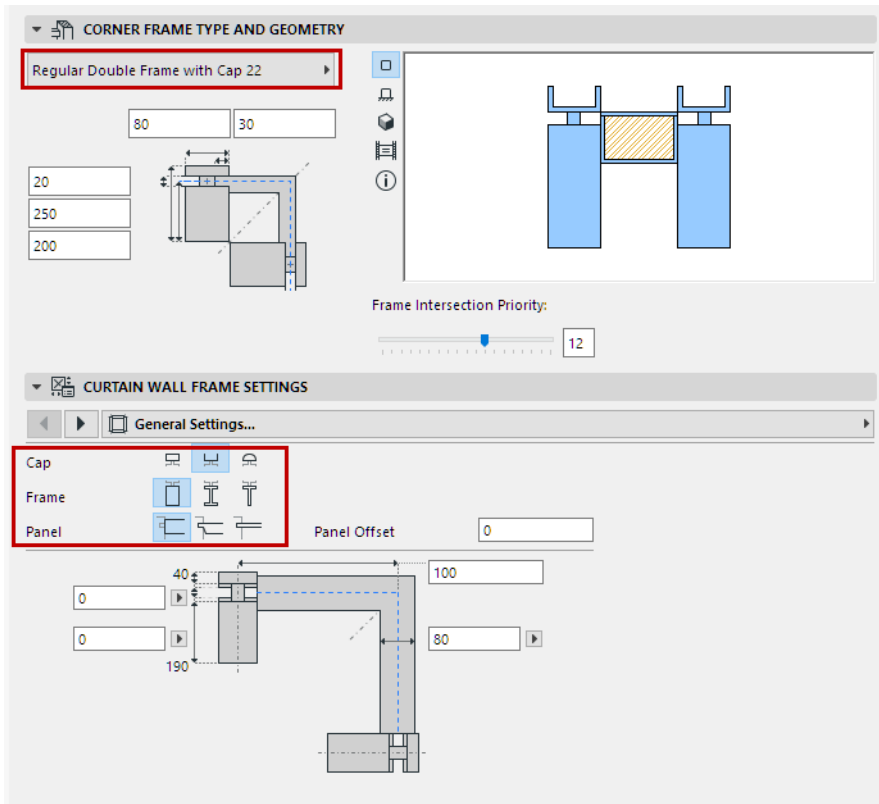


You can technically choose any Curtain Wall Frame as your default Corner Frame, but the Archicad library includes specific Corner Frame objects, such as:

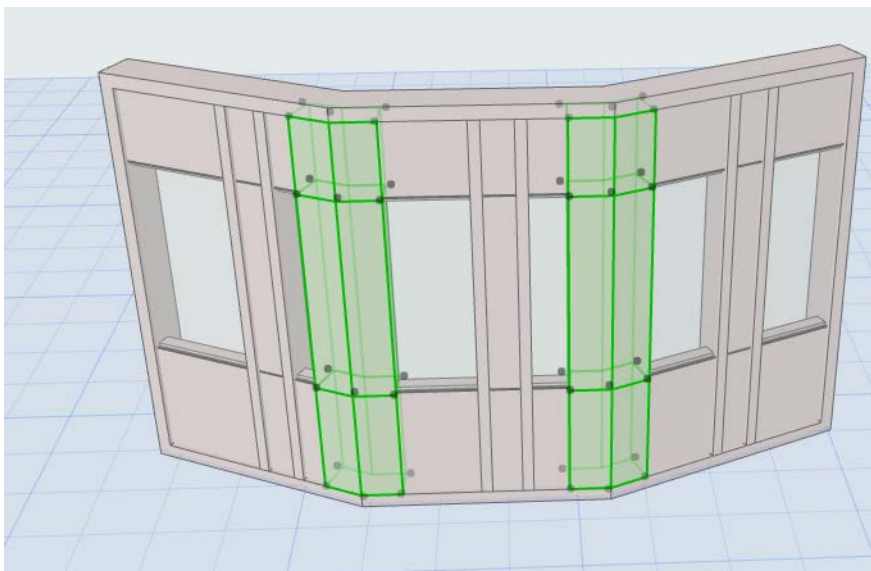
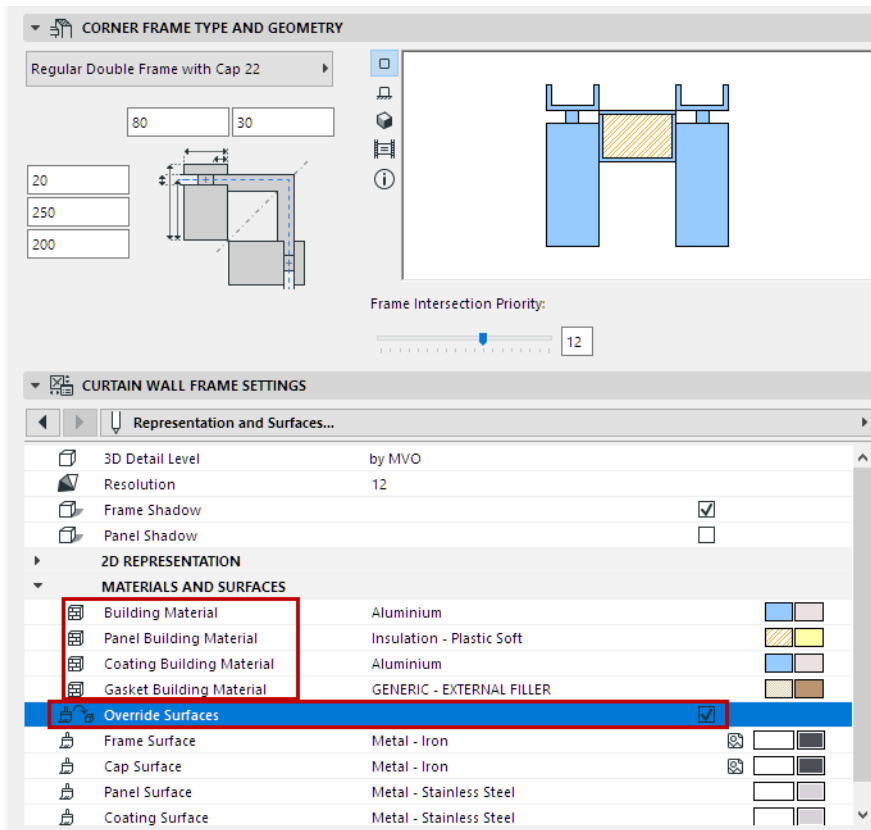
- Regular Double Frame with Cap
- Regular Double Butt-Glazed Frame
- Block Frame with Cap
- Block Butt-Glazed Frame
- Diagonal Butt-Glazed Frame

- Diagonal Frame with Cap

Depending on the selected Corner Frame type, set options and variations in Curtain Frame Settings:



Each Corner Frame uses separate building materials for its components: Frame, gasket and panel, where applicable. You can override surfaces for each as needed.



Corner Frames: Complex Profiles

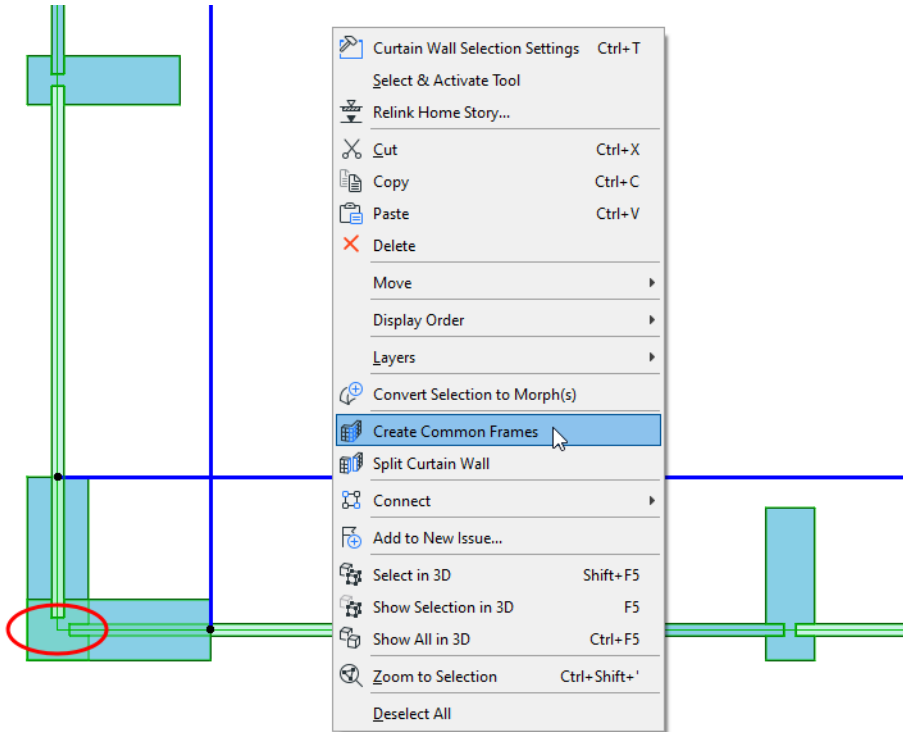
By default, Archicad includes Profile Corner Frames. You can create custom Profile Frames using the usual methods in Profile Editor. Note that the predefined Profiled Frames include a user origin which ensures the expected behavior for these Frames if the Curtain Wall geometry changes.

[See Profile Origin in Profile Editor Window](#)

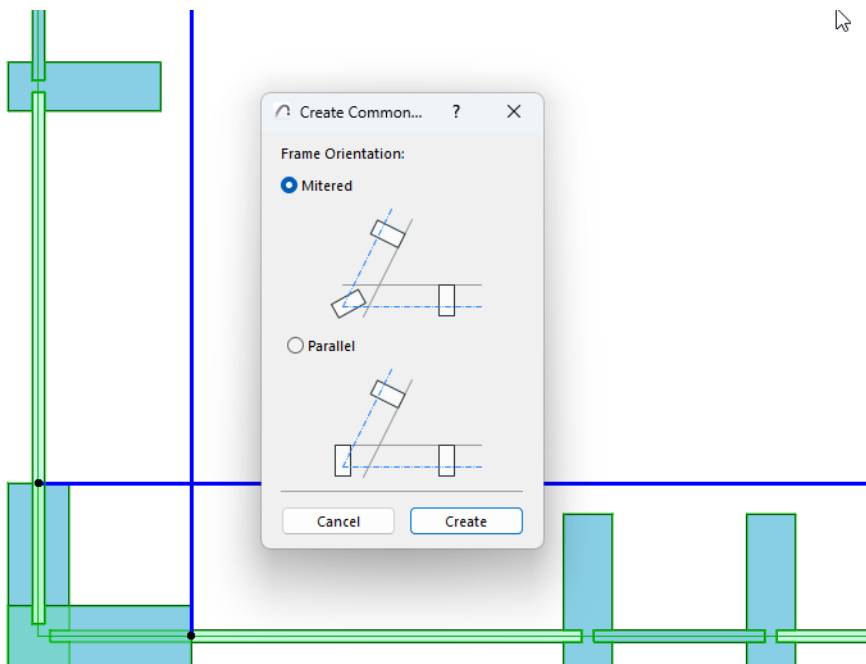
Create Common Frame for Adjoining Curtain Walls

If two Curtain Wall Base Surfaces join or overlap, use this command to clean up their common corners.

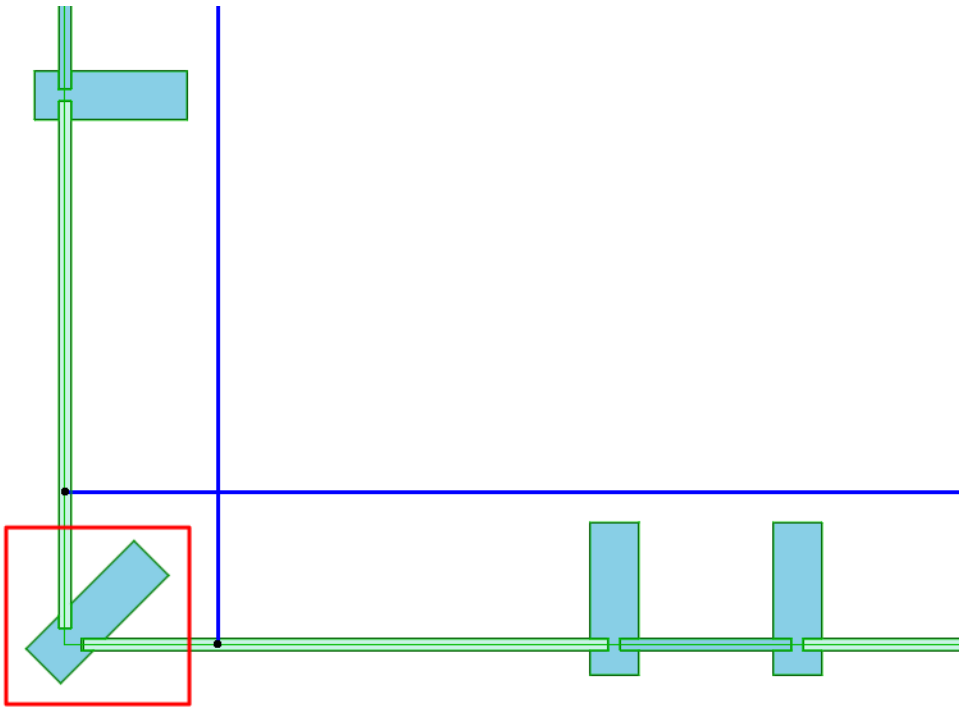
1. Select both Curtain Walls.
2. Use **Design > Modify Curtain Wall > Create Common Frames** (or the same command from the context menu).



3. Choose the preferred orientation for the common frame (mitered or parallel to an existing frame).



4. As a result, the first joining Frame selected will be transformed into the default Corner Frame, and the other one will be transformed into an Invisible Frame.

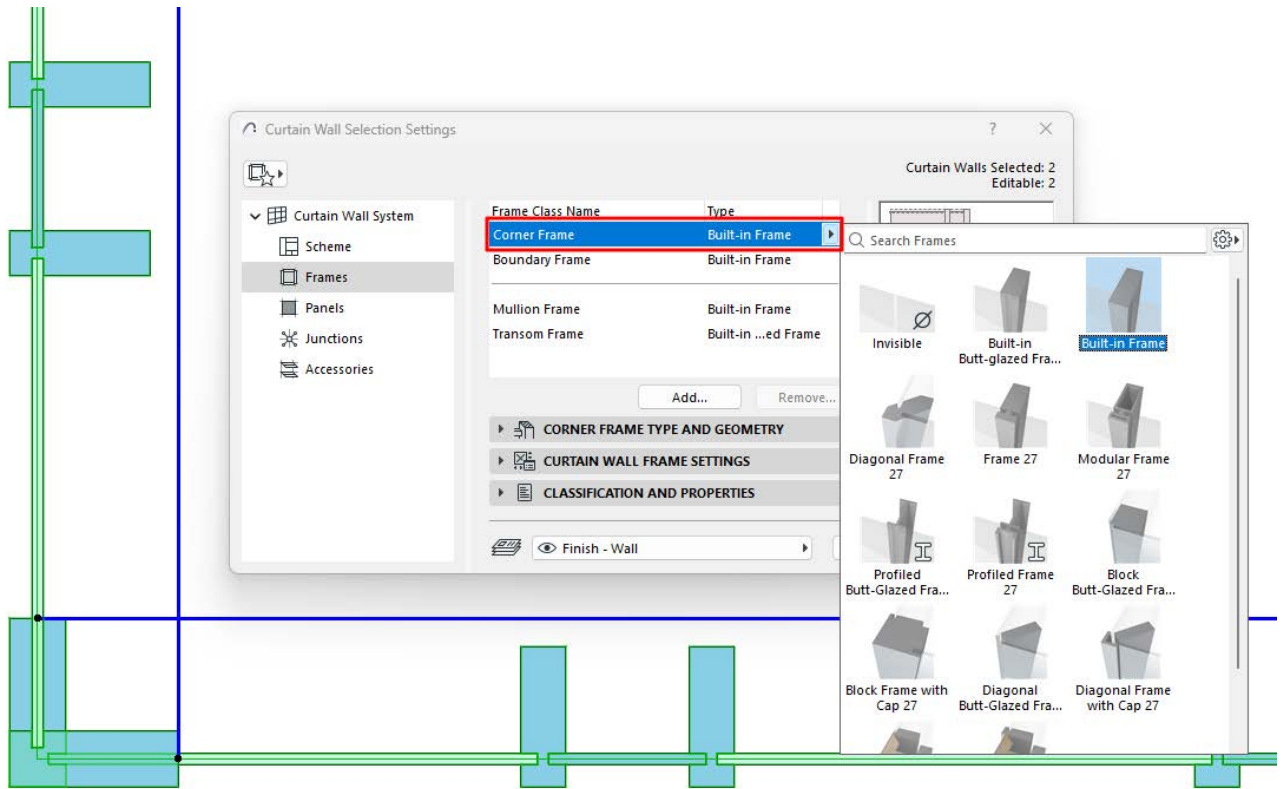


Notes on Common Frames

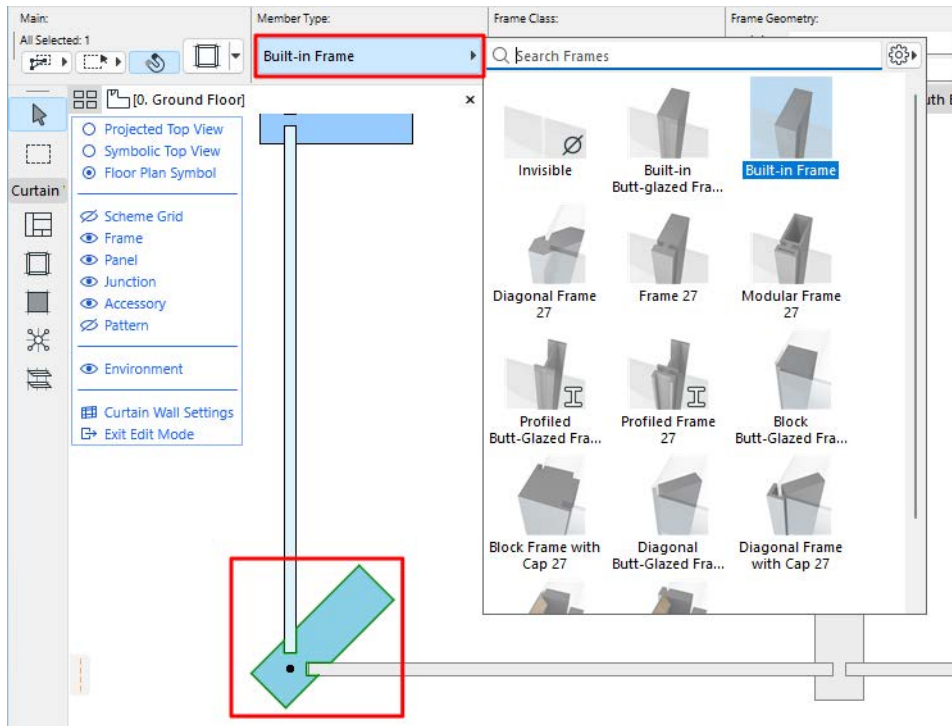
- This command does not create a connection between the Curtain Walls (they can be moved or edited separately). If you need a *unified* element consisting of two adjoining Curtain Walls, it is better to use the Chained geometry method and create these elements in one step to handle them as a single building part.
- Note that common frames can only be created at the common nodes of Curtain Wall *Base Surfaces*, not the common nodes of their Reference Lines, which are usually offset from Base Surfaces. In the images above, the Reference Lines are blue and the Base Surfaces are green.

[See Curtain Wall Components.](#)

- This command places the default Corner Frame defined for the Curtain Walls. Set this default Corner Frame component before using the command.



- You can change a Corner Frame component at any time using Edit mode.



Curtain Wall Frame Settings

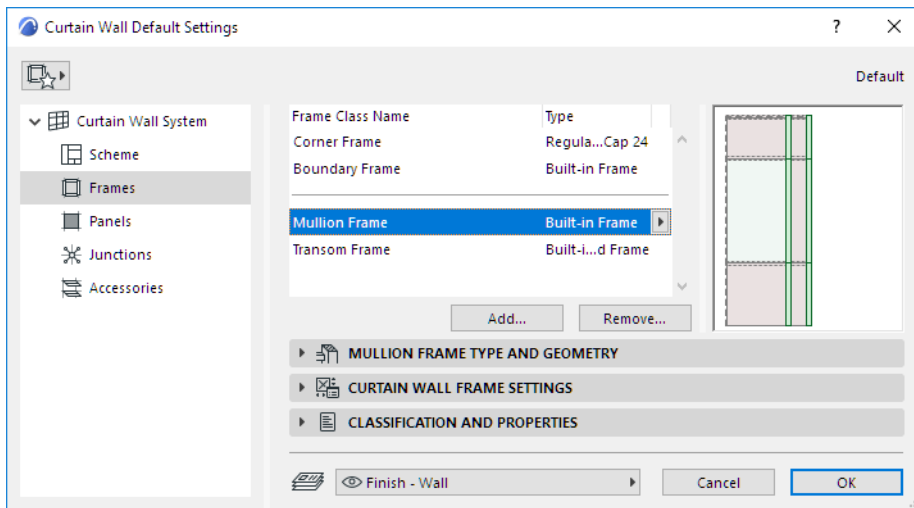
The Frames page of Curtain Wall Settings contain the settings for each Frame Class.

Frame Tool Settings (in Edit mode) contains nearly identical controls.

List of Frame Classes

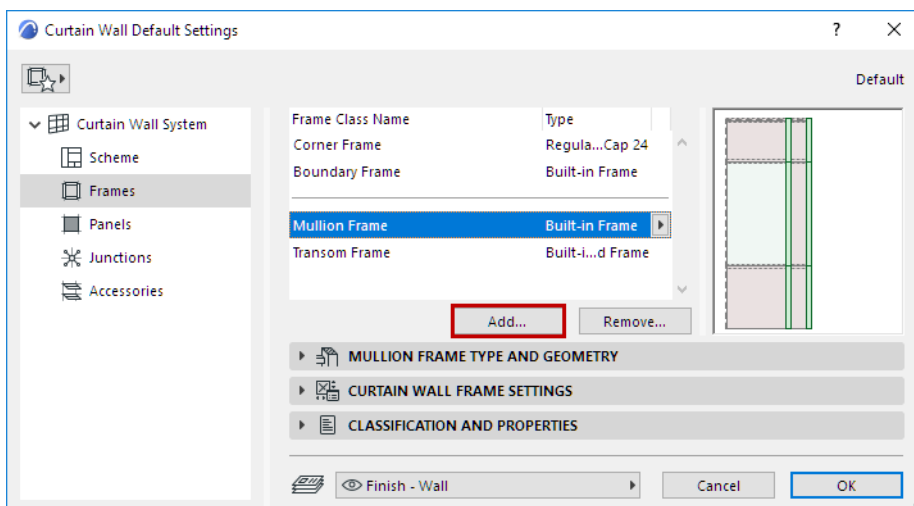
The top of the dialog lists the available Frame classes, by name and type. By default, Frame classes include: Corner, Boundary, Mullion and Transom.

The selected class is highlighted in the preview window. The preview window also displays the Frame's surface color.

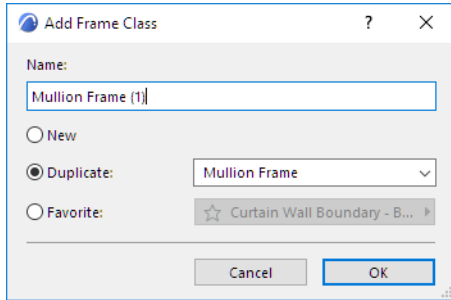


Add Frame Class

1. Click the **Add** button.



2. In the appearing dialog:

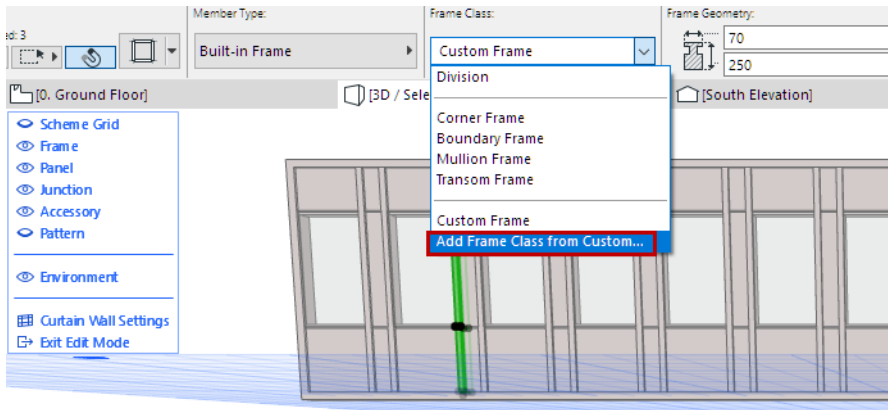


- **New:** Creates a new Frame class based on the default Frame (e.g. Built-in)
- **Duplicate:** Duplicates the currently selected Frame class under a new name
- **Favorite:** Click the pop-up to choose a Frame Favorite: the new Frame class will use these Favorite definitions.

Add Frame Class Based on Customized Frame (Edit Mode)

Another way to create a new Frame Class is to customize a single Frame in Edit Mode, then save its settings as a new Frame class.

1. Select the Curtain Wall and enter Curtain Wall Edit mode.
2. Select a Frame to customize it. Its class is now "Custom."
3. From the Frame Class pop-up in the Info Box, click **Add Frame Class from Custom...**



4. Use the upcoming dialog to create a new Frame class.

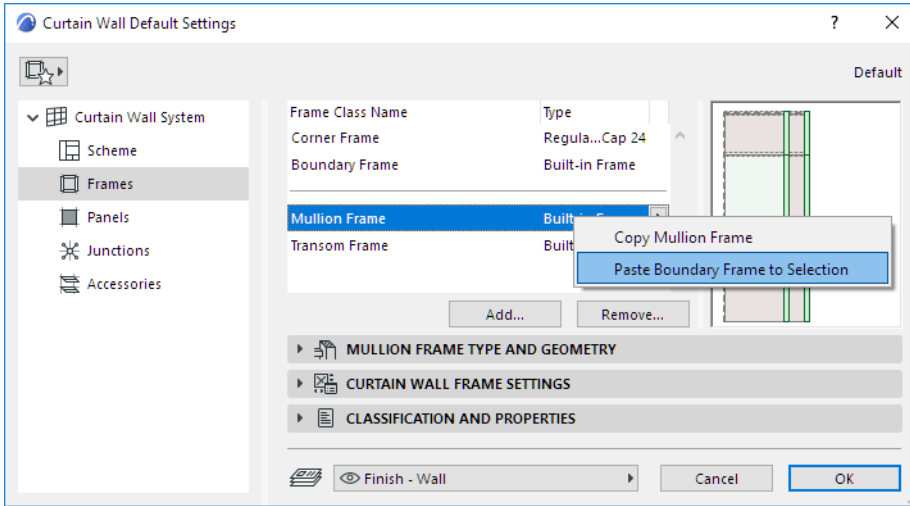
Rename Frame Class

Double-click the Class in the list, and rename as needed.

Copy-Paste Frame Settings Between Classes

In Frame Settings, right-click the name of any Frame to copy its settings.

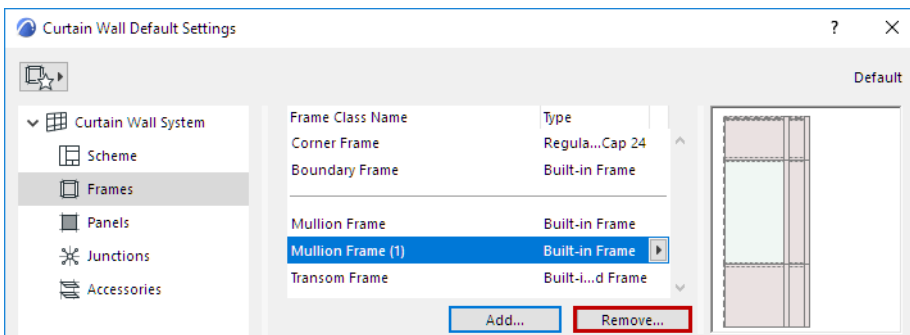
Right-click any other Frame to insert the copied settings for that Frame.



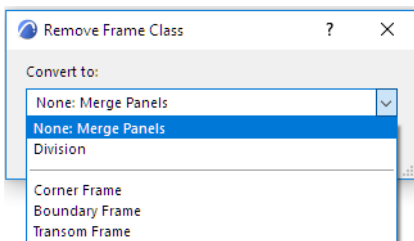
Remove Frame Class

1. Select one or more Frame Classes to remove.
2. Click **Remove**.

Note: The Boundary and Corner Frame classes cannot be removed.



3. In the appearing dialog:



- Choose how to convert the Frames in the class that is being removed:
 - **None: Merge Panels:** Frames will be deleted, and the adjoining panels merged together.
 - **Division:** Frames will be deleted, but the adjoining panels will remain divided.
 - **Convert to Another Frame Class:** Choose another, existing Frame Class.

“Division” Frame Class

Division Frames are a predefined class, with fixed settings. They function as an invisible Frame; the adjoining Panels remain divided. You cannot edit Division Frames on this page; this class cannot be deleted.

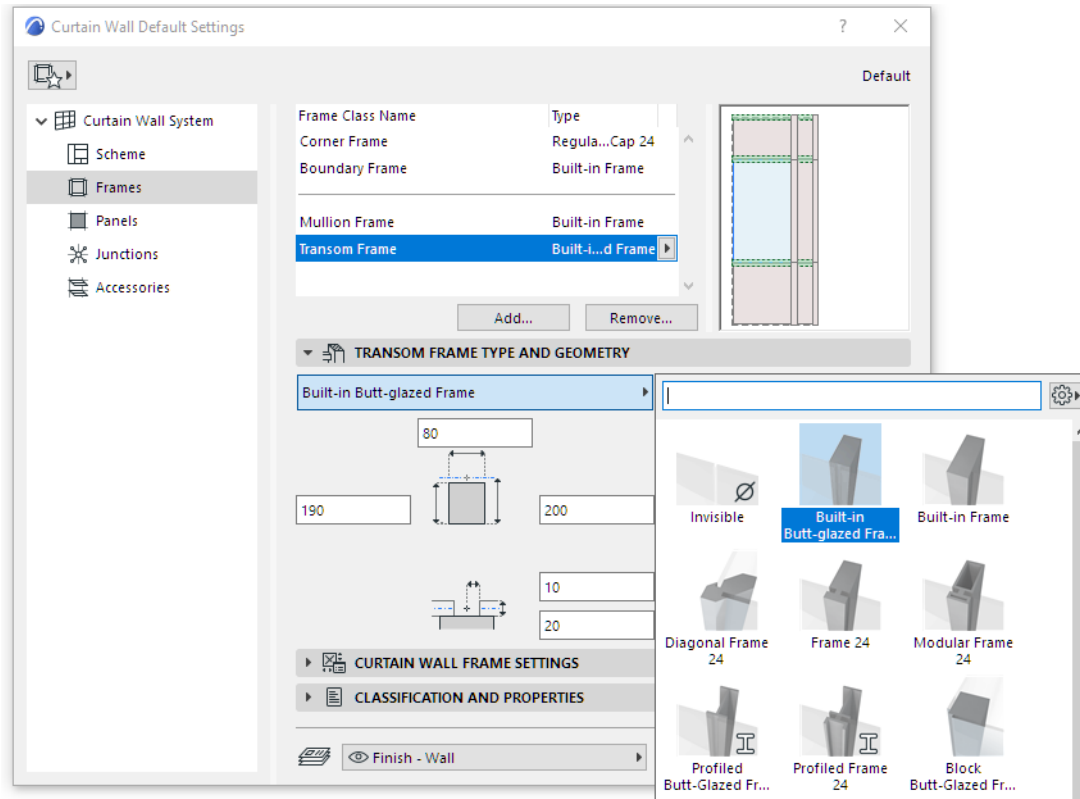
Division Frames are available

- as an optional replacement whenever you delete a Frame Class (replace the deleted Frames with a Division), and
- in Scheme Settings, as a Frame class option for a selected Frame in the Grid Pattern

Select Frame Type and Geometry

For the Class selected in the list at the top of the dialog: choose a Frame Type from the pop-up list, in the Frame Type and Geometry panel.

Then set its geometric parameters as applicable.



- **Invisible Frame:** This Frame exists as an object, but it is not visible.
 - Use this Frame if you want two neighboring panels to remain separate, but without displaying a Frame between them.
 - Set an optional gap between the two adjoining Frames.
 - Invisible Frames are also appropriate when you don't want to burden the model with the Frame details, or when other structures (e.g. junctions) are used to support the panel connections. A seamless panel surface can be achieved if Frames are deleted from grid lines.
- **Built-in Frame:** This is the most common type of Frame, with a rectangular profile. From the outside of the wall, a "cap" is visible. Assign its Building Material in Frame Settings (the panel below).
- **Butt-Glazed:** A rectangular profile, used for inner support of a butt-glazed Curtain Wall surface. The panels will connect to each other in the front of a butt-glazed Frame, so the Frame will not be visible on that side (unless the panels are transparent or a gap is specified.) Assign its Building Material in Frame Component Settings (the panel below).

- **Modular Frame:** Add an additional module to this Frame: a custom-created frame or cap profile object that you draw and save as a specialized component.

See [Example: Create Custom Frame or Cap Profile for Curtain Walls](#).

See also [Curtain Wall Corner Frames](#).

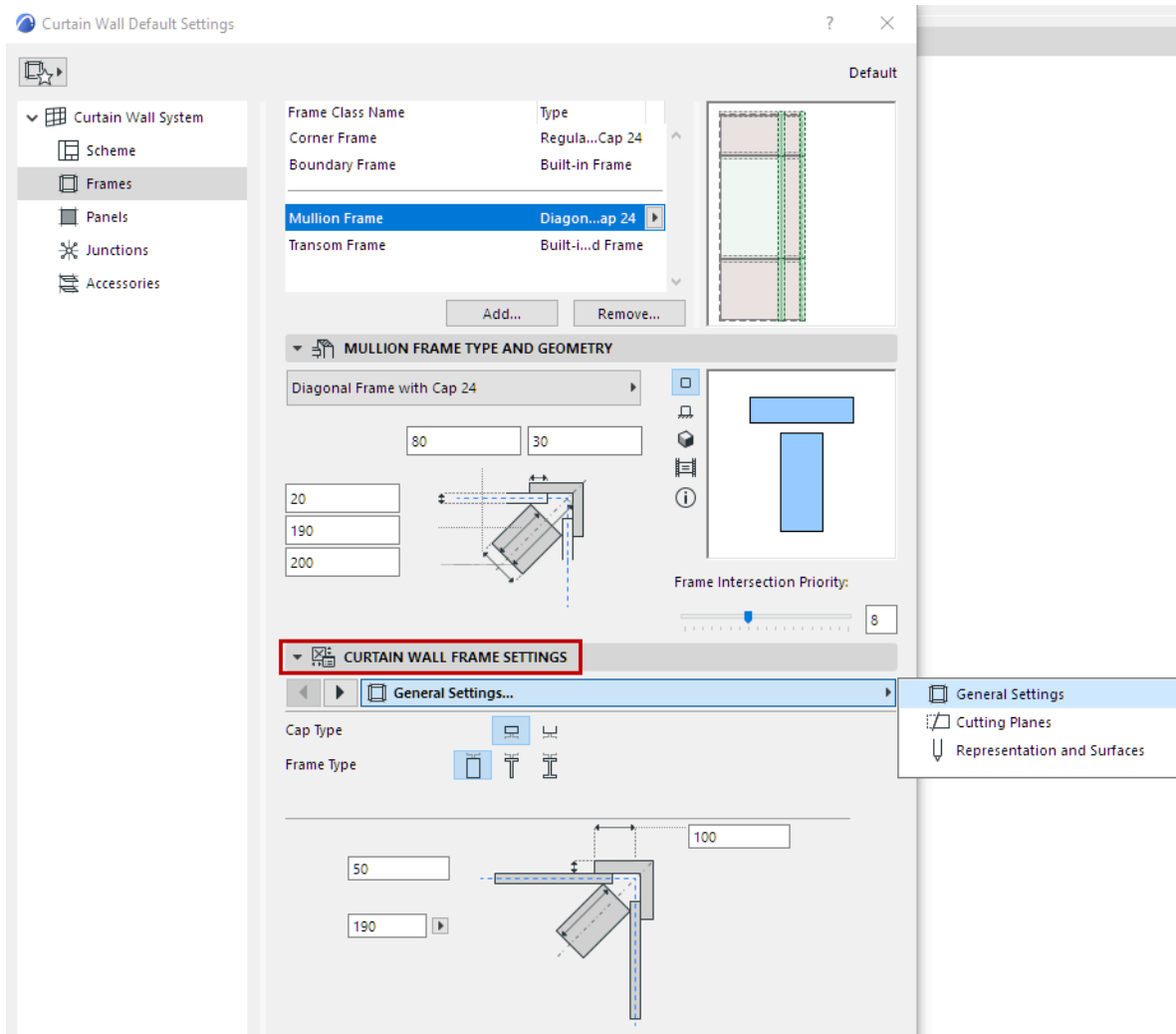
Frame Intersection Priority

See [Curtain Wall Frame Intersections](#).

Frame Component Settings

For the chosen Frame type, you can set various parameters, as usual for GDL-type components.

Note that you can override the Model View Options settings for its 2D and 3D Detail Levels, as applicable.



Curtain Wall Panels

Panels are the flat surfaces - generally glazed - of the Curtain Wall. Panels are automatically placed between Frames and onto the Base Surface.

A Curtain Wall has two predefined classes of panels, Main and Distinct. However, you can add more Panel classes as needed: [see *Curtain Wall Panel Settings*](#).

Panel Classes and their properties (type, surface, width) are defined in Curtain Wall Panel Settings.

The Curtain Wall's pattern of Frames and Panels can be defined in Scheme Settings. ([See *Set up Grid Pattern in Scheme Settings*](#).)

Most of your Curtain Wall panels will be "fixed" glazed surfaces. However, you can insert object-type Panels which serve as Door or Window openings.

When a **Panel** is selected in Edit mode, you can

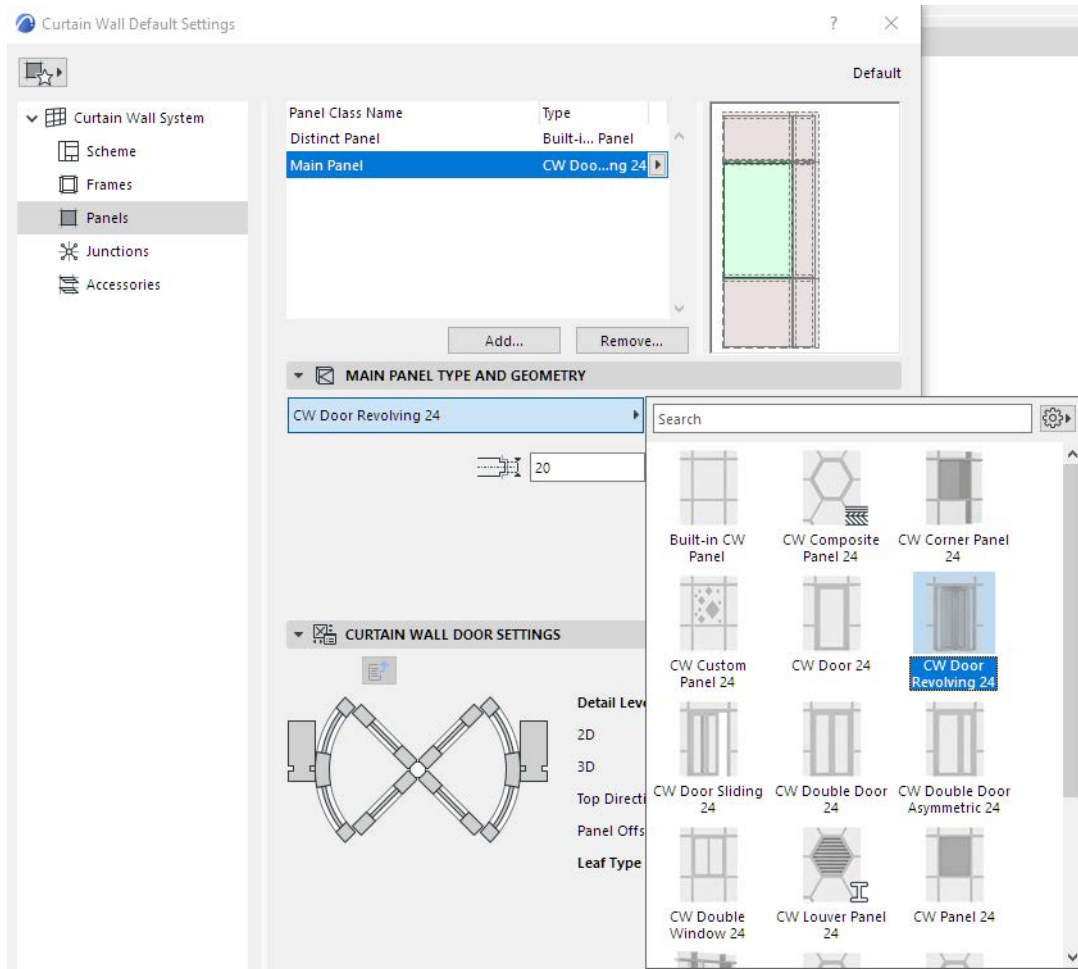
- Open Panel Selection Settings and change them
- Delete the Panel
- Edit its parameters to create a Custom Panel class
- Create a new, named Panel Class based on a customized Panel

Related Topics:

[Place Door- or Window-Type Panel in Curtain Wall](#)
[Curtain Wall Panel Settings](#)

Place Door- or Window-Type Panel in Curtain Wall

You can place a GDL-based panel which behaves like a window or a door. Some of these are predefined in Archicad and are listed in the Type pop-up list in the Panel Type and Geometry panel:



Even though these Curtain Wall Panels function as Doors or Windows, they do not use Door/Window Markers. However, you can use the Label tool to place a Curtain Wall Panel Label.

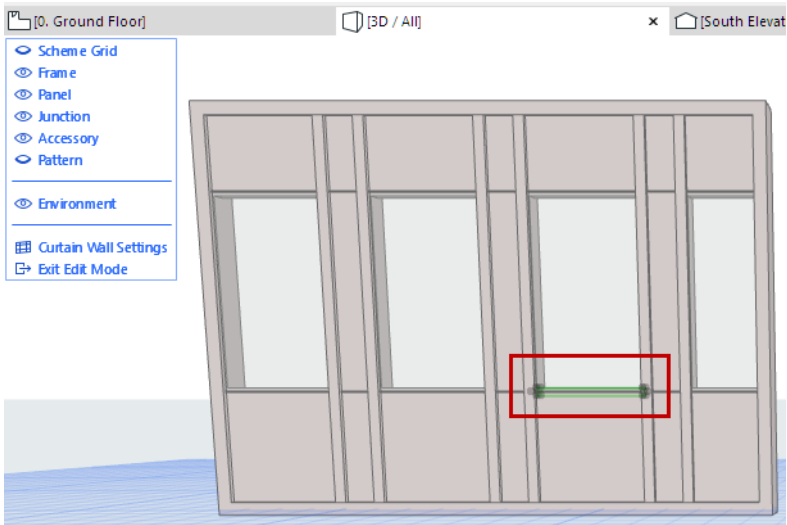
[See Labeling Curtain Walls.](#)

You can define opening-type Panels as part of the Curtain Wall's pattern, using Scheme Settings. ([See Set up Grid Pattern in Scheme Settings.](#))

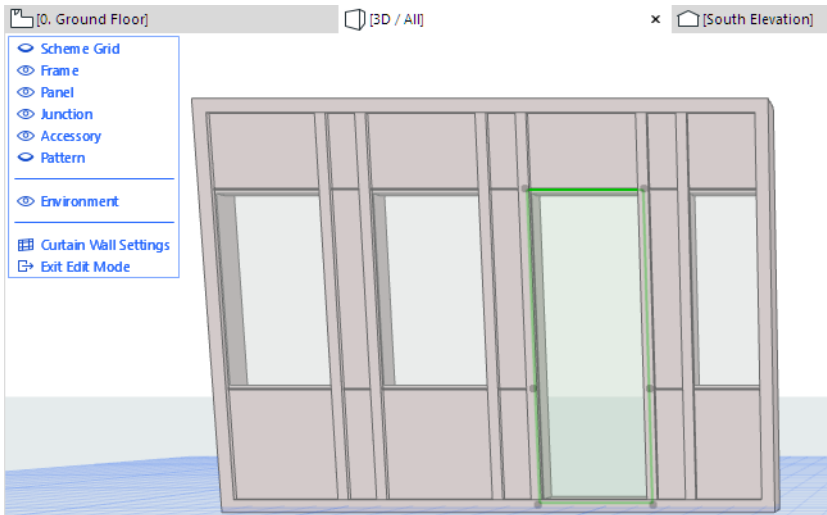
In an already placed Curtain Wall, use Edit Mode to replace regular panels with Door- or Window-Type panels, as described below.

To place a door-type panel into a Curtain Wall:

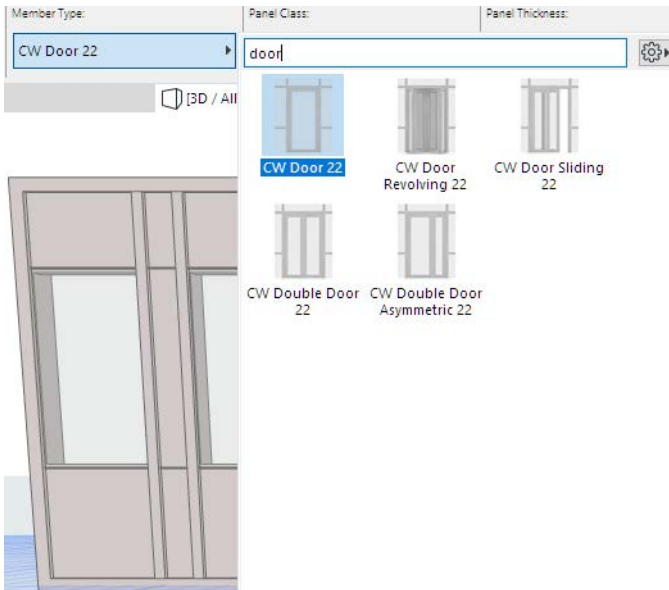
1. Select the Curtain Wall and go into Edit mode.
2. At the place where the Door should go, merge two existing panels by deleting the Frame between them:



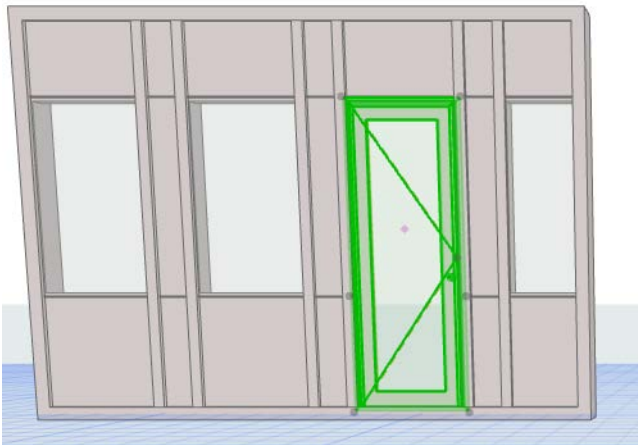
3. Select the newly merged panel.



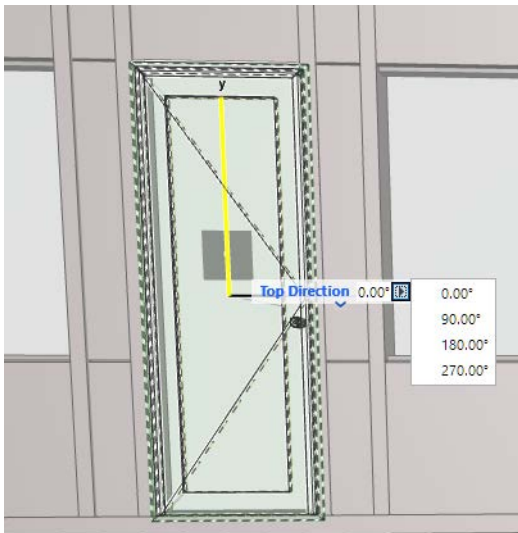
4. In the Info Box, from the Member Type pop-up, select the desired Door Type. (You can narrow the choices by name, by typing “Door” into the search field.)



The selected panel is replaced with a door-type panel, which you can open and close like other GDL doors.



Note that this door-type Panel object has an editable parameter called “Top Direction.” This lets you redefine the “top” of the door by 90 degree increments in case you later free-rotate the Curtain Wall and the “top” of the door changes its position.



Curtain Wall Panel Settings

The Panels page of Curtain Wall Settings contain the settings for each Panel Class.

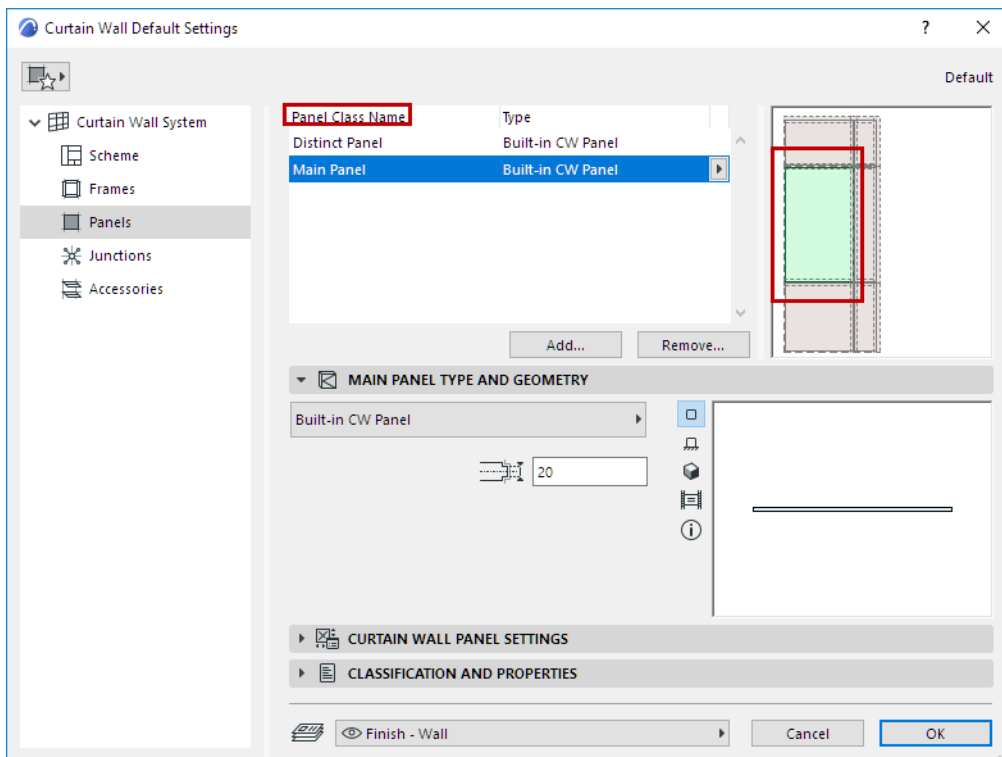
Panel Tool Settings (in Edit mode) contains nearly identical controls.

List of Panel Classes

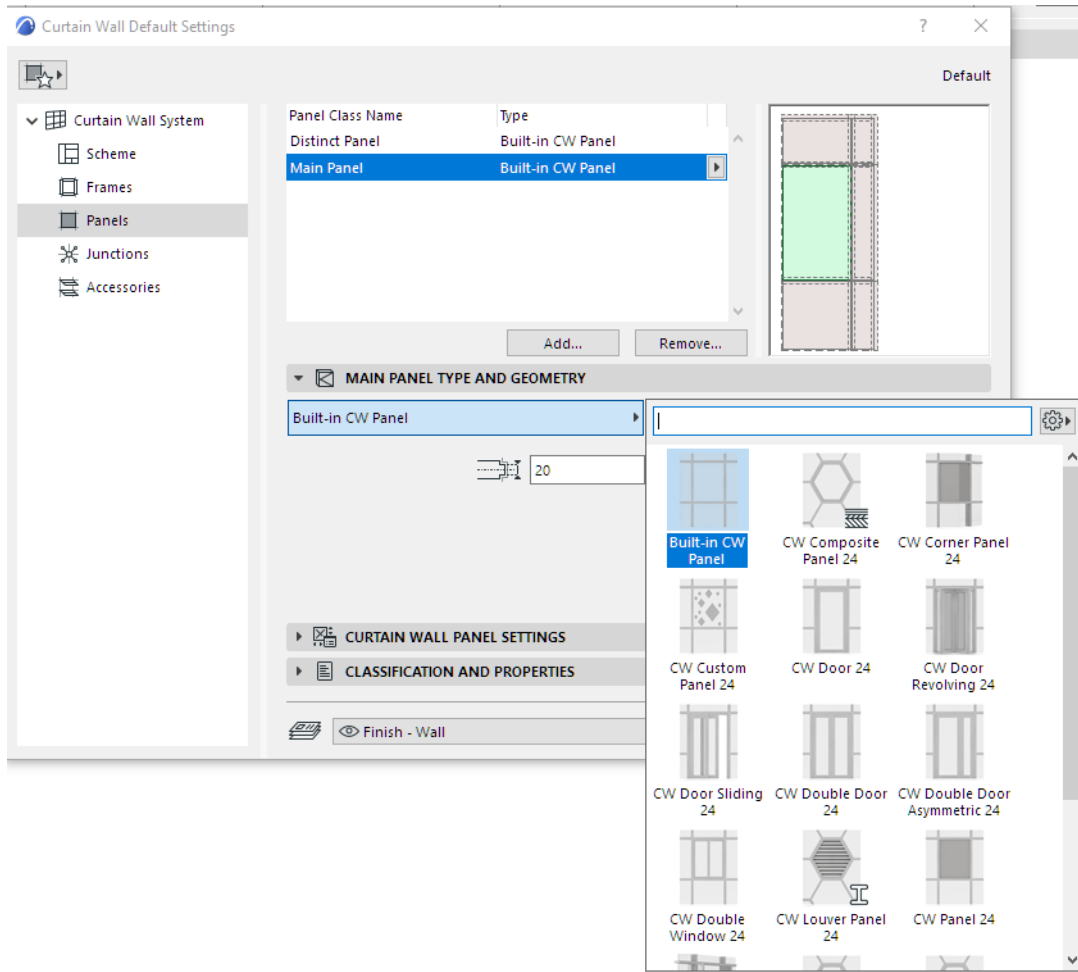
The top of the dialog lists the available Panel classes, by name and type.

Panel Classes are user defined (or provided by default - e.g. Main, Distinct). New Panel classes can be added or removed.

The selected class is highlighted in the preview window. The preview window also indicates the Panel's color.

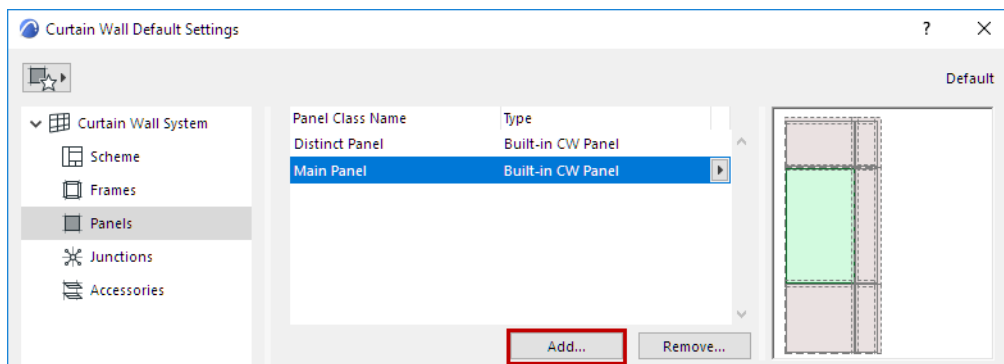


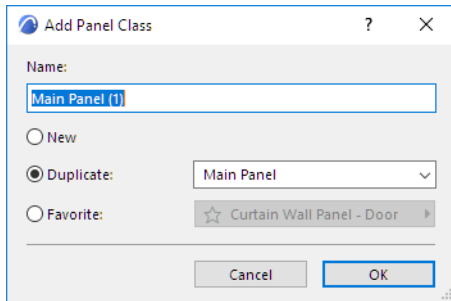
You can add or delete Panel classes - see below.



Add Panel Class

1. Click the **Add** button. In the appearing dialog:





- **New:** Creates a new Panel class based on the default (e.g. Built-in)
- **Duplicate:** Duplicates the currently selected Panel Class under a new name
- **Favorite:** Click the pop-up to choose a Panel Favorite. The new Panel class will use the Favorite settings.

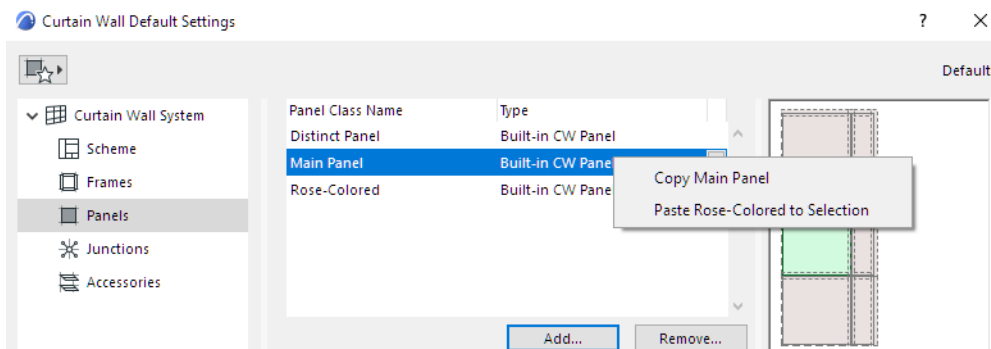
Rename Panel Class

Double-click the Class in the list, and rename as needed.

Copy-Paste Panel Settings Between Classes

In Panel Settings, right-click the name of any panel to copy its settings.

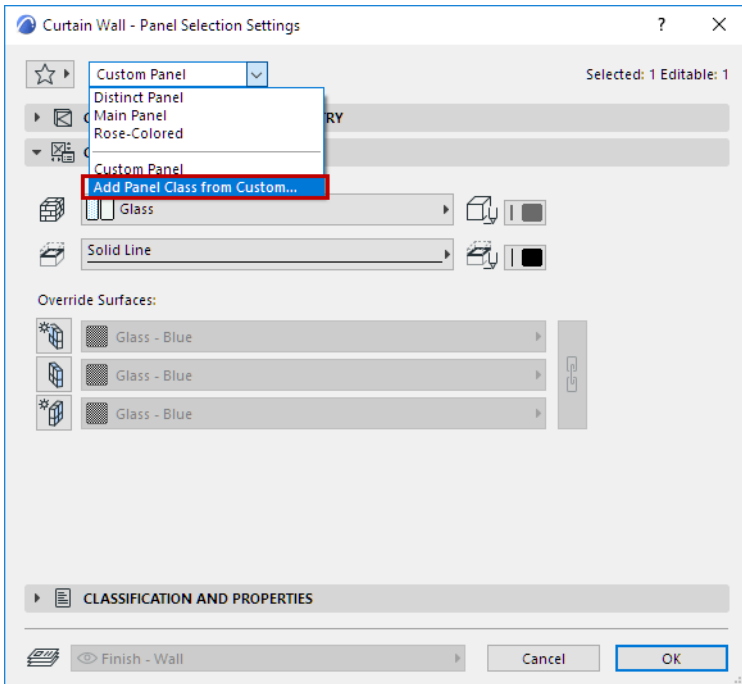
Right-click any other panel to insert the copied settings for that panel.



Add Panel Class based on a Customized Panel

Another way to create a new Panel Class is to customize a single Panel in Edit Mode, then save its settings as a new Panel Class.

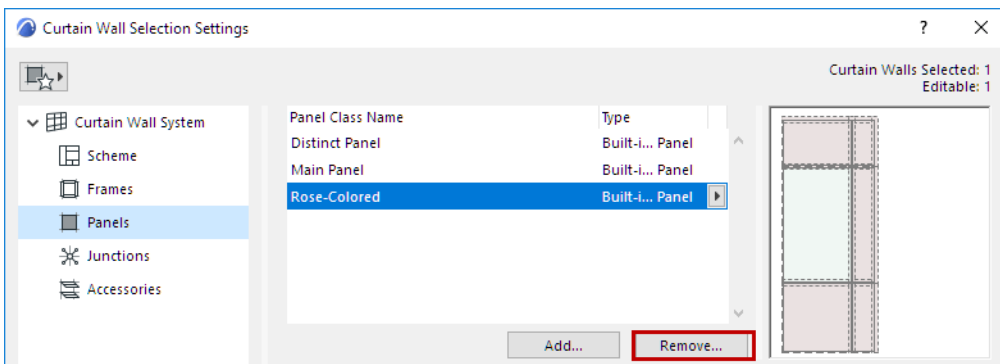
1. Select the Curtain Wall and enter Curtain Wall Edit mode.
2. Select a Panel to customize it. Its class is now "Custom."
3. From the Panel Class pop-up in the Settings dialog or Info Box, click **Add Panel Class from Custom...**



4. Use the upcoming dialog to name the new Panel class.

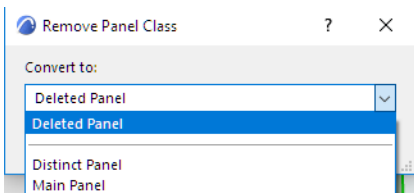
Remove Panel Class

1. Select one or more Panel Classes to remove.



2. Click **Remove**.

3. In the appearing dialog:

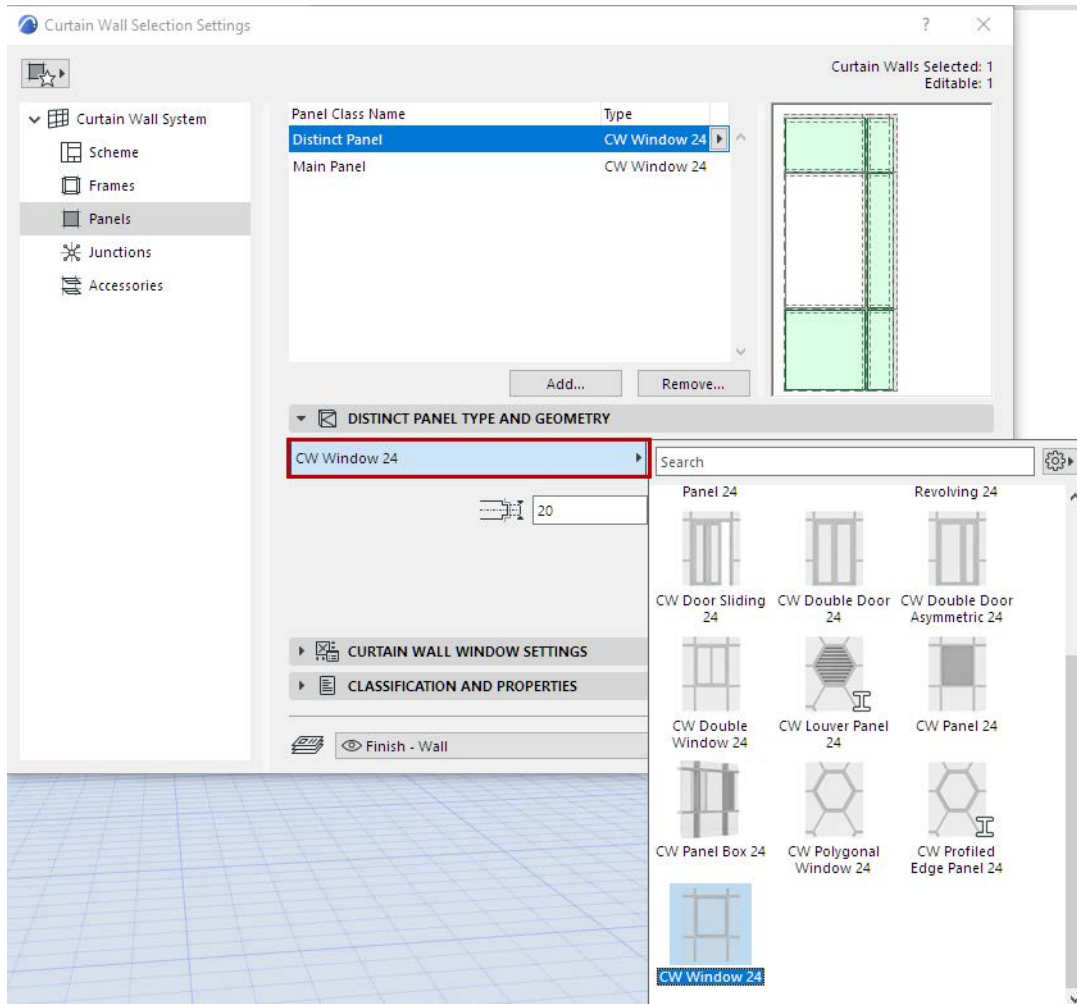


- From the pop-up, choose how to convert the Panels in the class that is being removed:
 - **Deleted Panel**
 - **Convert to Another Panel Class:** Choose an existing Panel Class.

Select Panel Type and Geometry

For the Panel Class selected in the list at the top of the dialog: choose a Type from the pop-up list, in the Panel Type and Geometry panel.

Then set its geometric parameters as applicable.

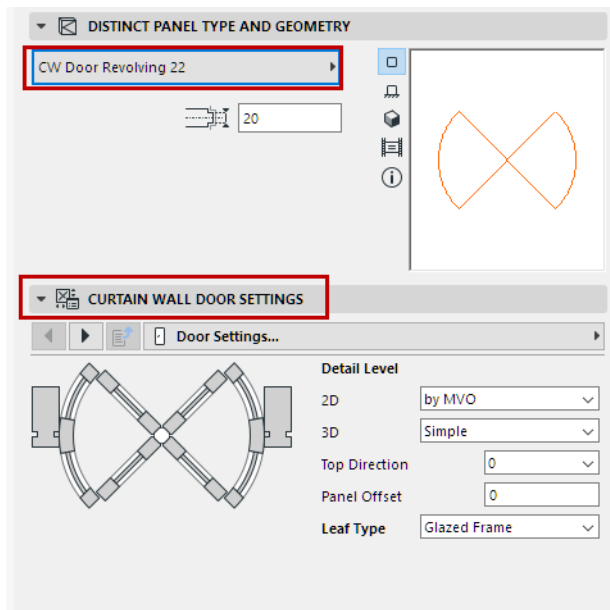


- Some Panels are designed to behave like doors and windows.

See [Place Door- or Window-Type Panel in Curtain Wall](#).

See also [Create Custom Components for Doors, Windows, Curtain Wall, Stair, Railing](#).

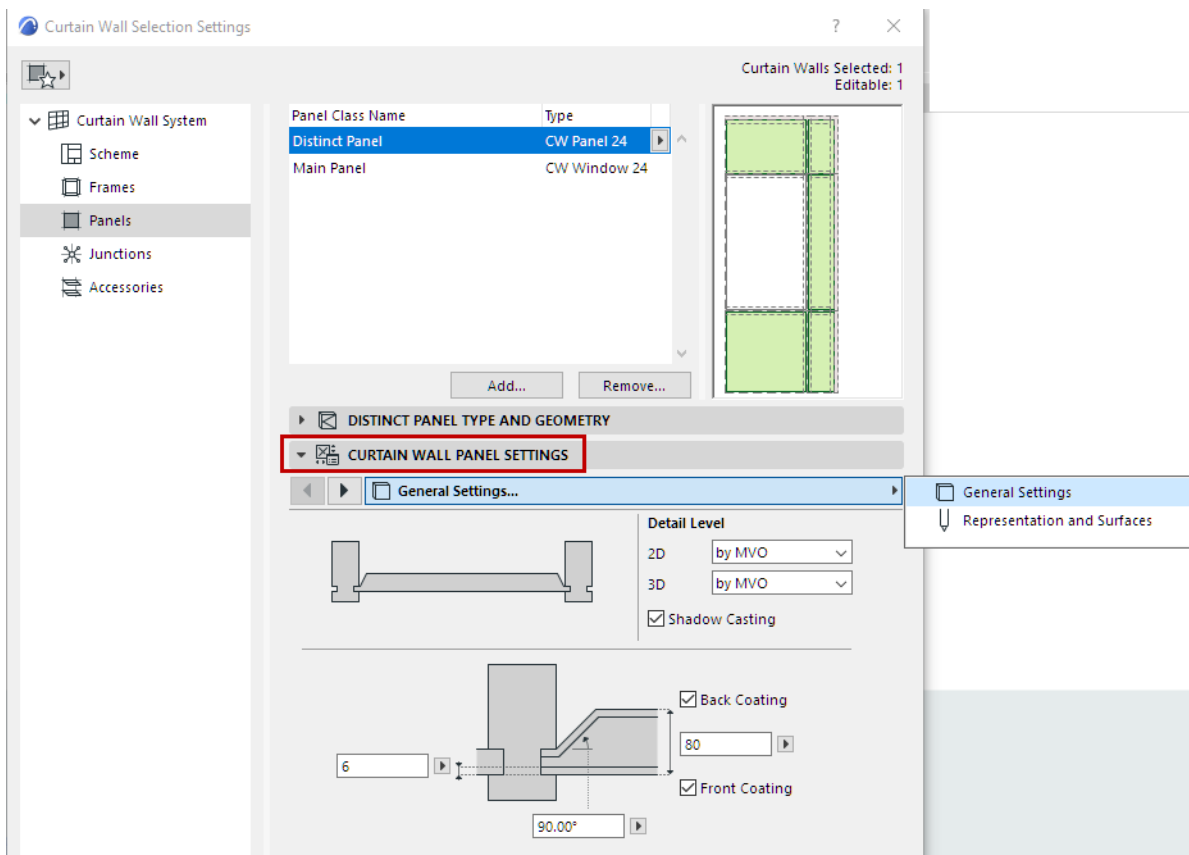
Available settings in the panel below depend on the chosen Panel type.



Panel Component Settings

For the chosen Panel type, you can set various parameters, as usual for GDL-type components.

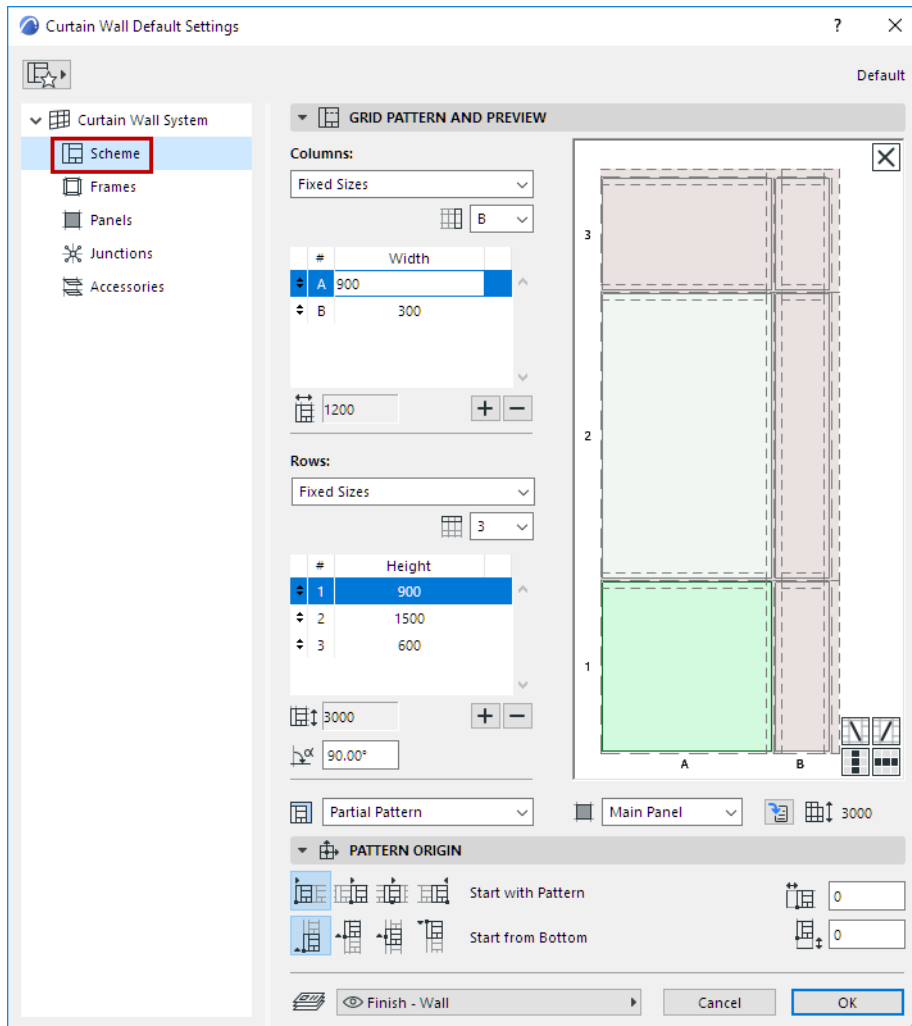
Note that you can override the Model View Options settings for its 2D and 3D Detail Levels, as applicable.



Curtain Wall Scheme Settings

This is a page of Curtain Wall Settings. Use these controls to define the pattern of the Curtain Wall Scheme Grid, by columns and rows. You can also set the class of any or all Frames and Panels here.

This section reviews the controls of the Scheme Settings page. For details and examples of Grid Pattern setup, see [Set up Grid Pattern in Scheme Settings](#).



Topics in this Section:

- [View/Edit Pattern in Preview Window](#)
- [Select Frame or Panel in Preview Window](#)
- [Set Frame or Panel Class](#)
- [Edit Panel Size](#)
- [Add/Delete Row or Column](#)
- [Scheme Width/Height](#)
- [Add Crossing Frame](#)
- [Clear Pattern](#)

View/Edit Pattern in Preview Window

The preview window displays one unit of the Curtain Wall pattern, consisting of Frames and Panels. The pattern shown here will be repeated horizontally and vertically within the Curtain Wall boundary, when it is placed in the project.

[See details and examples at Set up Grid Pattern in Scheme Settings.](#)

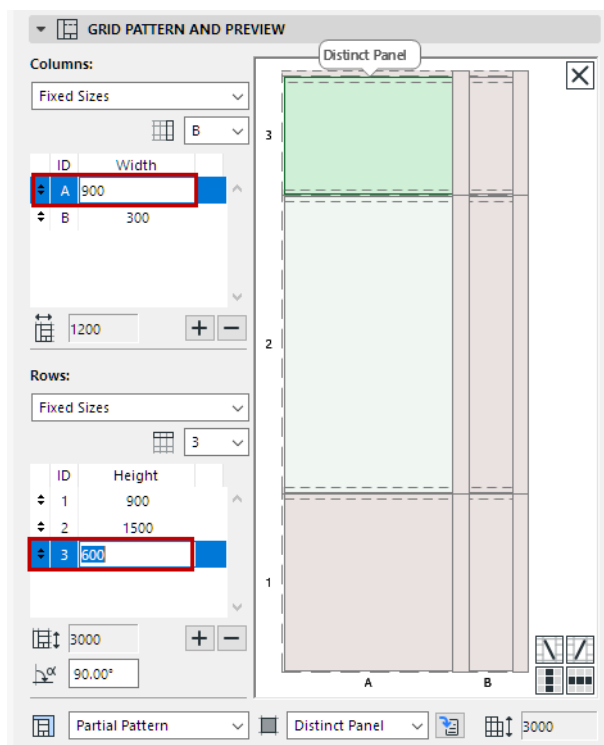
The Grid pattern's precise starting point, and the logic by which it should be repeated, are defined in the Pattern Origin panel of this Scheme Settings dialog.

Select Frame or Panel in Preview Window

Click on any Frame or Panel in the Preview: the clicked item is highlighted and a tag displays the item's class (e.g. Distinct Panel).

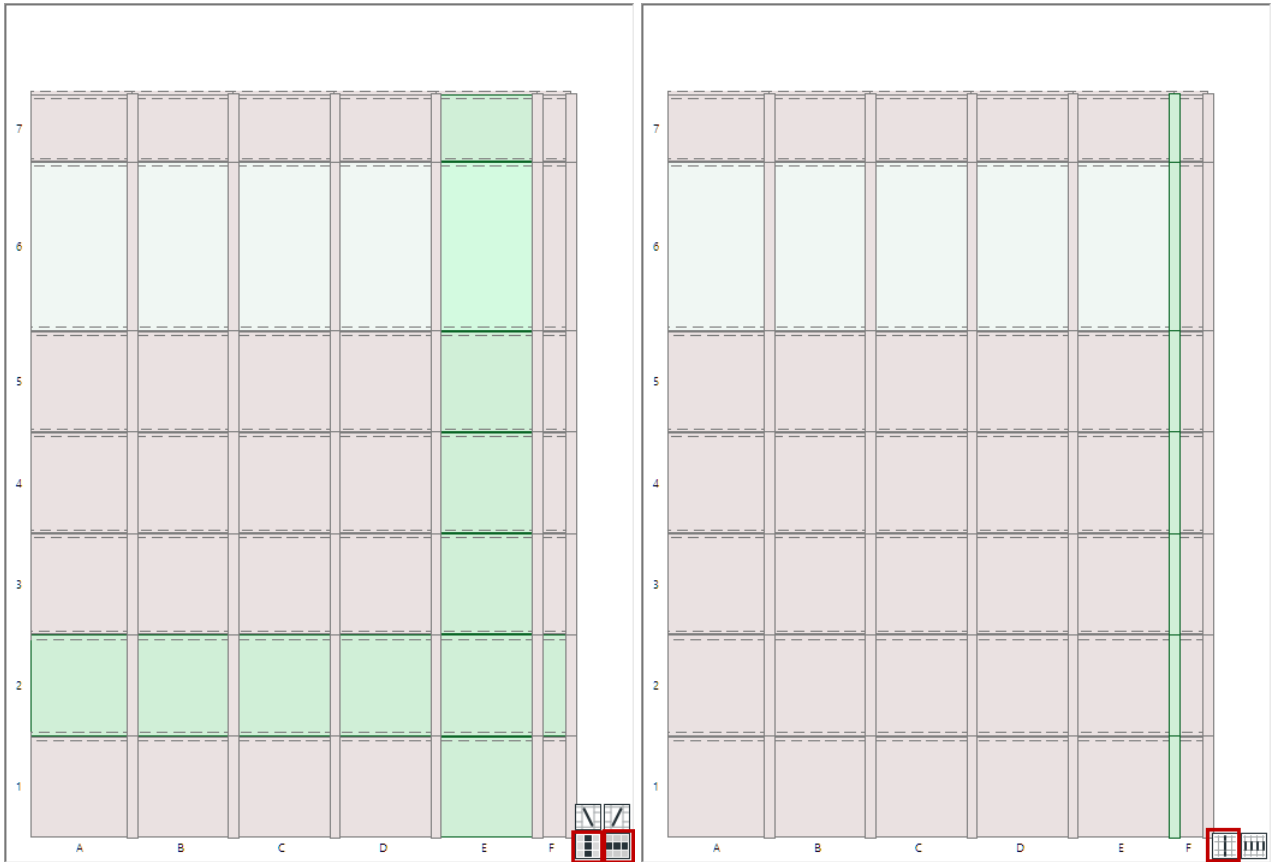
For a selected Panel, its position by Column and Row is also highlighted at left.

Here, the selected Panel is in the Distinct Class. The Panel is located in Column A, Row 3.



- Use shift-click to **multi-select** items in the Preview (i.e. when modifying a Frame or Panel class). In this case, the first-selected item is highlighted at left.

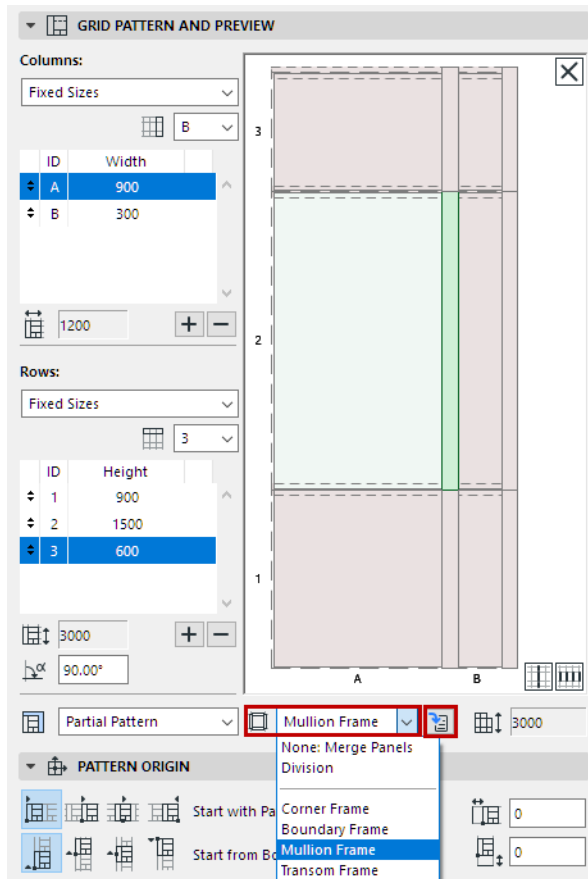
- Use the **shortcut multi-select buttons** at bottom right to select all the Frames or Panels in the same column (vertically) and/or the same row (horizontally) as the selected one:



Shortcuts: Multi-Select Panels and Frames

Set Frame or Panel Class

Use the pop-up below the Preview to change the selected Panel's or Frame's class, if needed.



Panel and Frame Classes are defined on the corresponding pages (Frames/Panels) of Curtain Wall Settings (where you can also define new classes as needed.)

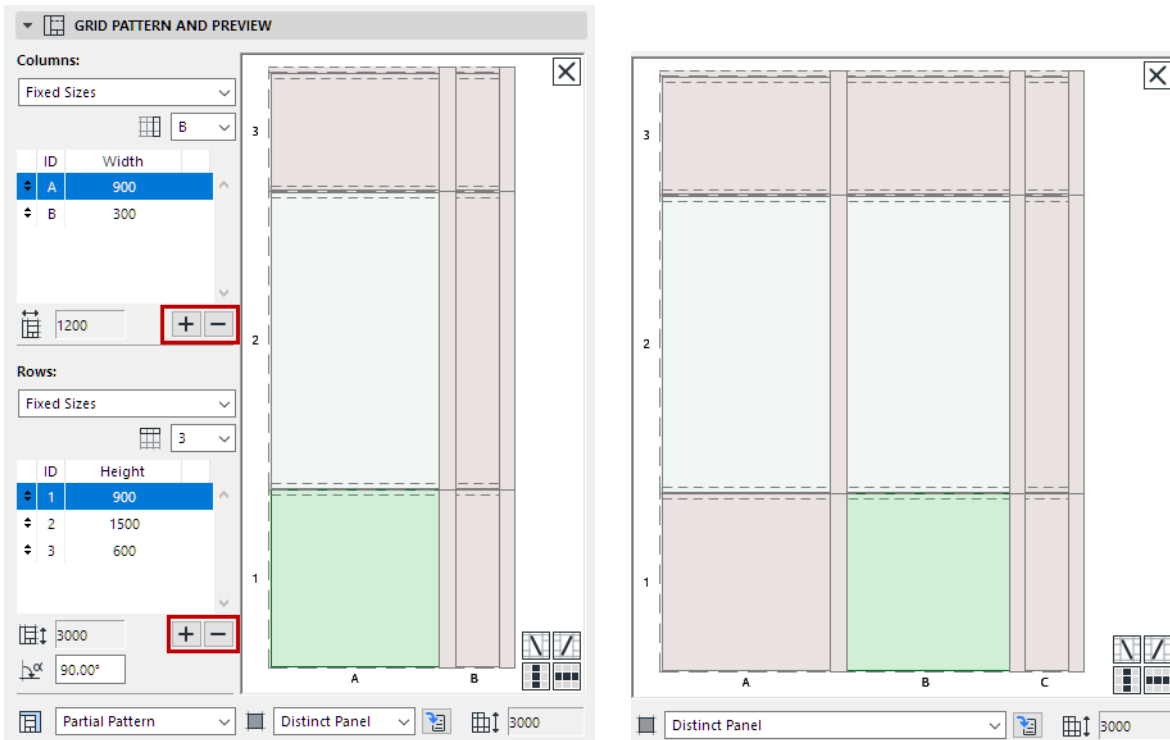
To access these pages from here, click the “Go to Settings” button.

Edit Panel Size

Enter values for the width and height of selected Panel(s). Notice how the Preview image changes accordingly.

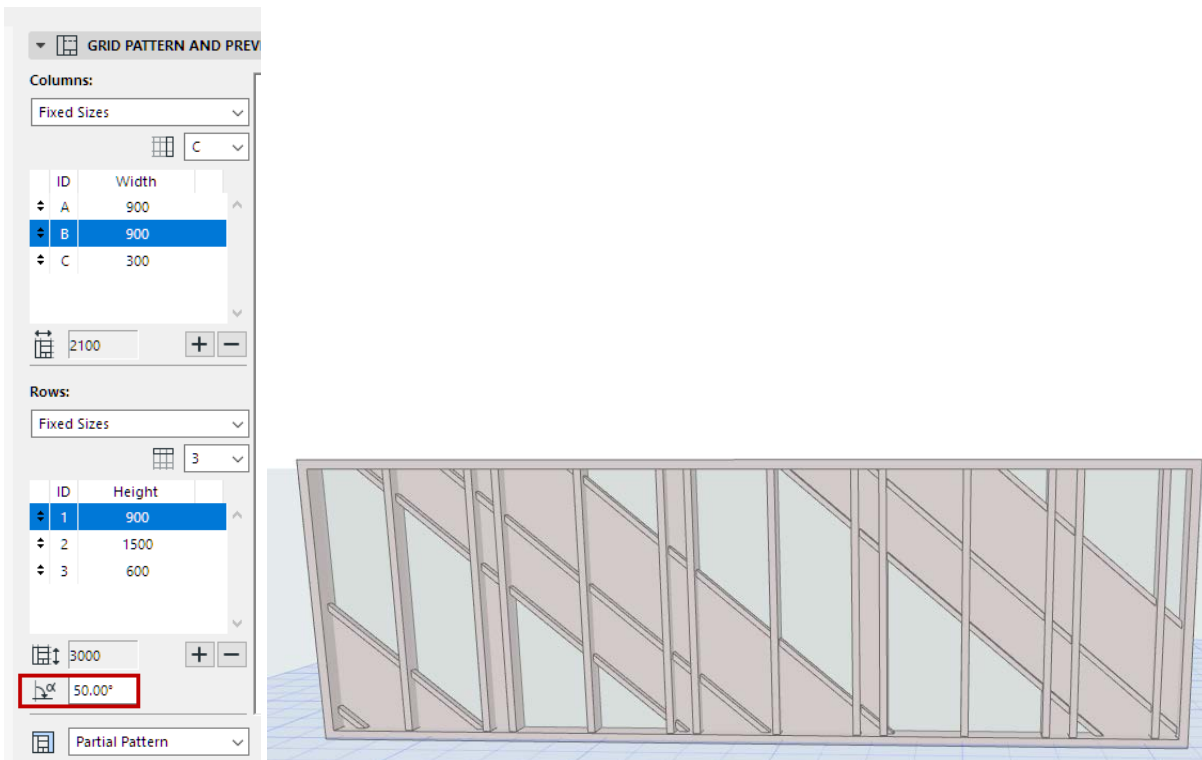
Add/Delete Row or Column

To create a new Column or Row of Panels, or to delete a selected one, click the Plus/Minus sign. The new Column/Row duplicates the currently selected one, and is inserted in front of it.



Rotation

Enter a Rotation value for the Rows of panels, relative to the Columns.



Note: Rotation angle will not be reflected in the Preview window.

Scheme Width/Height

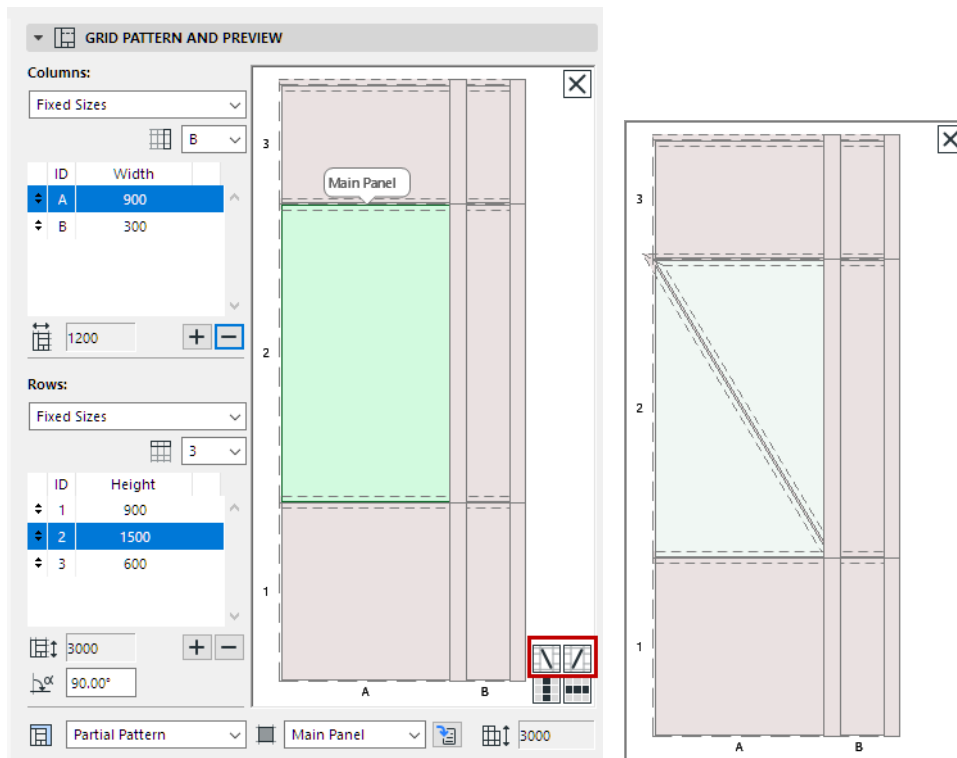
These values indicate the width and height of the Scheme Grid pattern. The placed Curtain Wall, depending on its final width and height, will repeat this Scheme pattern as many times as needed, and based on the logic defined in the Pattern Origin panel.

[See details and examples at Set up Grid Pattern in Scheme Settings.](#)

Add Crossing Frame

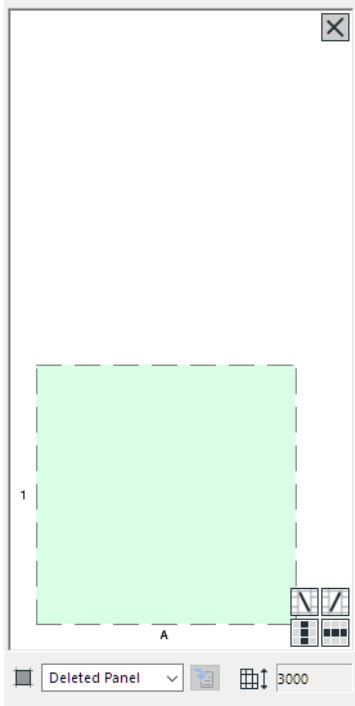
1. In the Scheme Preview, click on a Panel.
2. Click a Crossing Frame icon to add it.

Limitation: One diagonal frame per pattern cell.



Clear Pattern

In the upper corner of the Preview window, click the X button to clear all Frames and Panels. This results in a Curtain Wall with an empty Boundary.



Set up Grid Pattern in Scheme Settings

Use the Scheme page of Curtain Wall Settings to set up a Grid pattern in advance, before you place the Curtain Wall.

Note: Another approach is to place an empty Curtain Wall boundary and define the pattern graphically. See [Create Grid Pattern Graphically](#).

There are three main Pattern Distribution options for both rows and columns:

- Fixed Size
- Best Division
- Number of Divisions

Each logic is described below.

Topics in this Section:

[Fixed-Size Rows and Columns](#)

[Define Infill for the “Leftover” \(Fixed Size Row and/or Column\)](#)

[“Best Division” Pattern Distribution](#)

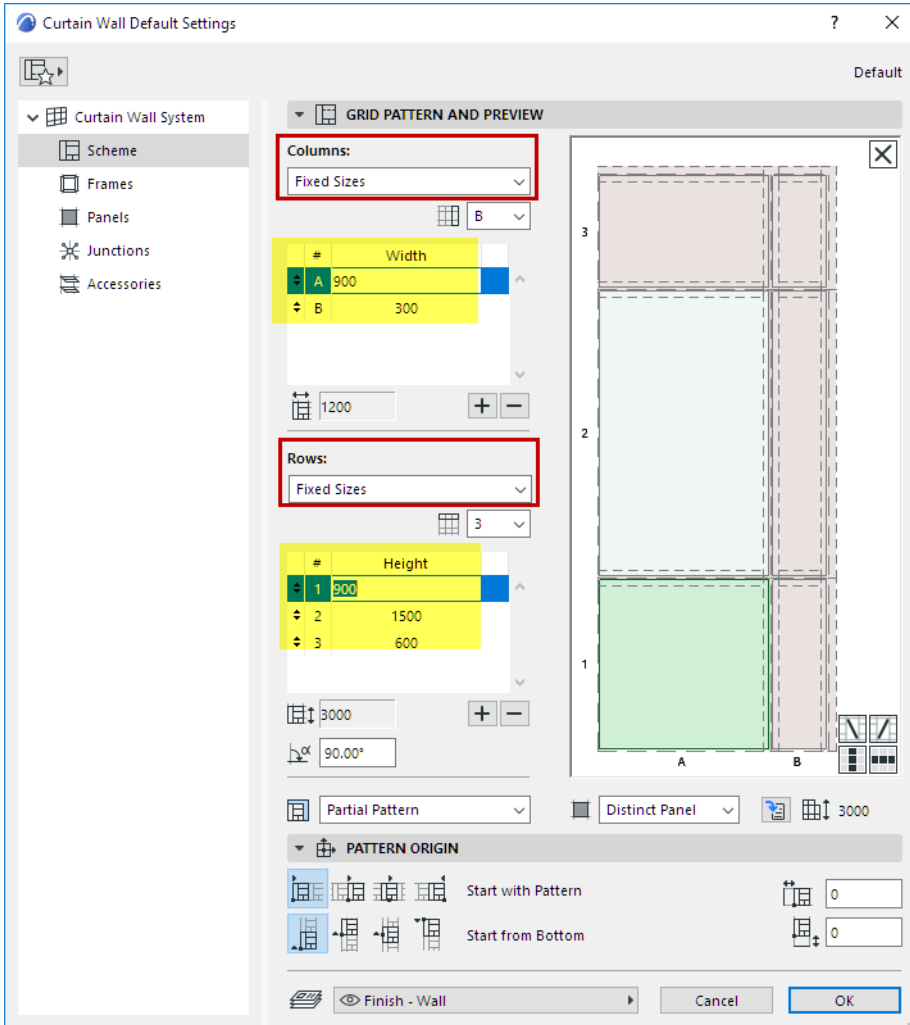
[Set Number of Divisions](#)

[Pattern Origin \(for Fixed Size patterns\)](#)

[End with \(Column/Row\)](#)

Fixed-Size Rows and Columns

For the Scheme's Rows and/or Columns: choose the Fixed Size option. (This Pattern Distribution logic is the same as in pre-22 versions of Archicad. Set each Column width and/or each Row height to an exact size.)



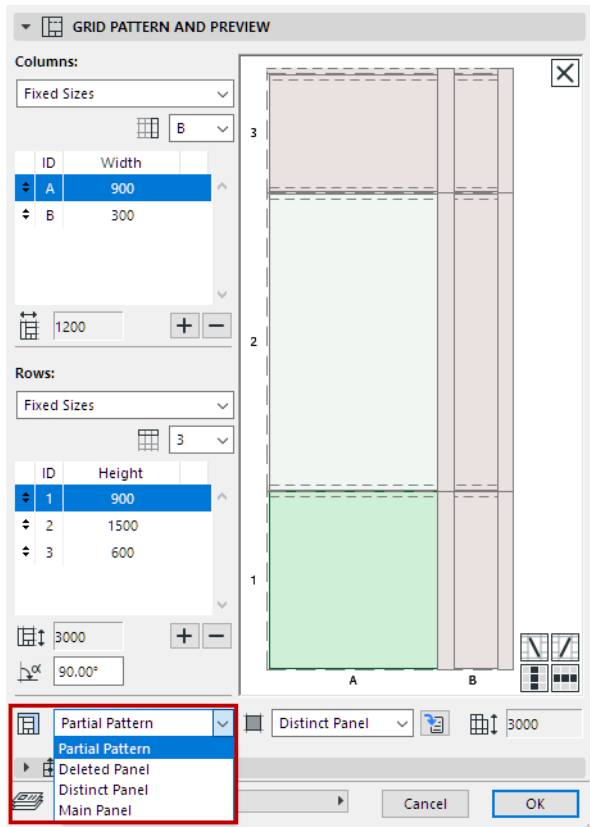
The resulting Curtain Wall in 3D, with fixed-size rows and columns:



The fixed pattern is repeated to the extent of the Boundary and then cut off.

Define Infill for the “Leftover” (Fixed Size Row and/or Column)

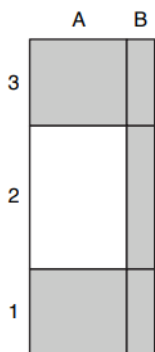
If either Columns or Rows is set to Fixed Sizes, use the “Infill with” control to define what to do with the “leftover” inside the Boundary, if the end of the pattern does not coincide with Boundary.



- **Partial Pattern** cuts off the pattern where it falls, at the end of the Curtain Wall Boundary.
- Or choose a **Panel class**: the leftover segments (in the horizontal and/or vertical directions) will be filled in with this panel type.
- The Pattern Origin setting affects the arrangement of the Infill (see examples below).

Examples of Infill Settings

This Curtain Wall uses a Grid pattern with fixed-size columns and rows:

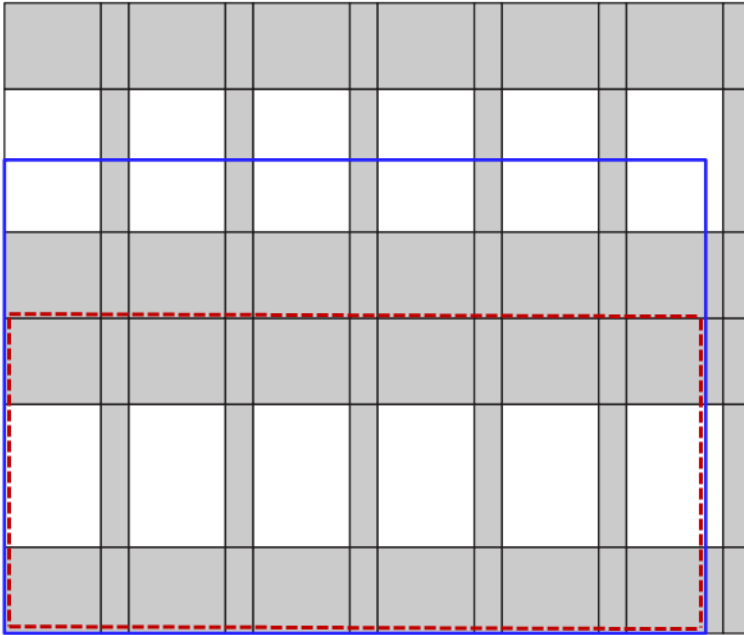


Curtain Wall Grid Pattern

See what happens with various Infill and Pattern Origin settings.

The blue line represents the Curtain Wall Boundary.

The red dashed line represents the repeated pattern.



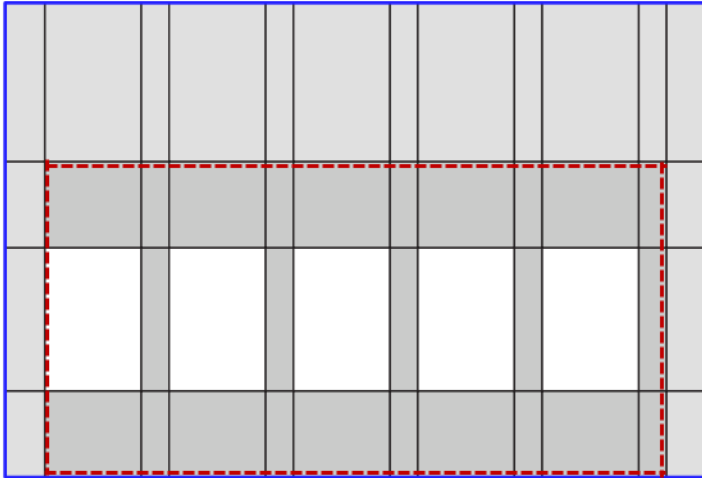
Infill with: Partial Pattern

Pattern Origin: Left Bottom



Infill with: Distinct Panel

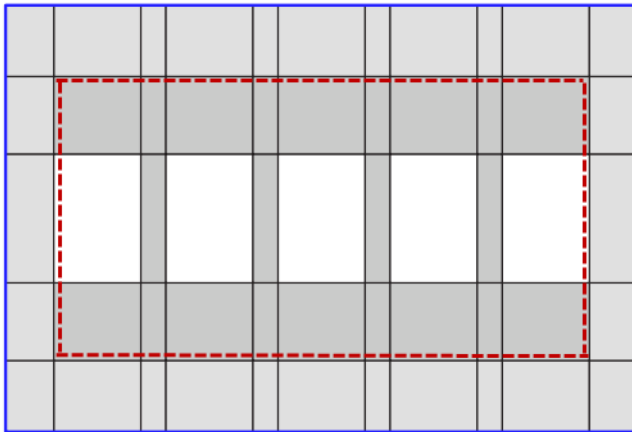
Pattern Origin: Bottom Left



Infill with: Distinct Panel

Horizontal Pattern Origin: Align to Center

Vertical Pattern Origin: Bottom



Infill with: Distinct Panel

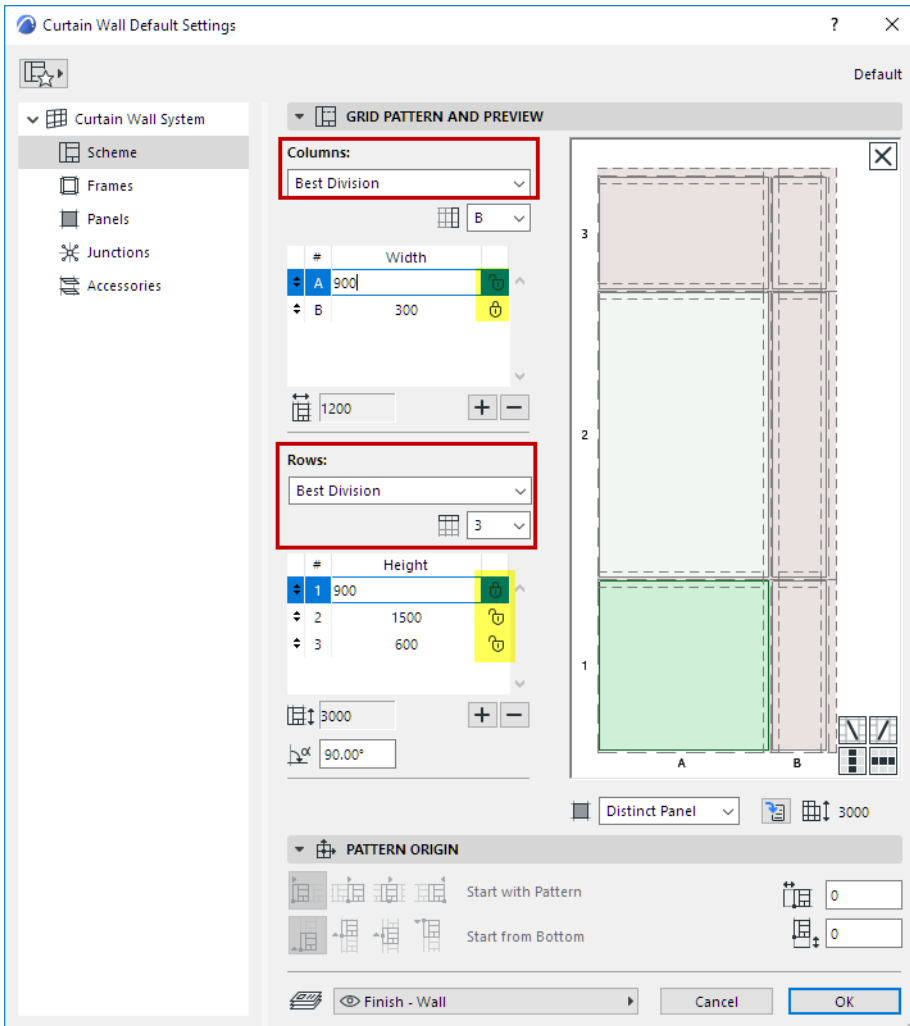
Pattern Origin: Align to Center (both directions)

“Best Division” Pattern Distribution

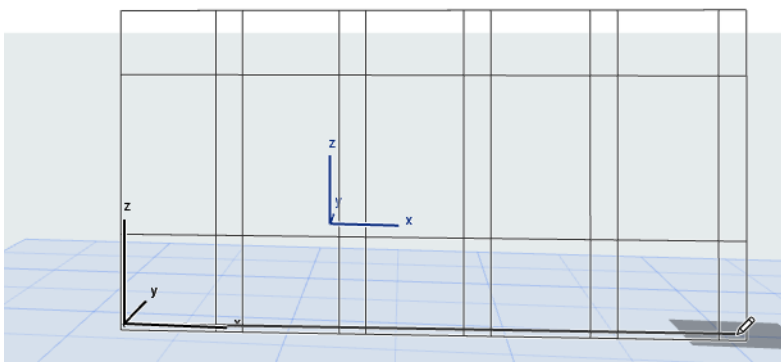
With this logic, the Curtain Wall pattern will exactly coincide with the end of the Curtain Wall segment, without any “leftover”.

For the Scheme’s Rows and/or Columns: choose the **Best Division** option.

- Set each Column width and/or each Row height to the desired size, then use the lock icons for each Row/Column to set whether its size should be fixed, or flexible.
- At least one Row or Column *must* be set to flexible. This means it might be stretched or shrunk relative to the size you entered.
- If you set a value to “flexible”, the resulting Row or Column will be sized as close as possible to this value. Several flexible values (e.g. two flexible column widths of 300 and 600) will maintain their relative sizes - that is, one flexible column will always be twice the width of the other.



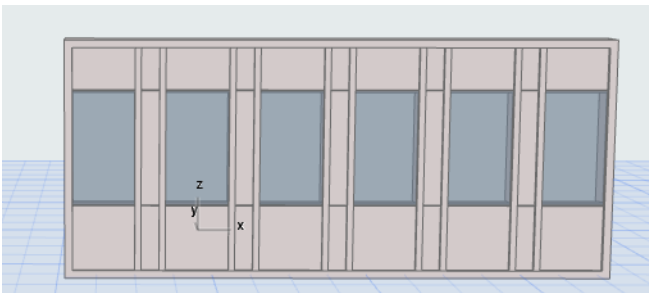
As you input a Best-Division Curtain Wall, the feedback shows you that the column and row patterns are automatically distributed evenly.



Use the “End With” control here, to use a different panel in the last column:



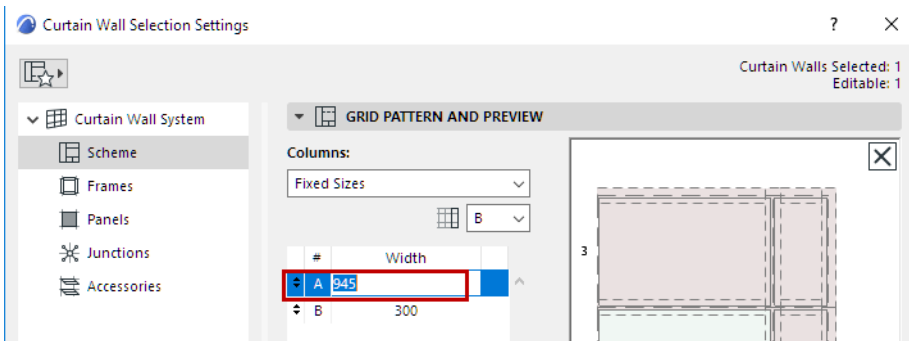
Best Division (End with Column B)



Best Division (End with Column A)

To get feedback on how the flexible row or column (e.g. Column A set to 900 mm, flexible) was actually sized in the resulting Curtain Wall:

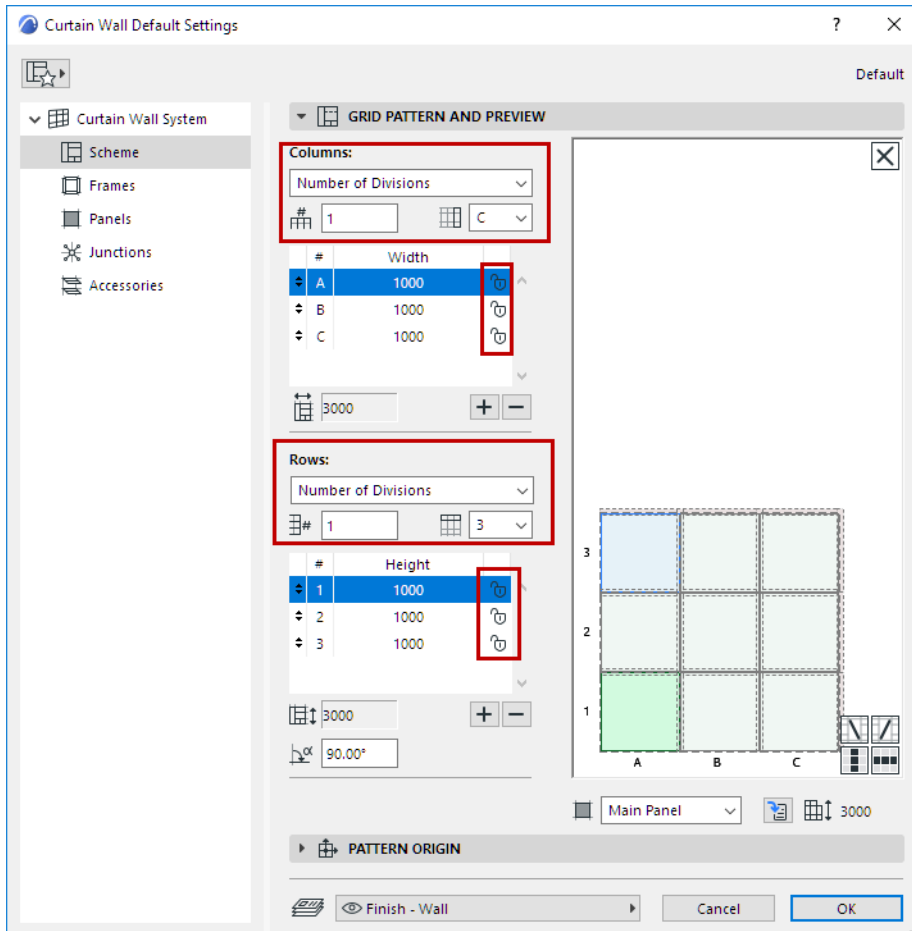
go to Curtain Wall Selection Settings, and switch the pattern to “Fixed Sizes” to see the value (here, Column A is actually 945 mm).



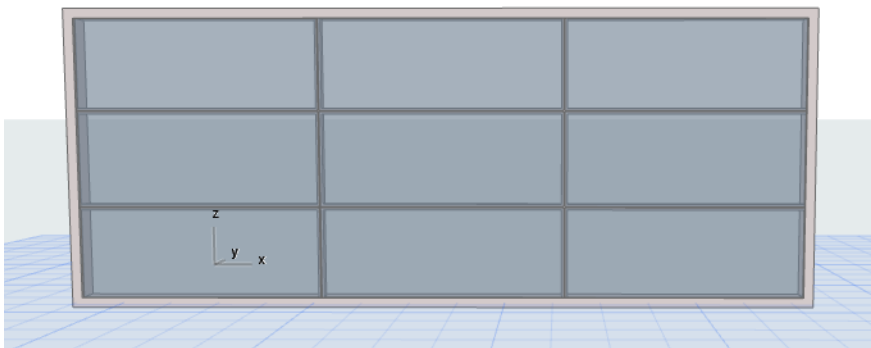
Set Number of Divisions

Use this option to specify that the pattern be repeated a fixed number of times within the Curtain Wall segment.

You must set at least one row/column to have a flexible size, to allow it to stretch.



Here, all the rows and columns are flexible. No matter how high and how wide the Curtain Wall is drawn, the pattern (three rows by three columns) will be repeated exactly once.



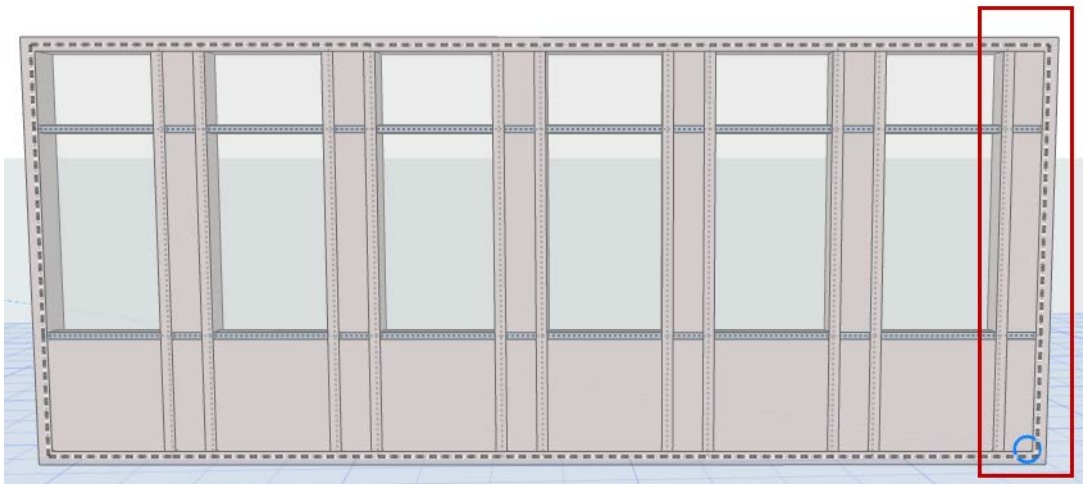
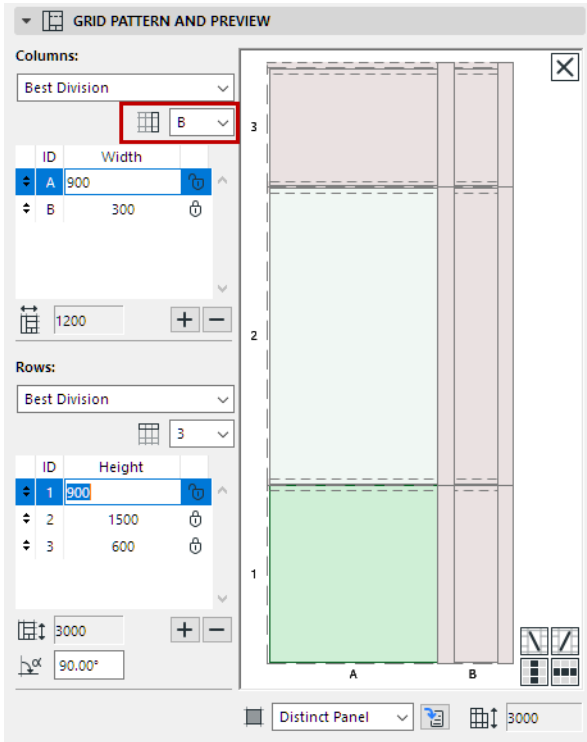
End with (Column/Row)

For both Columns and Rows, choose one to end the pattern with.

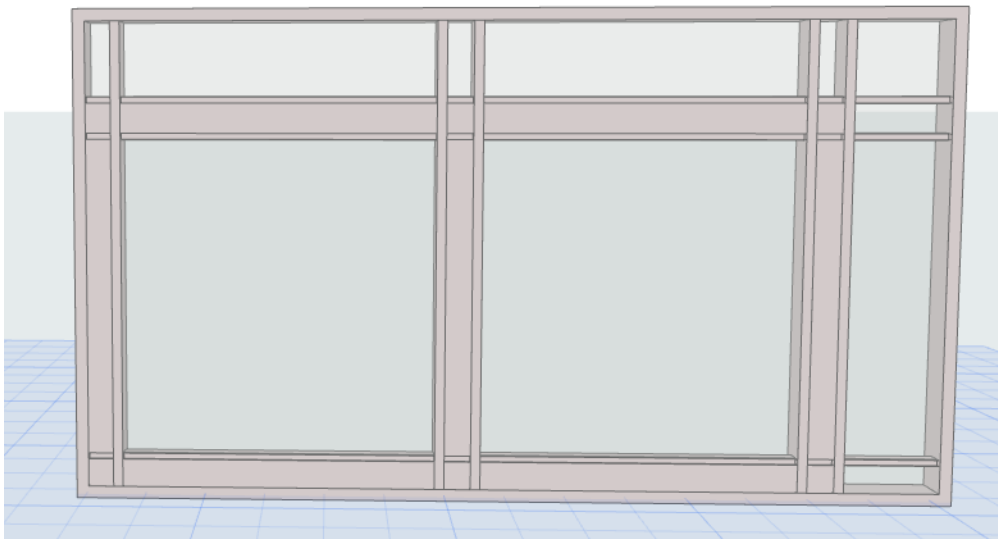
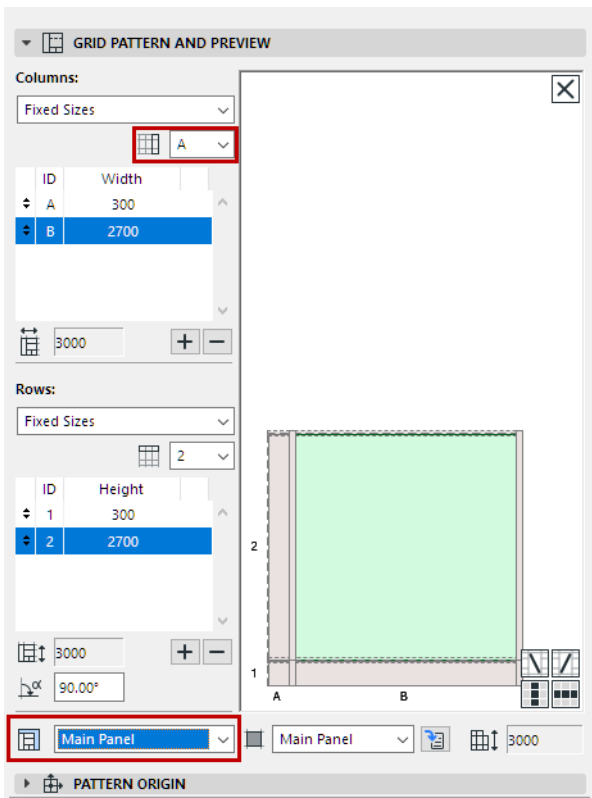
For example: Your pattern contains two column types - A and B.

Set End Column to B.

When you place the Curtain Wall, the AB pattern will be repeated using their fixed sizes, but the segment's last column will be the B column. (followed by the Infill, if applicable).

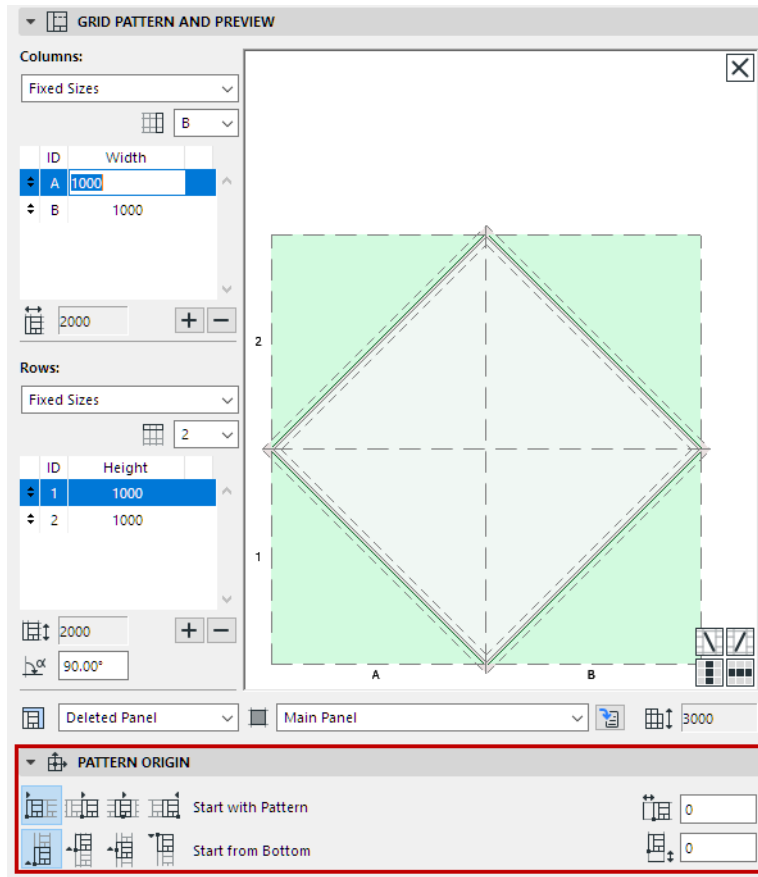


End Pattern with Column B



End Pattern with Column A - Then the Infill

Pattern Origin (for Fixed Size patterns)



The Pattern Origin definition locks the origin of the grid pattern (in both the horizontal and vertical directions) to a specific point on the Curtain Wall segment, plus or minus an optional offset.

Consult the tool tips and icons to pick the one you need.

Note: Pattern Origin controls not available if the row/column logic is set to “Best Division” or “Number of Divisions”.

Horizontal Pattern Origin

- Origin at the start of the segment
- Origin at the middle of the segment. The horizontal pattern will begin at the center of the segment, and will be distributed outward in both directions. (The center of the Curtain Wall will always have a vertical gridline running through it.)
- Pattern midpoint is at the segment midpoint
- Origin at the end of the segment

Vertical Pattern Origin

- Origin at the bottom of the segment
- Origin at the middle of the segment
- Pattern midpoint at the segment midpoint
- Origin at the top of the segment

See [Examples of Infill Settings for illustrations](#).

Create Grid Pattern Graphically

Use this workflow to graphically create a custom Curtain Wall grid pattern, in the 3D or Section-type window (using Edit mode).

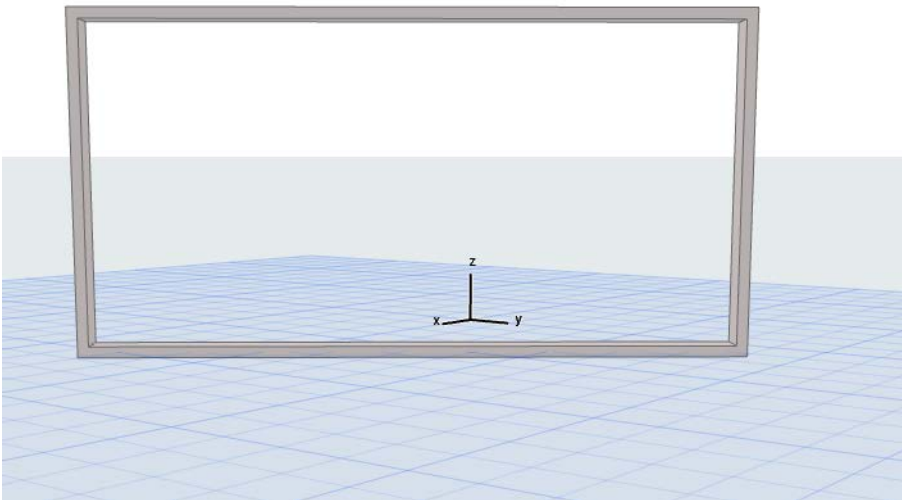
Note: Another approach is to predefine the pattern in Curtain Wall Scheme Settings before placing the Curtain Wall. See [Set up Grid Pattern in Scheme Settings](#).

Place Frames and Panels in any pattern of Frames and Panels on the Scheme Grid. Then activate the Pattern box, and use it to define the extent of the pattern. The pattern will be automatically repeated within the Curtain Wall segment.

See the detailed workflow below.

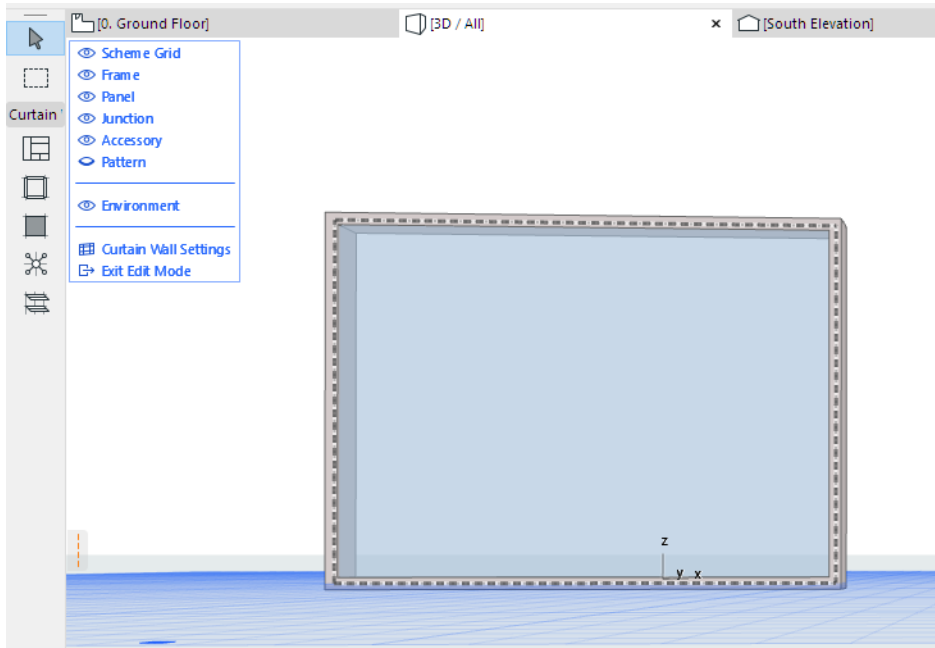
1. Place a new Curtain Wall without any Frames or Panels.
 - Use the Curtain Wall Scheme Favorite “Empty” or “Empty with Grid”. (For a curved Curtain Wall, use “Empty with Grid”.)

or draw a Curtain Wall Boundary in Section view.



2. From the 3D window or Section view, open the Curtain Wall in Edit mode.

3. From the Edit mode display menu, make sure the Scheme Grid is visible.

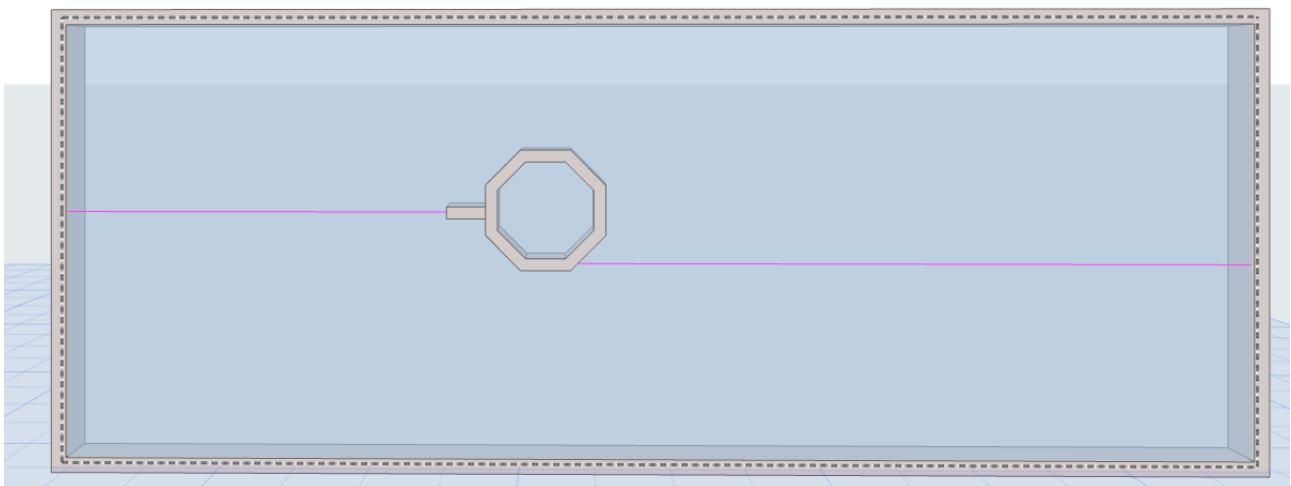


4. Use the Frame and Panel tools to draw Frames and place Panels in any pattern, anywhere within the Boundary. (To place a Panel, activate the Panel tool and click inside a Frame.)

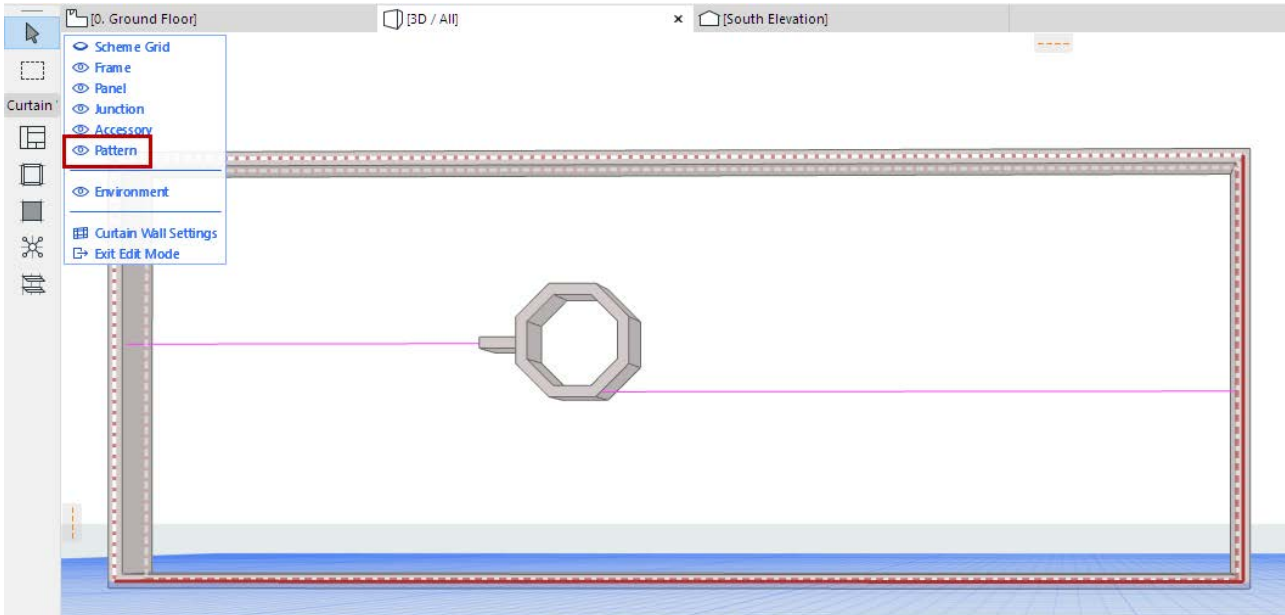
Note on Temporary Frames

All Frames must be connected at both ends to another Frame. If you place a Frame that is unconnected to another Frame at either end, a temporary Frame is automatically placed - as shown in the image below.

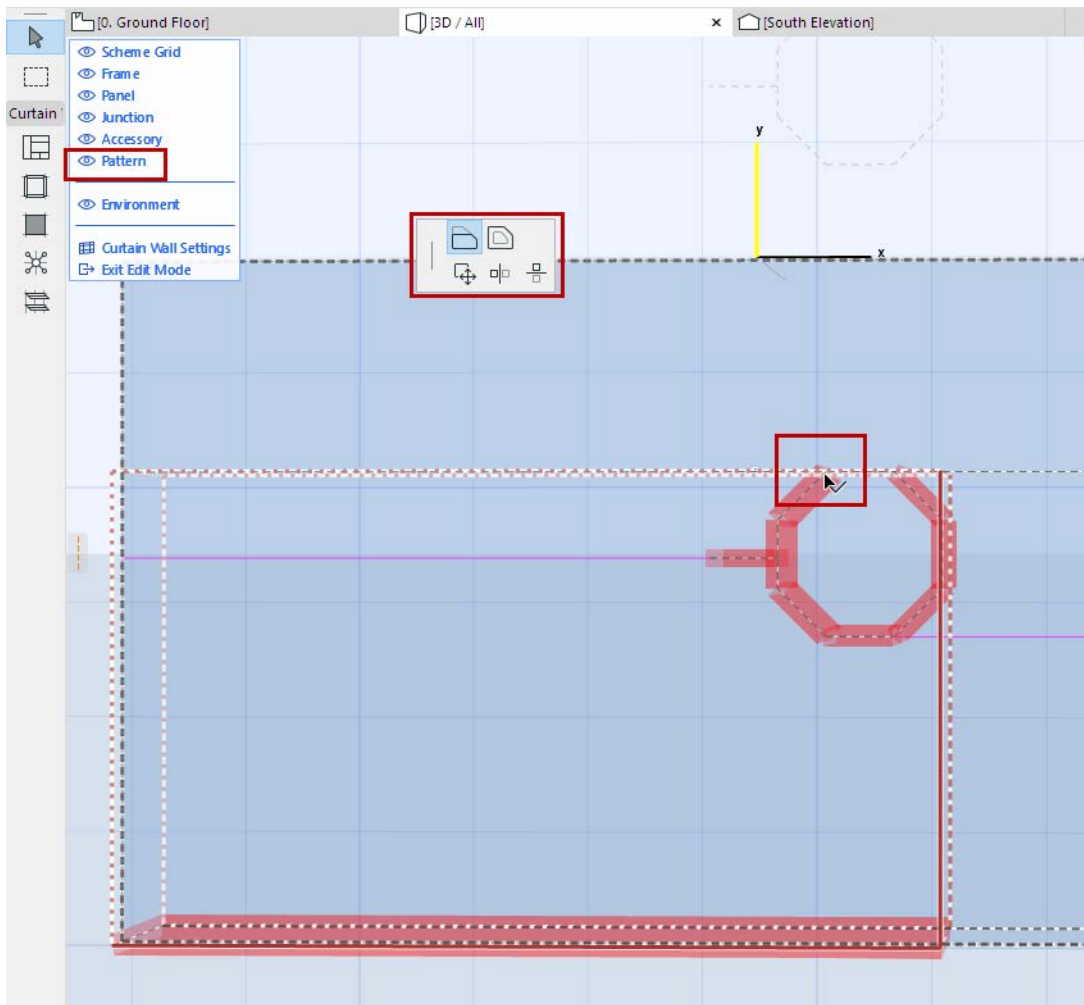
Temporary Frames disappear as soon as the Frame is connected to another one. If the Frame remains unconnected, the temporary Frame remains, and acts as a Division Frame: adjoining Panels will be divided at the Division Frame.



5. In the Edit mode display menu, click the Pattern item to display it. An editable red Pattern Box appears on your Curtain Wall.



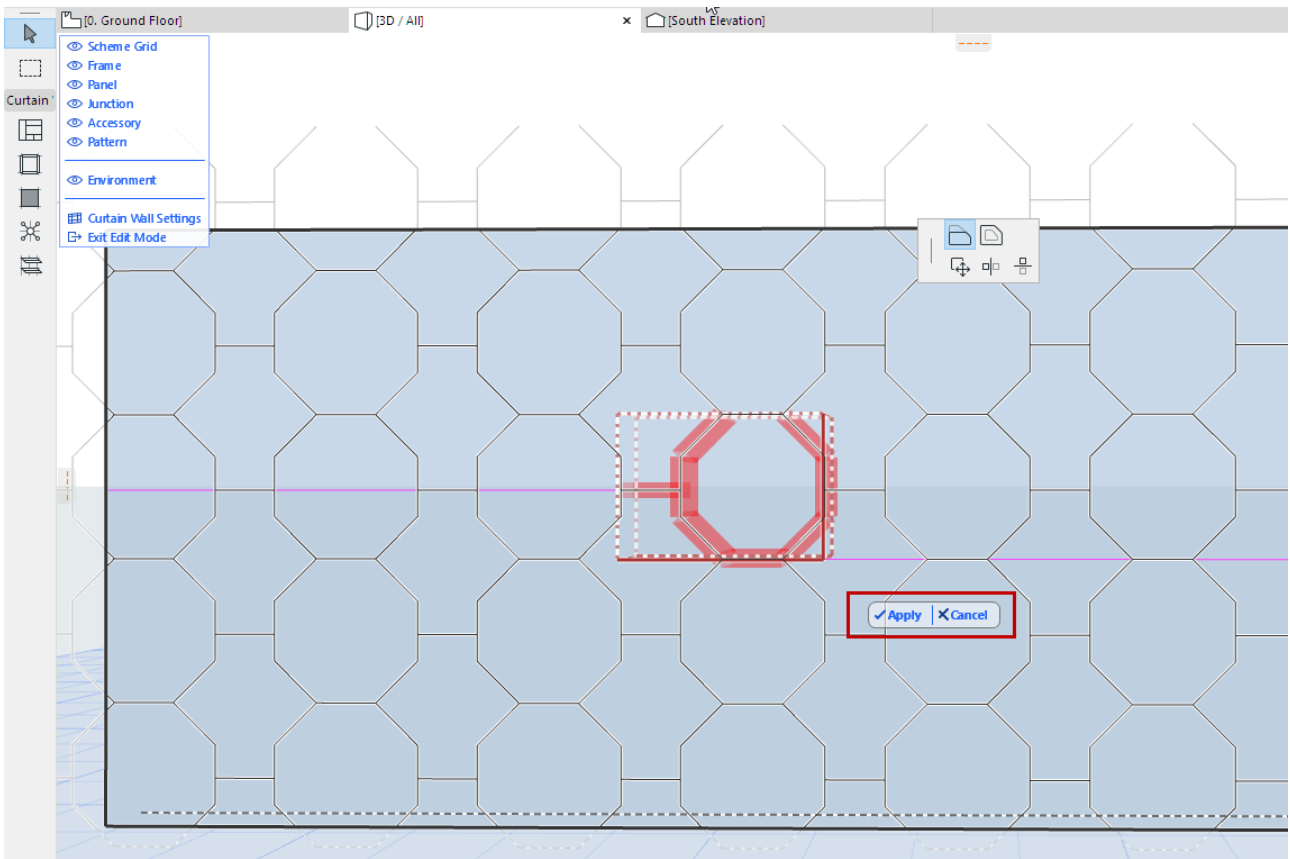
6. Use the Arrow tool to select the Pattern box edges and drag them (use Pet palette editing commands) to define the limits of the Pattern that you want to repeat.
7. Snap the Pattern box edges to Frame nodes. Frames and Panels within the Pattern box will constitute the Scheme Grid pattern.



Notes:

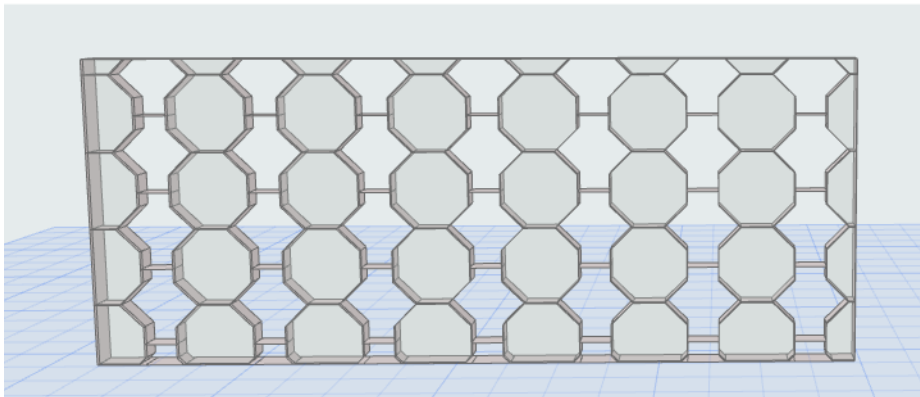
- To make it easier to snap the Pattern box, use a Model View Options Combination in which Curtain Wall components are displayed as Schematic or Simplified, or (for Frames) Axis only.
- Even with the Pattern Box visible, you can continue to edit any Curtain Wall components as usual.
- On a multi-segment Curtain Wall: in Edit mode, the Pattern box is used on one segment at a time. As you move your cursor over the different segments, the Pattern box jumps to the current segment. However, once you finish the Pattern, you can apply it to all segments with one click - see below.

8. When the Pattern Box encloses the pattern you need, click **Apply**.



For a multi-segment Curtain Wall: You have the option to apply this pattern to all segments.

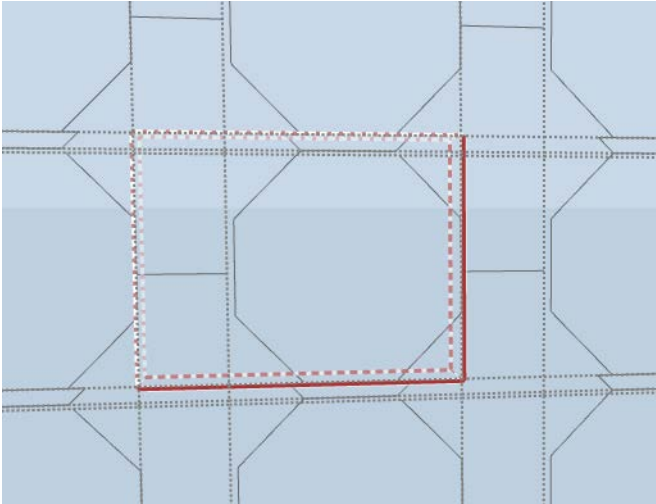
- 9. The pattern is now repeated over the Curtain Wall segment (or the entire Curtain Wall).



Curtain Wall Pattern Box

The Pattern Box is displayed on two sides with solid lines, and two sides with dotted lines.

With the box, the Frames that coincide with the solid lines will be repeated, while those that coincide with the dotted lines will be discarded. (This is relevant if the Pattern Box includes Frames with different settings.)



Curtain Wall Input Methods

The Curtain Wall is placed on or relative to the input plane.

- In the **Floor Plan**, the input plane is the Story level.
- In **Section/Elevation/IE**, the input plane is identical to the Section plane.
- In **3D**, the default input plane for Curtain Walls (as for any construction element) is Project Zero or the User Origin, but you may use several methods for defining any other input plane.

[See *Curtain Wall Input Plane in 3D Window*.](#)

Topics in this section:

[Create Curtain Wall on the Floor Plan](#)

[Draw Curtain Wall Boundary in the Section Window](#)

[Create a Curved Curtain Wall: Geometry Methods](#)

[Positioning Curtain Wall in Section](#)

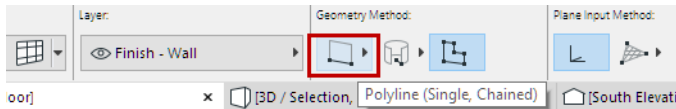
[Curtain Wall Input Plane in 3D Window](#)

[Create Curtain Wall in the 3D Window](#)

[Extrude Curtain Wall from a Slanted Input Plane](#)

Create Curtain Wall on the Floor Plan

1. Activate the Curtain Wall tool.
2. Choose the Polyline - Single geometry method from the Info Box.

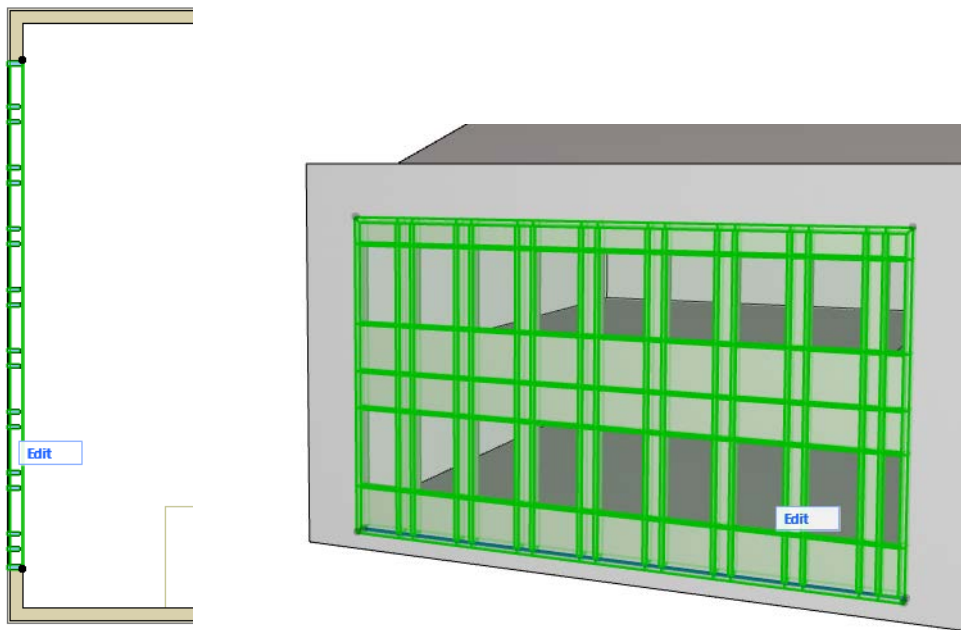


3. Draw a Reference Line representing the length of the Curtain Wall.

Note: If your Scheme has a fixed-size grid pattern, the ghost feedback and pencil/snap point indicate where the next pattern segment will be complete.

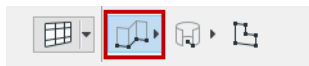


4. Click to complete the Reference Line.
5. View the result in the 3D window.



Chained Curtain Wall

Use the Chained geometry method to create a multi-segment Curtain Wall. The reference line is a polyline.

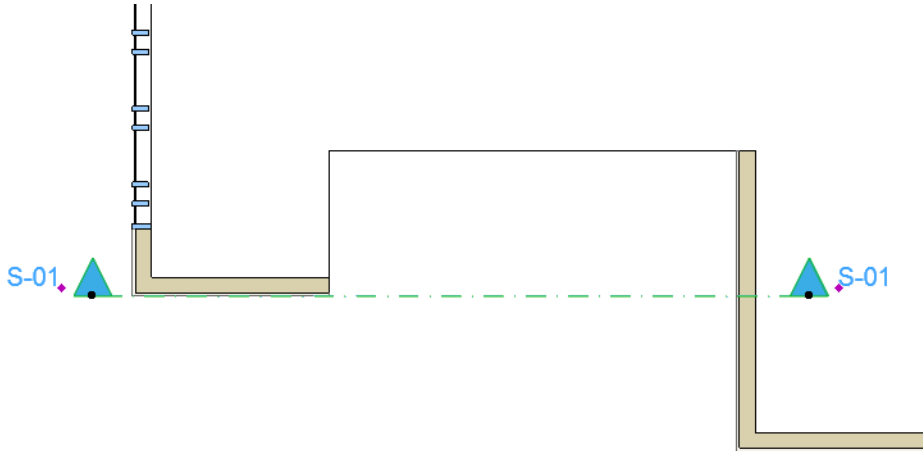


- For each segment, you can set up a separate Scheme, with a distinct pattern for each - in Edit mode.
- A Frame is always placed where two adjacent segments are joined. Such a Frame cannot be deleted. (If you wish, you can set its type to Invisible, by selecting the Frame in Edit mode and adjusting its Type in Frame Settings.)

Draw Curtain Wall Boundary in the Section Window

In this example, create a Curtain Wall by drawing a freehand Boundary in Section.

1. Go to the Floor Plan.
2. Draw a Section Line in the Floor Plan to define the Curtain Wall's input plane.



3. Open the Section.
4. With the Curtain Wall tool active, choose the Boundary geometry method from the Curtain Wall Info Box.



5. Draw the Curtain Wall Boundary along the desired contour, clicking at each node as when drawing any polyline.
6. Double-click to close the polygon.

Note: The Reference Line of this Curtain Wall is the first Boundary segment you draw.

Related Topic:

[Positioning Curtain Wall in Section](#)

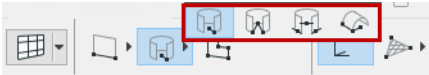
Create a Curved Curtain Wall: Geometry Methods

You can create a curved Curtain Wall in the Floor Plan, Section/Elevation/IE window, or 3D window.

When defining the geometry of a curved Curtain Wall, you must first define the **cylindrical Base Surface**, then mark off the parts of this cylinder onto which the Curtain Wall will be placed.

Note: Curved Curtain Walls are always regularly shaped: the axis of its cylindrical Base Surface is either horizontal or vertical to the input plane when you create it, although subsequently you can tilt it in any direction.

Choose one of the four geometry methods for a curved Curtain Wall. For the most part, these are the same methods used to draw circles or arc walls in Archicad.



1. Centerpoint and Radius:

- Click to place the centerpoint.
- Click a second time to define the radius and one of the endpoints of the Curtain Wall.
- Click a third time to define the other end of the wall.

2. Three Points: Click three times in succession to define the beginning point, another point, and the endpoint of the Curtain Wall. By defining these three points, the reference cylinder's radius and centerpoint are also calculated.

3. Tangential: Use this geometry method to place a Curtain Wall that forms a full circle. Click three times in succession to define three points on the Curtain Wall. By defining these three points, the reference cylinder's radius and centerpoint are also calculated. A temporary reference circle appears.

If the three tangential points enable more than one geometrical possibility for a full-circle, the eyeball cursor appears. Move the cursor to view the possibilities, then click to choose which geometry you need.

4. Radius and Axis: Use this geometry method to create a regular arc-shaped Curtain Wall. This Curtain Wall will always be a half-cylinder shape placed on the input plane.

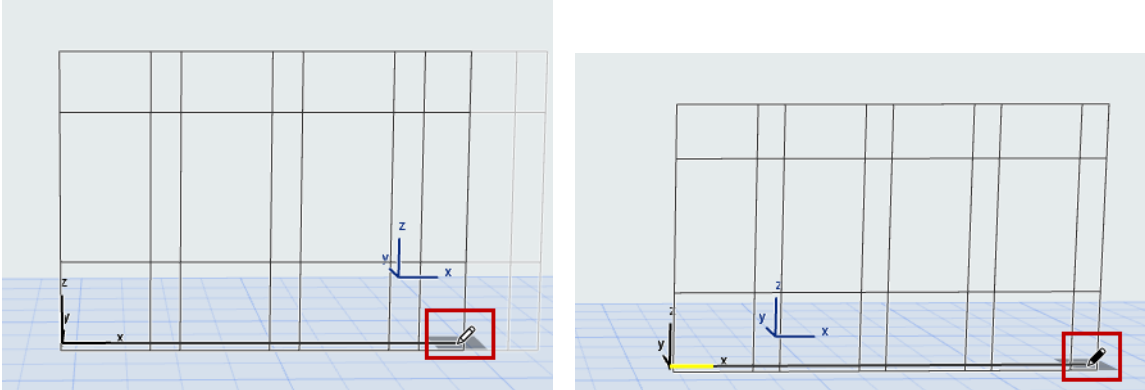
The first two clicks define the Curtain Wall's beginning and endpoints.

Click a third time to define the extrusion length of the arc.

Create Curtain Wall in the 3D Window

Input the Curtain Wall using the usual methods for a construction tool.

Note: If your Scheme has a fixed-size grid pattern, the ghost feedback and pencil/snap point indicate where the next pattern segment will be complete.



In the 3D window, the default Curtain Wall input plane is the same as the current 3D input plane.

To define other input planes, see [Curtain Wall Input Plane in 3D Window](#).

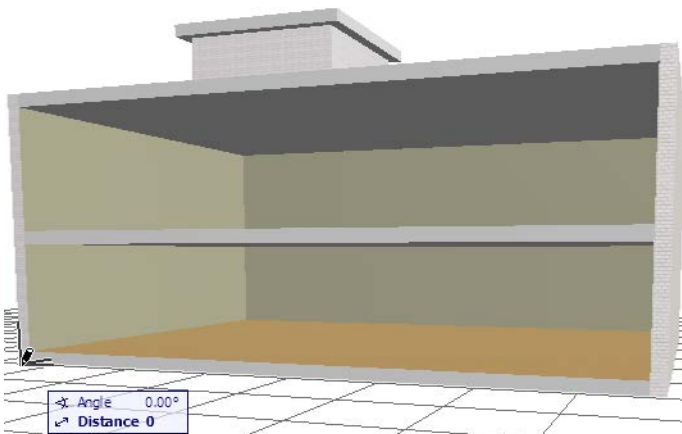
Polyline Method

1. With the Curtain Wall tool active, choose the Polyline - Single geometry method and the Default Input Plane from the Info Box.

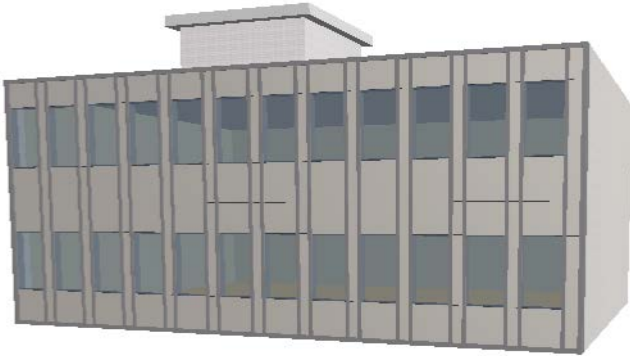


2. Click to begin drawing the Curtain Wall Reference Line.

In our example:



3. Click at the two front corners of the building to define the Reference Line length.
View the result.

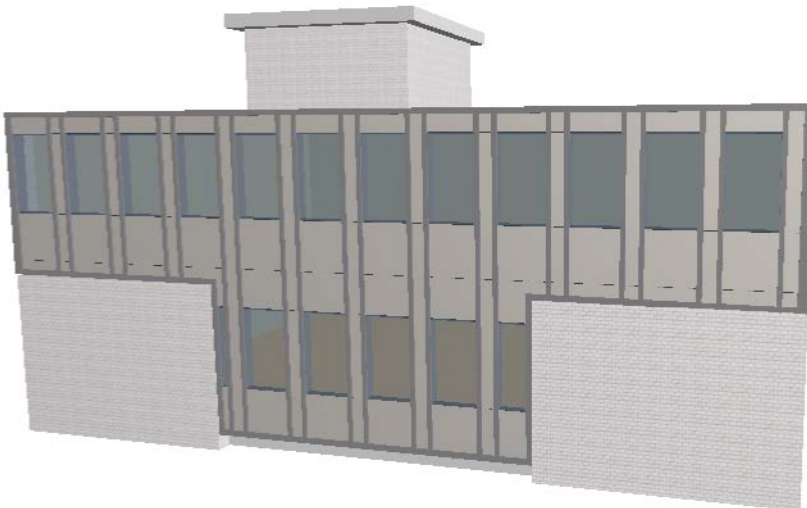


Boundary Method

1. With the Curtain Wall tool active, choose the Boundary geometry method and the Intelligent Input Plane Method from the Info Box.



2. Define the Input Plane by a series of clicks.
3. Begin drawing the desired Curtain Wall Boundary on the input plane, clicking at each node as when drawing any polyline.
Note: The Reference Line of this Curtain Wall is the first Boundary segment you draw.
4. Double-click or click with the Hammer cursor to close the polygon.
View the result.



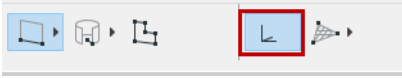
Related Topics:

[Curtain Wall Input Plane in 3D Window](#)

Curtain Wall Input Plane in 3D Window

In the 3D window, the Curtain Wall Info Box provides the following options for defining the Curtain Wall's input plane:

Default Input Plane

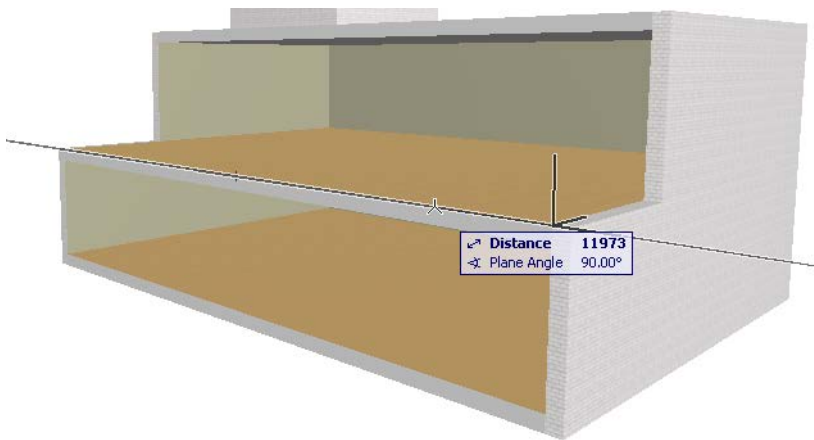


Intelligent Input Plane (Choose a plane)

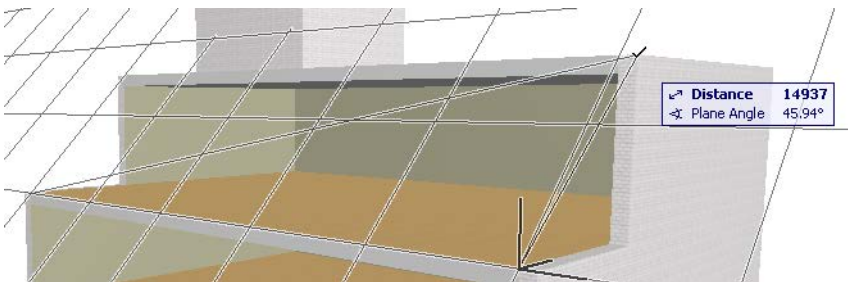


Use one of the following "intelligent" methods to define an input plane by a series of clicks:

- **Surface:** Click any surface: the plane of the surface will be your input plane.
- **Edge plus point:** Click any edge (Mercedes cursor) to define a line.

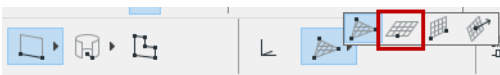


A plane appears on screen, rotating around the line as its axis. Then click a point (arrow cursor) to fix the input plane.



- **Point plus edge:** This is just the reverse of the method above. Click a point, then an edge to define the input plane.
- **Three points:** Click any three points to define the input plane.

Horizontal Input Plane



Click on a point to place a horizontal input plane at that point.

Vertical Input Plane



Click on a point to locate a vertical input plane, then click again to define the plane's orientation.

Normal Input Plane



Define a slanted input plane by defining a vector that is perpendicular (“normal”) to the desired plane.

1. Click two points to define the normal vector to the input plane, or click on an edge to define the normal vector to the input plane.
2. Then click a point through which the input plane will run.

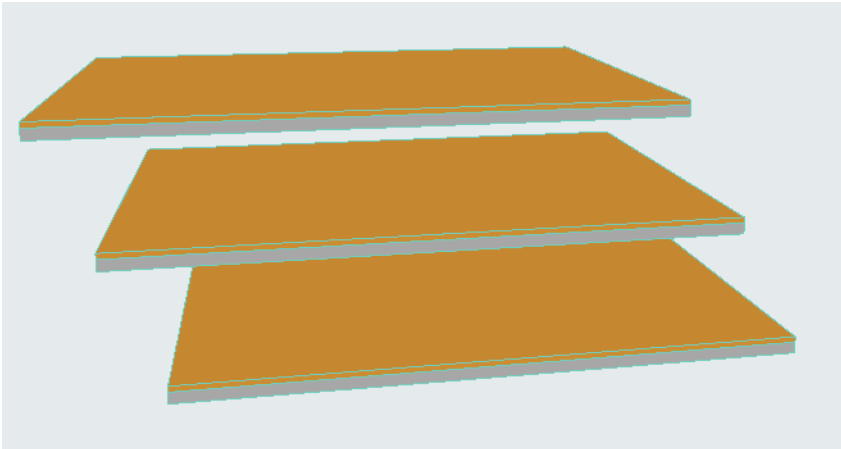
For an example, see [Extrude Curtain Wall from a Slanted Input Plane](#).

Extrude Curtain Wall from a Slanted Input Plane

In the 3D Window, you can define any input plane.

[See Curtain Wall Input Plane in 3D Window.](#)

Consider the following model in 3D:

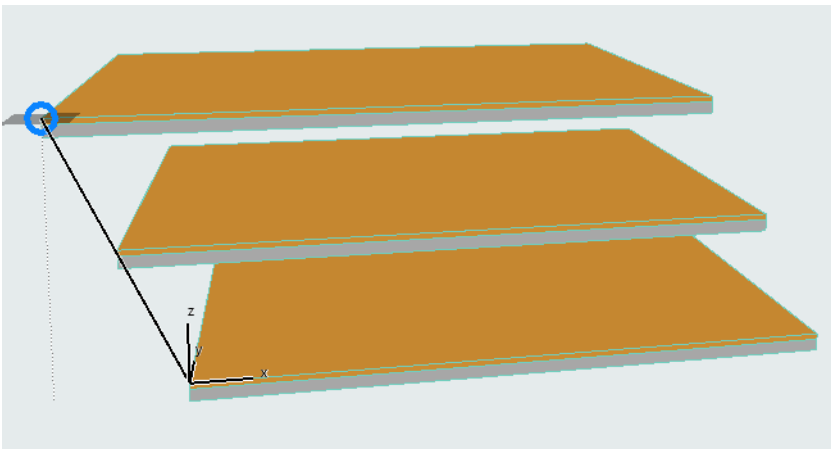


Suppose you want to draw the Curtain Wall so that it is extruded perpendicularly to an input plane which corresponds to the shape of the building. You need an input plane perpendicular to the building's slant.

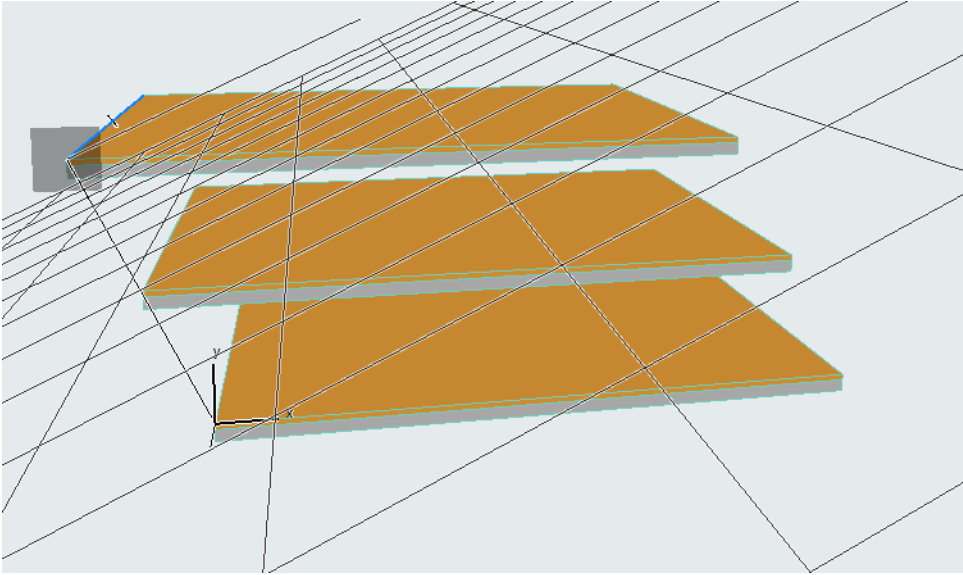
1. Activate the Curtain Wall tool and choose the Single geometry method combined with the Normal Vector input plane method:



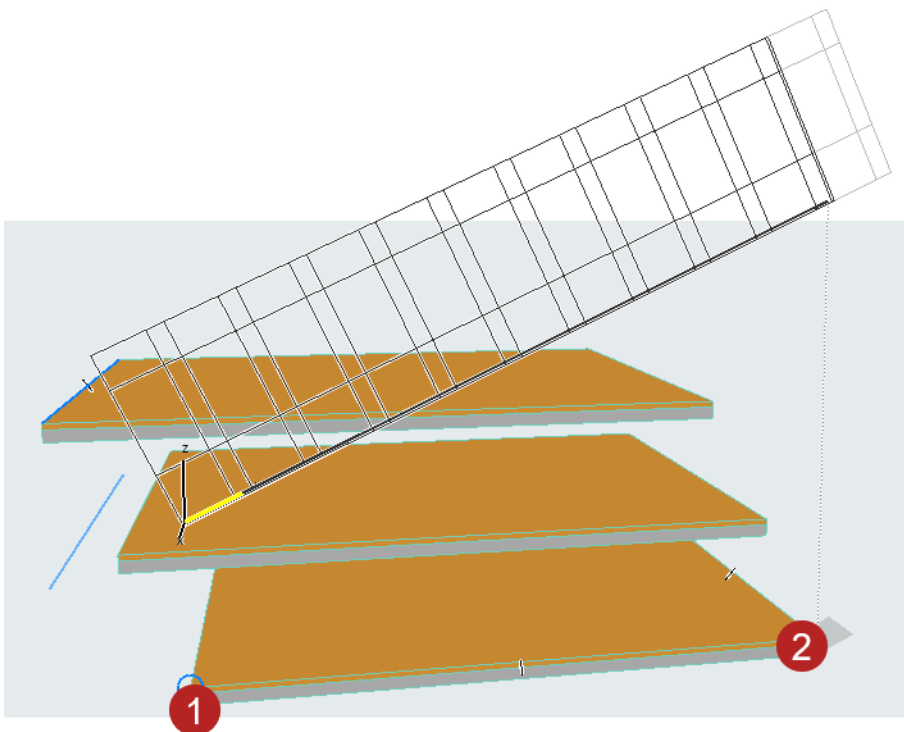
2. In the 3D window, draw a Normal Vector corresponding to the slanted shape of the building (you want the input plane to be perpendicular to this vector). Here, we draw the vector joining the corners of the top and bottom slabs:



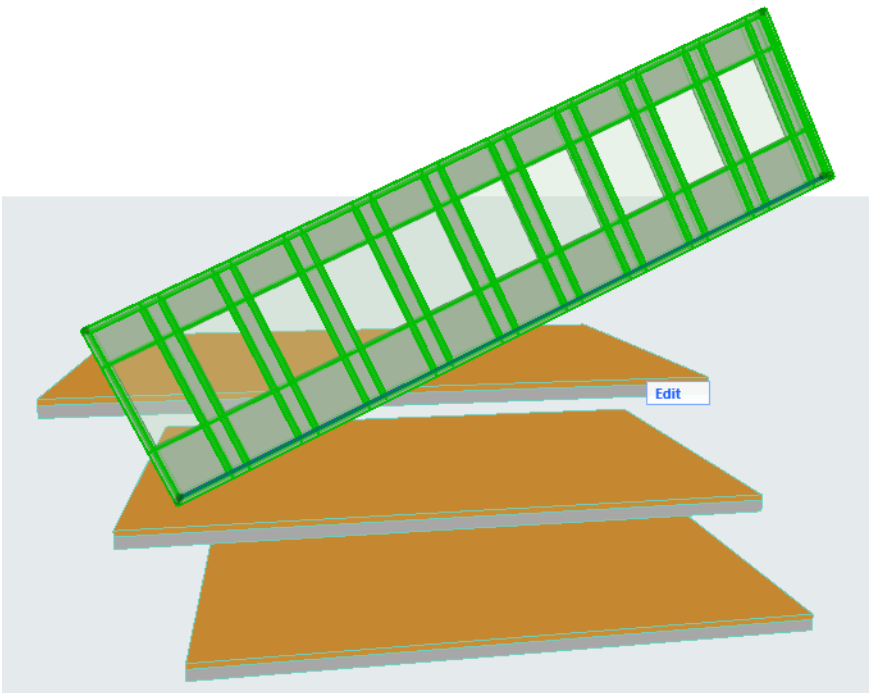
- An input plane appears, which is perpendicular to the vector.



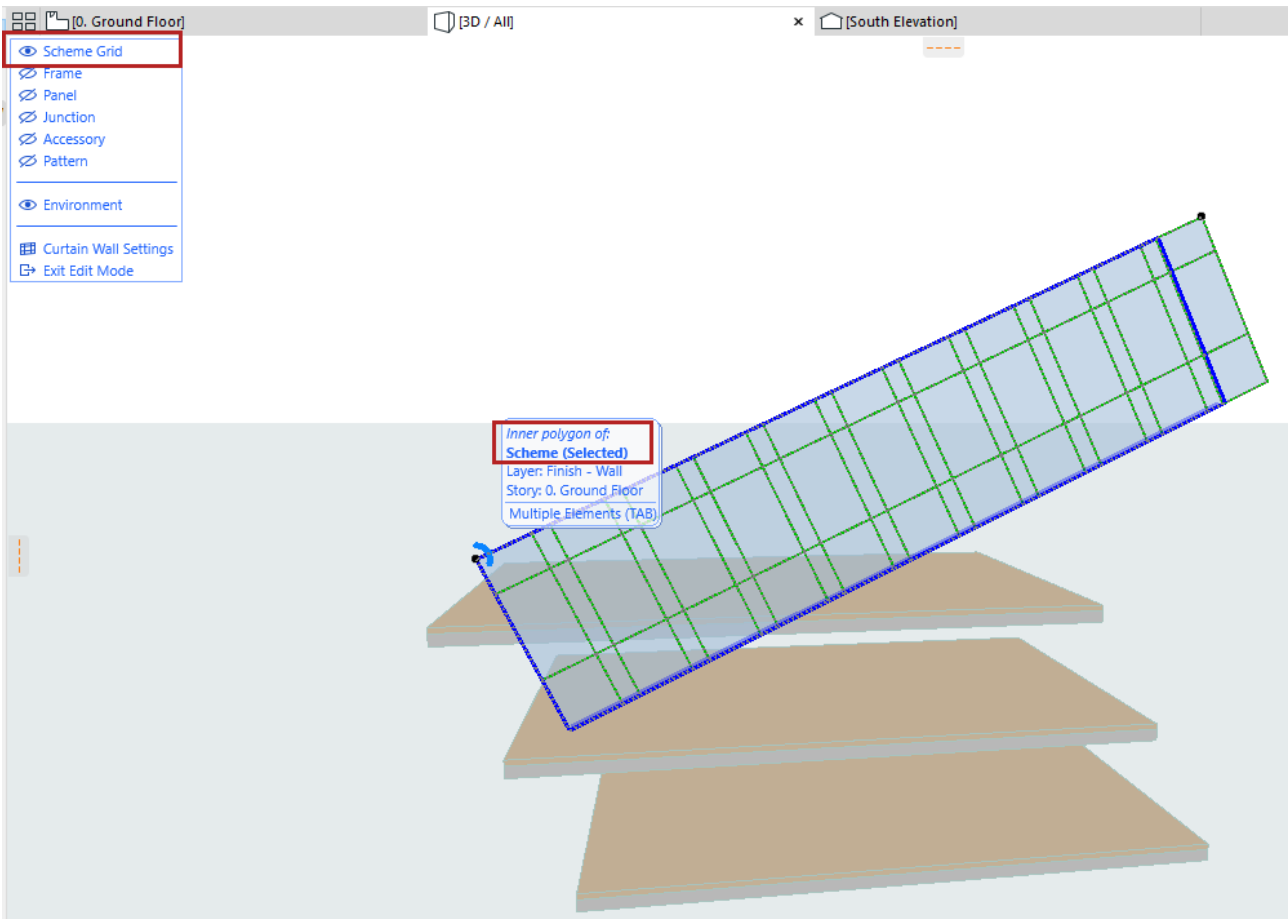
- Click anywhere to place the input plane.
- Now you are ready to draw the input line. The input line (as indicated by the rubber band) is constrained and is projected perpendicularly to the input plane. Click at the two points where you want the Curtain Wall to begin and end: in our case, at the two front corners of any of the slabs.



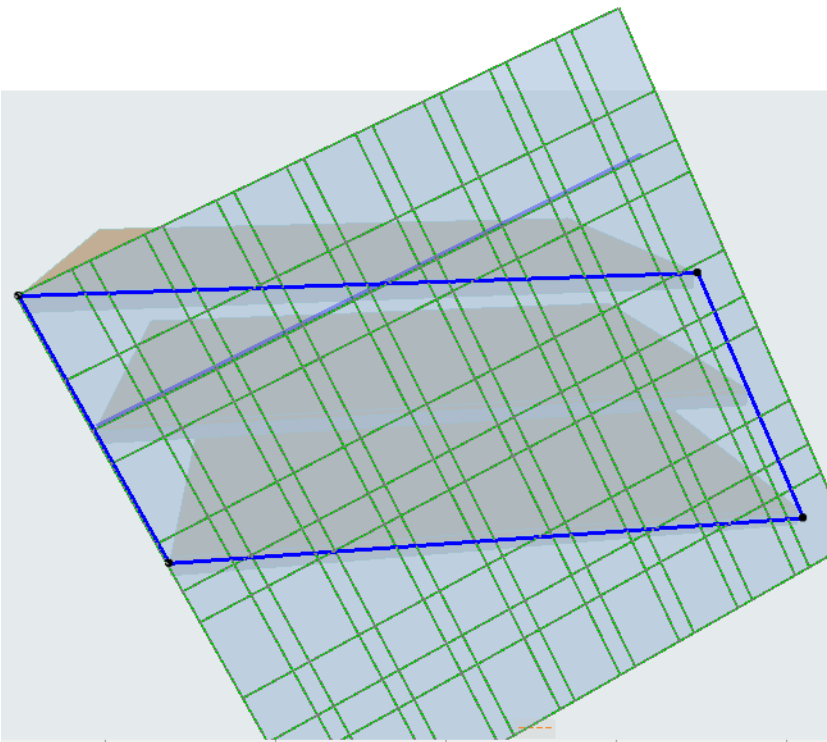
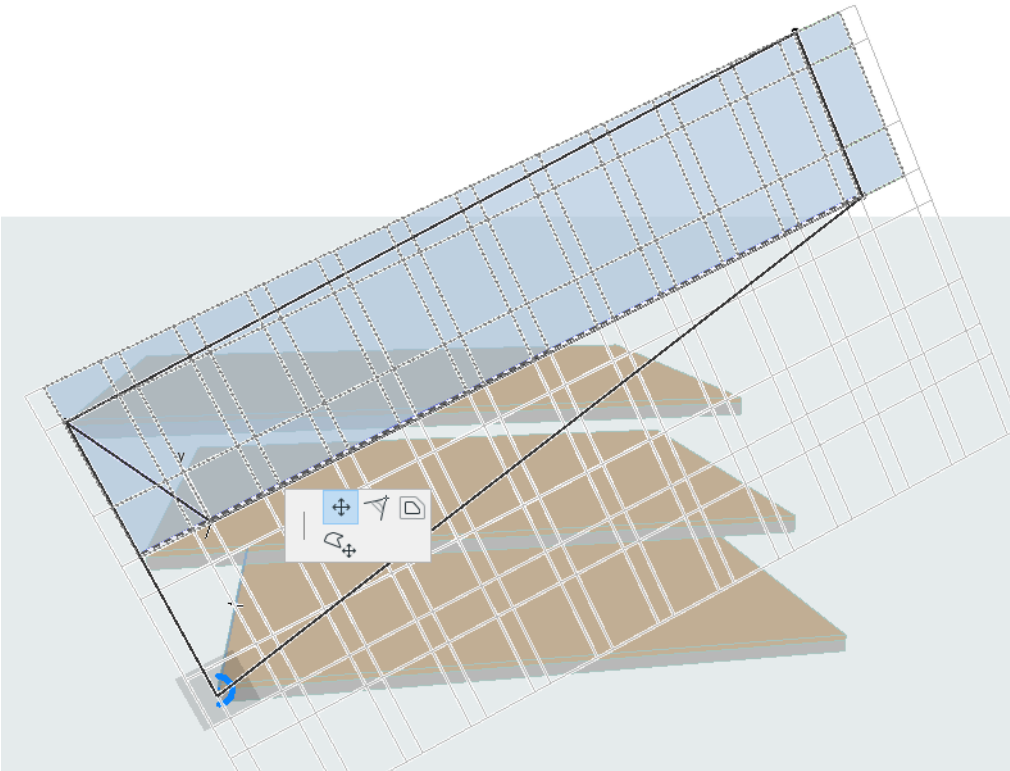
- A Curtain Wall appears, but you still have to edit the Scheme Grid. Select the Curtain Wall and enter Edit Mode.

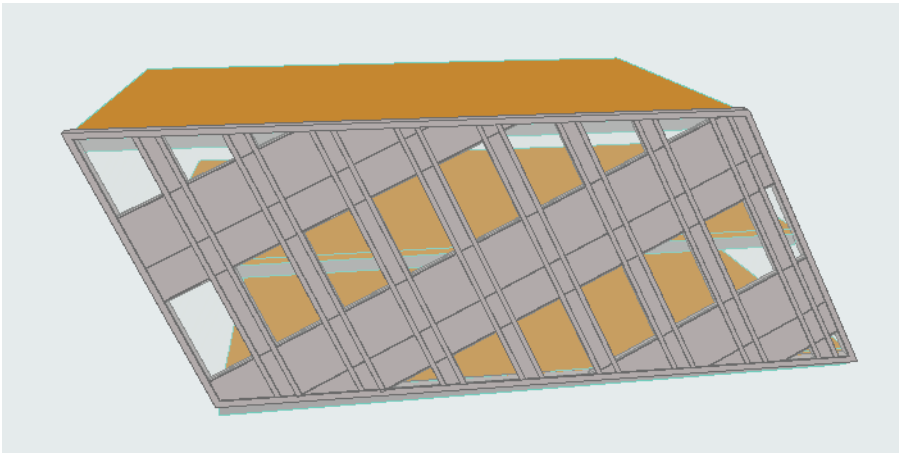


7. In Edit mode, turn off everything except the Scheme Grid.



8. Drag the corners of the Scheme Grid polygon into place - onto the top and bottom front corners of the slabs. (To drag the polygon, make sure you select the inner polygon of the Scheme Grid.)



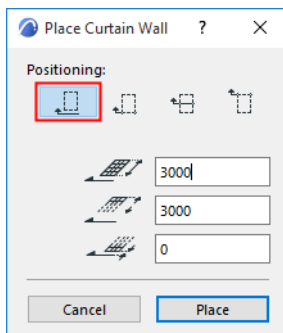


Positioning Curtain Wall in Section

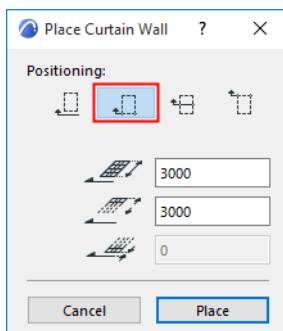
Use this workflow in a Section-type window when extruding a Curtain Wall from a Reference Line.

The input plane in the Section window coincides with the Section Plane.

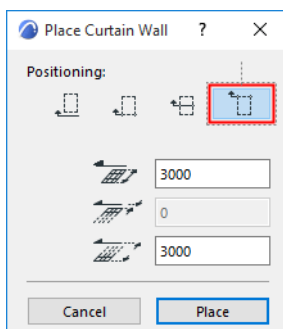
1. Activate the Curtain Wall tool
2. Choose an input method from the Info Box.
3. Draw the Reference Line as usual.
4. the **Place Curtain Wall on Section** dialog box appears. Choose an option:
 - **Freely position** the extrusion starting point relative to the input plane. The first field is the absolute length of the extrusion; the third field is its offset from the input plane. (The sum of the two values is shown in the middle field.)



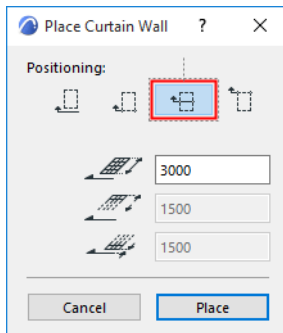
- To extrude the Curtain Wall **“away” from you**, starting from the input plane (Section line): choose the second method and enter the length of extrusion. There is no offset (the Curtain Wall begins right at the input plane).



- To extrude the Curtain Wall **“toward” you**, starting from the input plane (Section line): choose the fourth method and enter the length of extrusion.



- To place the **midpoint of the extrusion** vector at the input plane, choose the third option. Enter the total extrusion length in the first field below.



Model View Options for Curtain Walls

Use Model View Options to define view-based Curtain Wall display.

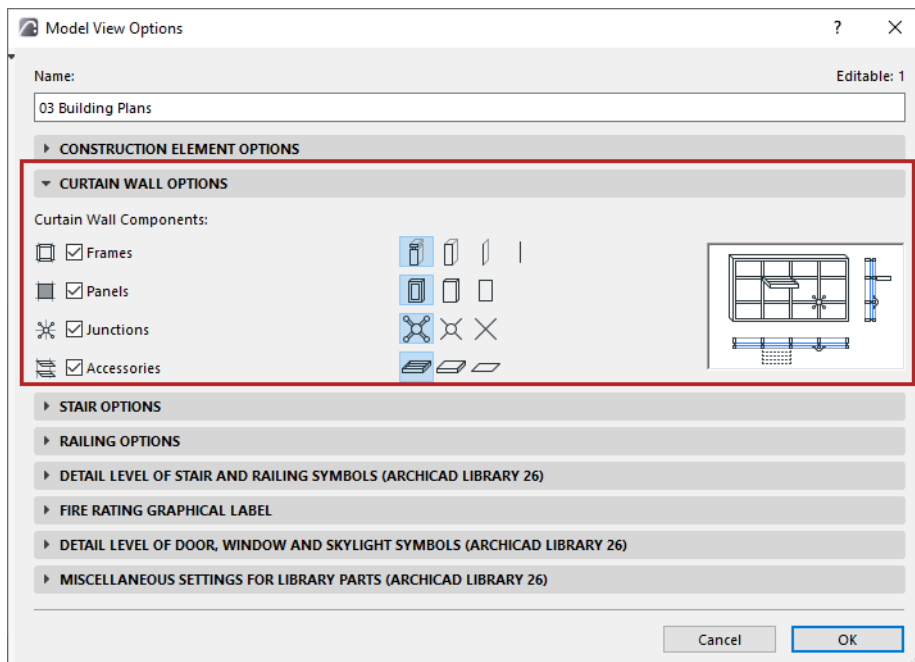
Using Model View Options, show/hide Curtain Wall members separately in each view, and set a separate display preference for each.

Go to **Document > Model View > Model View Options**.

Go to the **Curtain Wall Options** panel:

Show/Hide Components

By default, all Curtain Wall components (Frames, Panels, Junctions, Accessories) are displayed. Uncheck any component if you do not wish to display it in this Model View Options Combination.

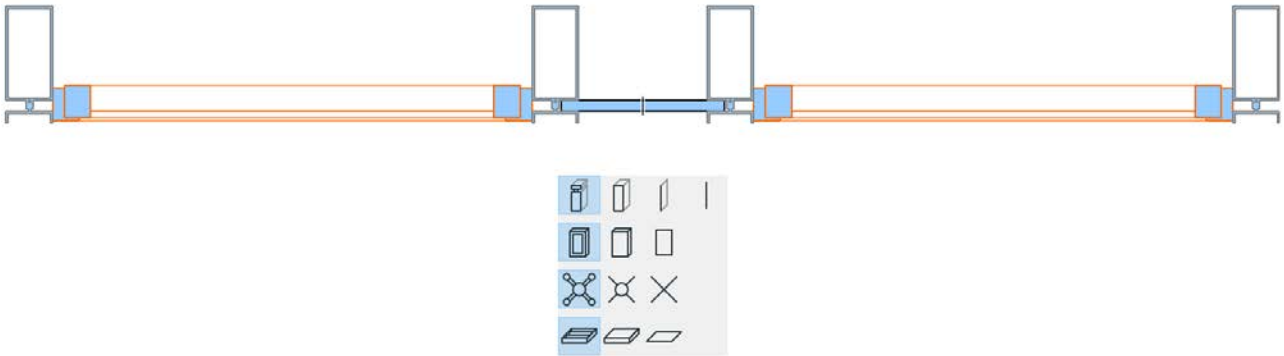


Detail Level Display (by Component)

For each component, click an icon to define its display: Detailed, Simplified, or Schematic.

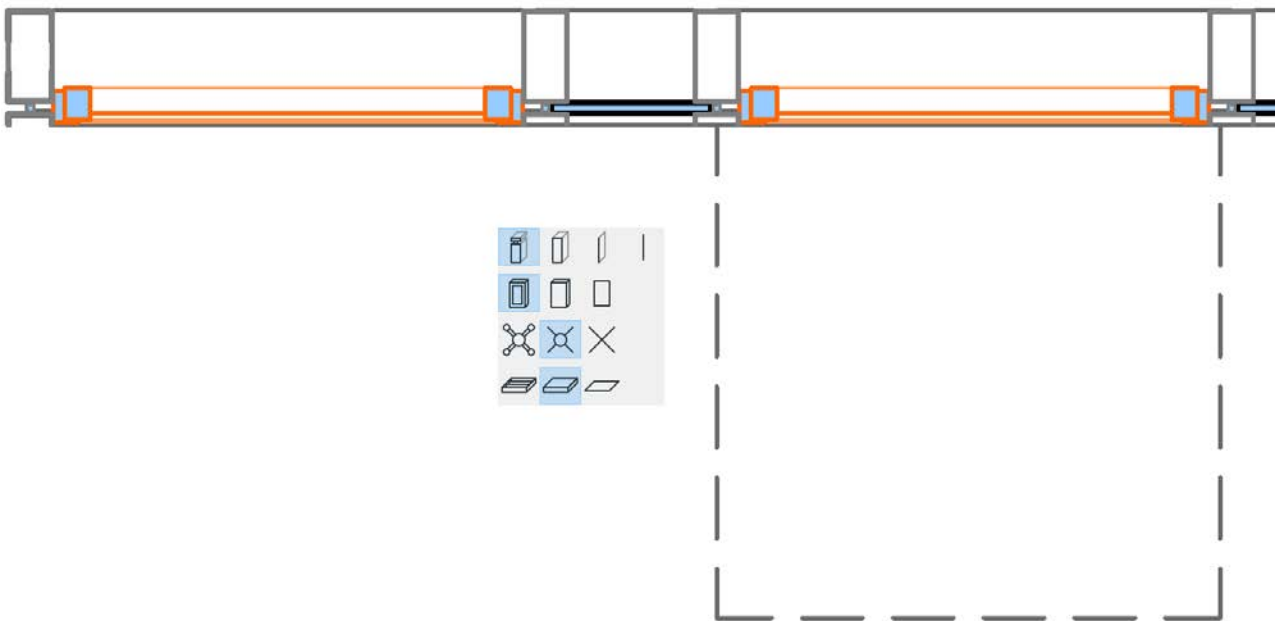
Frames have an additional option: Display with Axis only.

- **Detailed:** Entire geometry (e.g. of profiled components, with intersecting Building Materials) are displayed.



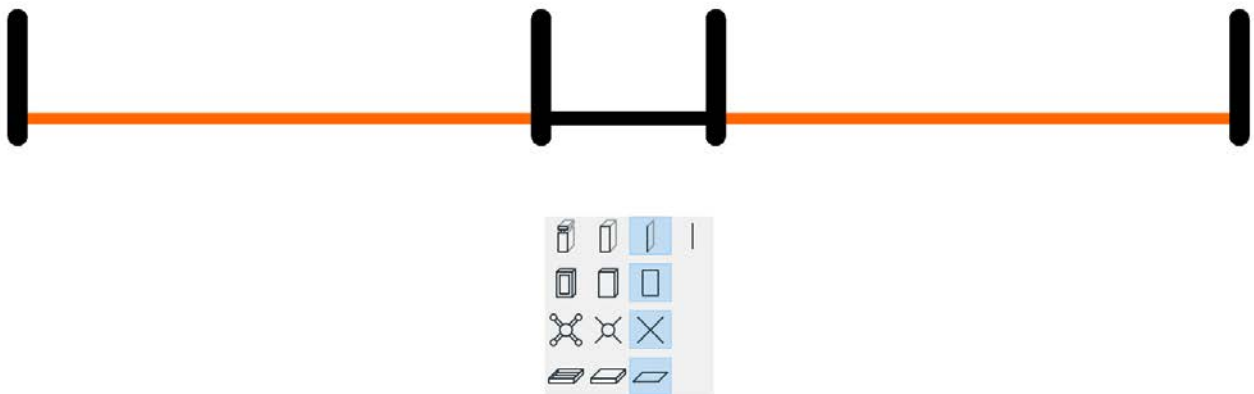
Curtain Wall in Full Detail

- **Simplified:** Displayed as rectangular boxes, using a single Building Material.

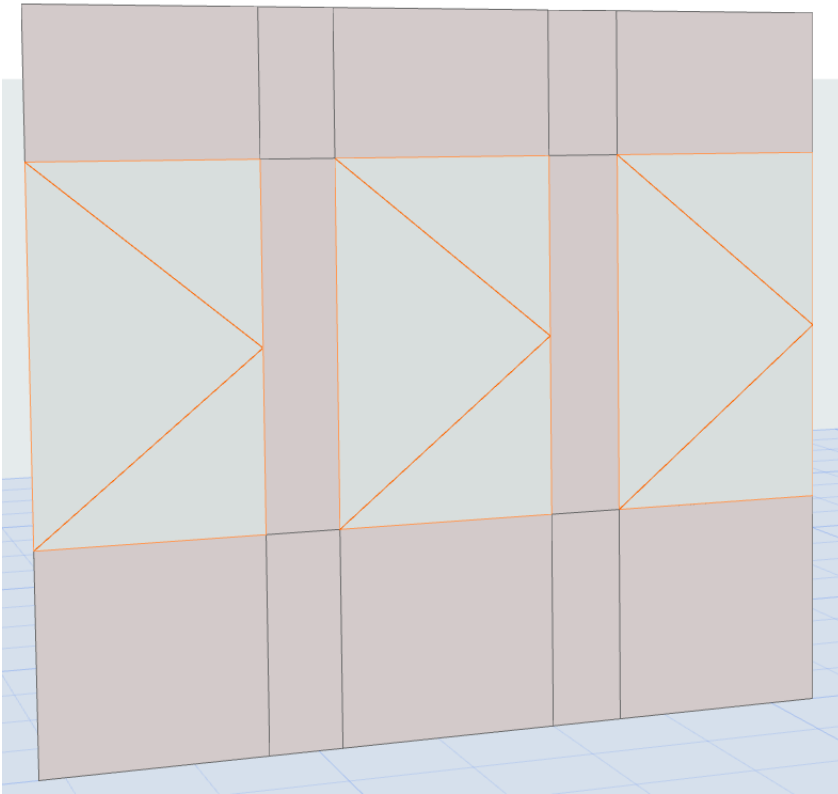
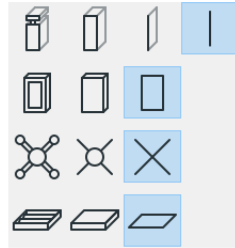


Detailed Frames/Panels, Simplified Accessories

- **Schematic:** Displays the scheme only (Gridlines and Boundary).



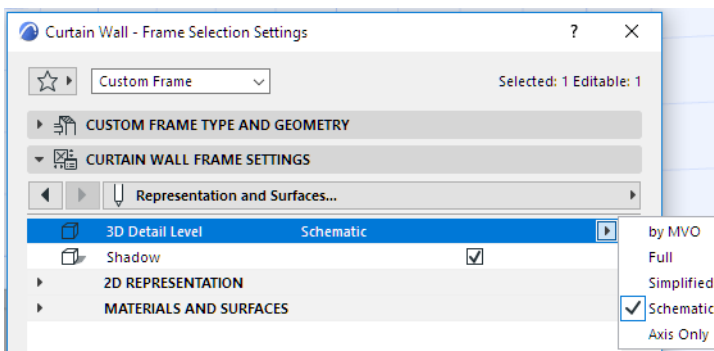
Curtain Wall in Schematic View (Floor Plan)



Curtain Wall in Schematic View (3D/Elevation)

Override MVO Definition

For any individual Curtain Wall member or Member Class, you can override its MVO-defined 3D Detail Level in its component-level Settings dialog:



Curtain Wall Edit Mode

Use **Edit mode** to access and change the individual members of a **selected Curtain Wall**, or to add new custom members to it.

In Edit mode, each Curtain Wall member (Scheme, Frame, Panel, Accessory, Junction) has its own Settings dialog box and its own tool, so you can modify settings and place new members in a selected Curtain Wall. (You cannot create a new Curtain Wall in Edit mode.)

Once you change the parameters of any selected Curtain Wall member in Edit mode, that member is no longer tied to Curtain Wall System Settings; it is a Custom member, and its parameters are locally defined, in its individual tool settings dialog box (e.g. Scheme, Frame, Panel, Junction, and Accessory Settings). Custom parameters are set in Edit mode only.

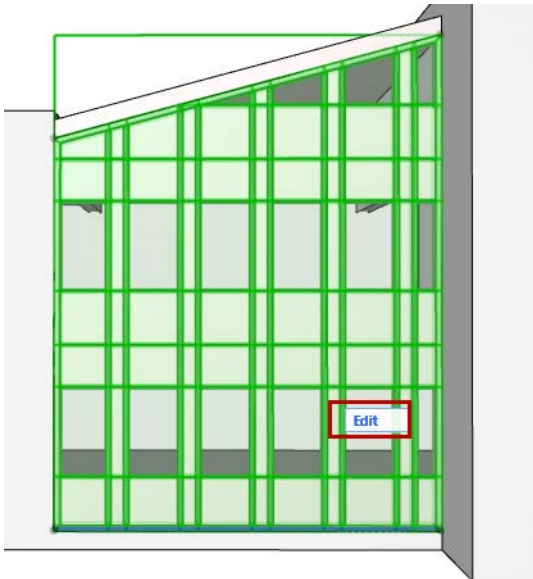
Enter Edit mode from Floor Plan, Elevation, Section and 3D Window. You can remain in Edit mode even when switching between views.

Enter/Exit Curtain Wall Edit Mode

Enter Edit Mode

To enter Edit mode, select a single Curtain Wall in any model view, then do one of the following:

- Click the **Edit** button which appears.
- Use **Design > Modify Curtain Wall > Enter System Edit Mode**.



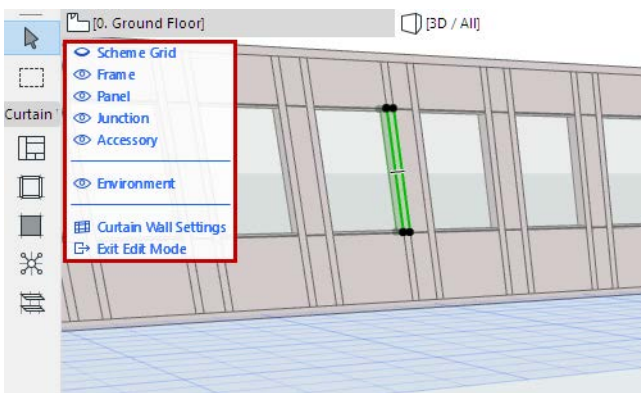
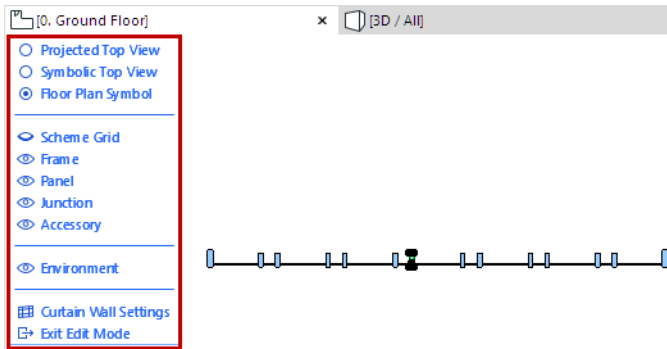
Exit Edit Mode

Do one of the following:

- Click **Esc**
- Click **Exit Edit Mode** from the display palette.
- Use **Design > Modify Curtain Wall > Exit System Edit Mode**

Edit Mode Display Palette

Once you are in Curtain Wall Edit mode, the regular Archicad toolbox is replaced by a special Curtain Wall toolbox and display palette



Edit Mode Palette - in Floor Plan and 3D Windows

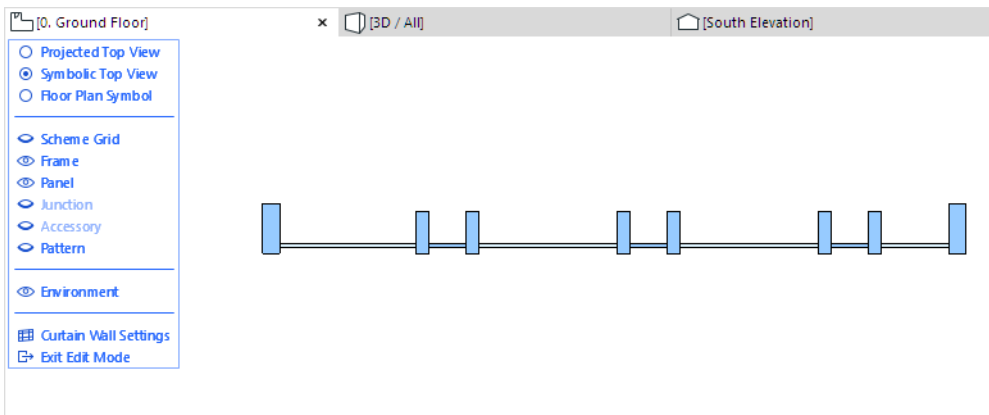
View Options in Edit Mode

When you enter Edit Mode from the Floor Plan, the Curtain Wall uses **Floor Plan Symbol** view option by default. It looks the same as in the regular Floor Plan view.

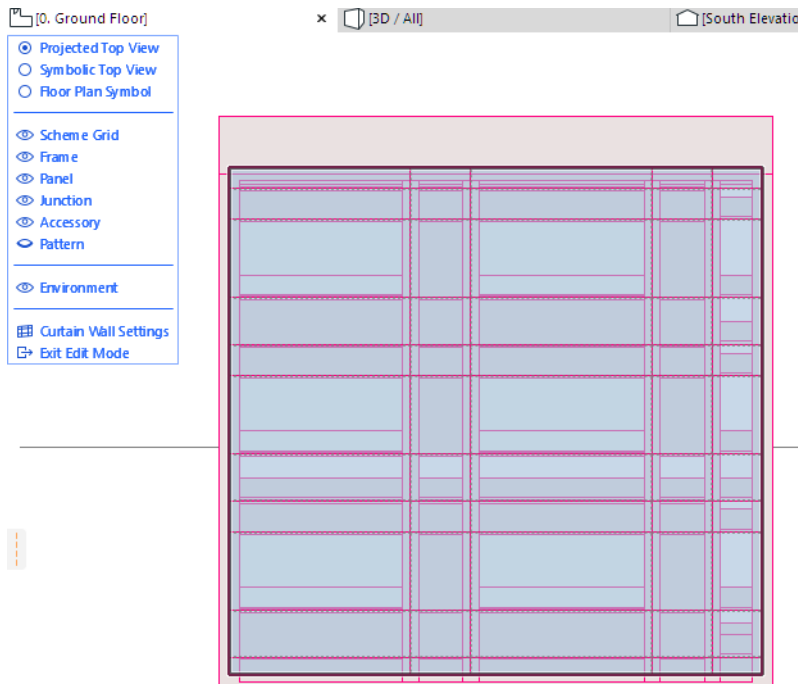
Note: These Edit Mode viewing options are for editing purposes only.

To get a better view of your editing results, you may prefer a different view options:

Symbolic Top View



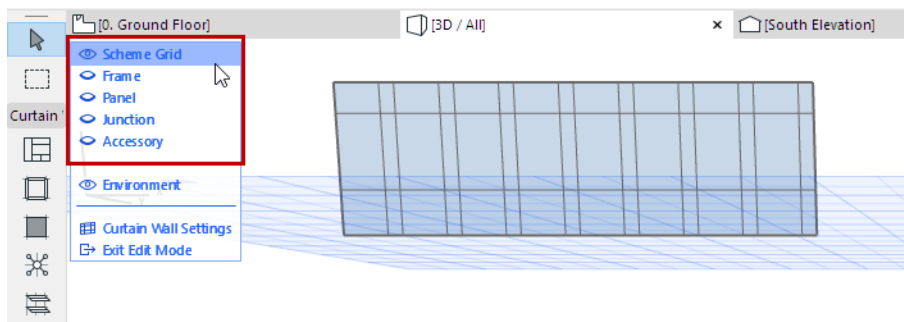
Projected Top View



Show/Hide Items in Edit Mode

Use the Edit mode display palette to show/hide Curtain Wall elements as needed (click the eye icons to open or close them), even in the middle of an editing operation.

To show a particular Curtain Wall Member while hiding the rest: Right-click the item you want to remain visible (e.g. “Scheme Grid”). The other members are hidden.



Show/Hide Environment

“Environment,” in this display palette, refers to the rest of the building model, which you can show or hide as needed to assist in editing. Preselection tags will identify the Environment’s model elements in dimmed form, but you cannot edit them.

Note: If you have trouble identifying the edges of these model elements shown as the Environment, try checking **Show Contours** in **View > 3D View Options > 3D Styles**.

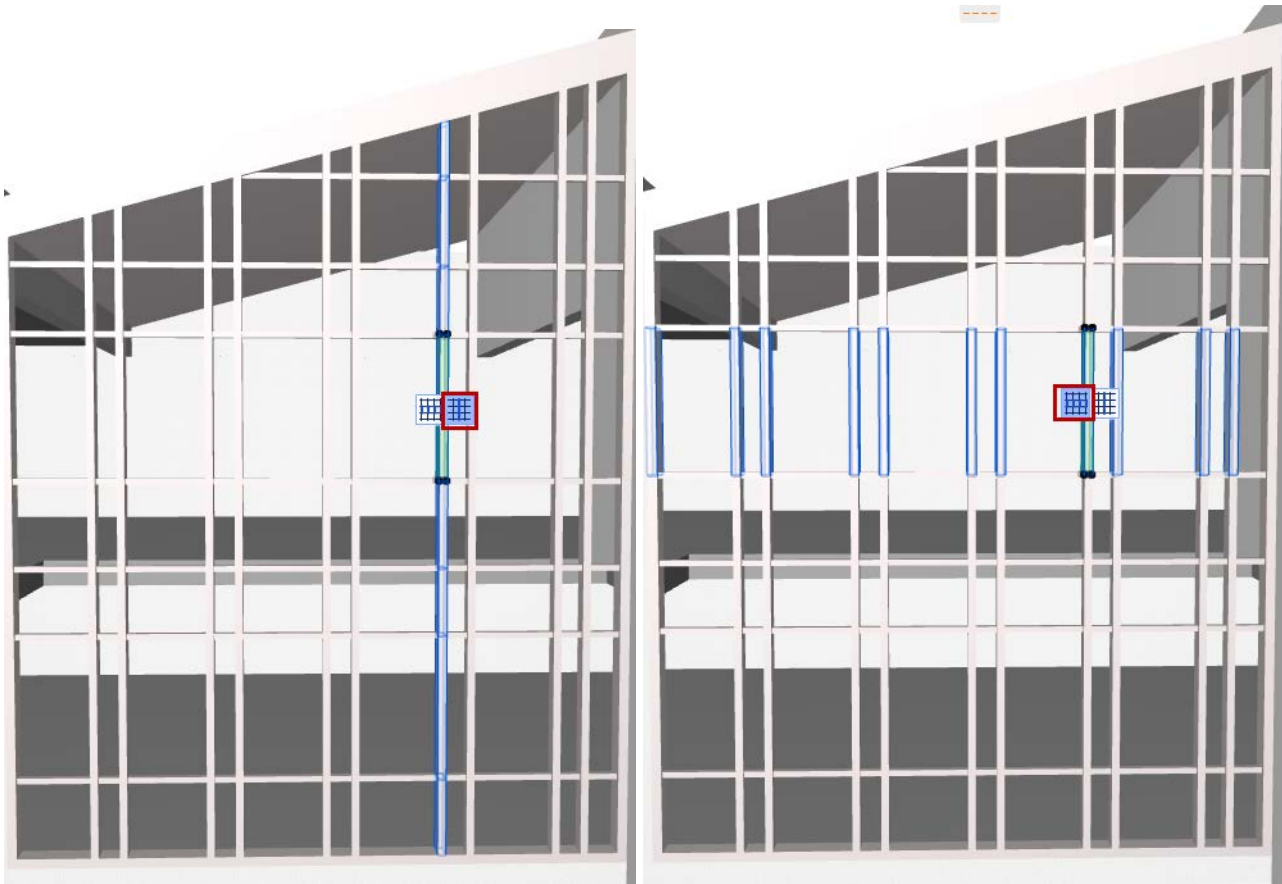
Edit Curtain Wall System Settings

To edit the System-level Settings without leaving Edit mode, click **Curtain Wall Settings** at the bottom of the display palette.

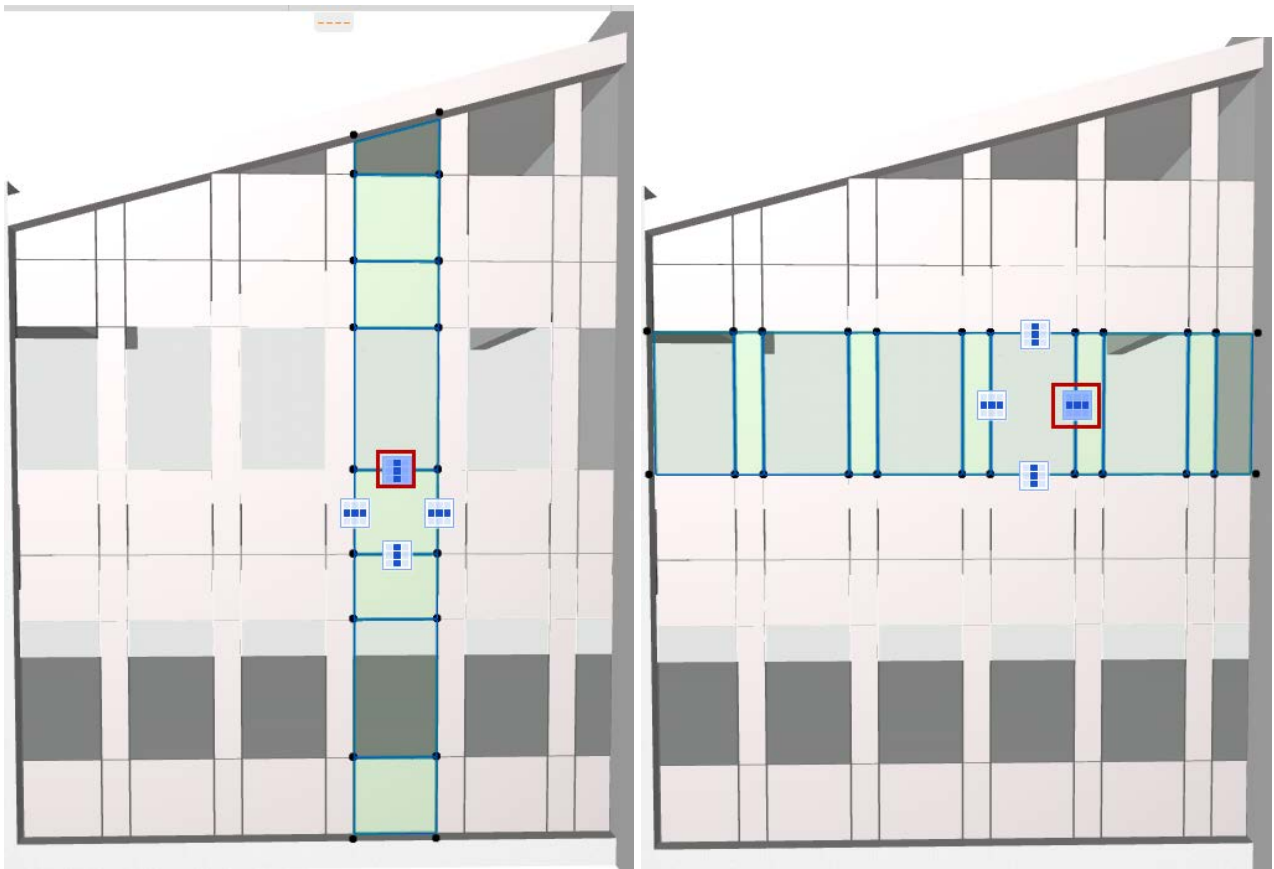
Multi-selection of Frames and Panels

When you select a single Frame or Panel of a Curtain Wall in Edit mode, a “Grip” appears with shortcuts for selecting multiple Frames or Panels.

- Select all Frames/Panels along the selected Row/Column.



Select All Frames Along Grid Column/Row



Select All Panels Along Grid Column/Row

[Watch the Video](#)

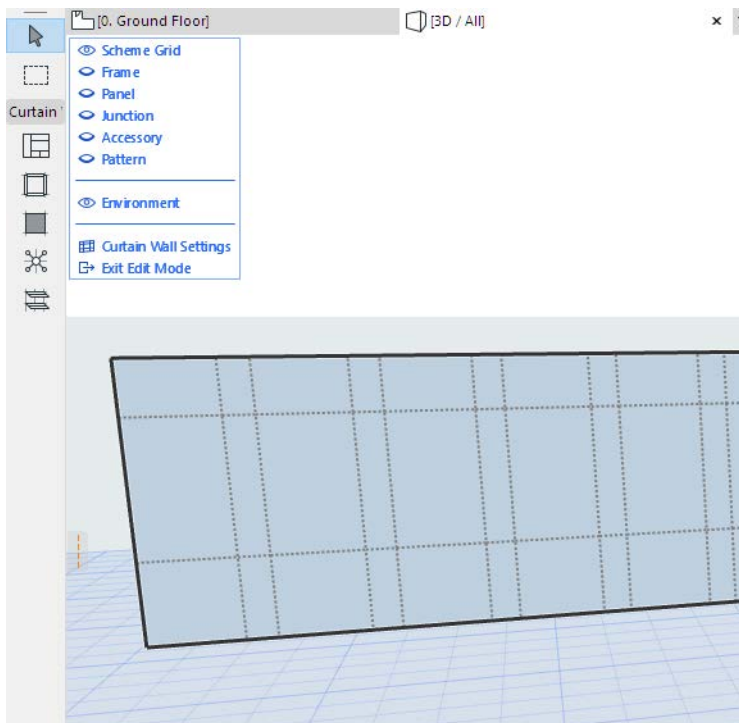
Edit Grid and Grid Lines in Edit Mode

In Curtain Wall Edit mode, the available operations with the Scheme Grid include:

- Move or delete a selected Grid Line
- Rotate all the Grid Lines of one of the Grid directions
- Add a new Grid Line
- Rotate or Move the Scheme Grid as a whole

Note that moving or rotating a Scheme Grid or Grid Lines means that the Frame(s) assigned to that Scheme Grid or Grid Line are moved or rotated along with it.

To make editing the Scheme Grid easier in Edit mode, make sure that the Scheme is visible (and turn off the other members' visibility as needed).



Topics in this section:

[Move Grid Line](#)

[Move Scheme Grid](#)

[Rotate Parallel Grid Lines](#)

[Rotate Entire Scheme Grid](#)

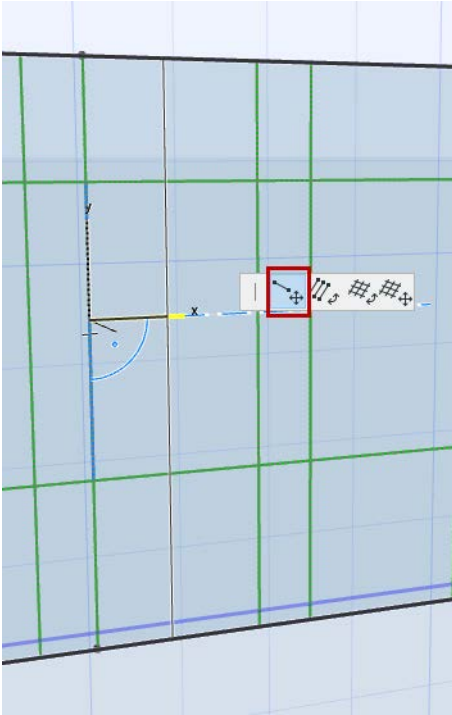
[Delete a Grid Line](#)

[Add a New Grid Line](#)

Move Grid Line

Enter Edit mode.

1. Select an individual Grid Line with the Mercedes cursor.
2. Choose the **Move Grid Line** command from the appearing pet palette:
3. Drag the Grid Line to its new position (it will remain parallel to its original position).

4. Click to place.**Move Scheme Grid**

Enter Edit mode.

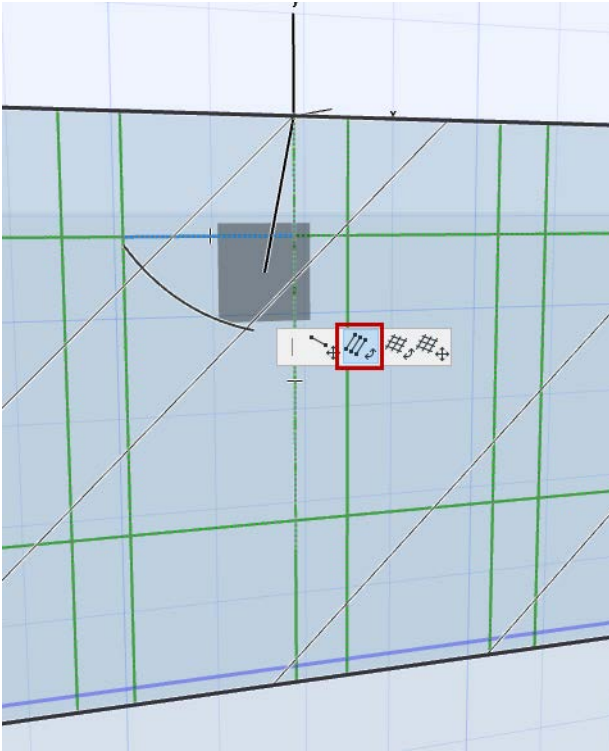
1. Select either a Grid Line or the entire Grid.
Note: To select the entire Scheme, click any node.
2. From the pet palette, use the **Move Grid** command.

Rotate Parallel Grid Lines

Enter Edit mode.

To rotate all the Grid Lines of one direction (e.g. all the Grid Line rows):

1. Select *one* of the Grid Lines you want to rotate.
2. From the pet palette, choose the **Rotate Parallel Grid Lines** command.



Note: If the Curtain Wall is Curved or Chained, you cannot rotate the column grid lines, just the rows.

3. Draw a rotation vector or enter a rotation angle in the Tracker.
4. Click to complete the transformation.

Rotate Entire Scheme Grid

Note: Not available for curved or chained Curtain Walls.

Enter Edit mode.

1. Select one Grid Line or select the entire Grid.

Note: To select the entire Scheme, click any node.
2. From the pet palette, choose the **Rotate Grid** command.
3. Draw a rotation vector or enter a rotation angle in the Tracker.
4. Click to complete the transformation.

Delete a Grid Line

In Curtain Wall Edit mode: Select the Grid Line and press Delete.

You cannot delete the Grid as a whole.

Note: Deleting a Grid Line deletes the Frame on it. The Panel(s) will change in size to conform to the resulting new Frame pattern.

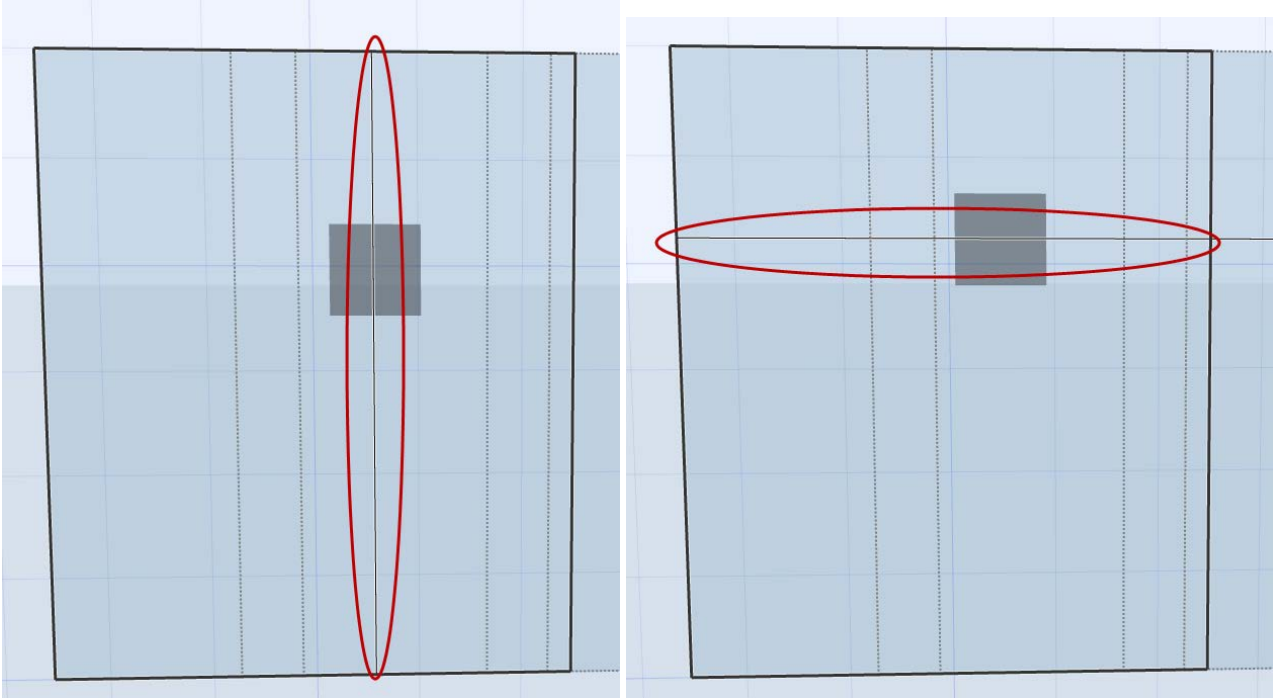
Add a New Grid Line

Enter Edit mode.

1. Activate the Scheme tool.
2. In the Info Box, select the Grid Input method:



Click on the place at which you want to place a grid line. Move the cursor around the point you clicked: use the feedback and click either of the orthogonal grid lines to place it:



Graphical Editing of Curtain Wall

To edit a placed Curtain Wall graphically, select it in Section or the 3D window, and use the pet palette commands. These are analogous to the commands used to graphically edit walls. (Some commands are not available in every window type.)

Stretch Height/Extrusion

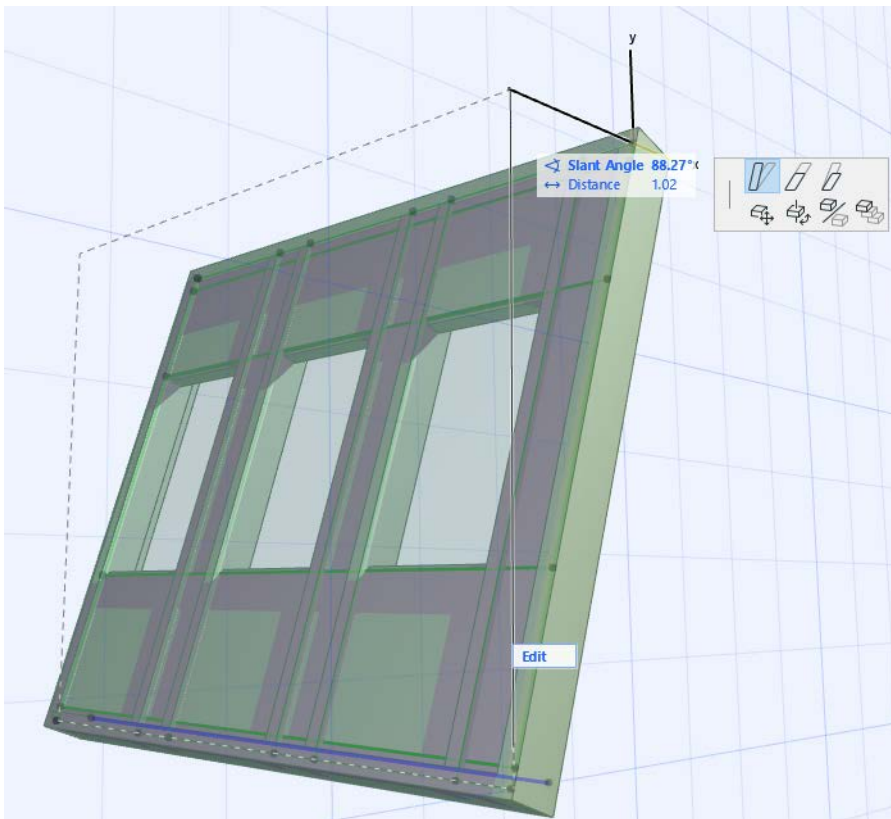
As you change the Curtain Wall height, its thickness remains the same but its slant angle is adjusted accordingly. The Curtain Wall's Reference Line stays in place.

Stretch Slanted Height

The Curtain Wall is stretched along its plane; its slant angle remains constant.

Modify Angle

The Tracker fields reflect the new slant angle as well as the horizontal distance that the selected Curtain Wall edge is moved. The Curtain Wall is slanted, pivoting around its Reference Line.



Another graphical option is to free-rotate the Curtain Wall, available in the 3D and in Section/Elevation-type windows.

Edit Curtain Wall Reference Line

Select the Curtain Wall, then click the Reference Line to edit it.

You can only select and edit the Reference line if it is on a plane that is perpendicular to the model view.

(In Curtain Wall Edit mode, make sure that Scheme visibility in Edit mode is switched to On, since the Reference Line is part of the Scheme.)

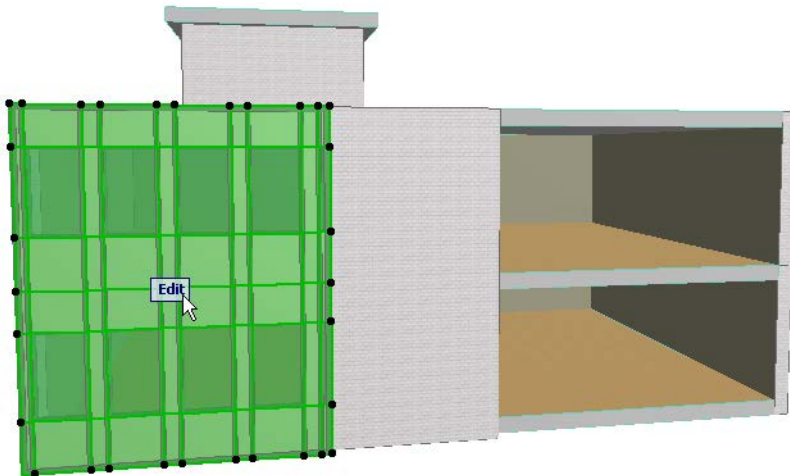
Editing the Reference Line will change the shape of the Curtain Wall Base Surface. (Grid Lines, Boundary, Panels and Frames will be modified accordingly.)

You can Flip the Curtain Wall orientation relative to the Reference Line, using the **Flip** command in the Info Box.

Add Additional Curtain Wall Boundary

A single Curtain Wall can include several Boundaries. The advantage is that you can create several separate shapes or sections within a single Curtain Wall, yet use an identical Scheme for all of them.

1. Draw a straight Curtain Wall on one of the surfaces using the usual methods.



2. Select the Curtain Wall and go into Edit mode. Make sure that the Environment display option is switched On.
3. Select the Scheme tool and the Boundary Input method.



Now click on the desired Curtain Wall segment in which you wish to draw.

4. Start drawing a new Boundary. You are drawing on the Base Surface of the currently edited Curtain Wall.

Note: Because the Base Surface of any Curtain Wall can be infinite in one or more directions, you always have “space” to draw any number of new Boundaries while still working on the same single original Curtain Wall.
5. Try editing a selected grid line, or edit the Panel pattern. Note that both “parts” of the Curtain Wall are handled in sync, because they are built upon a single Base Surface and a single Scheme.

Curtain Wall Junctions

A Curtain Wall Junction is an optional structure that serves to connect panels to each other, to support or (more often) to substitute for Frames, resulting in a seamless, more lightweight structure than the traditional Framework.

Junctions are placed at one or more grid points. Moving the grid lines will also move the Junction.

When a Junction is selected in Edit mode, you can

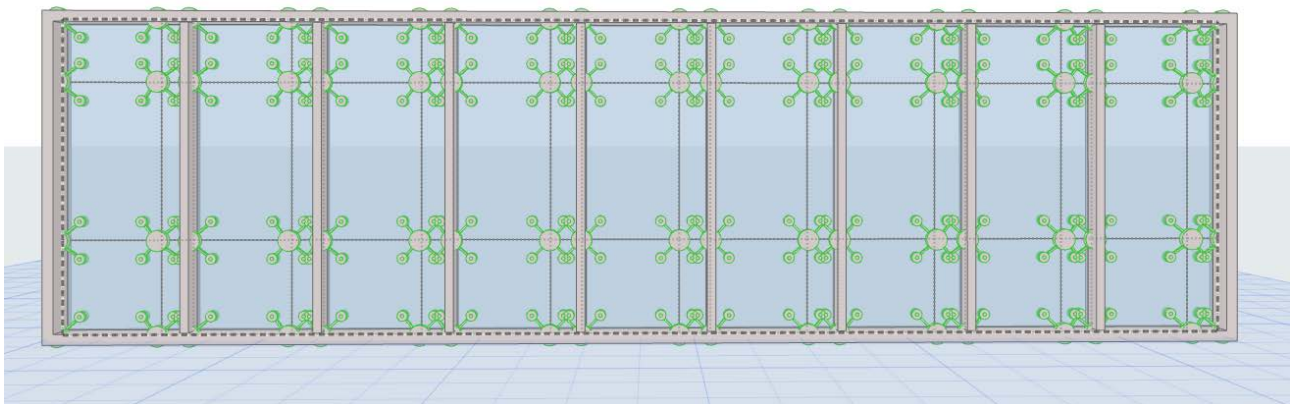
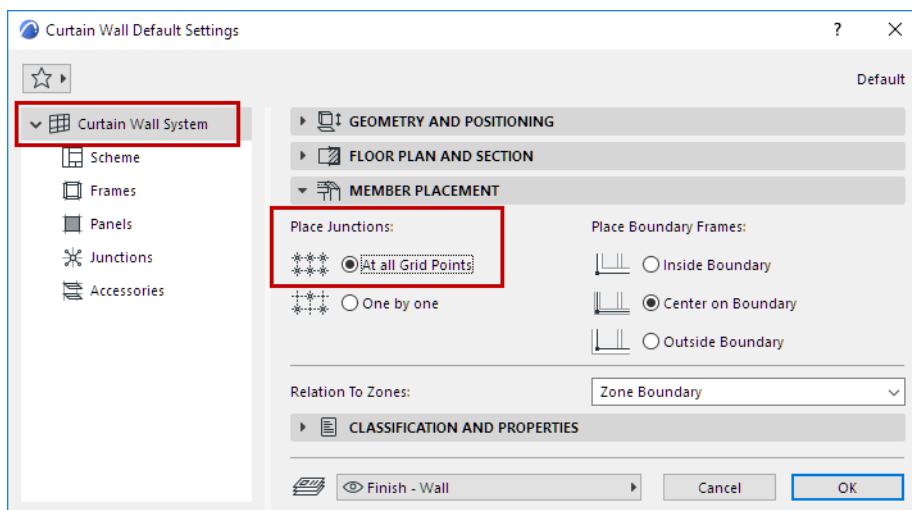
- Open Junction Selection Settings and change them
- Drag the Junction to another panel connection point (3D Edit mode only)
- Delete the Junction

Junctions are placed on the inside of the Curtain Wall by default. You can flip the Junction to the opposite side of the Curtain Wall (use the Flip button in Junction Tool Settings or Info Box).

Place Junctions at All Grid Points

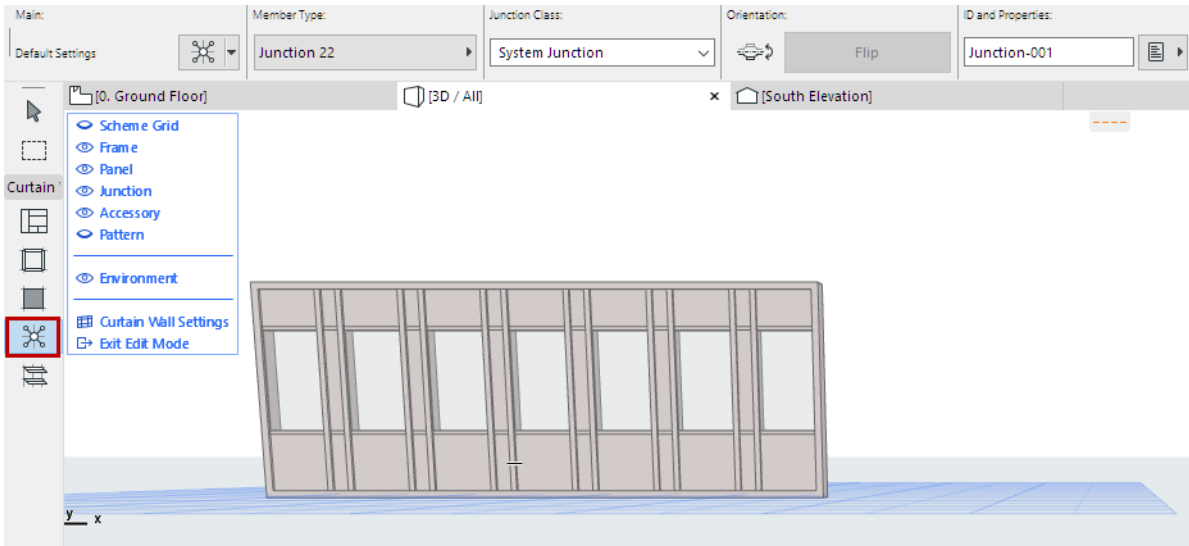
Junctions can be placed either one by one or automatically at every intersection.

To place them at all Grid points, choose this option under Member Placement in Curtain Wall System Settings.

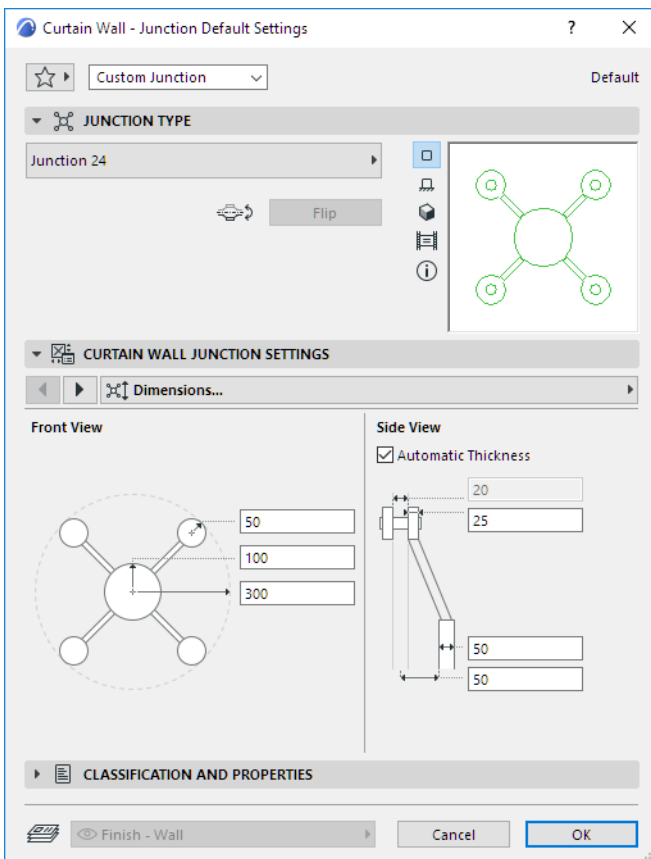


Add New Junction to the Curtain Wall

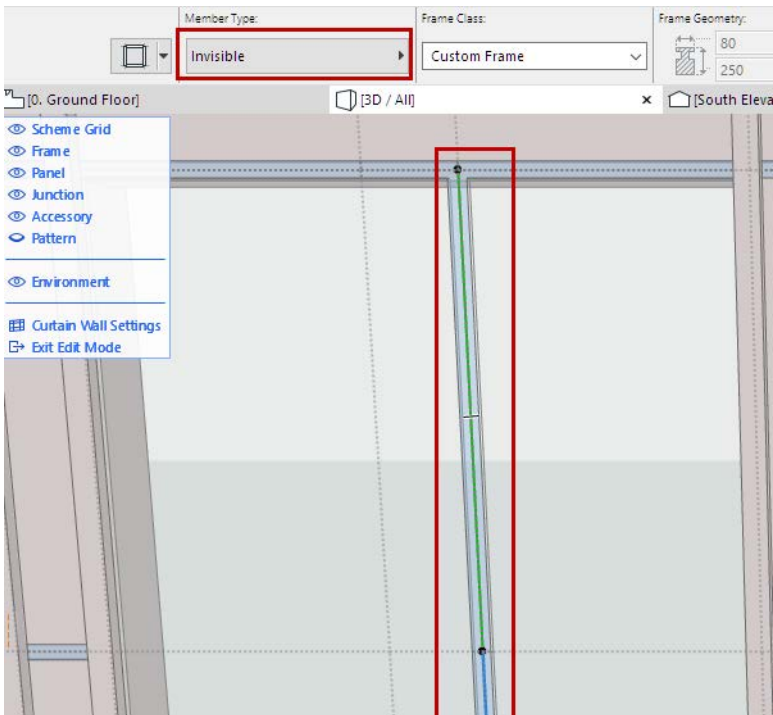
1. Enter Curtain Wall Edit mode from the 3D window.
2. Activate the Junction Tool.



3. To use Custom parameters, open the Junction Settings page and adjust the parameters as needed.

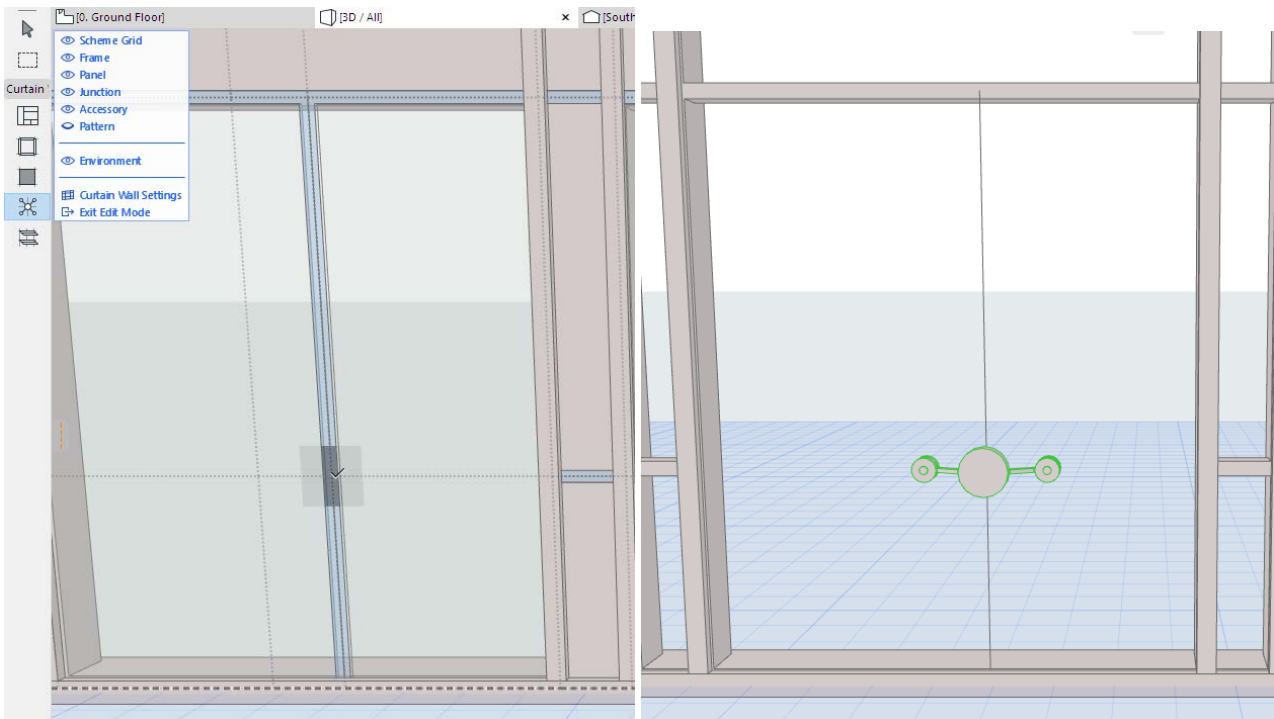


- To use the Junction as a substitute for Frames: select the Frames at a desired panel connection point and switch them to “Invisible” type.



- With the Junction tool active, click the hotspot in the middle of the connection.
- Click to define which side of the Curtain Wall you want to place the Junction. It will be placed at the nearest panel connection point to your click.

Note: To reverse its orientation, select the Junction and click **Flip** from the Info Box.



Curtain Wall Accessories

A Curtain Wall Accessory is an optional non-load-bearing member - such as a sun shade, a decoration strip or a canopy - attached to the Curtain Wall at any Frame.

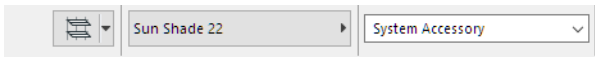
In Archicad, accessories can be placed one by one, in Curtain Wall Edit mode only, using the Accessory Tool.

An Accessory can be configured in Edit mode with custom properties, but you can also set Accessories to use the System-level Accessory settings from Curtain Wall Settings.

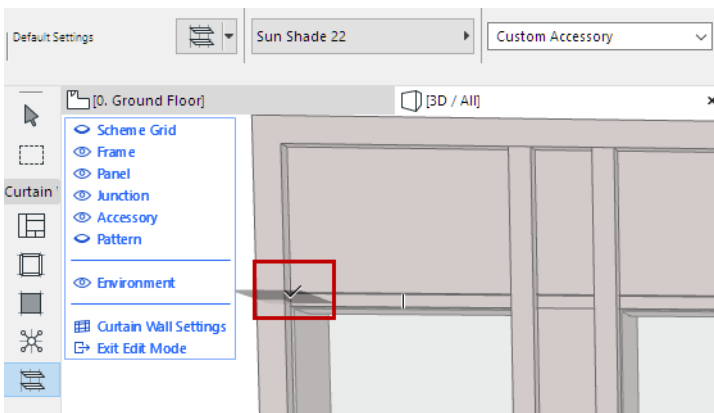
Moving Grid Lines or Frames will move the Accessory along with them.

Add New Accessory to Curtain Wall

1. Enter Curtain Wall Edit mode in the 3D window.
2. Activate the Accessory Tool.
3. Use the Info Box or Accessory Settings to define the Accessory type and Class.

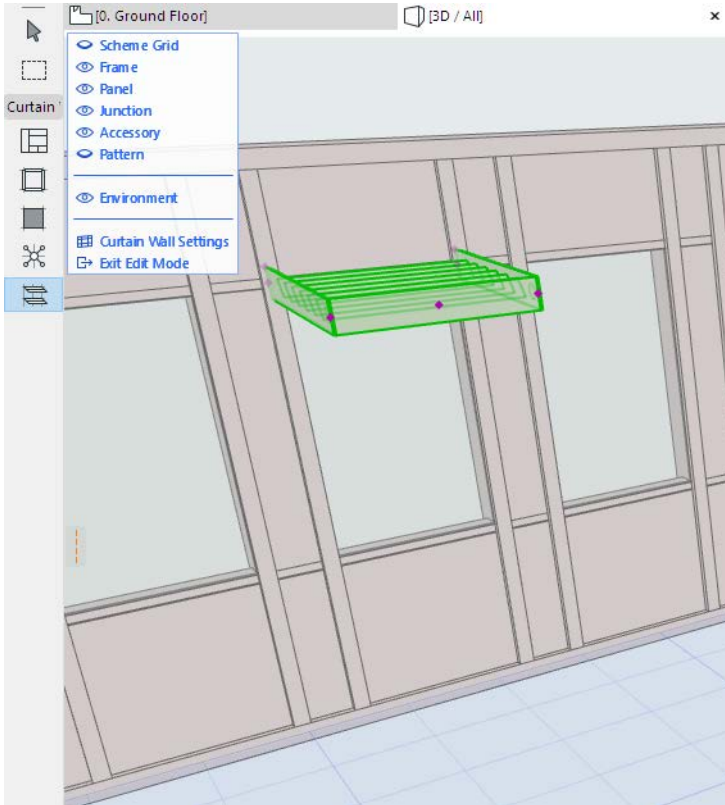


4. Click on a Frame or a Frame endpoint - the endpoint closest to your click is the Accessory startpoint.



5. Drag the cursor and click to define the endpoint.

6. Click with the eyeball cursor to define its direction: outside or inside the Curtain Wall.



Customize an Individual Accessory

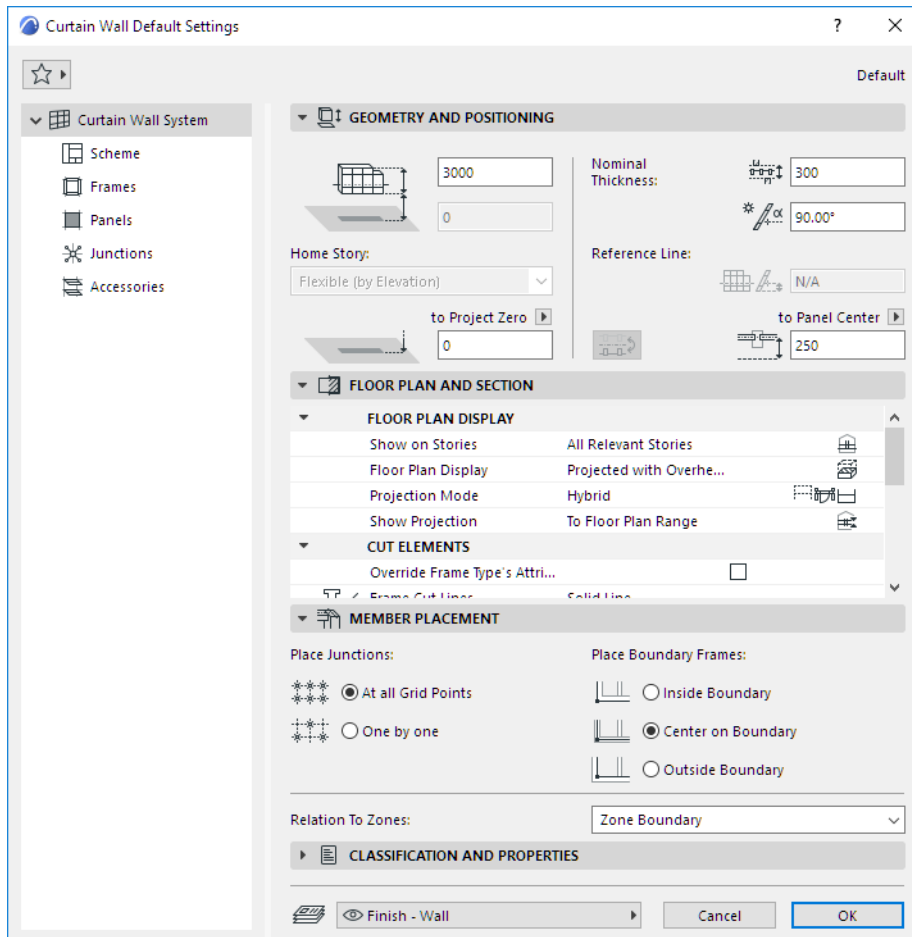
When an Accessory is selected in Edit mode, you can

- Open Accessory Selection Settings and change it
- Drag the Accessory onto another Frame (in 3D Edit mode only)
- Flip the Accessory to the opposite side of the Curtain Wall (Use the “Flip” button in Accessory Tool Settings or the Info Box)
- Delete the Accessory

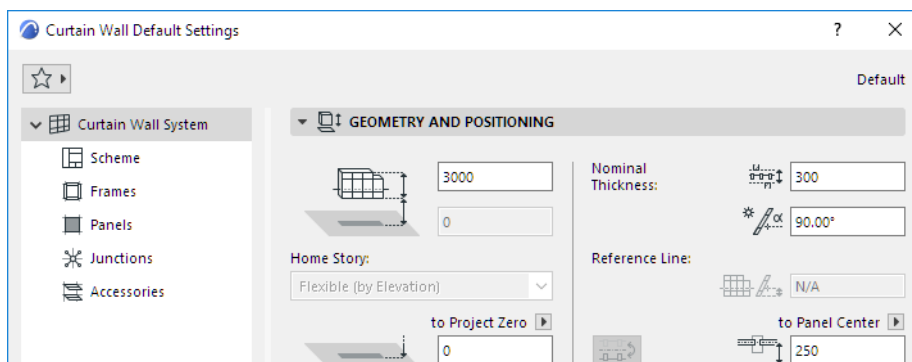
Set the visibility to make your work easier (for example, turn off the visibility of everything except for Accessories, Frames and the Scheme.)

Curtain Wall System Settings

Use System-level editing to set up or edit the Curtain Wall element as a whole, using a single dialog box. Open **Curtain Wall Default Settings** by double-clicking on the Curtain Wall Tool.



Geometry and Positioning Panel



- Curtain Wall Height
- Bottom Offset to Home Story
- Bottom Elevation [to Reference level]

For a Curtain Wall with a non-horizontal Reference Line: elevation at the Curtain Wall's lowest point.

- **Home Story:** The Curtain Wall's reference line is physically linked to its Home Story. If you later modify the Home Story's position (e.g. redefine the floor level), the Curtain Wall will change its position.
If you change an element's elevation so that its reference line is moved to a different story, you have the option to make the Home Story change to match the element's new location:

[See Change Home Story by Elevation.](#)

Nominal Thickness

Distance from the Reference Surface to the opposite side of the Curtain Wall.

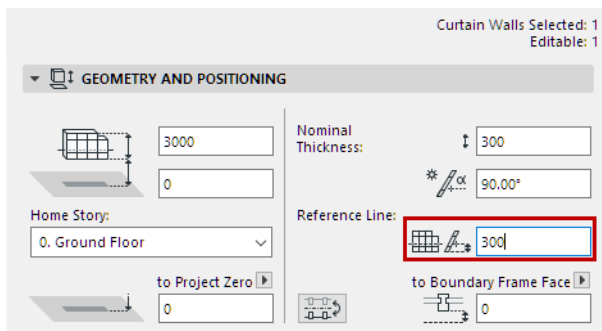
This Nominal Thickness value is used by elements connected to the Curtain Wall: Adjoining Zones and connecting Walls will extend to the Nominal Thickness by default.

[See also Join Wall to Curtain Wall.](#)

Slant Angle

Not available for multi-segment Curtain Walls.

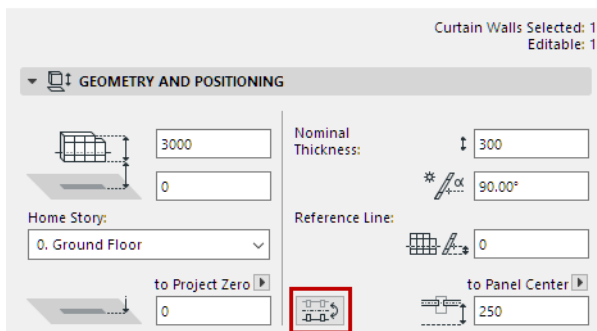
Vertical Reference Line Offset



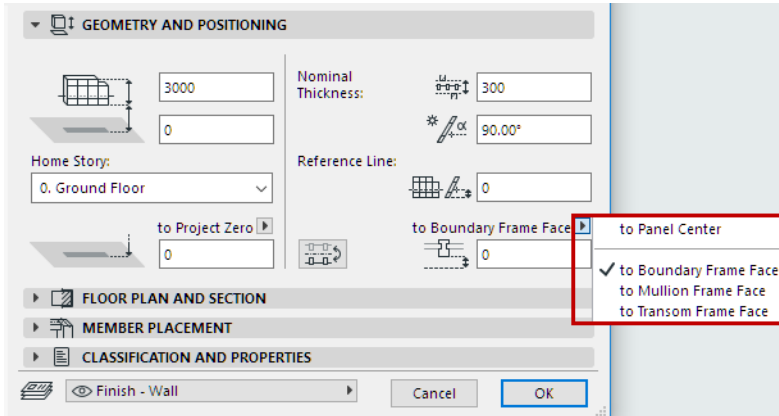
Flip Curtain Wall on Reference Line

Available for a selected Curtain Wall, or during input in the Info Box.

Click the Flip button to flip (or mirror) a selected Curtain Wall along its Reference Line: the body of the Curtain Wall flips to the other side. The Reference Line stays in place, and the pattern is not affected.



Distance from Panel (or Frame) to CW Reference Line



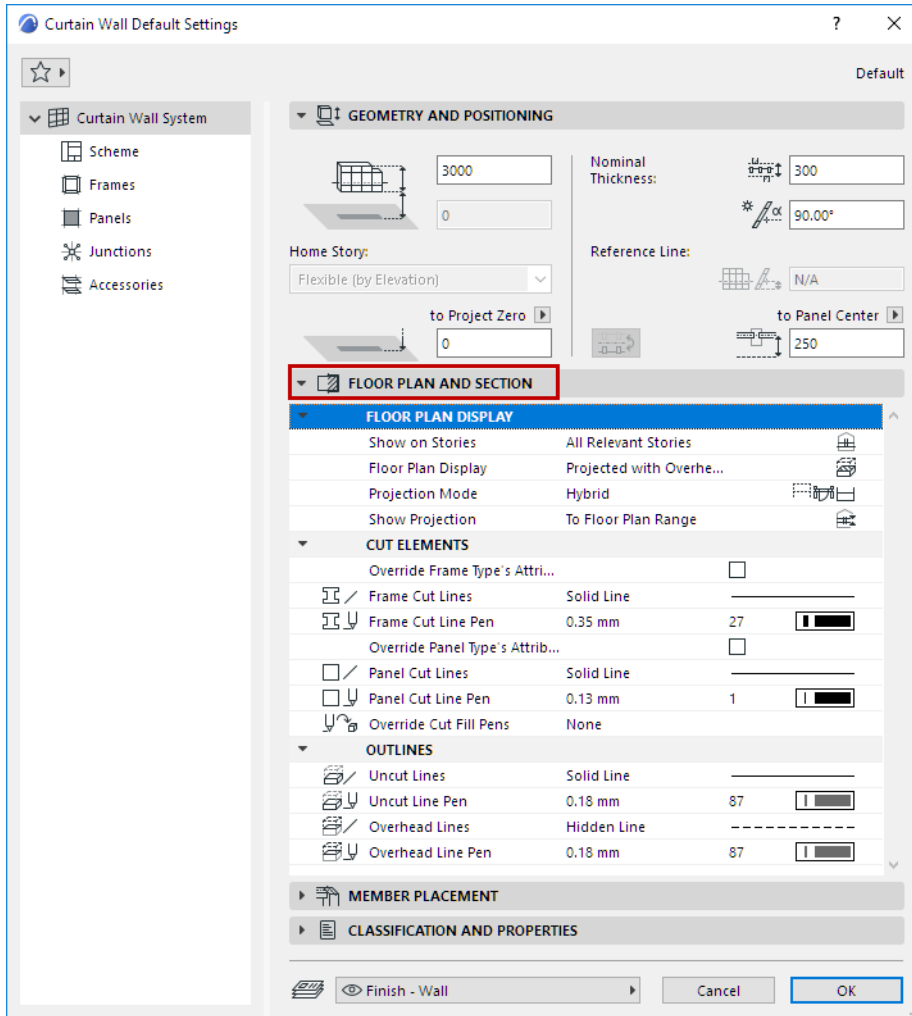
Use the pop-up and consult the image to define the offsets between the Reference Surface and each class of Frames (e.g. **Boundary**, **Mullion** and **Transom** Frames); or the center of the Panel.

The offset values here are interrelated; changing one will change all the rest.

Note: Frame geometry is defined, for each Frame class, on the Frame page of Curtain Wall settings.

[See Curtain Wall Frame Settings.](#)

Curtain Wall Settings: System Page: Floor Plan and Section Panel



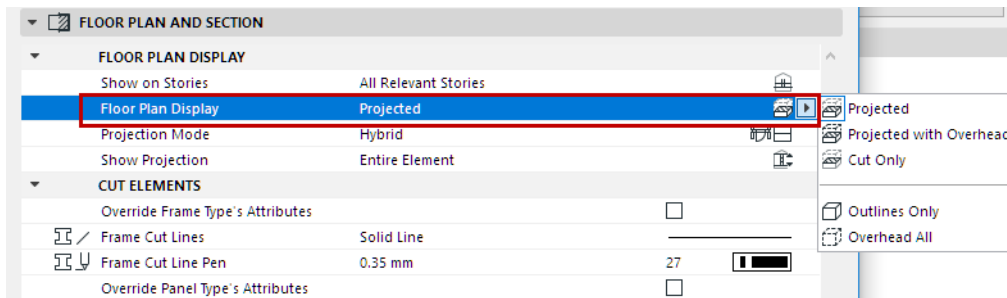
Use these controls to define the appearance of the Curtain Wall: its Floor Plan Display; its Cut Elements (appearance of cut frames and cut panels); and its Outlines.

Show on Stories

- **All Relevant Stories:** Display and edit on all stories which the element intersects.
- **Home Story Only:** Display on its Home Story only.

Floor Plan Display

Use this pop-up to define **which parts of the Curtain Wall are displayed** on Floor Plan.

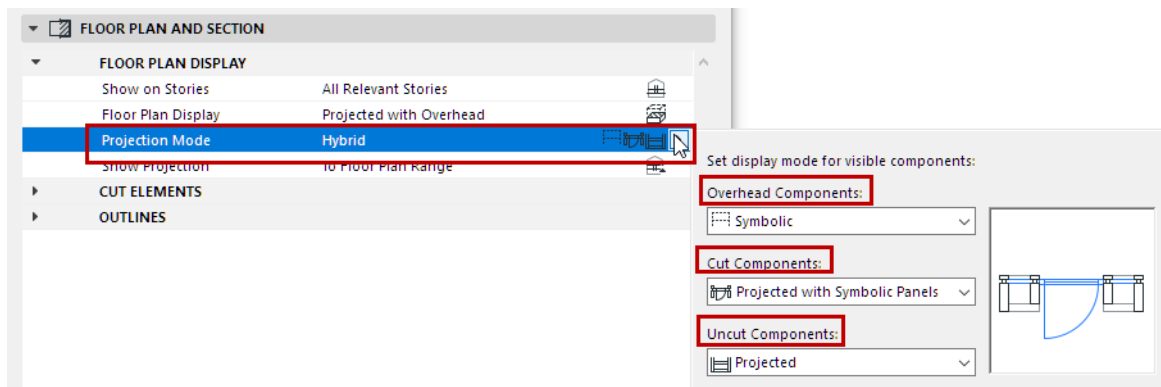


- **Projected**

- Cut part of 3D model (i.e., as cut at the level of the Floor Plan Cut Plane)
- Uncut (downward) part
- **Projected with Overhead:**
 - Same as Projected, plus Overhead part (above the Floor Plan Cut Plane)
- **Cut Only:** displays only the cut part, as cut by the Floor Plan Cut Plane.
- **Outlines Only:** the entire element's outline is shown using uncut attributes.
- **Overhead All:** the entire element's outline is shown using overhead attributes.

Projection Mode

Once you have set Floor Plan Display, use Projection Mode to fine-tune the displayed components (Overhead, Cut, Uncut) of the Curtain Wall:



- **Projected**
- **Symbolic**
- **Projected with Symbolic Panels** (for cut components only). Here, Cut Panels are Symbolic, while Cut Frames are Projected.

Consult the preview to visualize the result for each option. The Projection Mode is “**Hybrid**” by default - that is, it uses various settings for the three component types.

- Available options depend on the Floor Plan Display you chose above; some components may be invisible, and you cannot set a display mode for those.
- If Floor Plan Display is set to Outlines Only or Overhead All, then the Projection Mode becomes All Symbolic, and there are no further options.
- Curtain Wall display is also determined by Model View Options.

[See Model View Options for Curtain Walls.](#)

Show Projection

For projected parts of the Curtain Wall: the range of the projection. This control is only available if the element's Floor Plan Display is set (using the Floor Plan Display pop-up) to either “Projected” or “Projected with Overhead.”

- **Entire Element:** Element is displayed on all relevant stories.
- **to Floor Plan Range:** Show the Curtain Wall on a range of stories (the current story, plus a given number of stories above and below it).

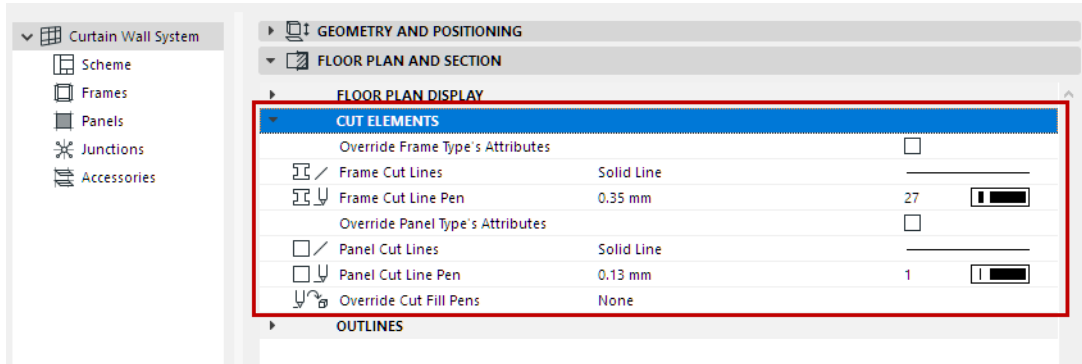
Set the desired range in **Floor Plan Cut Plane Settings**.

- **to Absolute Display Limit:** Set a fixed lower limit (by default, this is Project Zero), then show all parts of the Curtain Wall above this limit.

Set the Absolute Display Limit in **Floor Plan Cut Plane Settings**.

[See Floor Plan Cut Plane Settings Dialog Box.](#)

Cut Elements



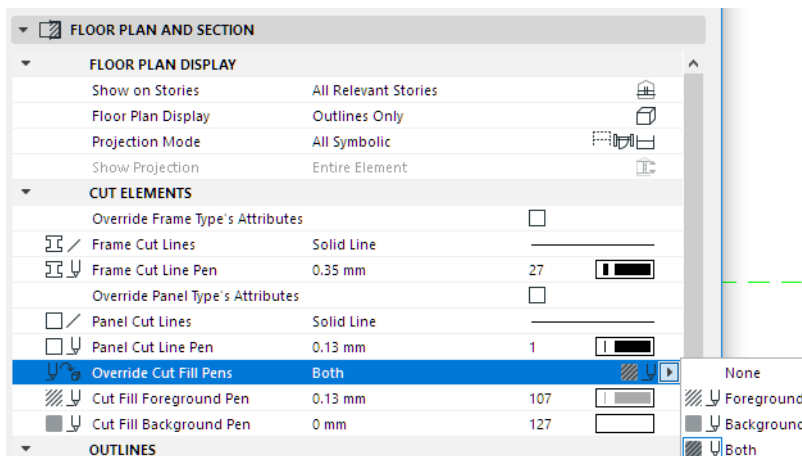
Override Frame Type's Attributes: Check this box to override the attributes defined in the GDL object, using the two controls below.

- **Frame Cut Lines**
- **Frame Cut Line Pens**

Override Panel Type's Attributes: Check this box to override the attributes defined in the GDL object, using the controls below.

- **Panel Cut Lines**
- **Panel Cut Line Pen**

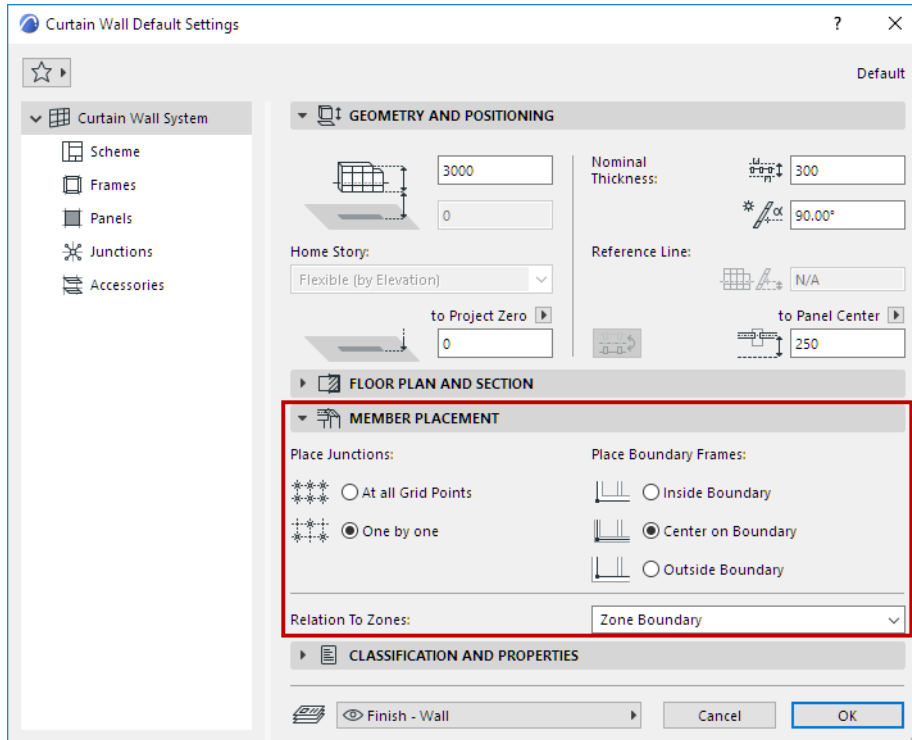
Override Cut Fill Pens: If needed, override the fill foreground/background pens of this element (defined by default in the Building Material). To do this, choose Foreground, Background or Both from the Override Cut Fill Pens pop-up to access the respective controls.



Outlines

Select Lines and Pens for the **Uncut** and **Overhead** lines of the Curtain Wall, as displayed on Floor Plan and Section.

Curtain Wall Settings: System Page: Member Placement Panel



Use the controls on this panel to define:

- how to place Junctions on your Curtain Wall, and
- where the Boundary Frame should be positioned relative to the grid.

Place Junctions

- **At all Grid Points:** Place Junctions automatically at all points of the grid. This works if a valid Junction Type is defined on the Junction page of Curtain Wall Settings.
- **One by one:** Place Junctions manually, using the Junction tool. (This is done in Edit mode only, after you have placed and selected a Curtain Wall.)

Related Topic:

[Curtain Wall Junctions](#)

Place Boundary Frames

- **Center on Boundary:** This is the default setting. The Boundary Frame centerline coincides with the grid line.
Note: The Curtain Wall Grid Pattern is defined on the Scheme page of this dialog box.
- **Inside Boundary:** Offsets the Boundary Frame to the inside of the Boundary grid line.
- **Outside Boundary:** Offsets the Boundary Frame to the outside of the Boundary grid line.

Relation to Zones

See these topics:

[Calculating Zone Area and Zone Volume](#)

[Relation to Zones](#)

Curtain Walls and Other Elements

The following sections describe how Curtain Walls can interact with other elements.

Join Wall to Curtain Wall

Managing Curtain Wall Joins

Curtain Walls and Adjacent Zones

Split Intersecting Curtain Walls

Create Common Frame for Adjoining Curtain Walls

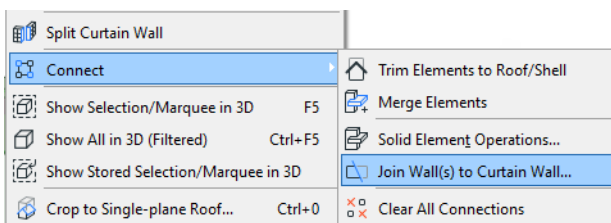
Join Wall to Curtain Wall

A Curtain Wall may be joined to one or more walls.

- If you move or rotate the Curtain Wall, then the joined wall will stretch accordingly.
- The Join command is also helpful in adjusting wall geometry to a slanted Curtain Wall.

To join a wall to a Curtain Wall:

1. Select the wall(s) and the Curtain Wall. (You may select any number of walls, but only a single Curtain Wall.)
2. Use the **Design > Modify Curtain Wall > Join Wall(s) to Curtain Wall...** command or the **Connect > Join Wall(s) to Curtain Wall** command from the context menu.



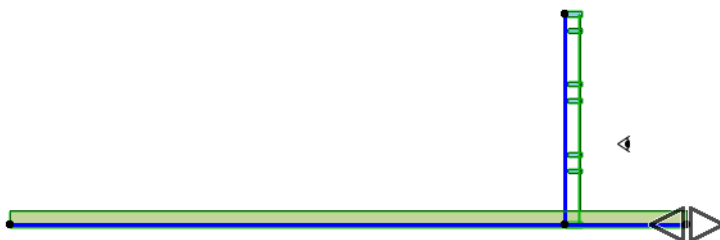
3. The **Join Wall(s) to Curtain Wall** dialog box appears. Use this dialog box to specify the exact connection point of the wall(s) to the Curtain Wall.

See details below.

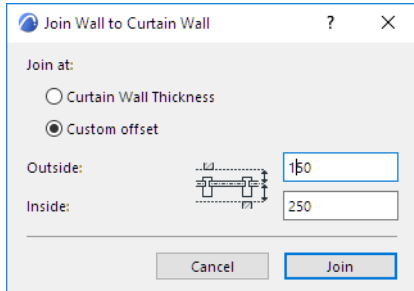
4. Click **Join**.

5. If the ends of the wall and Curtain Wall do not coincide, the **eyeball** cursor appears: click to define which end of the wall you want to join to the Curtain Wall.

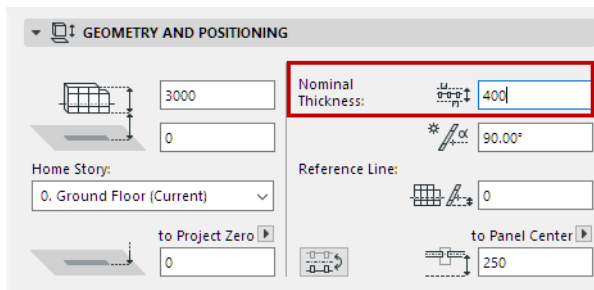
As you move the cursor to either end of the wall, a double-arrow appears on the wall, to indicate that this is the end that will be joined if you click now.



Join Wall to Curtain Wall Dialog Box



- **Join at Curtain Wall Thickness:** This is the default choice. These Inside and Outside offset values total the Curtain Wall's Nominal Thickness, as defined in Curtain Wall System Settings Geometry and Positioning Panel. In our example, the Nominal Thickness is set to 400.



- The **Outside** value: this defines the point at which walls coming from the “Outside” direction of the Curtain Wall will be joined to this Curtain Wall.
- The **Inside** value: this defines the point at which walls coming from the “Inside” direction of the Curtain Wall will be joined to this Curtain Wall.

Both the Inside and Outside wall joining points are expressed as *offsets from the Base Surface*, as shown in the “Join Wall to Curtain Wall” dialog box.

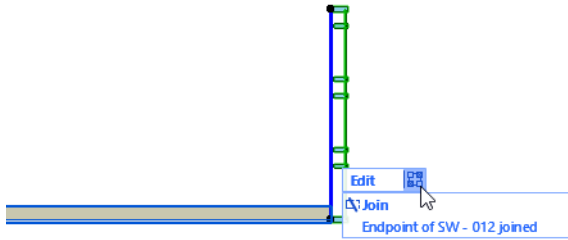
For the purposes of offsetting joined walls from this Curtain Wall, the “Inside” portion of this Nominal Thickness is equal to the Panel offset from the Reference Surface (in our example, 250); the “Outside” portion is what remains of the Nominal Thickness (in our example, 150).

- **Custom Offset:** Click this to stop the adjoining wall(s) at points other than the Curtain Wall's nominal thickness. Enter values for the *offset from the Curtain Wall's Base Surface* to the Outside and Inside wall connection points.

To redefine the offset for an already joined wall, join it again.

Managing Curtain Wall Joins

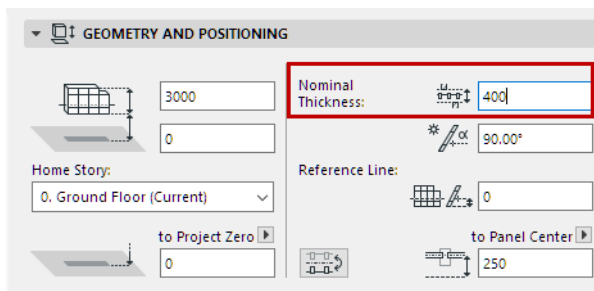
Selected elements that participate in a join between a Curtain Wall and Wall(s) are shown with a “grip”. Click the grip to see a list of the Connections (including Solid Operations and Trims, as well as joined Walls/Curtain Walls) which involve the selected element. Use this list to manage these relationships.



For more information, see [Managing Element Connections](#).

Curtain Walls and Adjacent Zones

Adjacent **Zones** extend up to the Curtain Wall's Nominal Thickness, as defined in the Nominal Thickness field of Curtain Wall System Settings (Geometry and Positioning Panel).



As with walls and columns, you have several options for defining how to calculate Curtain Walls which are bounded by Zones or which serve as a Zone boundary. Use the Relation to Zones pop-up on the Member Placement Panel of Curtain Wall Settings to define how to calculate these Zones.

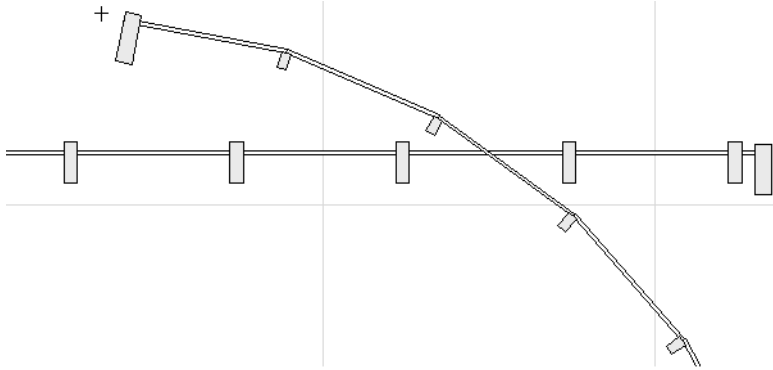
See [Relation to Zones](#).

Split Intersecting Curtain Walls

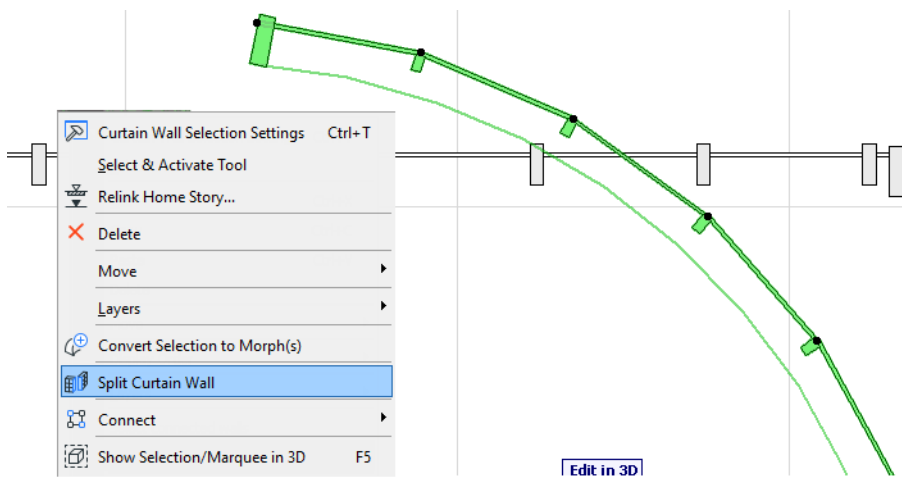
One way to precisely connect two Curtain Walls is to make them intersect, then split off the “extra” parts, using the **Design > Modify Curtain Wall > Split Curtain Wall** command.

This command works only for splitting one or more Curtain Walls with another Curtain Wall. It is available in any model window (but not in Curtain Wall Edit mode). It is analogous to the **Edit > Reshape > Split** command, which works on other Archicad elements.

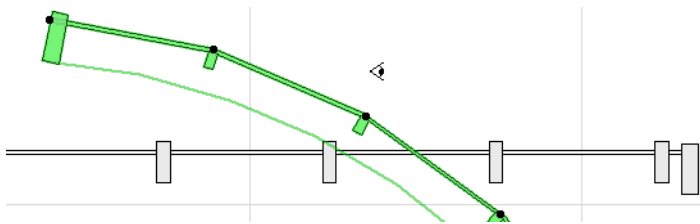
In this example, the two Curtain Walls intersect on the Floor Plan.



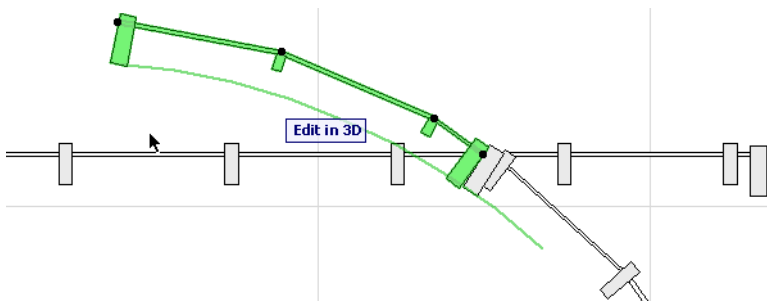
1. Select the curved Curtain Wall.
2. Use **Design > Modify Curtain Wall > Split Curtain Wall**, or the same command from the context menu.



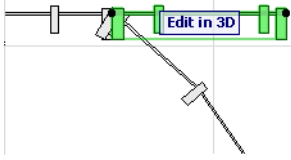
3. Click on the other Curtain Wall.
4. The eyeball cursor appears. Click either side of the curved Curtain Wall to define which part will remain selected after the Split takes place.



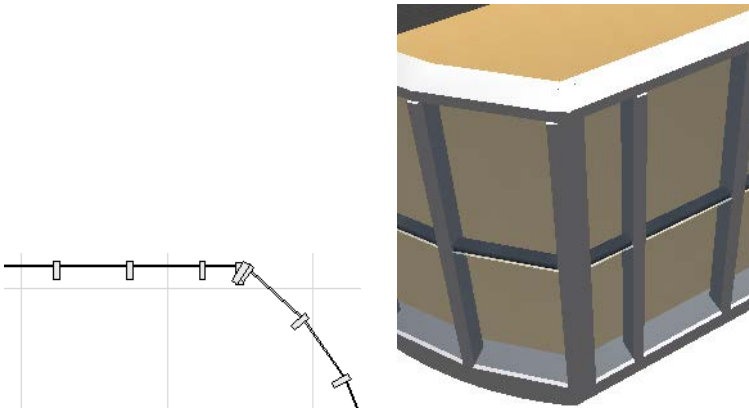
The curved Curtain Wall has been split into two separate Curtain Walls.



5. Now you can delete the one you don't need, and the remaining Curtain Walls are joined exactly.
6. Repeat the process to split off, then delete the protruding part of the straight Curtain Wall.



The result on the Floor Plan, and in 3D:



In such a case, the two adjoining Curtain Walls' Frames may overlap. To remedy this, create a common Frame for both of them.

[See *Create Common Frame for Adjoining Curtain Walls.*](#)

Stairs

Topics in this Section

[About the Stair Tool](#)

[Stair Input](#)

[Graphical Editing of Stair Scheme](#)

[Stair Turning Type](#)

[Stair Segment Type](#)

[Stair Tool Edit Mode](#)

[Automatic “Solver” Options to Resolve Geometric Conflicts](#)

[Stair Settings \(Main Element Settings\)](#)

[Toplinked Stairs and Changed Story Height](#)

[Tread and Riser Finishes](#)

[Flight and Landing Structures](#)

[Split Stair Structure](#)

[Custom Sub-Elements for Stairs and Railings](#)

[Stair Endings and Connections](#)

[Model View Options for Stairs](#)

[Detail Level of Stair and Railing Symbols](#)

[Floor Plan/Reflected Ceiling Plan Display \(Stair Settings\)](#)

[Labeling Stairs](#)

Related Topics:

[Ramp Object](#)

About the Stair Tool

Use the Stair Tool to design and document any typical or unique Stair, up to 1:50 design scale.

Input is available on the Floor Plan and in the 3D window, using methods similar to polyline or Wall input.

During the polyline path input, instant graphical feedback shows what the resulting Stair will look like.

- A Stair has one or more Segments, with Turnings (nodes) between them.
- Like the Curtain Wall, the Stair is a hierarchical element.
- The main Stair element consists of multiple components: Treads, Risers and Structures.
- Stair components are GDL elements, which use Building Materials and/or Custom profiles.
- Use Edit mode to access and customize the individual components of a selected Stair.
- Customizable Rules and Standards provide the constraints, and describe local regulations. The user can choose the rules that apply to his/her project, and set the allowed ranges. As the user places the Stair with graphical input, the algorithm automatically adjusts certain geometric values of the stair, within the user-provided ranges.
- In some cases, the input values and the polyline path cannot result in an adequate Stair. If such a conflict or error occurs, the 'Solver' pops up to help resolve it. The Solver presents multiple possible solutions to modify the Stair so as to meet regulations.
- Stairs can be linked to Stories, and so can follow changes in floor height.
- Stair Structure
 - Four structure types: monolithic, beam, cantilevered, and/or stringer.
 - Structure for flights and landings can be defined individually
 - Customize each structure for size, building material, profile, 2D appearance
- Stair Finish
 - Components include Treads and Risers. Treads and Risers for Flights and Landings can be defined individually, and any individual Tread and Riser can be customized in Edit mode.
 - Optionally define custom components and save them using one of the predefined Stair subtypes.
- Stair 2D sub-elements
 - Break Mark, Walking Line, Numbering, etc.
 - Symbols are 2D GDL components, completely separate from the 3D structure
 - Use Model View Options to define which 2D sub-elements should appear on the Stair and how to display them.

[Watch the Video](#)

Stair Input

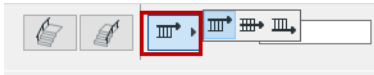
Basic Stair Input

To get a feel for the Stair Tool, start placing Stairs in the plan using the default settings, or one of the Stair Favorites.

Input is available on the Floor Plan and in the 3D window, using methods similar to polyline or Wall input. During the polyline path input, instant graphical feedback shows what the resulting Stair will look like.

The Stair is input along a Baseline. Note some of the key input preferences in the Info Box:

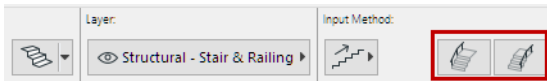
Input Methods (Info Box)



Stair Baseline Position: Left, Right Center



Input Stair Upward/Downward

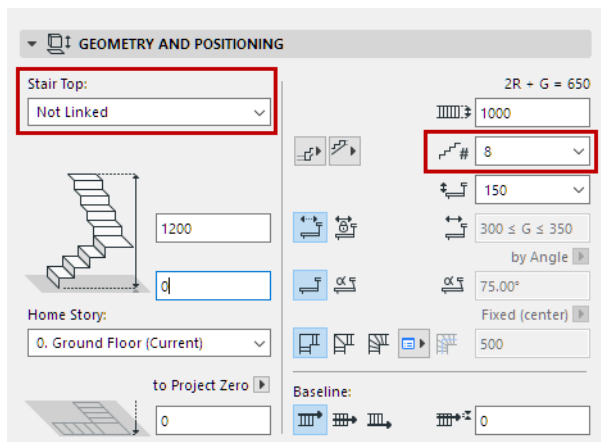


Add Railing to Stair (Right, Left, Both sides)

Example 1: Entrance Stair

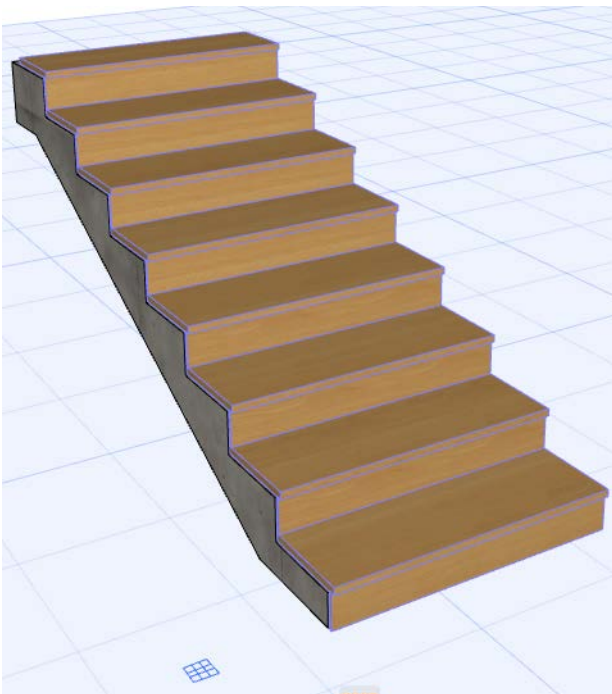
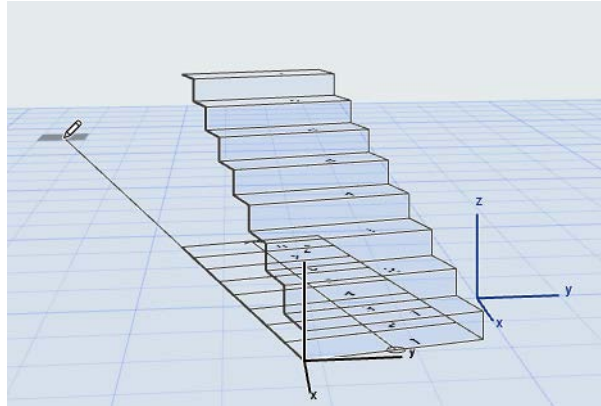
1. Open the Stair Settings dialog box, and review the Geometry and Positioning Settings.

This Stair is not toplinked, and it has a fixed number of 8 risers. It will be a straight stair with no turning, so the turning type is not relevant.



In the Floor Plan or 3D window:

1. Click to begin inputting the Stair.
2. Note the feedback: Since the number of risers is fixed, the length of the Stair is shown as a ghost outline during input.

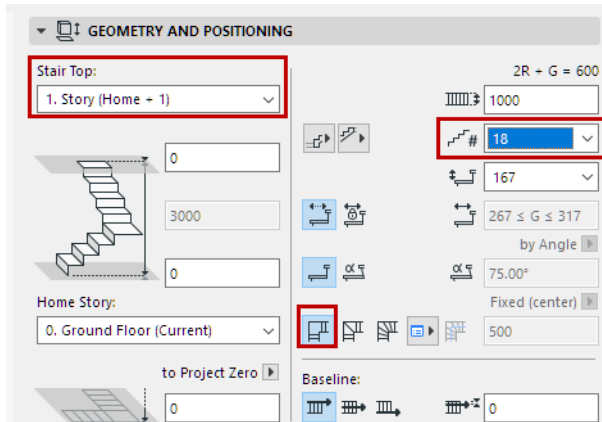
3. Double-click to complete the Stair.

[Watch the Video](#)

Example 2: Top-Linked Staircase

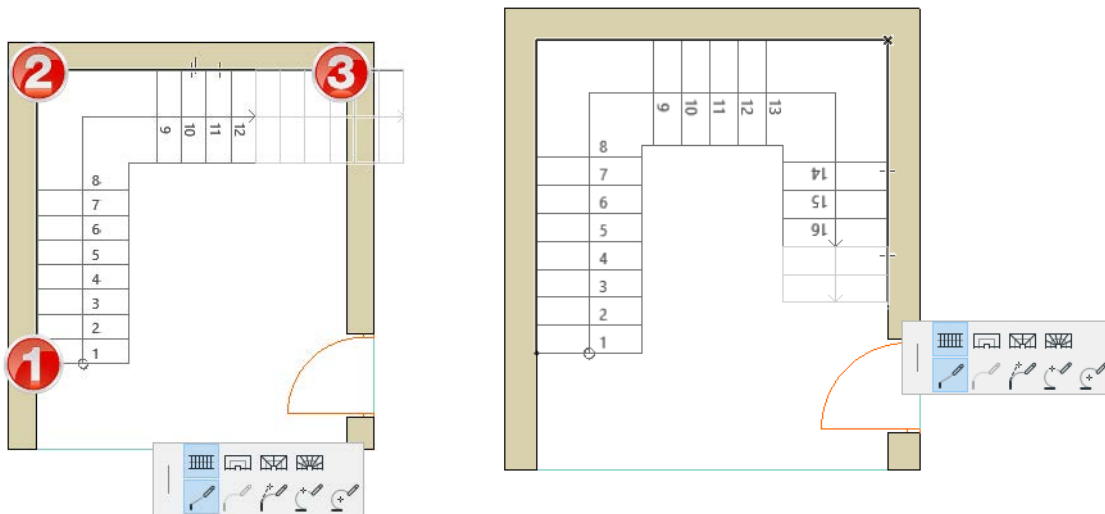
Here, we create an interior Staircase up to the next floor.

Set up the Stair Settings as shown. This Stair's top is linked to the story above, and it has a fixed number of 18 risers. This Stair will have Landings, and the Landing type is set to Automatic Landing.

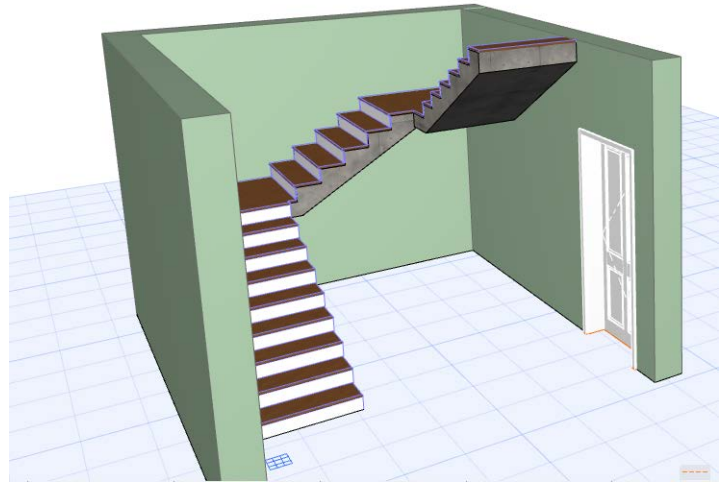
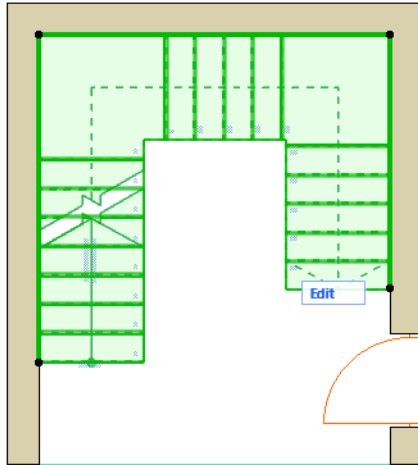


In the Floor Plan or 3D window:

1. Click to begin inputting the Stair along the Walls, as when drawing a polyline.
2. Click at each of the two Wall corners, then double-click to complete the Stair. The ghost feedback shows you that 18 risers will be placed in any case, according to the predefined settings.



The Stair is complete. The Turning type you defined (Landing) was placed at both turnings (points on the input baseline) automatically.



Turnings and Segments

Stairs consist of Turnings and Segments.

- Set Turning Type for the Stair as a whole in Stair Settings. (Choose Automatic Landing or one of two Winders.)

[See *Stair Turning Type*.](#)

- Segment Type is always “Flight” by default. You can change the segment type on the fly, for each segment, during input.

[See *Stair Segment Type*.](#)

- You can customize Turning Type options - for any particular segment or any particular Turning - after Stair input.

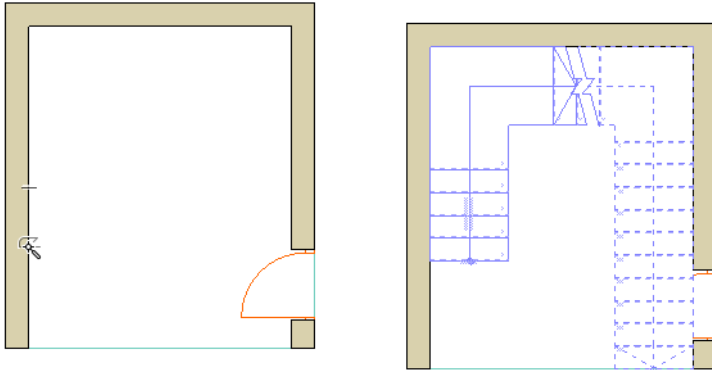
[See *Edit a Stair's Turning Type*.](#)

Create Stair with Magic Wand

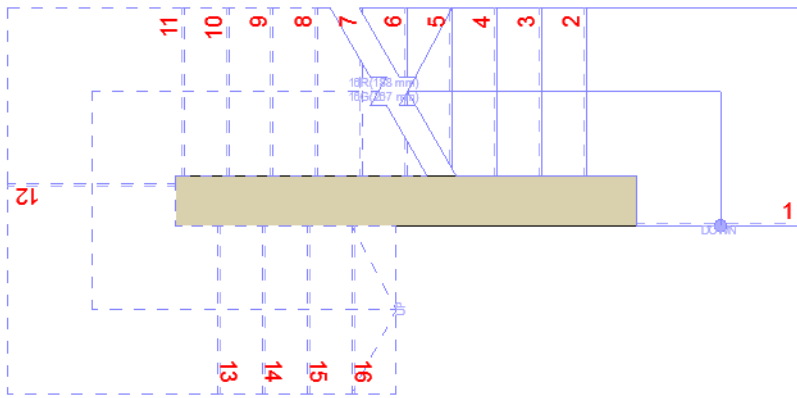
Use the Magic Wand method (Spacebar + click) to automatically generate a Stair along the clicked edge or polygon.

- Click at the point where you want the Stair to begin.
- The clicked point will be the bottom of the Stair (even if the input method is Downward).
- The Stair Baseline is placed along the traced polygon/polyline.

Change the Baseline if needed in the Info Box or in Stair Settings (Left, Right or Middle of Stair). This allows you to vary the Stair placement (inside or outside of the clicked polygon/polyline).



- To create a Stair around a free-standing Wall: click into the Wall.



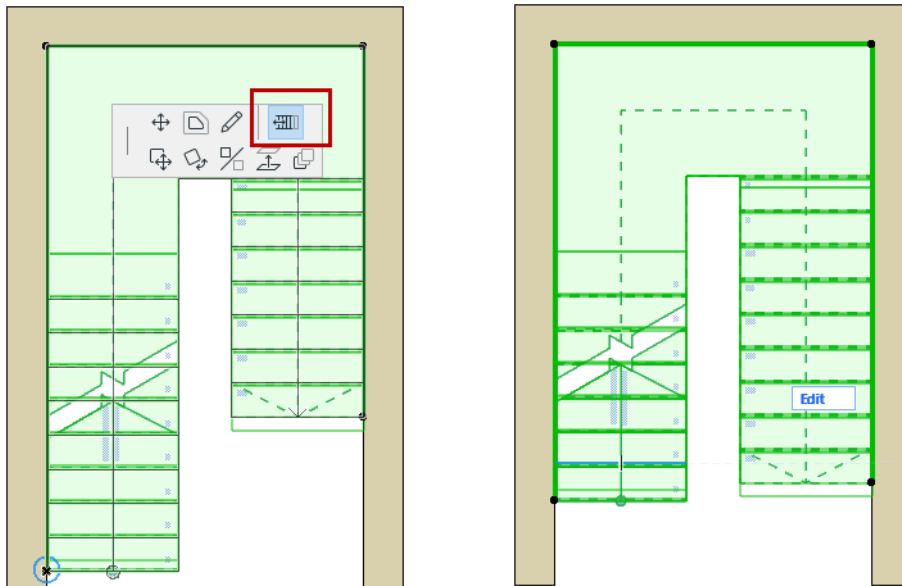
[Watch the Video](#)

Graphical Editing of Stair Scheme

The following functions let you adjust overall Stair geometry on the Floor Plan, without having to use Edit mode.

Rearrange Steps

Use this pet palette command to graphically rearrange the symmetry of a U-shaped stair:



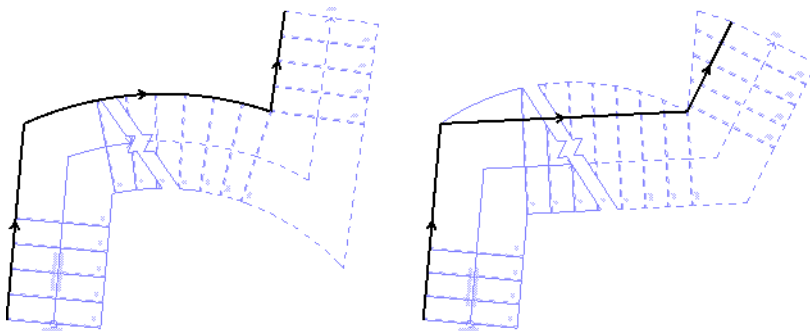
Edit Stair Baseline

The Stair Baseline is created during input. Use Stair Settings or the Info Box to set the Baseline position and any offset (either before or after input).

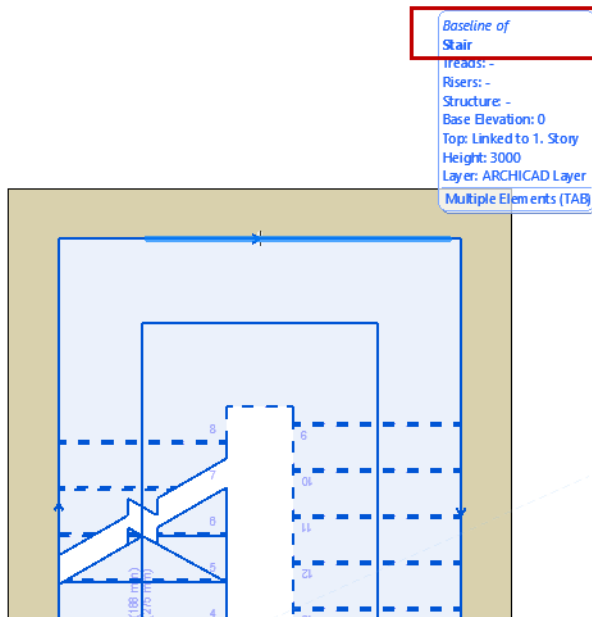
[See also Stair Baseline.](#)

You can edit the Stair Baseline in any window (regardless of Edit mode).

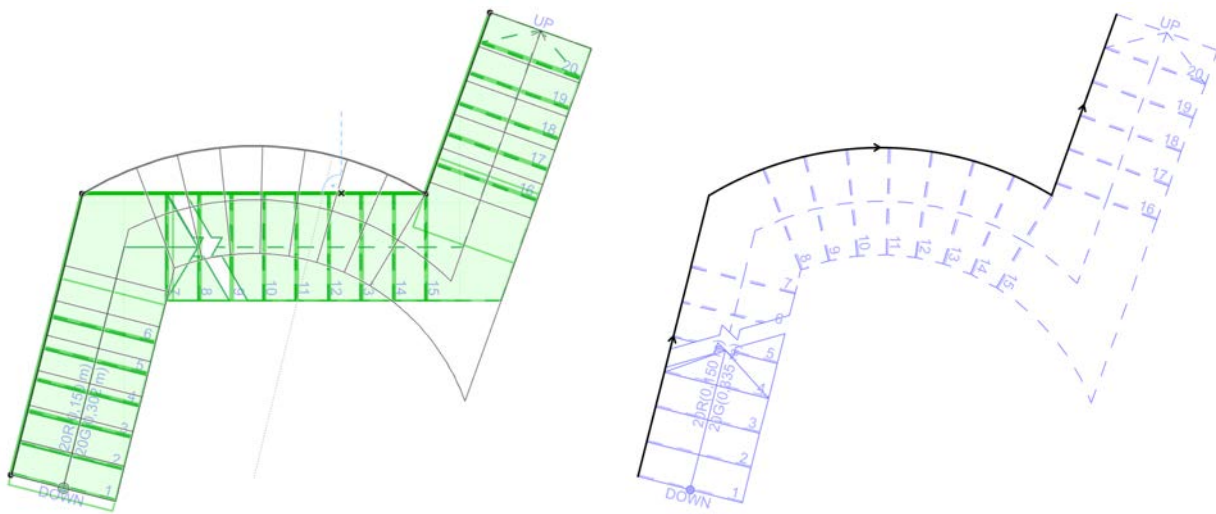
To show/hide the Stair Baseline on the Floor Plan, turn on **View > On-Screen View Options > Stair Baselines.**



Use the pre-selection highlight tag to make sure you select the Baseline (not the Boundary).



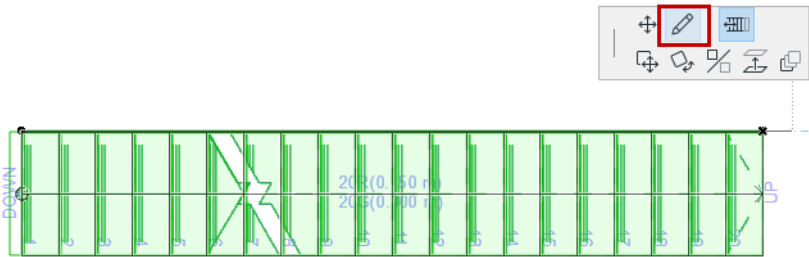
Baseline editing changes the overall Stair geometry, including the Walking Line.



Walking Line Follows Baseline Editing

Continue Stair

To continue a Stair Baseline (in Edit mode or in a model view), use the **Continue Stair** command from the selected Baseline's pet palette.

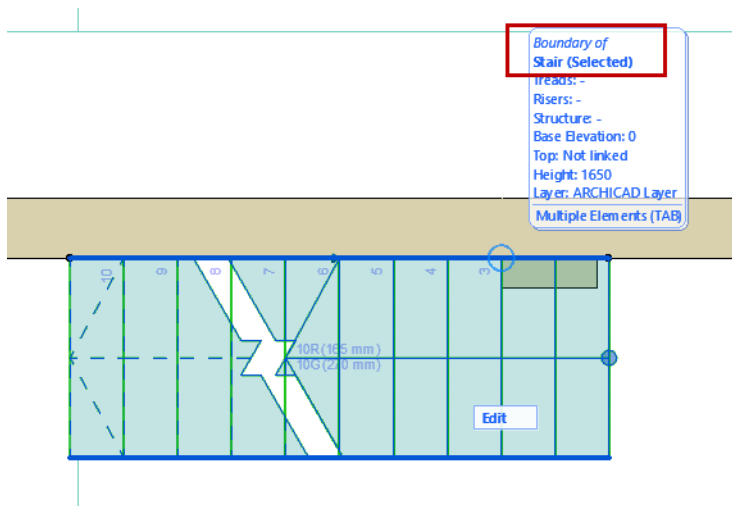


Edit Stair Boundary

Select any part of the Boundary to highlight it.

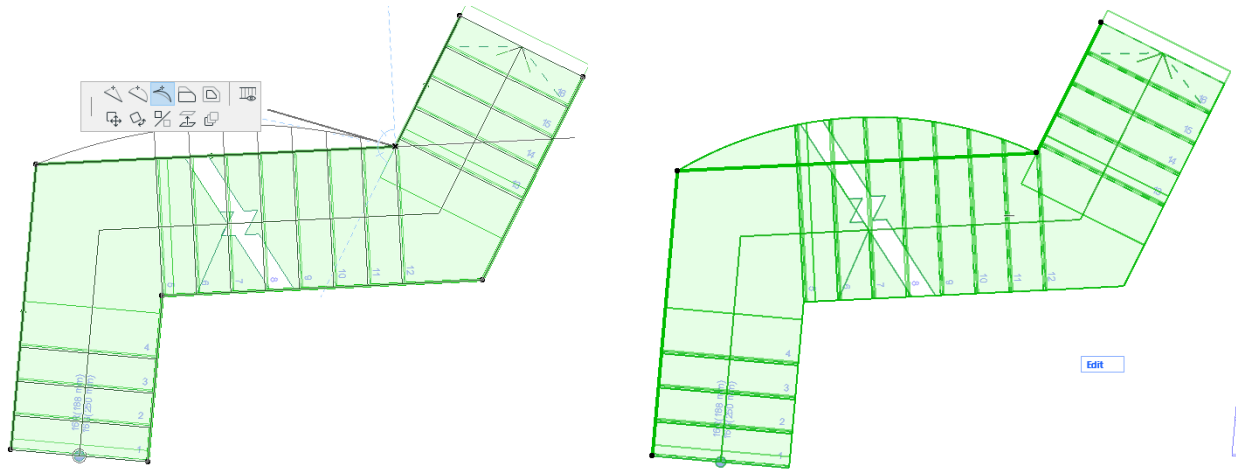
You can edit the Stair Boundary in any window (regardless of Edit mode).

Use the TAB button and the pre-selection highlight to make sure you select the Boundary, not the Baseline.



Edit the Stair boundary graphically, using the same pet palette commands as when editing a Fill.

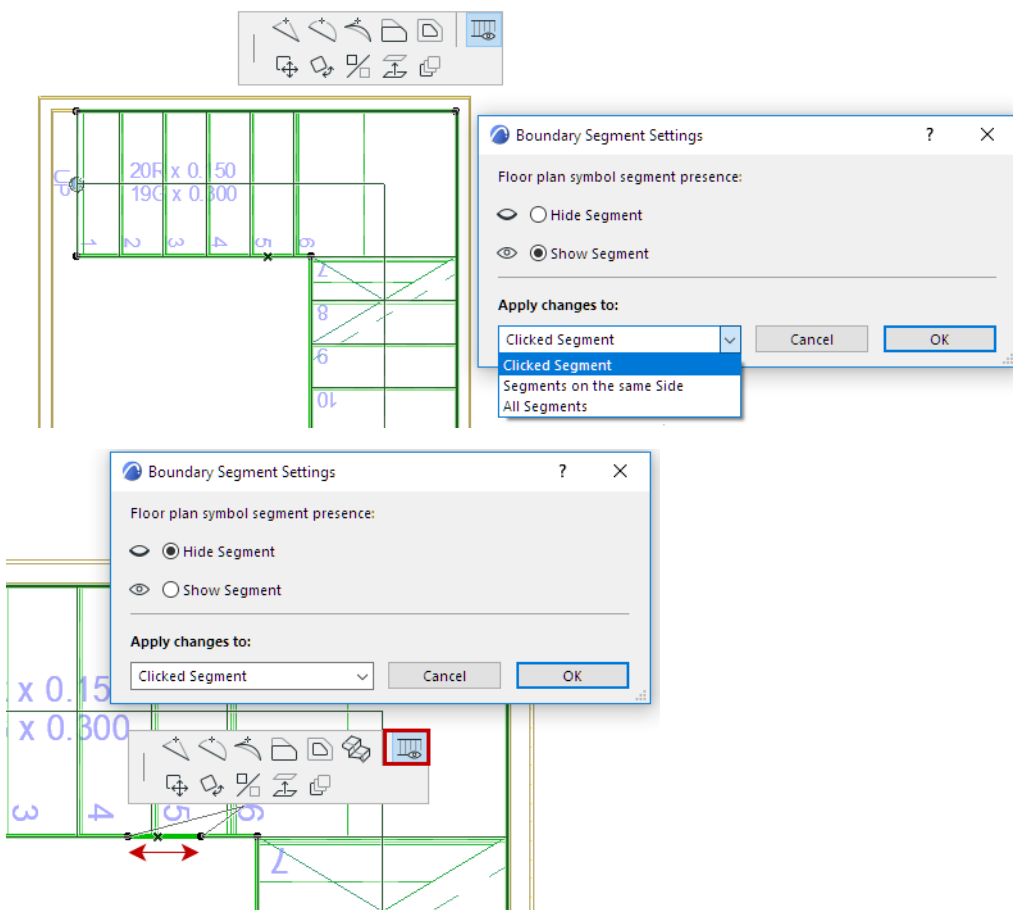
Boundary editing changes the outer shape of the Stair, but the Walking Line and Baseline stay put.



Edit Stair Boundary (Walking Line Unaffected)

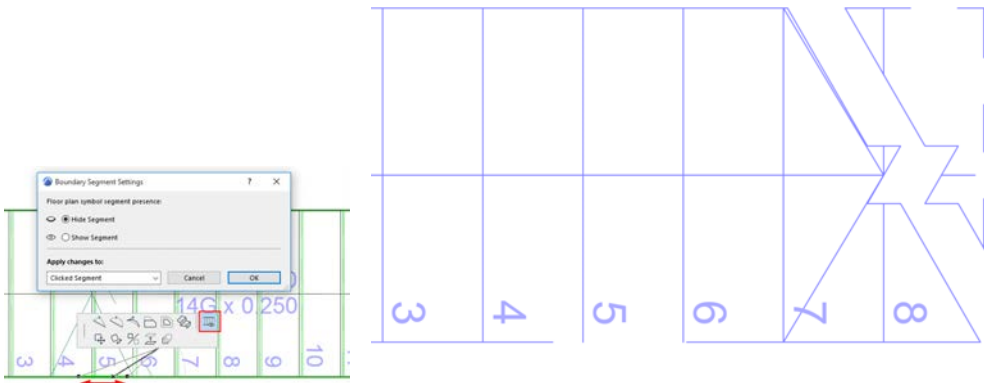
Show/Hide Boundary Segment on Floor Plan

For any selected Stair segment, use the Boundary Segment Settings pet palette command to open this dialog box.



Show or hide the selected segment (or apply to all or other Stair segments) as needed.

For example: you can add two nodes to a Stair segment, then hide the Stair Boundary between those two points.

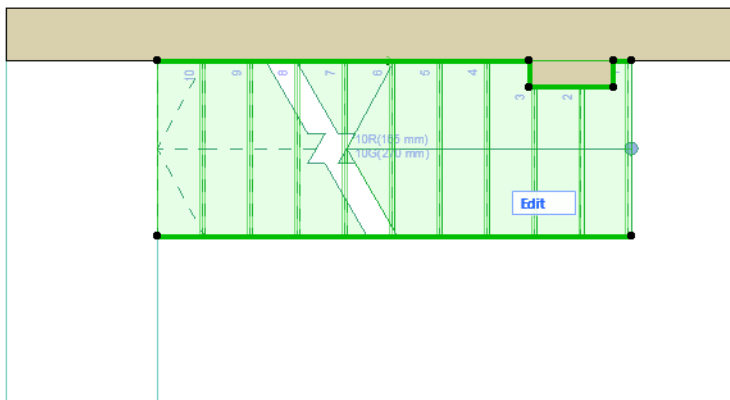
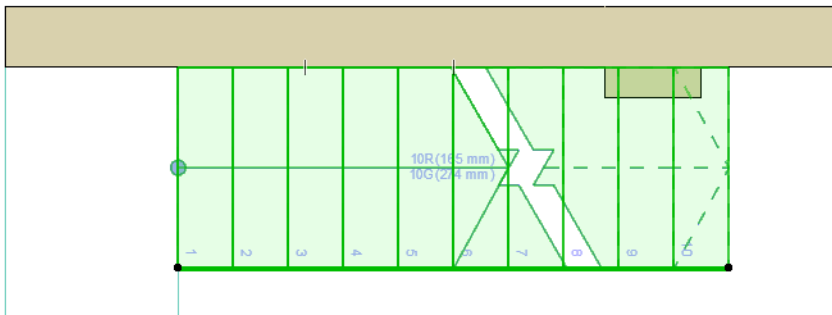


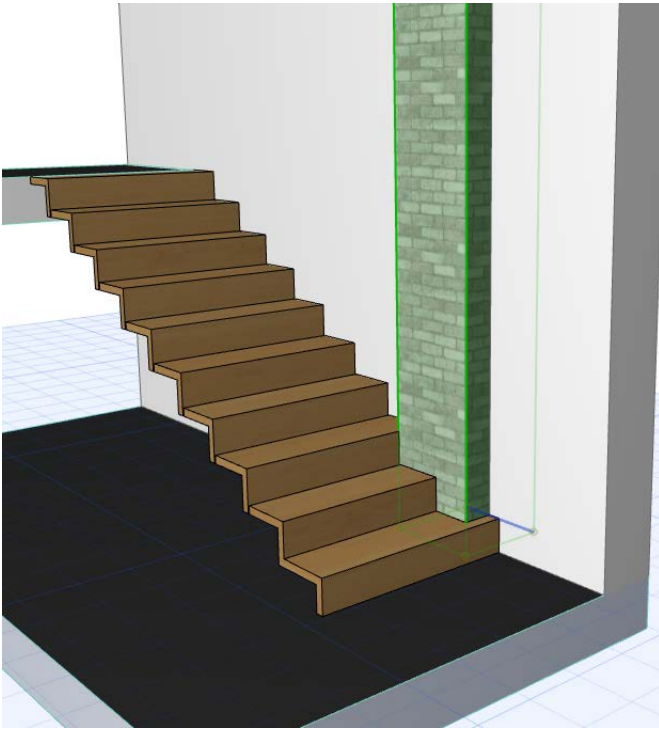
Insert Two Nodes to Create New Stair Segment... Then Select and Hide Segment

See also [Hide Any Edge of a Single Tread](#).

Edit Stair Boundary - Example

In this example, you can edit the Stair boundary to cut out enough room for a protruding Wall segment.





[Watch the Video](#)

Stair Turning Type

Set the Stair's Turning Type (Landing or a Winder) in Stair Settings, and define its options.

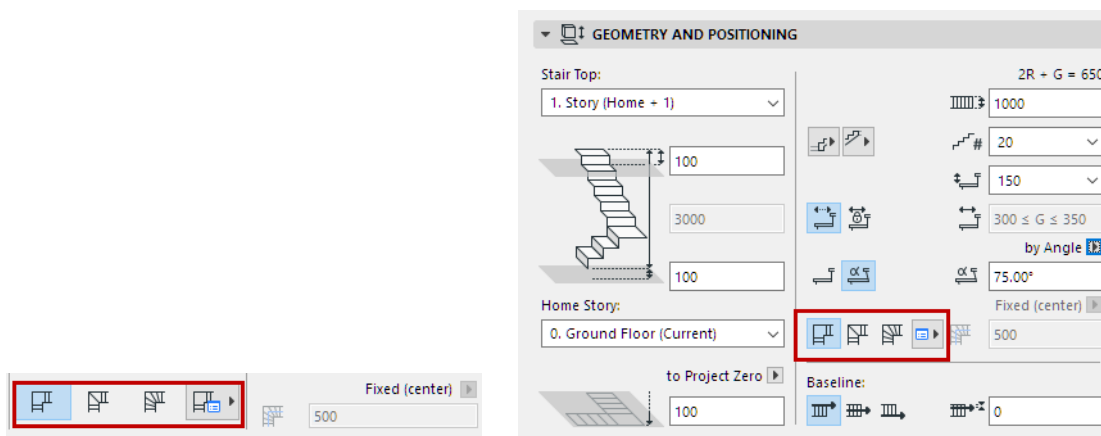
The selected Turning Type is placed at each turning (each point you click while placing the Stair baseline), between any two Stair Flight segments. All Turnings will thus have the same settings. However, for a placed Stair, you can customize each Turning type separately.

[See Edit a Stair's Turning Type, below.](#)

In Stair Settings or the Info Box, choose a preferred Turning Type for the entire Stair.

- Automatic Landing
- Winder with Equal Angles
- Winder with Equal Goings

Note: In the USA version of Archicad, "Goings" are known as "Runs".



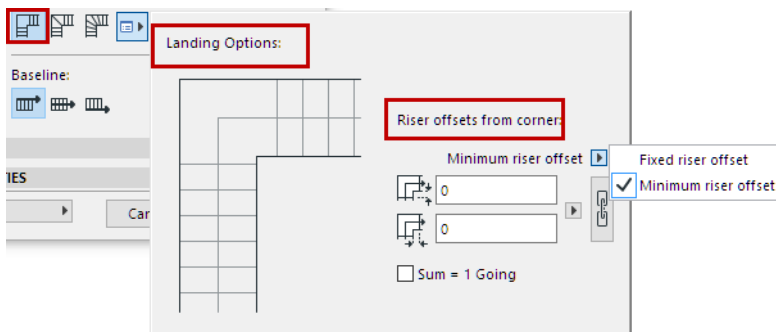
For each type, click the pop-up to adjust its specific settings. The settings differ for each Turning Type:

Landing Options

Winder with Equal Angles: Division Method

Winder with Equal Goings

Landing Options



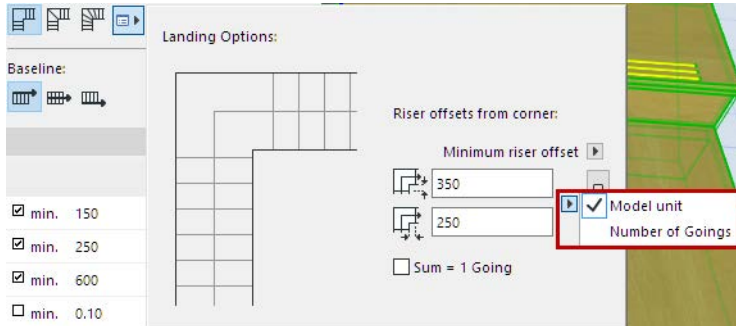
Riser Offsets from Corner

- Define the Landing offsets from the corner as either Fixed or a Minimum. (If you pick Minimum, the program will have some leeway to increase the offset, to adjust to the Stair input geometry.)
- Enter offset values for the Landing's Lead and Trail Edges.

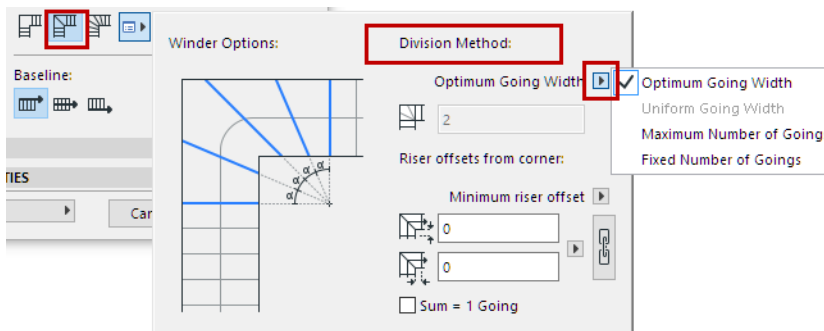
- Click the chain icon for a symmetrical landing (same offsets on both ends).

Note: The chain icon will ensure a symmetrical landing for each turning of the Stair. However, a Stair with multiple landings may use different offset values at different landings (if offset values are defined as a “minimum” rather than fixed).

- Define the offset value in terms of either Model Units, or as a certain number of Goings.



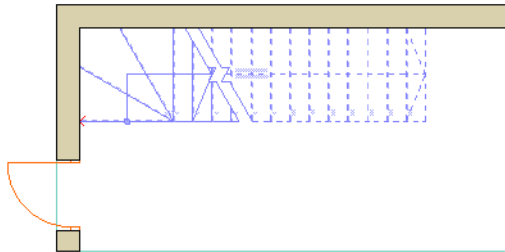
Winder with Equal Angles: Division Method



Set Division Method

Choose a method for dividing the Winder:

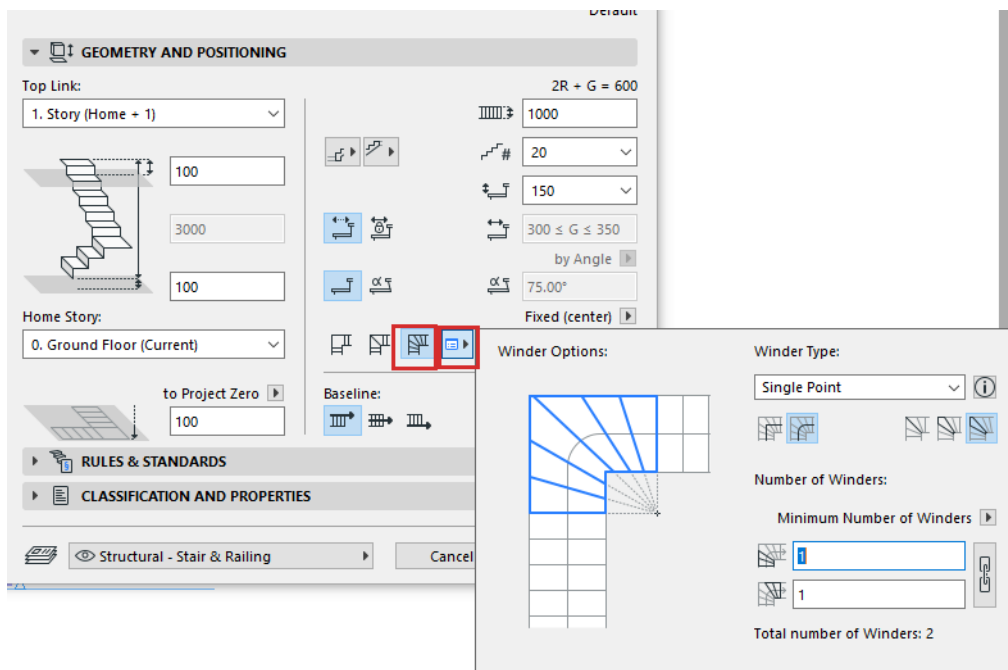
- Optimum Going Width:** The program automatically finds a Going Width for the Winder which is as close as possible to the Going Width on the Stair’s straight flight.
- Maximum Number of Goings:** The program automatically inserts the maximum number of goings into the Winder, within the limits of the current Stair Settings.
- Fixed Number of Goings:** In the field below, enter a fixed number of Winder divisions. The Winder will be divided accordingly, regardless of any other rules set in Stair Settings.



Just as for the Landing, set riser offsets at either end of the winder, defined as a fixed or minimum value.

[Read More](#)

Winder with Equal Goings

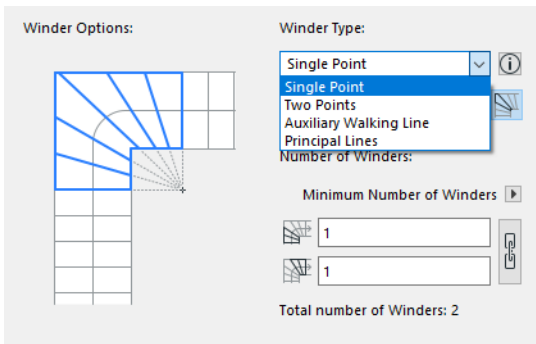


Equal Goings means that all goings in the Stair are equal, both on the straight and the winding steps.

Note: In the USA version of Archicad, “Goings” are known as “Runs”.

- Within a single Stair, all the Equal-Going Winders always use the same Winder Type (e.g. all are “Two Point” winders).
- For any Equal-Going Winder, you can offset the Calculated Walking Line, which affects Going depth. [See *Offset Walking Line on Turning \(Winder with Equal Goings\)*](#)
- For any Equal-Going Winder, you can set the Walking Line to be either Straight or Curved.

Choose one of the four Equal-Going Winder Types. They differ in how they are constructed. For each, consult the illustrations in the dialog for an indication of how it works.



See more illustrations and descriptions here:

[Winder Types for Equal-Going Winders](#)

[Read More](#)

Turning Type vs Segment Type

Stair segments - not just Turnings - can be defined as Landing or Winder-type segments (as well as Flight segment). Segment type is set not in Stair Settings, but on the fly, during Stair input.

[See Stair Segment Type.](#)

These Segment definitions can produce different Turning Types than the one you set in Stair Settings.

In any case, you can customize Turning Type options - for any particular segment or any particular Turning - after Stair input.

[See below.](#)

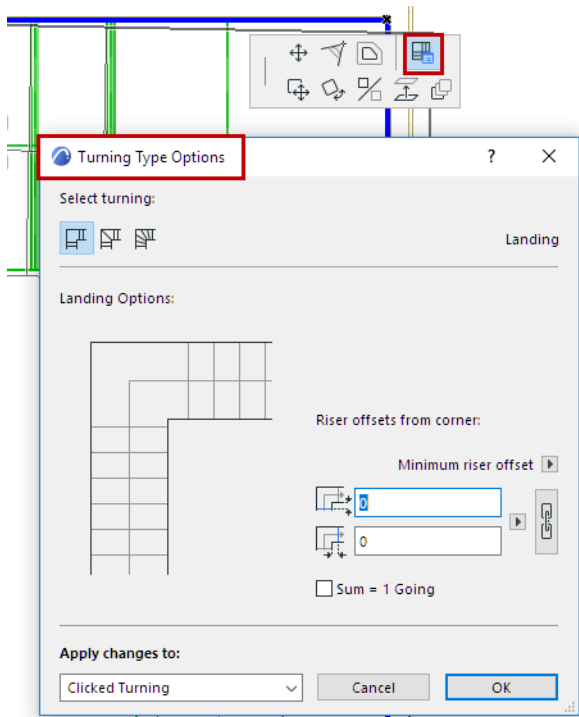
Edit a Stair's Turning Type

For a placed Stair, you can adjust Turning Type Settings for any specific turning, using the pet palette.

Note: Within a single Stair, all the Equal-Going Winders must use the same Winder Type. The only editable option for a selected Equal-Going Winder is the number of winders.

1. Select the Stair on the Floor Plan.
2. Click on a turning point, then choose Turning Type Options from the pet palette.
3. In the appearing dialog box, select the desired Turning type, and the options which apply.

Apply changes to: From the pop-up, select either the specific clicked Turning, or all turnings of the entire Stair.

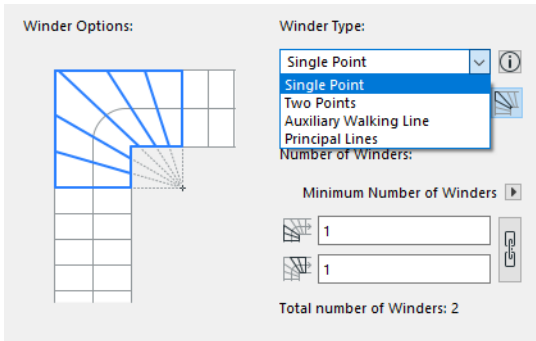


Related Topic:
[Stair Segment Type](#)

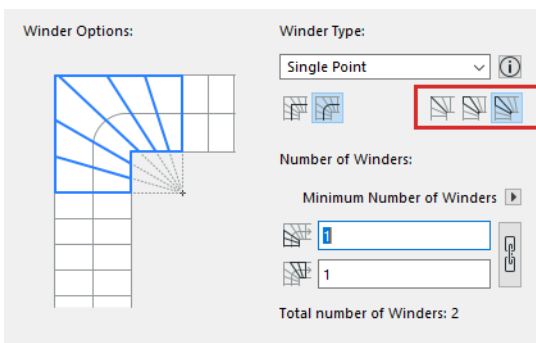
Winder Types for Equal-Going Winders

In Stair Settings - Geometry and Positioning Panel:

1. Choose one of four types of Equal-Going Winders from the pop-up: Single Point, Two Points, Principal Lines, and Auxiliary Walking Line.

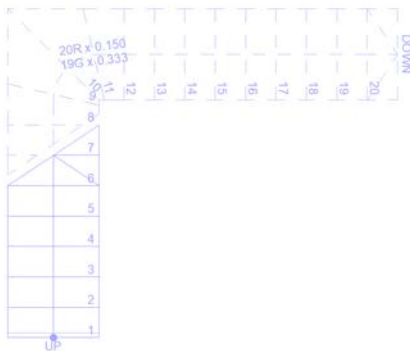


2. Click one of the three icons to set how to distribute the treads at the turning point, where the winder axis intersects the walking line:



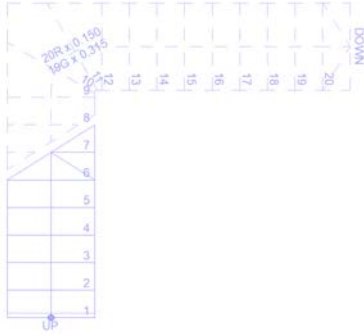
- Riser: place a riser at the turning point
- Symmetric Going: a tread is halved at the turning point
- Asymmetric Going: no restrictions on tread placement

See the following illustrations:



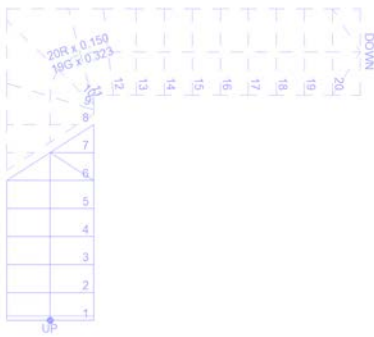
Winder Type: Auxiliary Walking Line

Turning Point: Riser



Winder Type: Auxiliary Walking Line

Turning Point: Symmetric Going



Winder Type: Auxiliary Walking Line

Turning Point: Asymmetric Going

Winder Segments

In case of a Winder Segment, the midpoint of the Winder part of the walking line is considered the starting point of the Winder.

- Bisectors are constructed as the perpendicular to the walking line through this point.
- Equal Goings are measured starting from this point.
- Tapered Tread Minimum Widths are measured starting from this point's projection onto the inner edge.

Stair Segment Type

A Stair segment can be defined as Flight, Landing, or one of two Winder types.

When you start inputting a Stair, the default segment is always “Flight”. You can change the segment type on the fly, for each segment, during input.

When inputting a Stair, you can select a different Stair Segment type (Landing, Winder, Flight) for each segment as you click to place it. These types determine the Turning that is placed at the end of each segment.

For example, a Flight segment connected to a Landing-type segment will use a Landing-type turning.

For a placed Stair - after input is complete - you can edit any of its Turning Types individually.

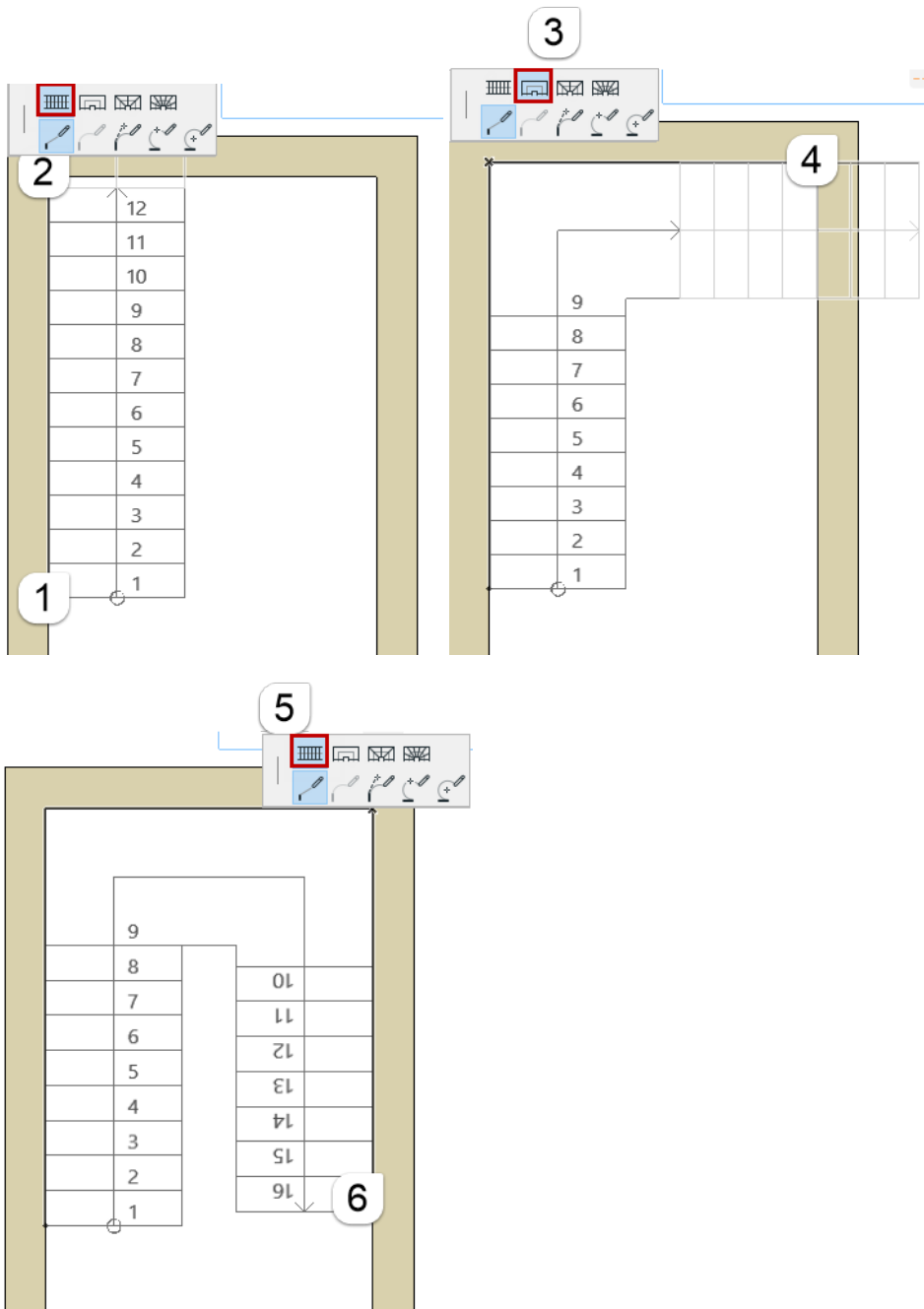
[See *Edit a Stair's Turning Type*.](#)

Define Segment Type (Flight, Landing, Winder) During Stair Input

During Stair input, the pet palette appears with available Segment Types: Flight, Landing, or one of two Winder types. You can change this Segment type on the fly during Stair input. After each click, set a Segment type for the next input segment.

For example:

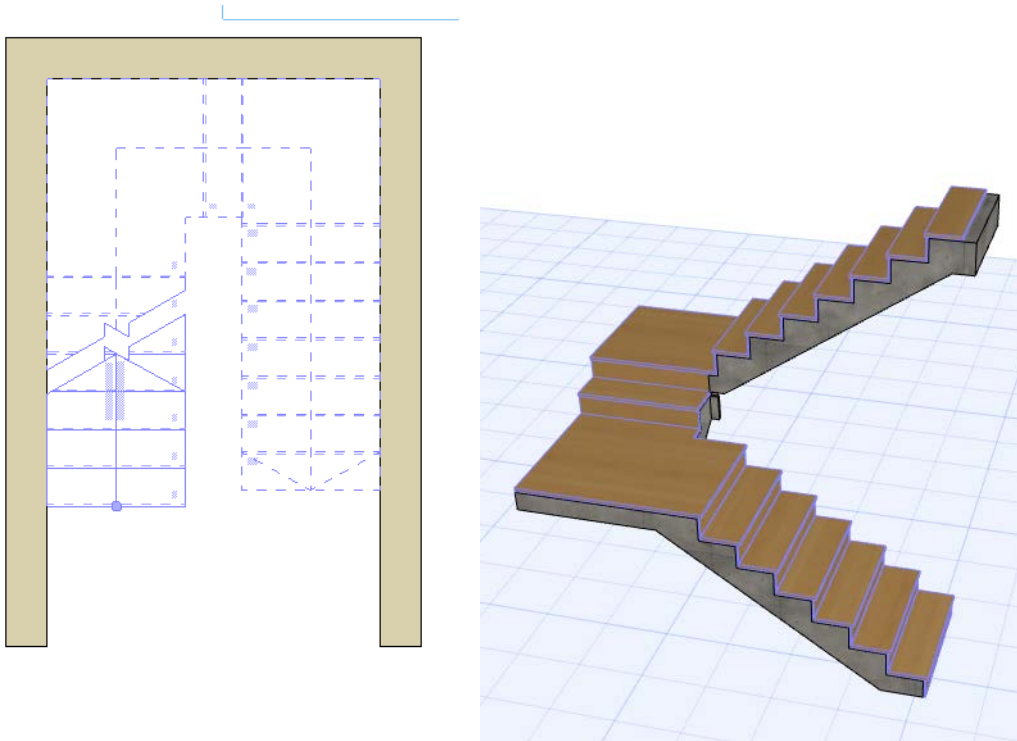
1. Click to start placing the first Stair segment: the pet palette shows that the current segment type is set to Flight, which is what you want.
2. Click to complete the **Flight** segment.
3. As you drag to input the next segment, choose **Landing** from the pet palette.
4. Click to complete the Landing segment.
5. As you drag to input the next and final segment, choose **Flight** from the pet palette.
6. Click to complete the Stair.



Switch Segment Types During Input: Flight – Landing – Flight

Redefine Segment Type: Landing, Flight, Winder

If you input a Stair whose default *Turning Type* is set to Landing in Stair Settings ([see Stair Turning Type](#)), you may find that two adjacent Landings within a single Stair segment have an unwanted Riser:

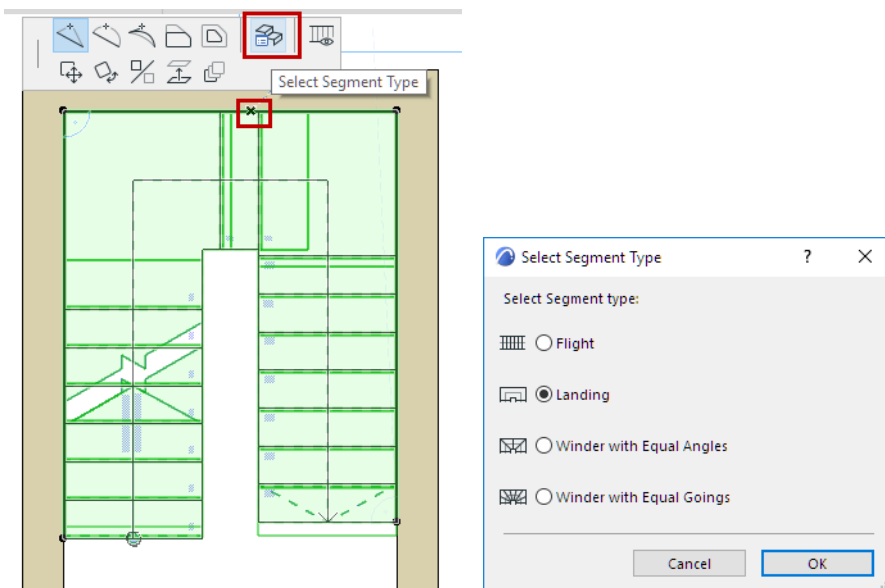


Turning Types = Landings, Segment Type = Flight

The reason is that the Stair segment that connects the two Landings is a Flight-type segment, which inserts a Riser between the two landings.

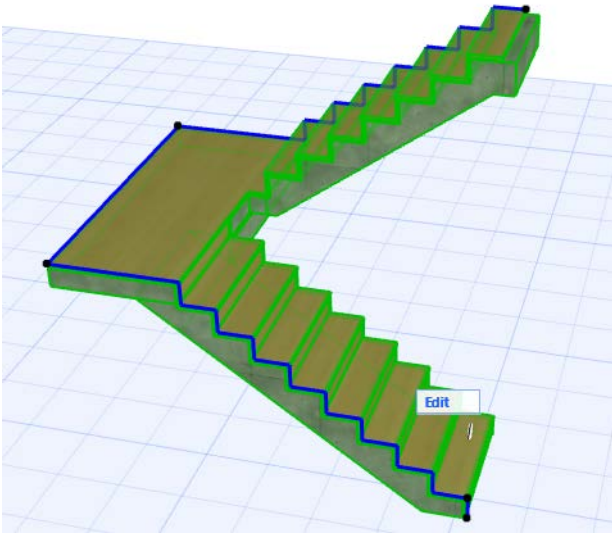
To resolve this, redefine the Stair segment as a Landing:

1. Select the Stair on the Floor Plan.
2. Click on the segment with the landings and choose **Select Segment Type** from the pet palette.
3. In the appearing dialog box, switch the segment type to Landing.



Change Stair Segment Type

The Stair segment is now a Landing, and its two Landing-type turnings meet in the middle.



Segment Type = Landing

Related Topic

[Stair Turning Type](#)

Stair Tool Edit Mode

Use the Edit mode of the Stair tool to access and edit any one or more Stair components: Treads, Risers, and Structures.

Note: You can edit the Stair's Baseline and Boundary regardless of Edit mode.

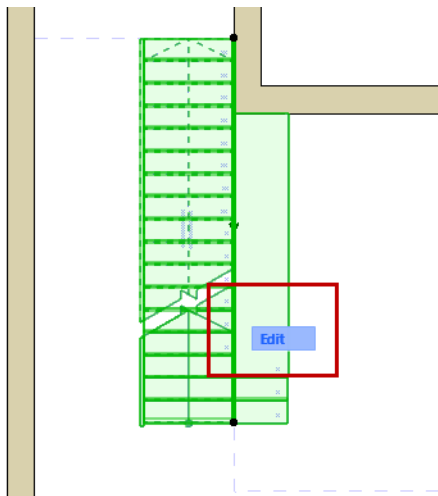
- Enter Edit mode from Floor Plan, Elevation, Section and 3D Window. You can remain in Edit mode even when switching between views.
- In Edit mode for the Stair (unlike Curtain Wall Edit mode) you cannot add new Treads, Risers or Structures - you can just edit existing ones, either graphically or through their component Settings dialogs.

Exception: In Edit mode, you can extend the Stair using **Continue Stair** from the Baseline's pet palette.

Enter Edit Mode

To enter Edit mode, select a Stair in any model window, then do one of the following:

- Click the **Edit** button which appears.
- Use **Design > Enter Stair Edit Mode**.



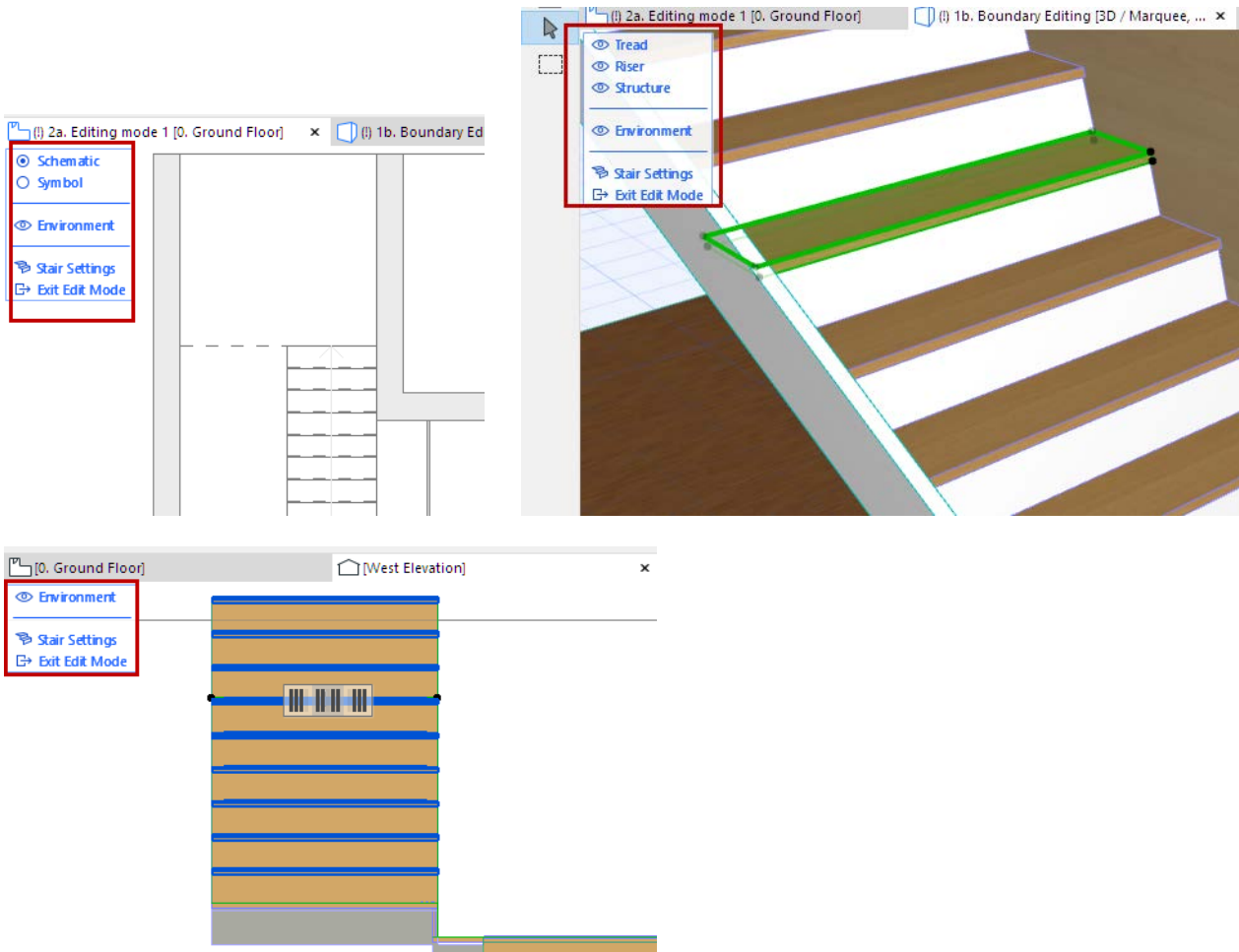
Exit Edit Mode

Do one of the following:

- Use **ESC** (with nothing selected)
- Click **Exit Edit Mode** from the display palette (see below).
- Use **Design > Exit Edit Mode**.

Edit Mode Display Palette

Edit mode has its own Display Palette, which varies depending on the window you are in (2D, 3D, Section/Elevation).



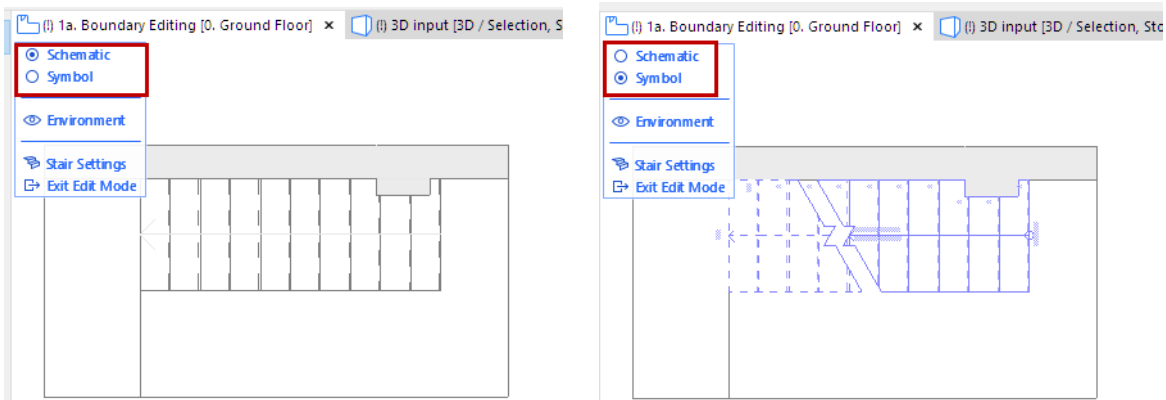
Edit Mode Display Palette

Use the Edit mode Display Palette to show/hide the Stair components as needed (click the eye icons to open or close them), even in the middle of an editing operation.

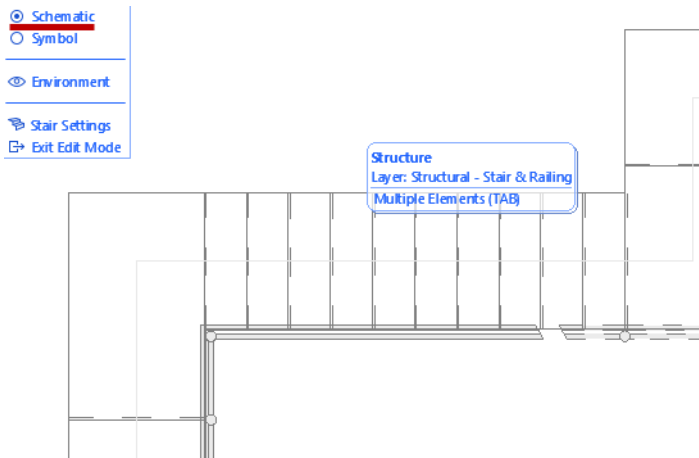
Right-clicking any of the components (e.g. “Tread”) in the display palette means that component will be displayed and the rest hidden.

Schematic vs. Symbol View (2D Edit mode)

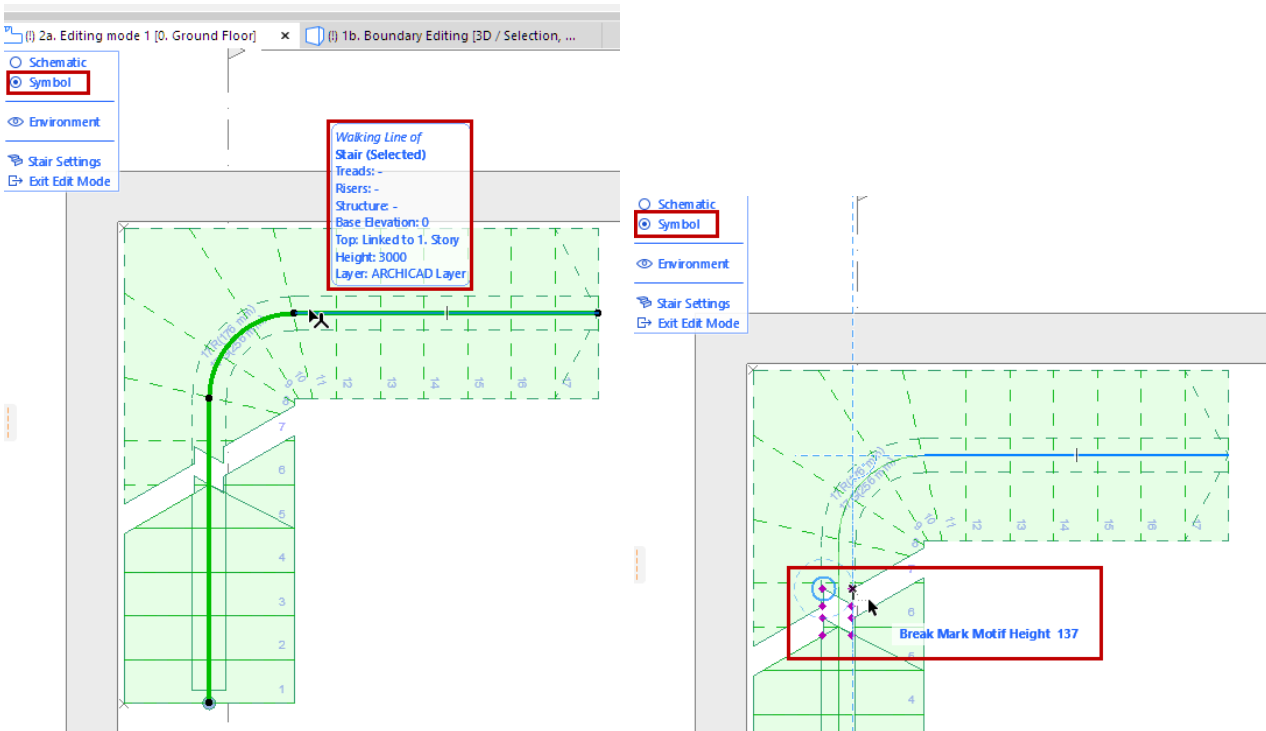
The 2D view of Stair Tool Edit mode has two options to display the Stair for optimal editing:



- Schematic:** shows the Stair as a simplified 3D transparent surface. Use this mode to select Structures, as well as Treads and Risers (use Tab to cycle among multiple highlighted elements):



- Symbol:** Use this to access the Stair's the 2D symbol elements (e.g. Walking Line)



Use Symbol View to Edit Walking Line or Break Mark

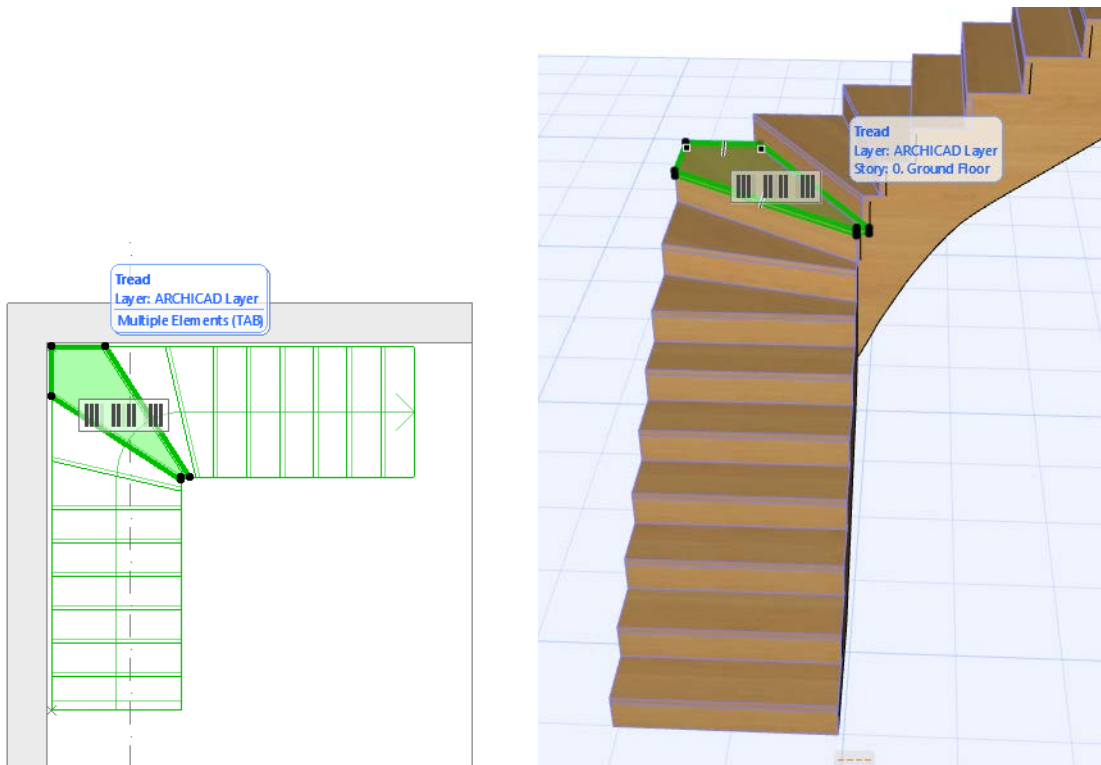
Related Topics

[Edit Stair Symbol Components Graphically](#)

[Walking Line Symbol: Position and Display Settings](#)

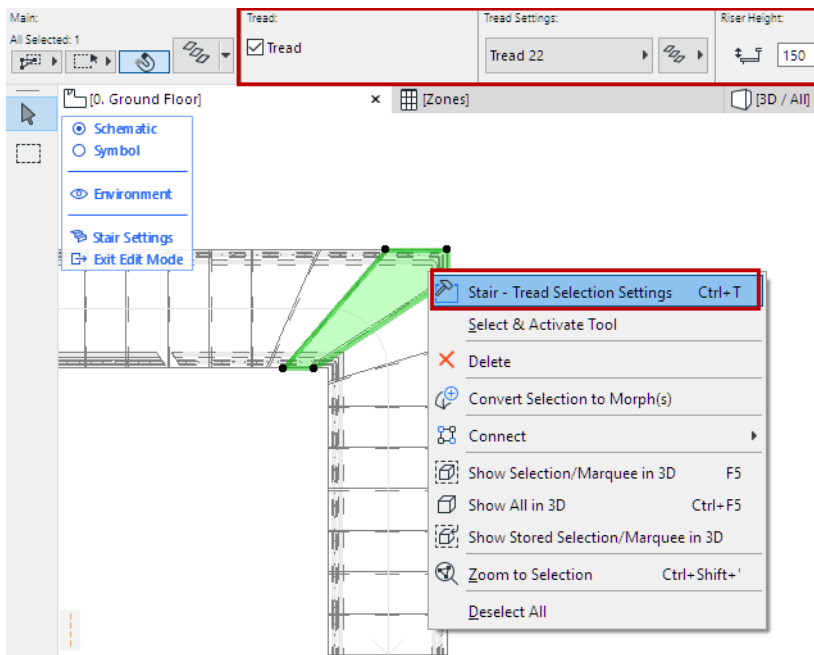
Select Stair Component for Editing

In Edit mode, select a Tread, Riser or Structure component to edit it graphically and to access its settings.

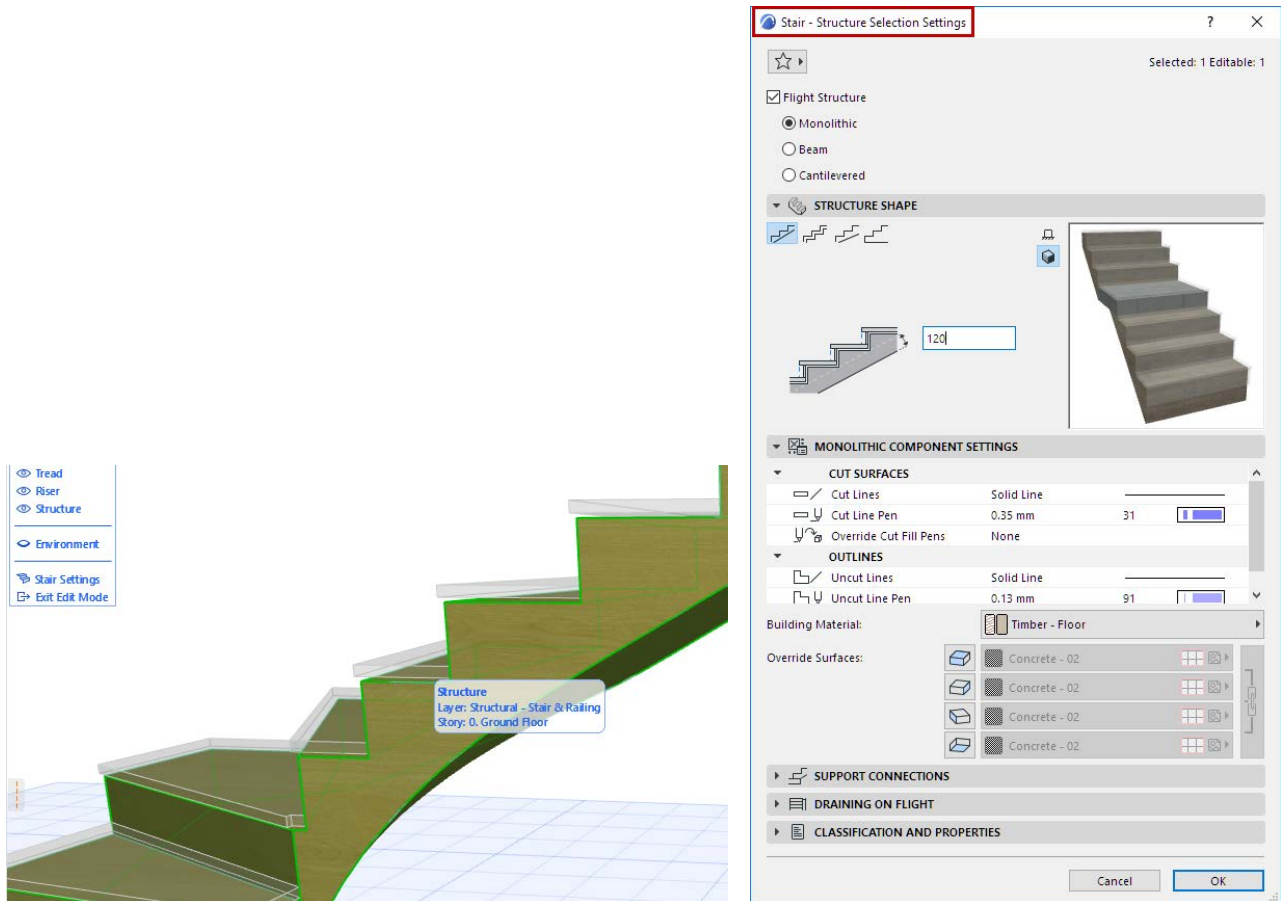


- Use the Element Info tag to make sure you select the component you need. In case of multiple overlapping elements, use the Tab key to cycle among selection options.
- The selected component’s Settings are available from the Info Box (click Settings) and the context menu.

Note: Unlike Curtain Wall Edit mode, there is no Stair toolbox available from Edit mode, because you cannot create new Stair components in Edit mode.



- When selecting a Structure component in 3D Edit mode, you may have to navigate “underneath” the Stair, or turn off the display of Treads/Risers.



Multi-selection of Treads and Risers

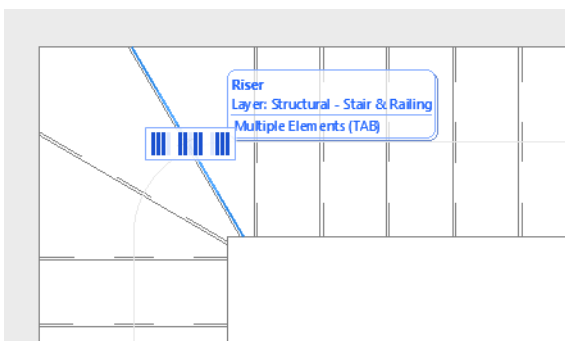
When you select a single Tread or Riser in Edit mode, a “Grip” appears with shortcuts for selecting multiple Treads or Risers.

Click on one of the Grip options to select one or more Treads/Risers, in addition to the selected one:

- all Treads/Risers above
- all Treads/Risers below
- all Treads/Risers altogether



A Riser in 2D Edit mode is shown as a single line. (Hover your cursor and use Tab if you have trouble clicking on the Riser line.)



Selected Riser (Line) in 2D Edit Mode

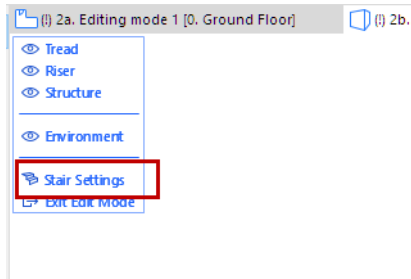
Show/Hide Environment

“Environment,” in this display palette, refers to the rest of the building model, which you can show or hide as needed to assist in editing. Preselection tags will identify the Environment’s model elements in dimmed form, but in Stair Edit mode, you can edit only the components of the Stair you selected to enter Edit mode.

Note: If you have trouble identifying the edges of these model elements shown as the Environment, try checking **Show Contours** in **View > 3D View Options > 3D Styles**.

Access Stair Settings from Edit Mode

To edit the System-level Settings without leaving Edit mode, click **Stair Settings** at the bottom of the display palette.



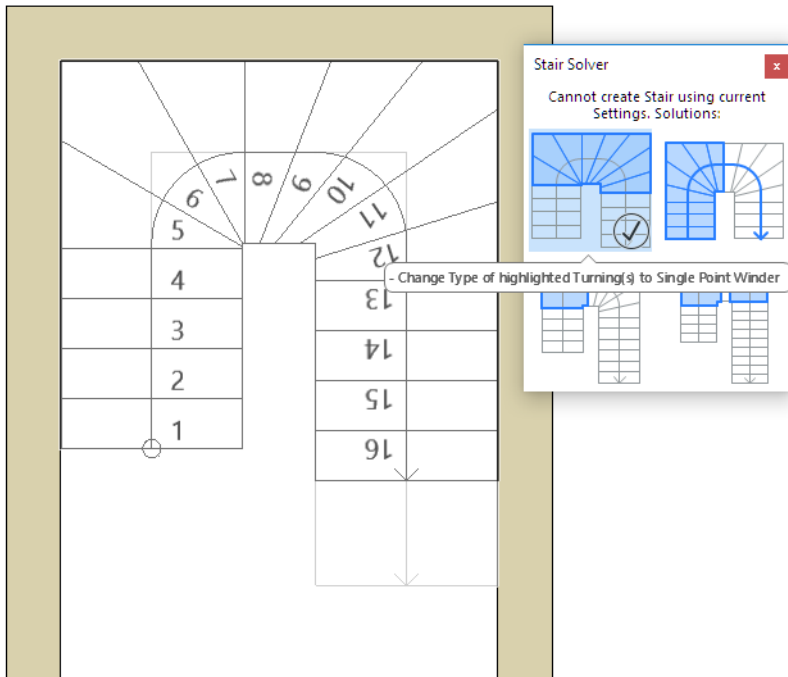
[Watch the Video](#)

Automatic “Solver” Options to Resolve Geometric Conflicts

If the Stair input geometry is not compatible with the Rules of the current Stair Settings, the Solver feedback appears automatically, with multiple possible solutions to modify the Stair.

The Solver may appear during or after input - e.g. when you enter or modify Stair geometry values or the Rules and Standards, in Stair Settings.

It can also appear when you change a Turning Type or a Segment Type.



Hover over each solution in the Solver dialog to see how it will change the Stair. Note the feedback on the Stair itself, as well as within the dialog box:

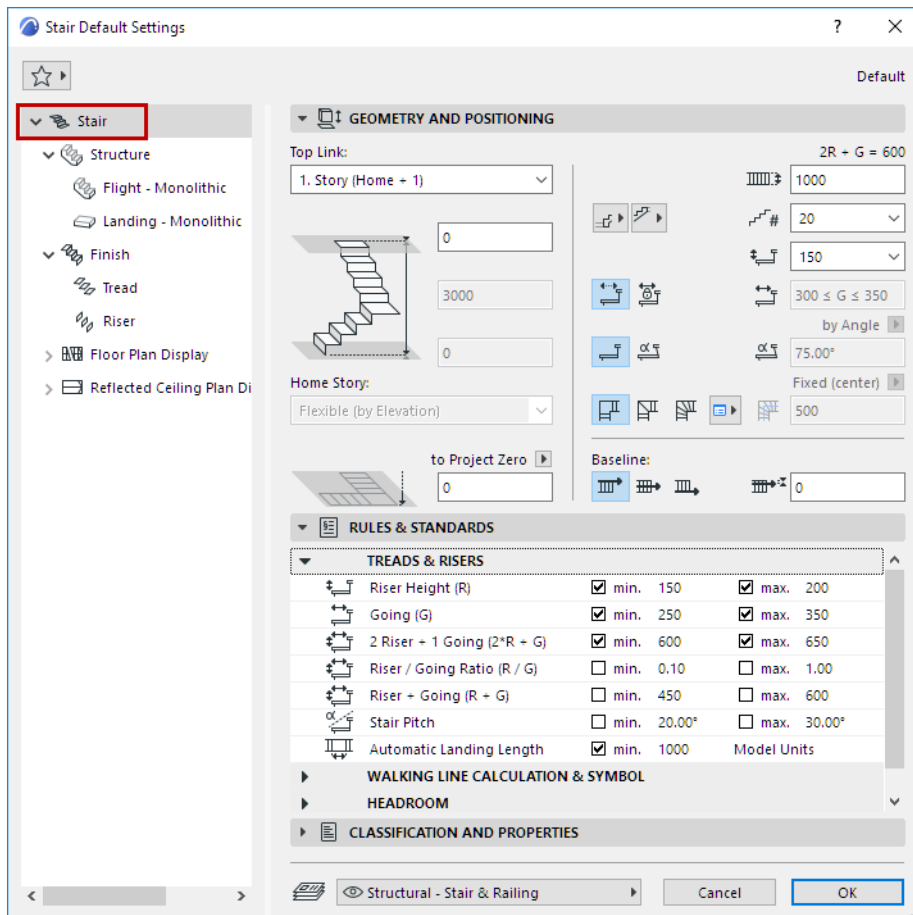
- Read the pop-up note to find out exactly what will be altered for each solution.
- The colored fill and/or line indicates the part of the Stair that will be altered.

Click the checkmark of the solution you prefer: the result is a Stair whose geometry has been altered, using the solution you choose, to conform to the Rules and Standards.

[See *Stair Settings: Rules and Standards*.](#)

Stair Settings (Main Element Settings)

Choose the main Stair node from the tree structure at left to access these settings.



Related Topics:

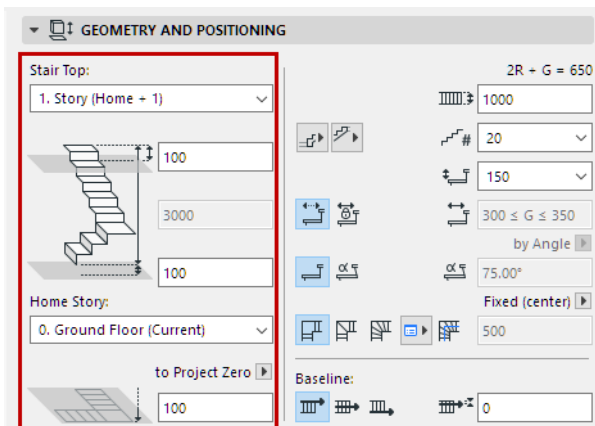
[Stair Settings: Geometry and Positioning](#)

[Stair Settings: Rules and Standards](#)

[Winder Types for Equal-Going Winders](#)

Stair Settings: Geometry and Positioning

Use this panel of the main Stair Settings page to define essential Stair geometry.



Stair Toplink

- Use this control to toplink the Stair to a particular Story.

If you later modify the positions and heights of stories in your projects, the heights of any linked Stairs will follow automatically, if the change still allows the Stair to follow its Rules and Standards. If there is a conflict, you are prompted to resolve it.

See [Toplinked Stairs and Changed Story Height](#).

- Offset:** Optionally **offset** the Stair top from its top linked story (the Stair height changes accordingly). This offset value can be positive, negative or zero.
The offset field is not available if the Stair has no top link.
- Not Linked:** Choose this to make it a fixed-height Stair instead of toplinked.
- Flexible:** This option is available before you begin Stair input (not for an existing Stair). The Stair is not linked, and the Stair height is not defined; it will depend on your input.
Use this option, for example, if you are creating a Stair using a fixed Riser height, but the number of Risers will be determined when you are done with input.

Stair Height

Value is affected by the top and bottom offsets, if any.

For a toplinked Stair, this field is not editable.

Bottom Offset

Offset the Stair bottom in either direction.

Home Story

Choose one of the following Home Story settings. The bottom of the Stair is physically linked to its Home Story. If you later modify the Home Story's position (e.g. redefine the floor level), the Stair will change its position (while respecting the defined offset, if any).

- Current story: Home Story is the current story, on which the Stair is placed.
- Click **Select Story** to bring up the full list of stories in the project, if they are not all shown in the list.
- Optionally, define an **offset** for the Stair bottom from its Home Story.

If you delete the Stair's Home Story, the Stair will be deleted, along with all other elements located on the story.

If you change a Stair elevation so that its baseline is moved to a different story, you have the option to make the Home Story change to match the element's new location.

[See Change Home Story by Elevation.](#)

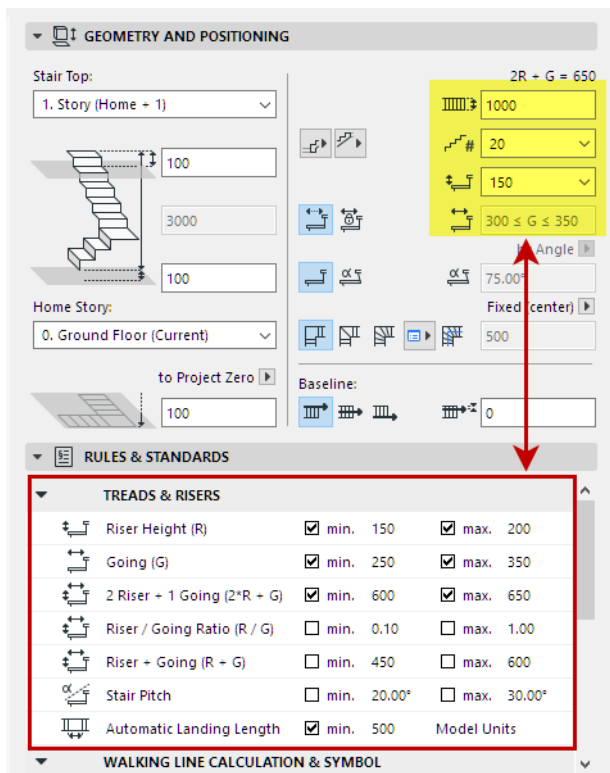
Bottom Elevation [to Reference level]: Calculates the current elevation of the Stair's baseline, as measured from the Reference level (by default, this Reference level is Project Zero). Click the pop-up arrow to change the Reference level, if needed.

Note: Reference levels are defined at **Options > Project Preferences > Reference Levels.**

[For general information on levels, see Reference Levels Preferences.](#)

Stair Geometry Controls

Use these fields to set essential Riser/Tread geometry (each setting is described below). Available values are tied to the range, if any, set in the Treads & Risers section of the Rules and Standards panel below.



Stair Tread/Riser Settings Depend on Ranges Set in Rules & Standards

Stair Width

Enter a value for the Stair width.

Number of Risers

Choose a value from the drop-down list. The available values depend on the ranges you set in the Rules and Standards panel, below, and on the Riser Height setting.

Note: If the Stair is being placed with Flexible Stair Height, the Riser Height control is greyed.

Riser Height

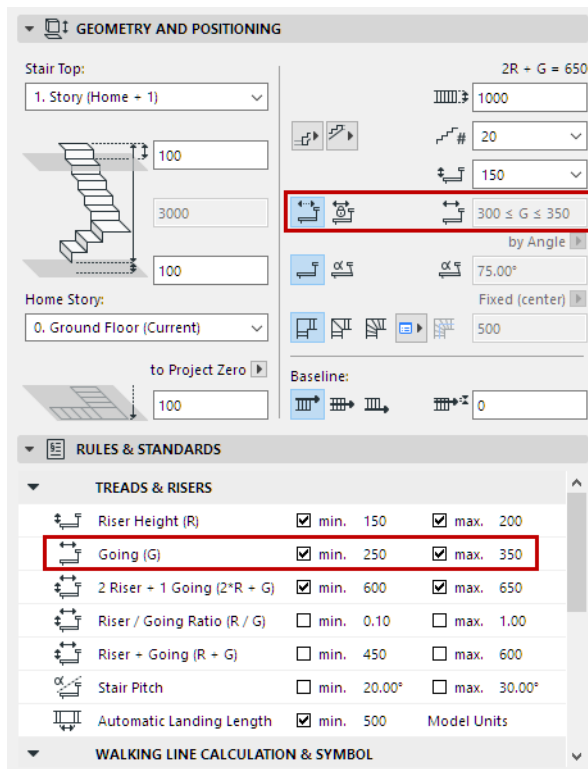
Choose a value from the drop-down list. The available values depend on the ranges you set in the Rules and Standards panel, below, and on the number of Risers.

Going Depth

Note: In the USA version of Archicad, “Goings” are known as “Runs”.

Choose either **Flexible** or **Fixed**.

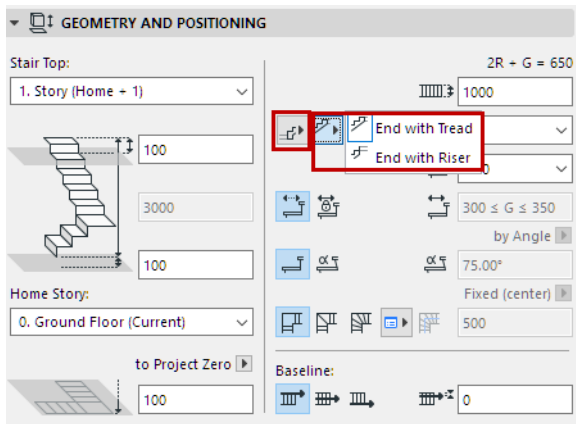
- **Flexible:** the Going value will be defined by Archicad automatically, based on the input geometry, and within the range (if any) set for the Going in the Rules and Standards panel.
- **Fixed:** Enter a fixed Going value here, within the range (if any) set for the **Going** in the Rules & Standards panel.



Start/End Stair with Tread or Riser

Use the pop-up controls to define how to begin and end the Stair: with a Tread or a Riser.

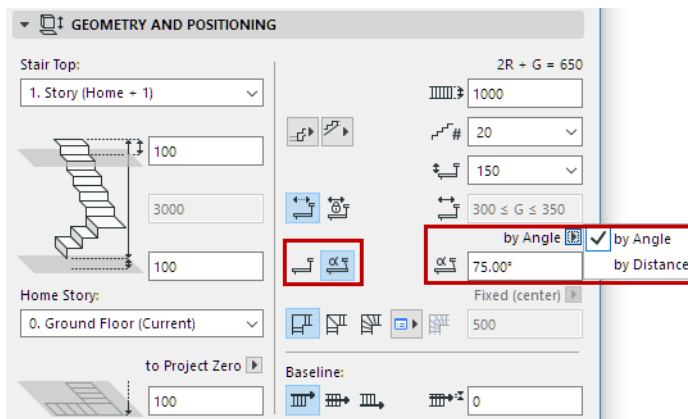
You can also switch this preference at any time on a placed Stair.



Riser Geometry: Straight or Slanted

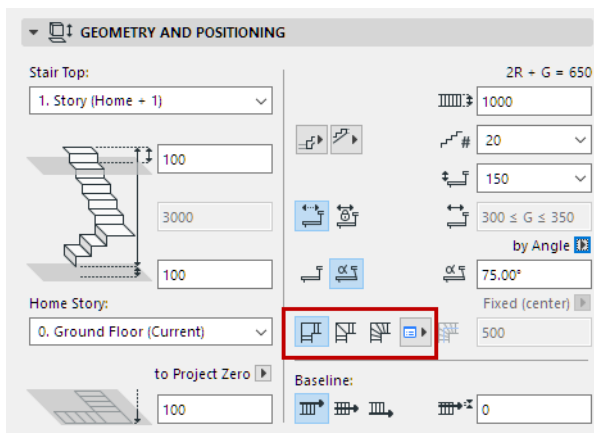
Choose either **Straight** or **Angled** for the Riser.

For angled: Define the angle value in the field at right, either with an Angle degree (of slanted Riser, relative to horizontal) or by Distance (that Riser bottom is set back from Riser top).



Turning Type

Choose a Turning Type for the Stair, then click the pop-up at right to set its specific geometry options.



- Automatic Landing
- Winder with Equal Goings

- Winder with Equal Angles

For details, see [Stair Turning Type](#).

Calculated Walking Line

The Stair’s Walking Line position is calculated automatically, and is normally in the center: its offset is half the Stair Width value. This is the **Calculated Walking Line**.

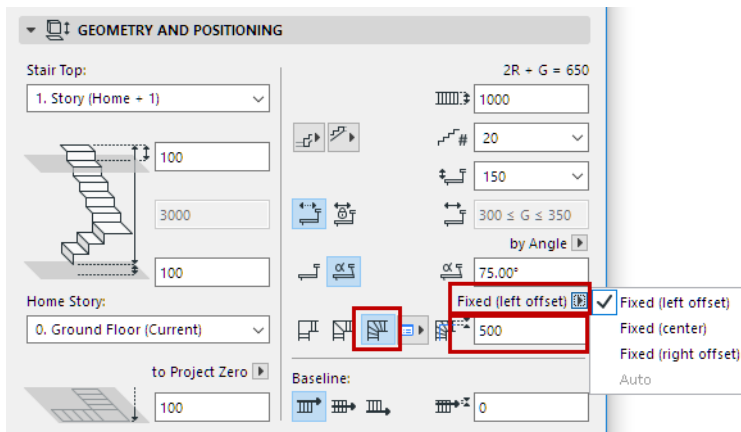
Note: To show/hide the Calculated Walking Line, use **View > On-Screen View Options > Stair Calculated Walking Lines**.

For a Stair Winder with Equal Goings, you have the additional option to offset the Calculated Walking Line:

Offset Walking Line on Turning (Winder with Equal Goings)

Use the pop-up to specify an offset for the Calculated Walking Line from the right or left edge of the tread, on the Stair Turning. This offset will affect Tread geometry.

This control is editable only if the turning is set to “Winder with Equal Goings”.

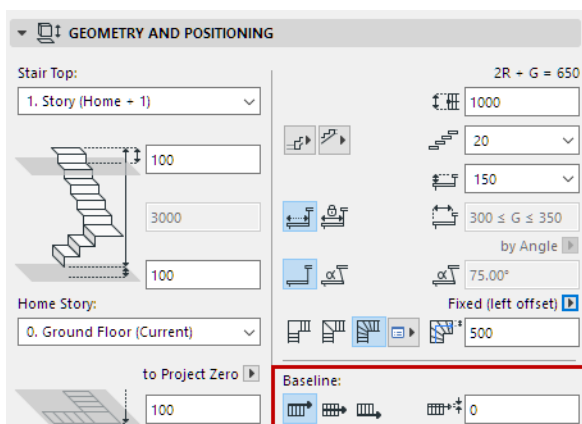


The Walking Line offset range is defined by the Rules & Standards for Walking Line Calculation.

Note: If you set a range for the Walking Line Offset (in the Rules and Standards panel below), then the Stair width has to be greater than or equal to two times this offset.

Stair Baseline

Define the Stair’s Baseline (Reference Line) position, and an optional offset.



The Baseline serves as the basis for the Stair’s Home Story, and as a reference for editing Stair geometry.

To show/hide the Stair Baseline on the Floor Plan, to make editing easier, turn on **View > On-Screen View Options > Stair Baselines**.

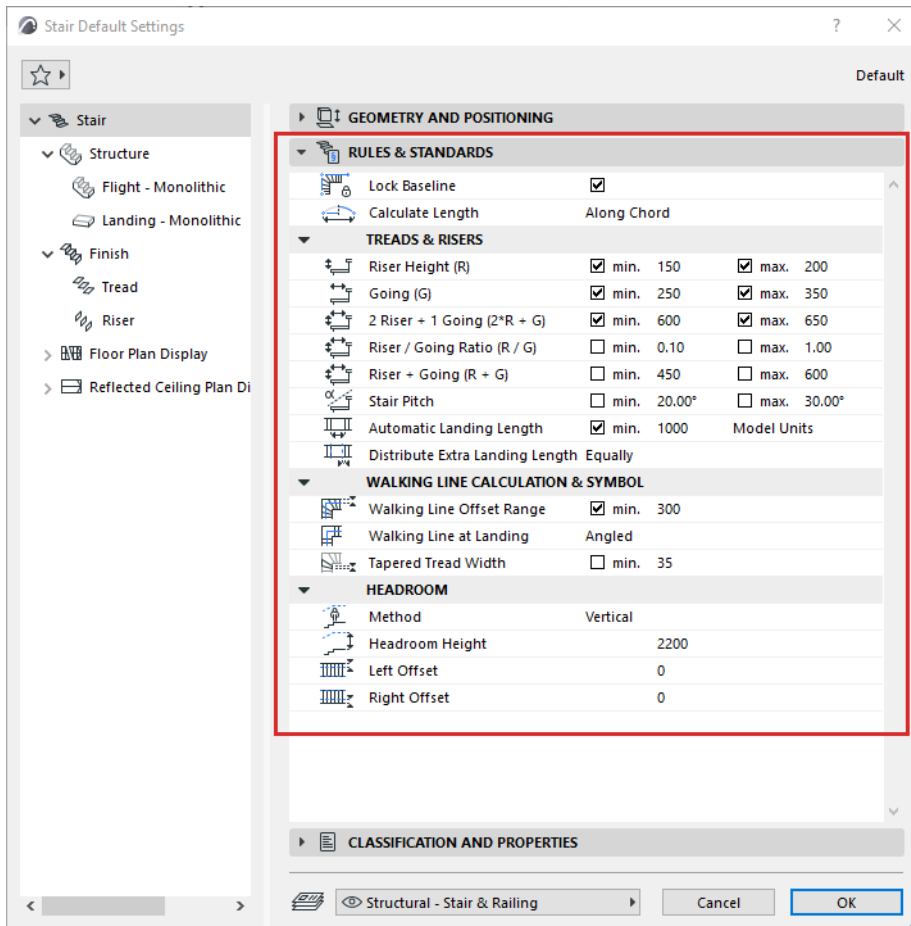
See also [Edit Stair Baseline](#).

[Watch the Video](#)

Stair Settings: Rules and Standards

Use this panel of the main Stair Settings page to optionally set a preferred range of values for Stair geometry variables, e.g. to meet local Stair regulations.

During Stair input, Archicad will automatically create a Stair that adheres to Rules and Standards you define here.



Lock Baseline

Note: Not available with “Along Arc (Legacy)” length calculation method.

Stair flights can automatically stretch or shrink as needed during Stair placement or editing - thus, the points clicked during input might be moved. To enable this, uncheck “Lock Baseline”.

If you unlock the baseline, flights between turns are automatically selected and unlocked. This extra flexibility helps you avoid impossible geometrical configurations, yet still place unusual stairs without needing the Solver

Calculate Length

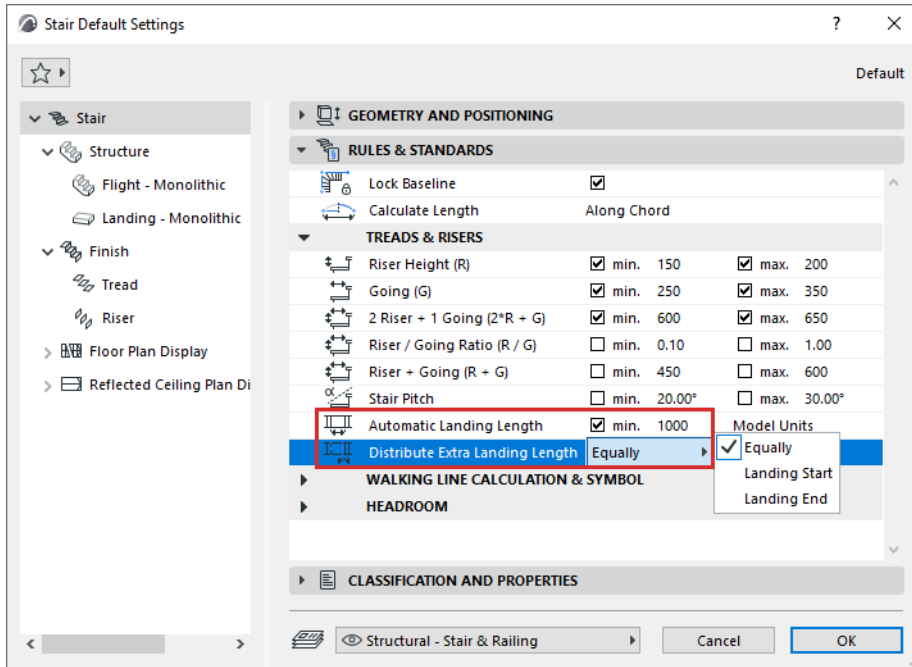
Choose a method to calculate the length of each Stair segment along turnings.

- Along Arc: segment measured along its arc
- Along Chord: segment measured along its chord (e.g. for German standard).
- Along Arc (Legacy): This method is used by all Stairs in pre-Archicad 25 versions.

Note: With the Legacy method, the “Lock Baseline” and “Distribute Extra Landing Length” options are not available.

Automatic Landing: Minimum Length and Distribution

For a Stair whose Turning Type is Automatic Landing: you can set a minimum Landing length here, expressed either in length units or number of goings.



Distribute Extra Landing Length

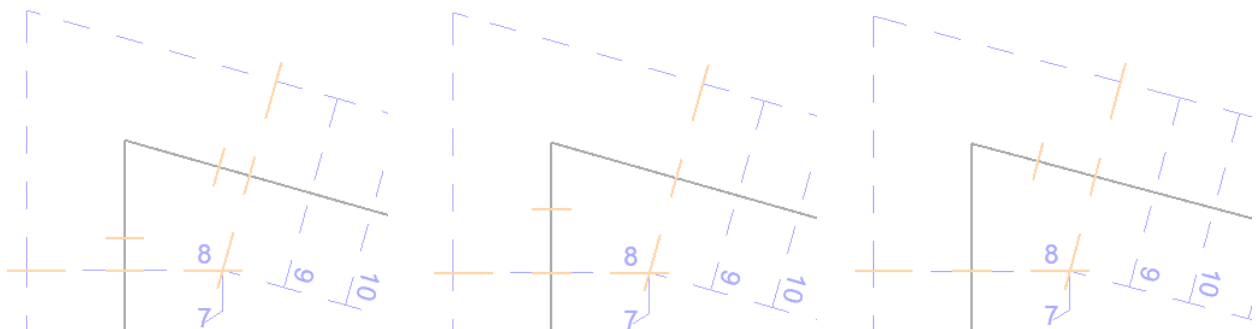
Note: Not available with “Along Arc (Legacy)” length calculation method.

For the extra landing length (over the minimum value that you set), choose which end of the landing to insert it, or to divide it equally on both ends of the landing.

To verify these automatic landing geometry results, turn on Stair Landing Calculation Guidelines (View > On-Screen View Options).

Note: These guidelines are visible only on automatic landings.

Guidelines highlighting different options: Equally, Landing Start, Landing End



Treads & Risers

These values define the range of possible input values in the Geometry and Position Panel - for example, you cannot input a Riser Height value that is less than the minimum you define in Rules and Standards.

In case these defined Rules & Standards conflict with the Stair geometry you input graphically or via Settings dialogs, Archicad will automatically propose alternatives and let you pick one.

[See *Automatic "Solver" Options to Resolve Geometric Conflicts.*](#)

Walking Line Calculation and Symbol

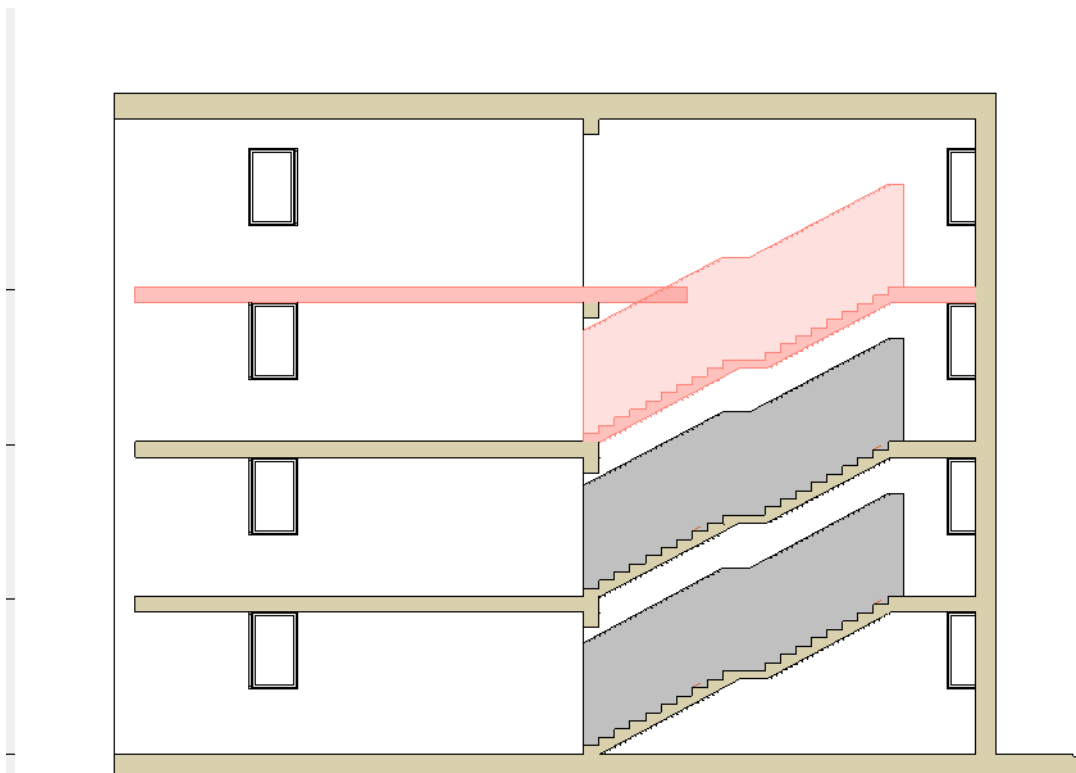
The **Walking Line Offset Range** defined here limits the values you can enter when defining an offset for the Calculated Walking Line.

[See *Offset Walking Line on Turning \(Winder with Equal Goings\).*](#)

Headroom

Set rules for displaying a Headroom symbol in 3D and Section/Elevation.

- Method: Choose to define the height either vertically, or perpendicular to the walking line.
- Headroom Height
- Offsets from Stair edges



Turn Headroom display on/off at Model View Options.

[See *Model View Options for Stairs.*](#)

You can run Collision Detection to locate headroom conflicts.

[See *Collision Detection.*](#)

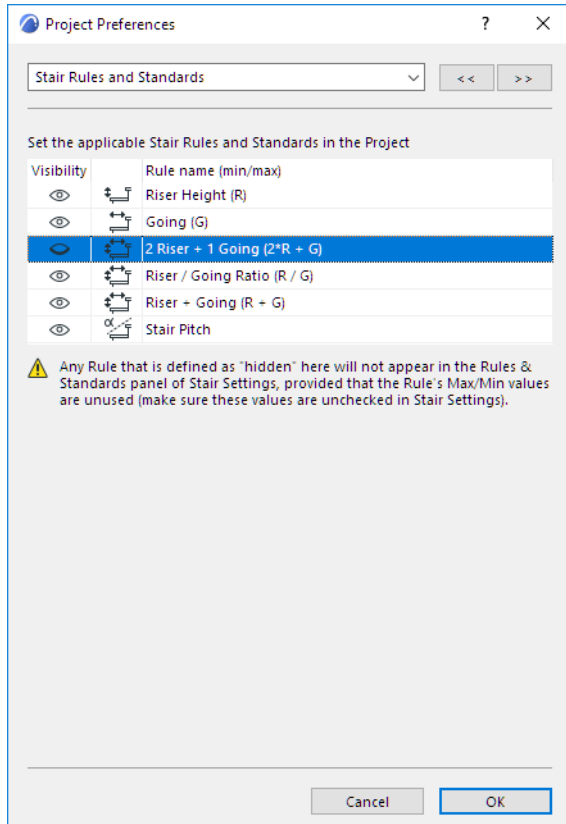
Save Rules and Standards in Project Template

The Rules and Standards settings are part of an Archicad Template file. Thus, you can set up company-specific and/or region-specific settings here, and re-use them via a template on any number of projects.

Stair Rules and Standards: Project Preferences

Use the Visibility (show/hide) icons at **Options > Preferences > Stair Rules and Standards** to show or hide particular rules for your project.

Any item that is set to be hidden (in Preferences) will not appear in the Rules & Standards panel of Stair Settings, provided that the rule is not in use.



If you hide a Stair Rule name in Project Preferences, yet the Rule still appears in the Rules and Standards panel of Stair Settings, there is a reason for it:

- The Rule has values assigned (to remedy this, uncheck the Min/Max values for that Rule in Stair Settings), or
- The Rule is in use by an existing Stair in the project

[Watch the Video](#)

Toplinked Stairs and Changed Story Height

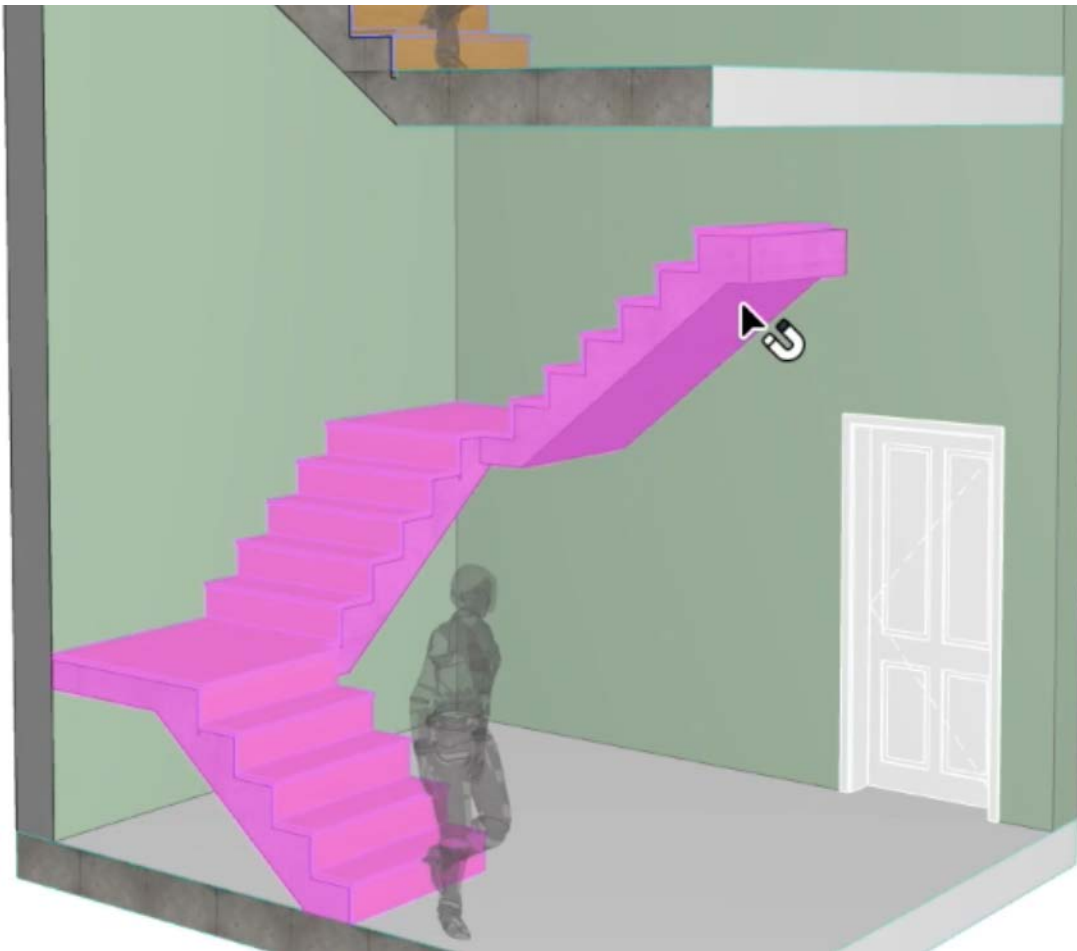
Like Walls, Stairs can be linked to Stories, and so can follow changes in floor height.

See [Stair Toplink](#).

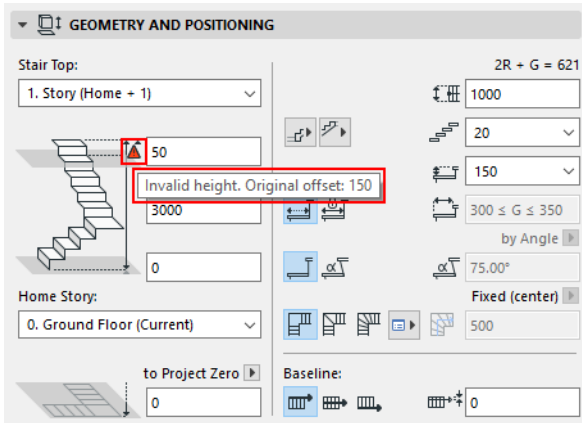
If the Story height changes only by a small amount, the Stairs will follow automatically (the number of risers will remain fixed, and only the height of the risers will increase – within the set Rules).

If Story heights change by a significant amount, then Archicad highlights the affected Stairs in the whole project until the user changes any of the given data (e.g. number of steps or Riser heights).

Note: To set the highlight color of invalid Stairs, use the control for Inverted elements at **Options > Work Environment > On-Screen Options**.



Stair Settings provides feedback that indicates the changed parameters:



To make the Stair valid (i.e. to extend to the upper story), enter the original offset value into the Settings dialog, then change the number of Risers if necessary.

Find Invalid Stairs

To easily locate **Invalid Stairs** in a project, use the built-in **Invalid Stairs** Criteria set with Find & Select.

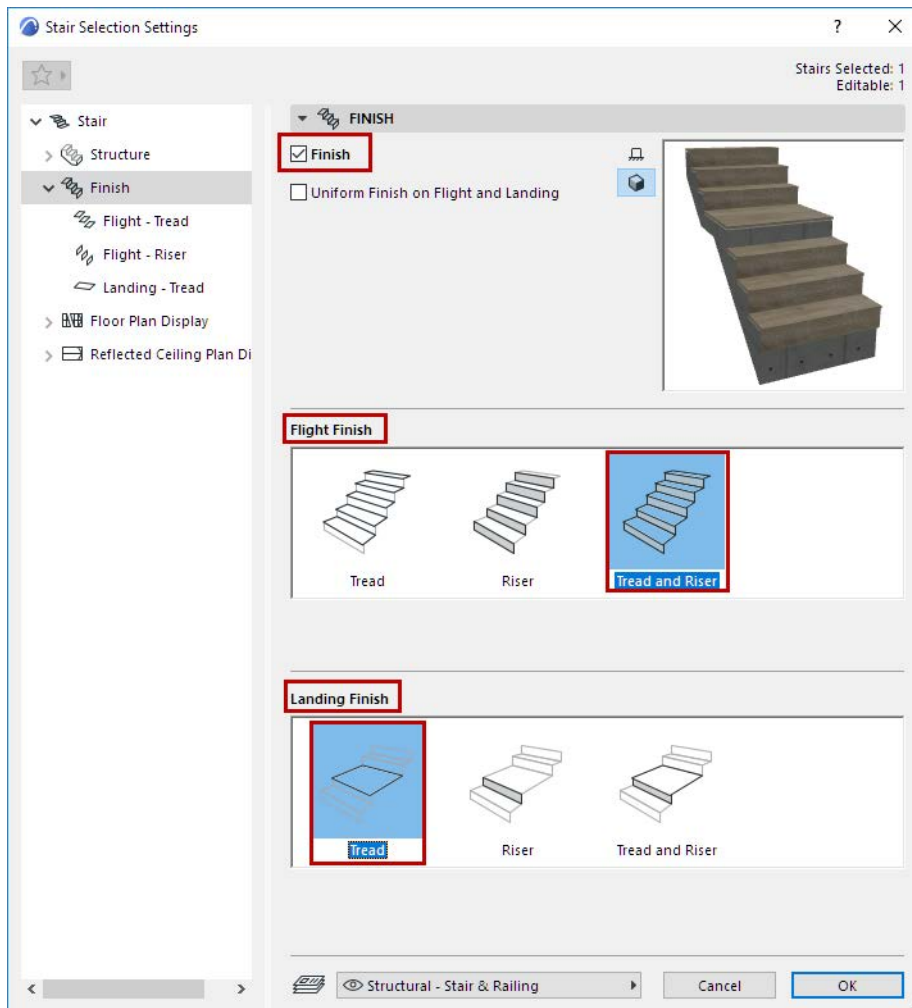
Tread and Riser Finishes

From Stair Settings, select **Finish** from the tree to open this page.

Check **Finish** to add Tread/Riser Finish components to the Stair.

Below, click the components you need:

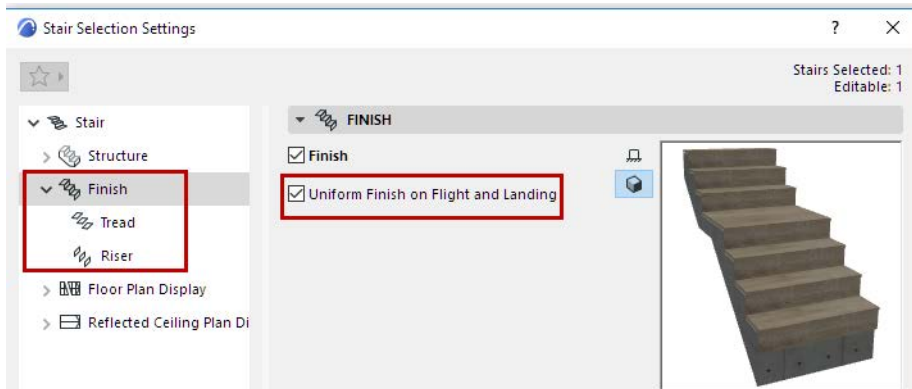
- **Flight Finish:** Apply to Tread, Riser, or both
- **Landing Finish:** Apply to Tread, Riser, or both



The components you choose appear in the tree structure: a separate settings page for each of the Tread/Riser/Landing/Flight finishes.

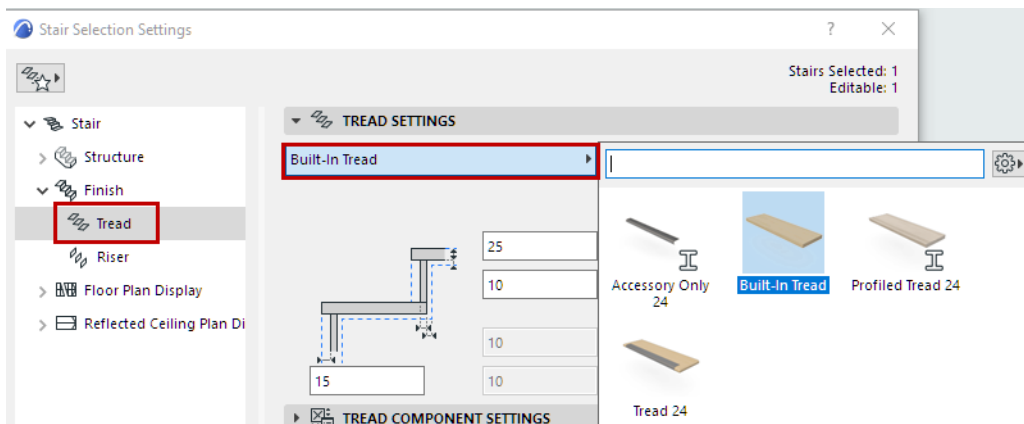
Uniform Finish on Flight and Landing

Check this option to use the same Finish settings for all the Stair's Treads and Risers, on both Flights and Landings. The number of settings pages is thus reduced.

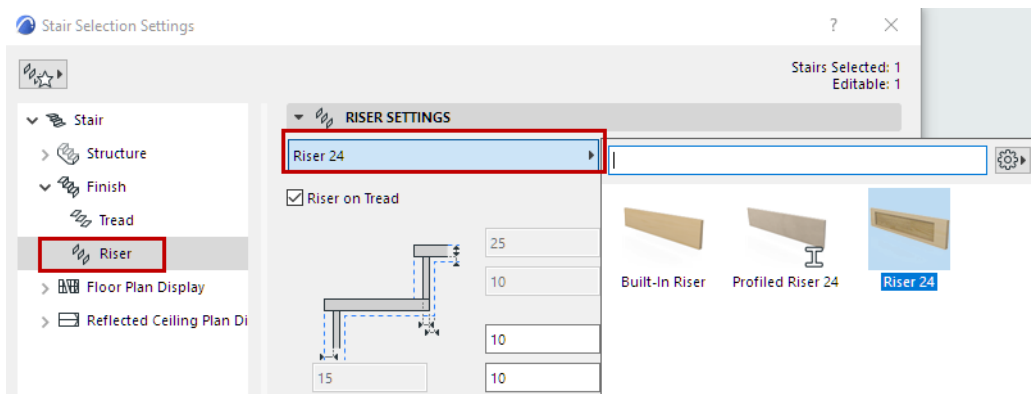


Tread and Riser Settings: Choose Component

On the Tread and Riser Settings pages, as applicable, choose a GDL component from the pop-up to serve as the Finish:



Choose Component as Tread Finish

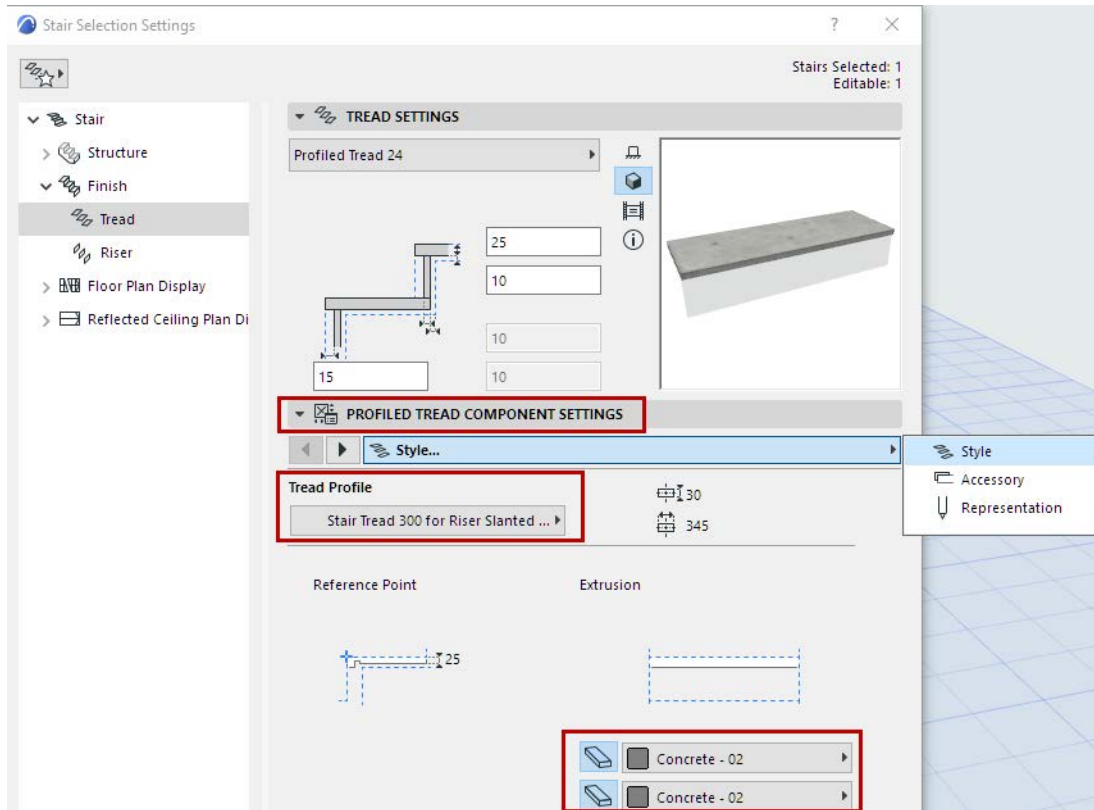


Choose Component as Riser Finish

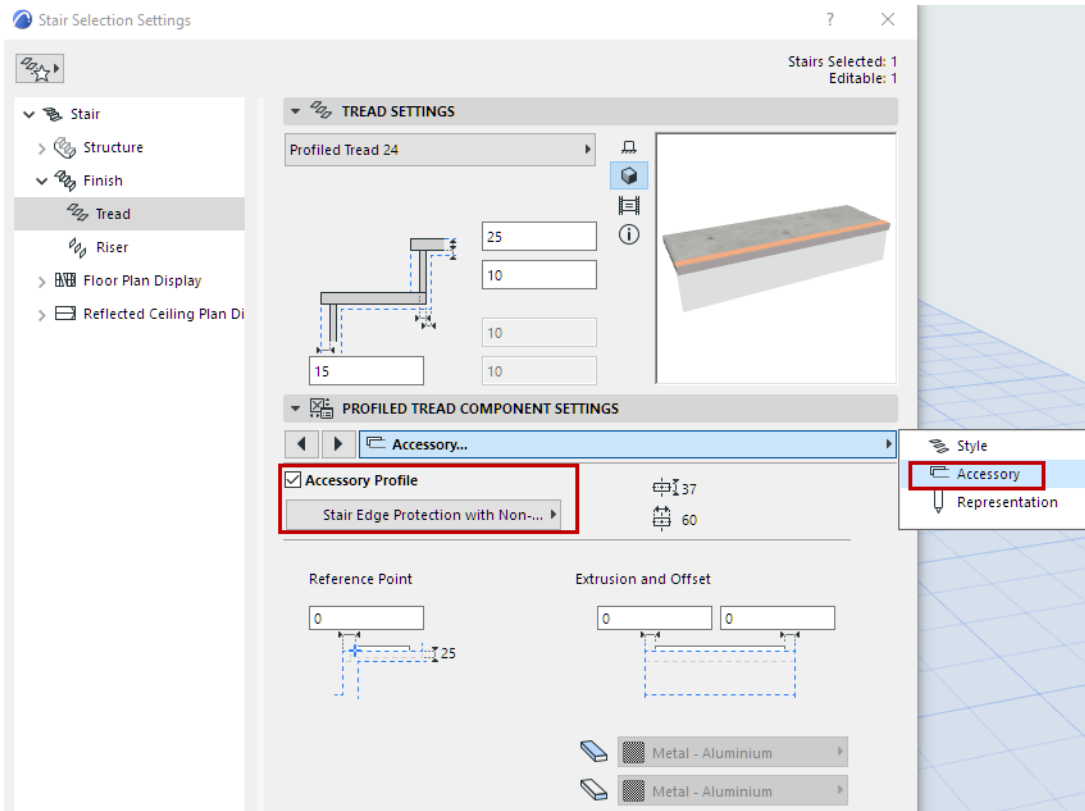
Riser on Tread: Check this to place the Riser on top of the Tread. (If this is unchecked, the Riser extends to the bottom plane of the Tread - see the diagram.)

Tread and Riser Component Settings

Use the Component Settings panel to specify the shape and appearance of the chosen finish. The settings vary depending on the chosen component:



Choose Tread Profile and Building Materials



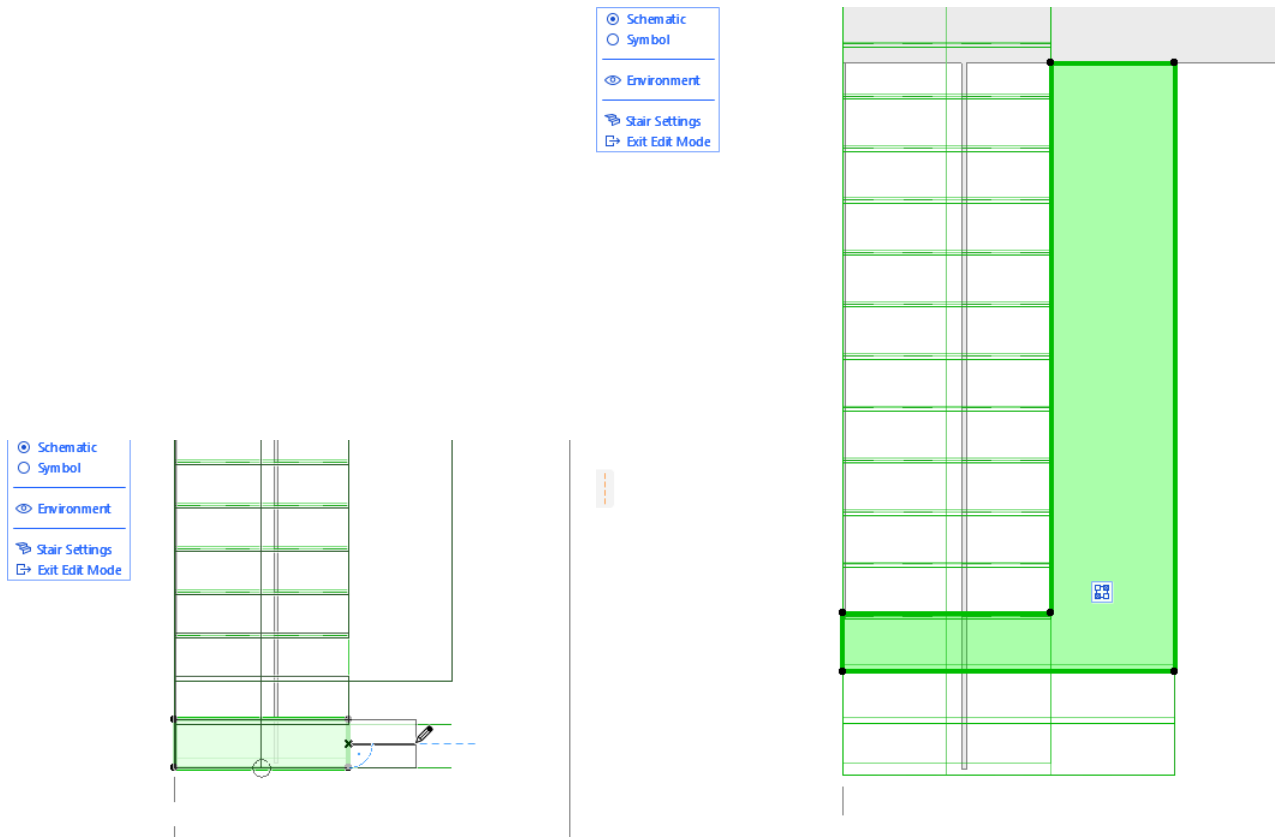
Choose a Separate Accessory Tread Profile

Edit Individual Treads or Risers

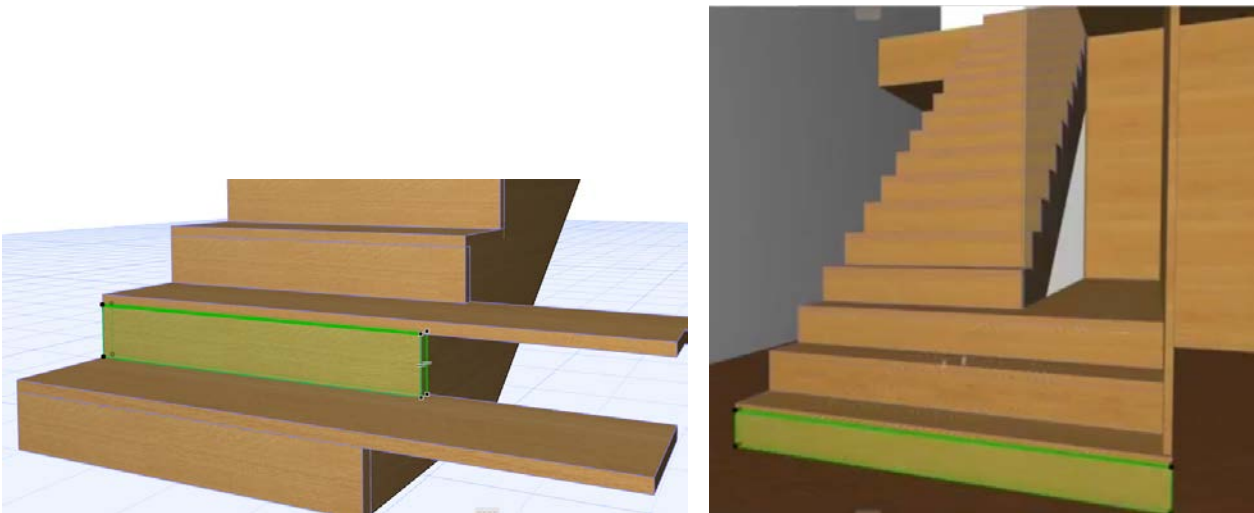
Use Edit mode to edit individual Tread and Riser components. You must edit Treads and Risers separately. For example, edit the shape of certain Treads and Risers of a Stair that has a built-in cabinet structure.



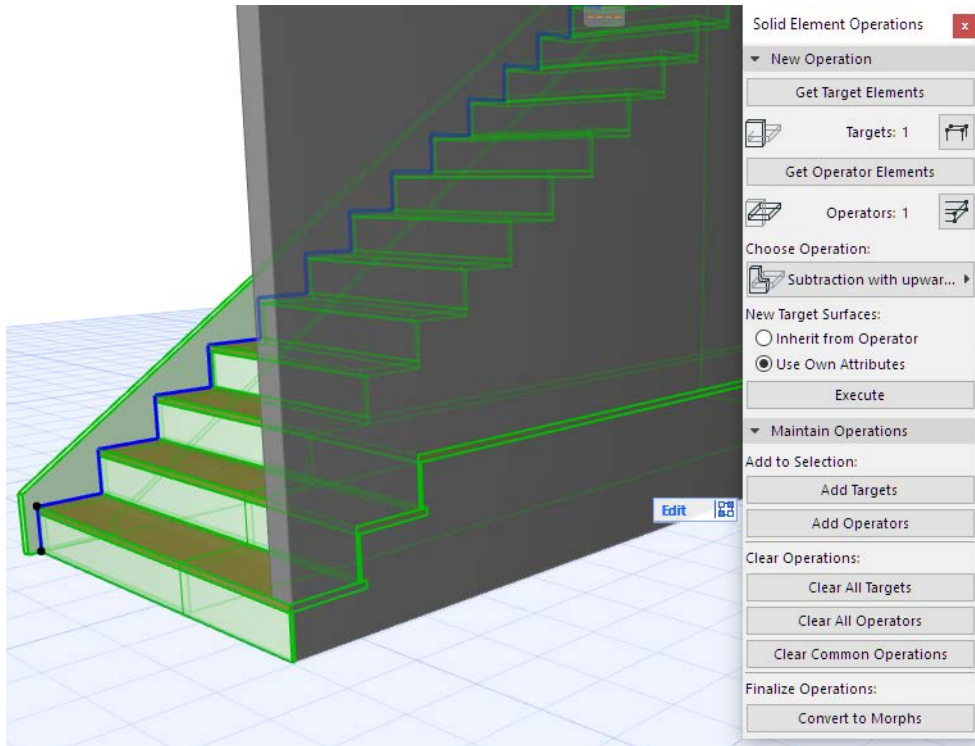
1. Select the Stair and enter Edit mode. ([See Enter Edit Mode.](#)) Here, use the Schematic view.
2. Select each Tread individually. Use the familiar pet palette commands to edit them.



- 3. Switch to the 3D window: note that you are still in Edit mode.
- 4. Continue editing the Stair components - here, edit the Risers individually.



5. When the Stair is complete, use Solid Element Operations to connect it to the Wall.



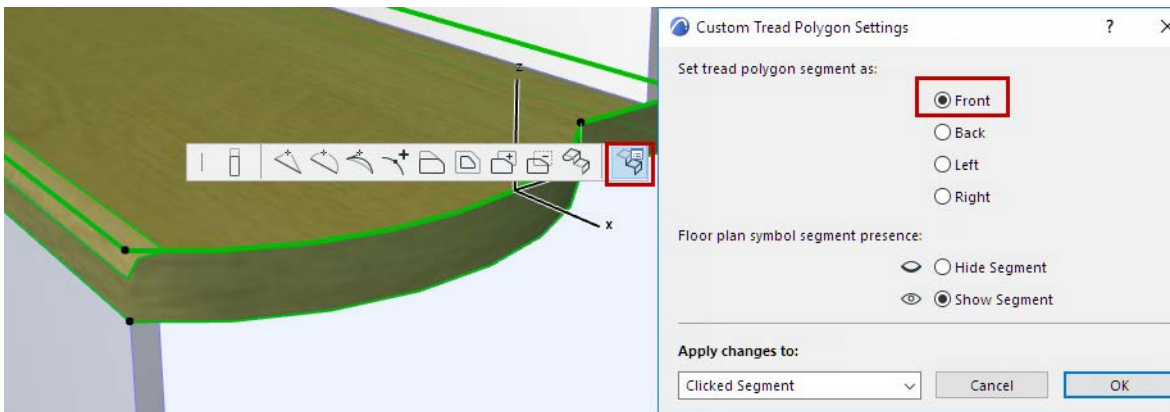
Define Custom Tread Segment: Front, Back, Left, Right

When you edit a Tread to change its shape, the Tread accessories - such as the nosing - might not extend to the customized parts.



In this case, you can customize any Tread segment to behave like the front, back, left or right side of the standard tread shape.

1. Select the Stair, then enter Edit mode.
2. Select a Tread, then click on one of its side segments.
3. From the pet palette, choose Custom Tread Polygon Settings.



4. Set tread polygon segment as “Front”.

Note: To change all tread segments, choose **All Segments** from the **Apply changes to** pop-up.

5. The customized Tread segment now behaves like the front of the Tread and also displays the nosing.

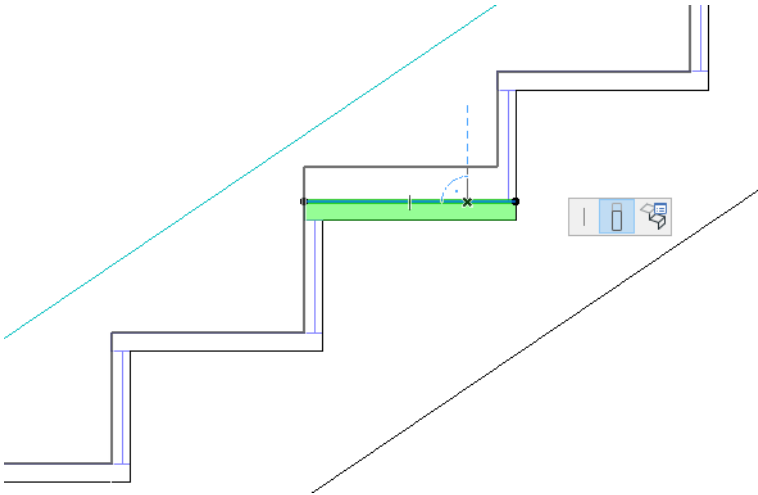


Edit Individual Riser Heights in Edit Mode (Section/Elevation)

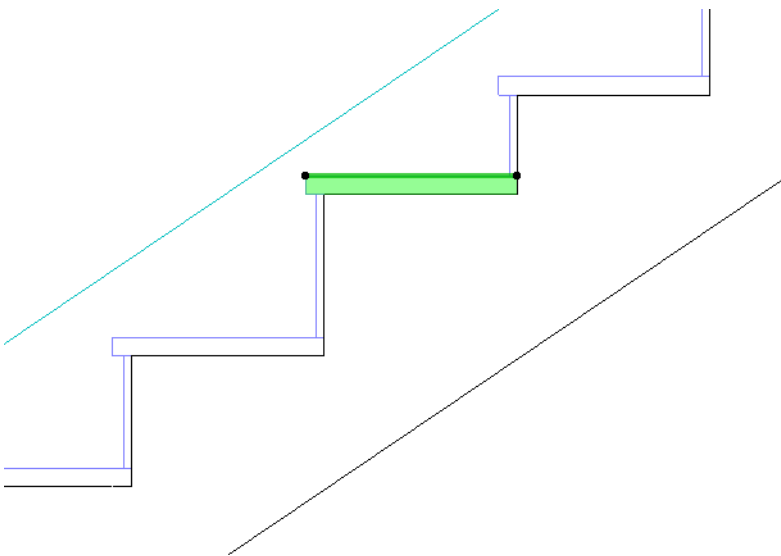
To edit the height of each Riser separately, you can use Edit mode from the Section or Elevation window. (For example, to document a historic Stair with Risers of various heights.)

1. Select the Stair in the Section/Elevation window, then enter Edit mode.
2. Select a Tread.

3. Drag the top of the Tread to the desired location.



The Riser Height is adjusted. Note that the entire Stair shape is affected (including the connecting Structure).

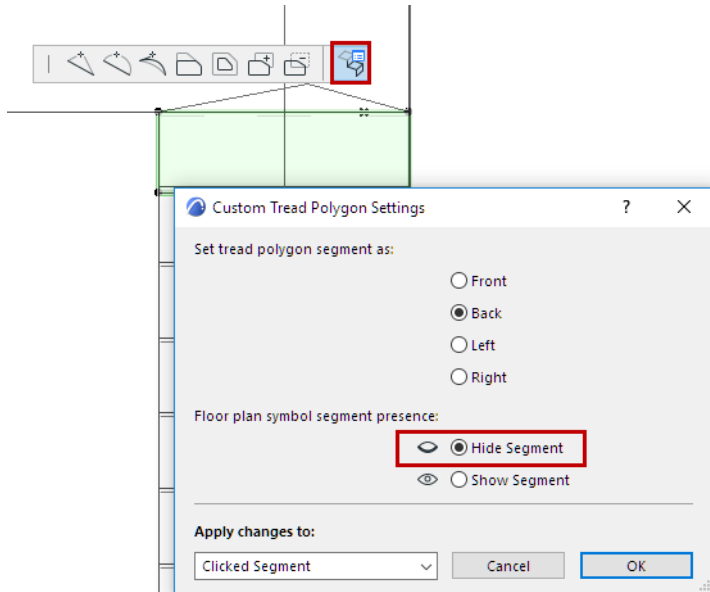


Hide Any Edge of a Single Tread

You may wish to hide a tread line in 2D. For example, you can hide the top tread line, if the last tread is on the same level as the slab of the landing.

1. Select the Stair in the Floor Plan window, then enter Edit mode.
2. Make sure that Edit mode is using the Schematic view
3. Select a Tread, then click on one of its side segments.

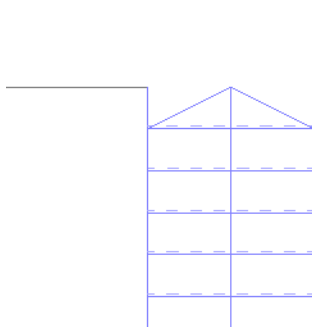
- From the pet palette, choose Custom Tread Polygon Settings.



- Click **Hide Segment**.

Note: To hide all segments: Choose **All Segments** from the **Apply changes to** pop-up.

- View the result in Symbol view.



See also [Walking Line End Symbol: End at Second to Last Tread.](#)

[Watch the Video](#)

[Watch the Video](#)

[Watch the Video](#)

Flight and Landing Structures

In Stair Settings, select the Structure node in the tree to open the Structure tab page.

Use this page to select Structure types for the Stair Flights and for the Landings.

Check the **Structure** box to use a Stair Structure.

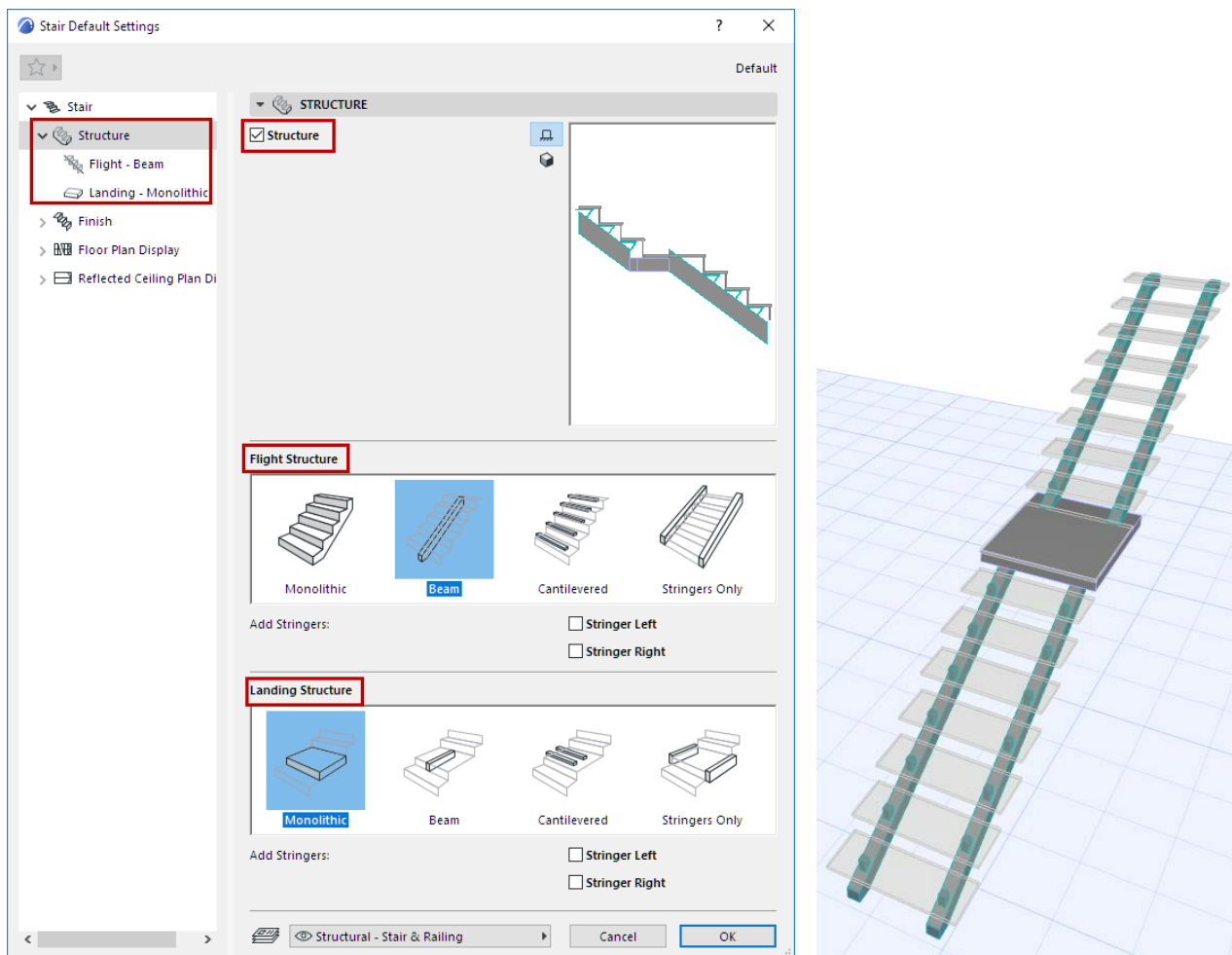
Note: You can turn the Structure on or off for any single selected Flight or Landing, using Edit Mode. ([See Select Stair Component for Editing.](#))

Choose Structure Type

Structure types include Monolithic, Beam, Cantilevered, or Stringers Only.

Note that the contents of the Settings tree structure reflect your choices here.

As a rule, it is recommended to use the same structure type for both Flight and Landing: this will ensure that all Stair sub-elements will be documented correctly.

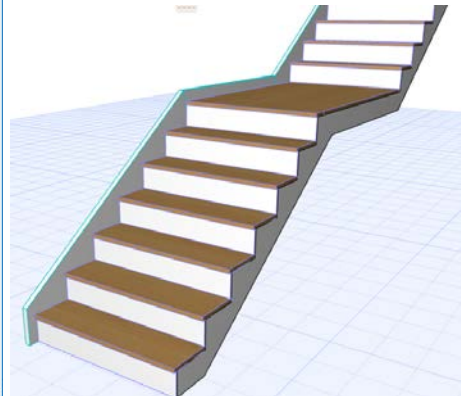
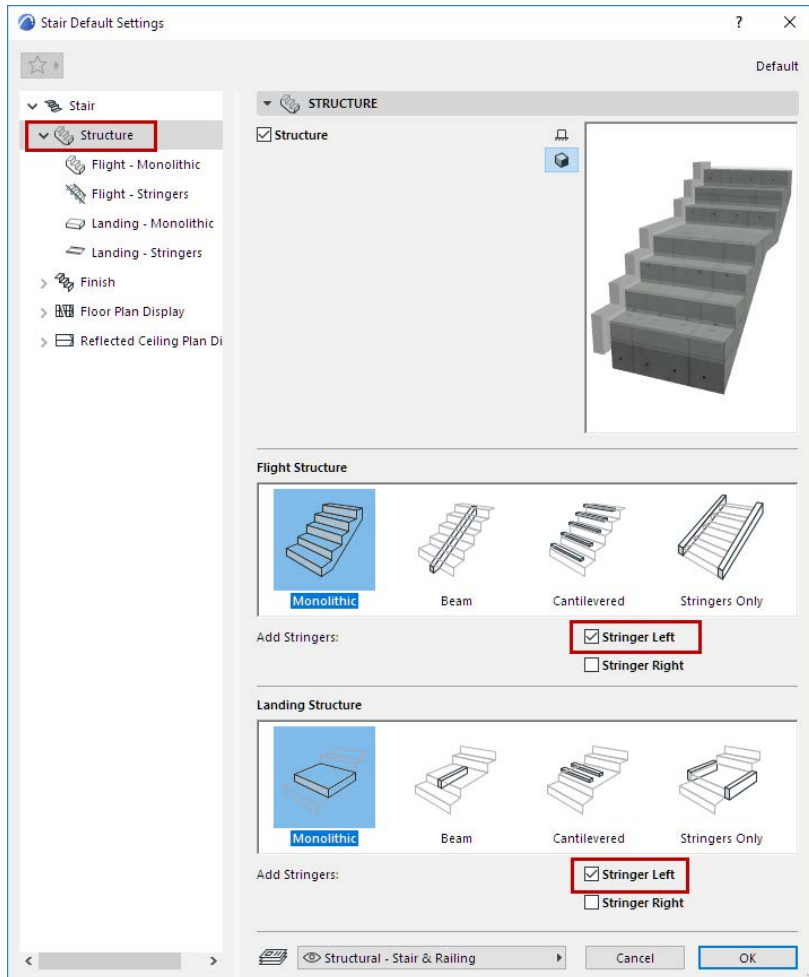


Note: It is also possible to use two different Structure types for a single Flight.

See [Split Stair Structure](#).

Add Stringers

Optionally, add Stringers to the Left and/or Right side of the Flight or Landing. Notice that Stringers can be added to every other Structure type. Stringers tab pages are added to the Structure node accordingly.



In the tree structure, click on a component (Flight or Landing) to open the respective Structure Settings page. The settings here correspond to the Structure type (e.g. Monolithic) you chose on the main Structure page.

Related Topics:

[Monolithic Flight and Landing Structures](#)

[Beam Flight Structure](#)

[Beam Landing Structure](#)

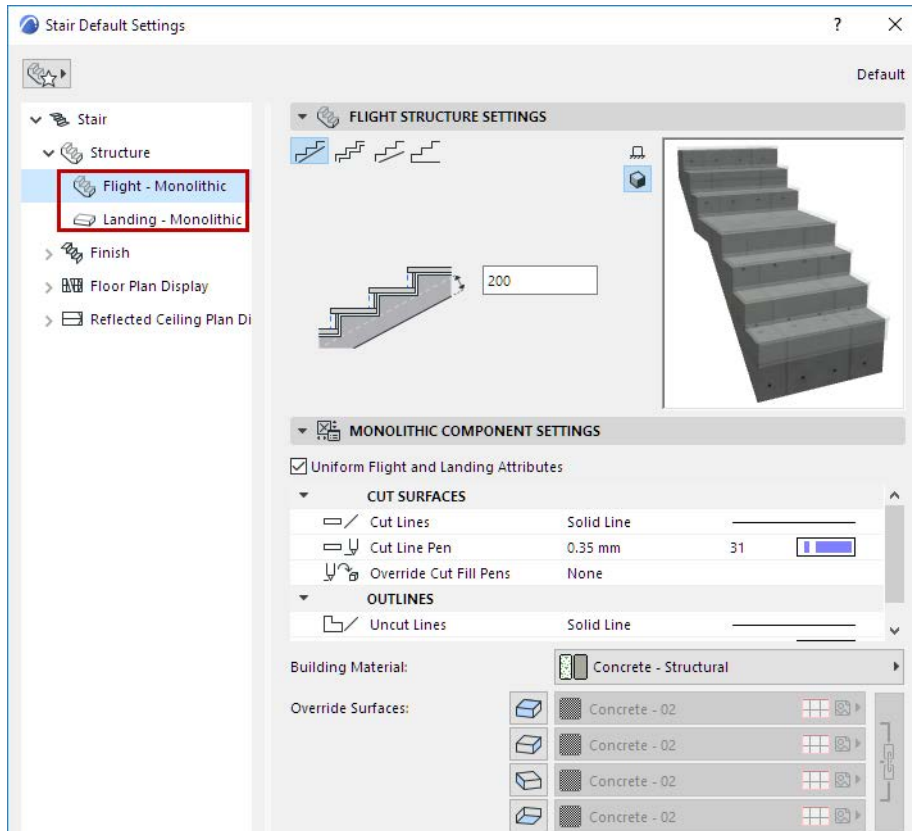
[Cantilevered Flight and Landing Structure](#)

[Stringers Flight and Landing Structure](#)

Monolithic Flight and Landing Structures

These settings are available if you chose a Monolithic Structure type for the Flight and/or Landing.

See [Choose Structure Type](#).

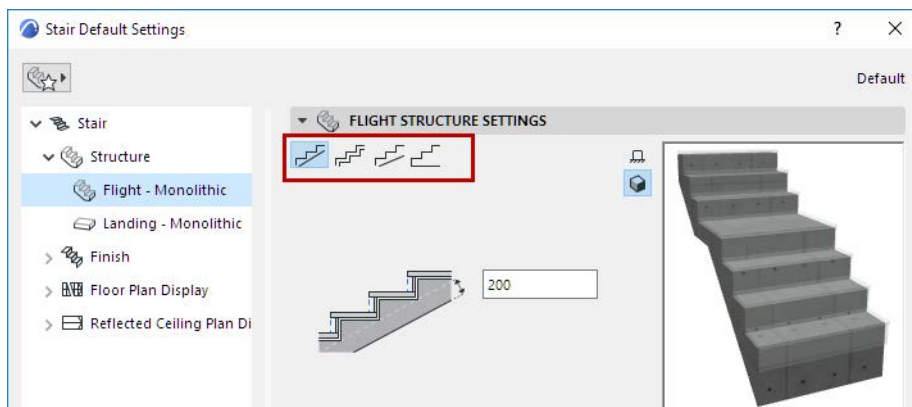


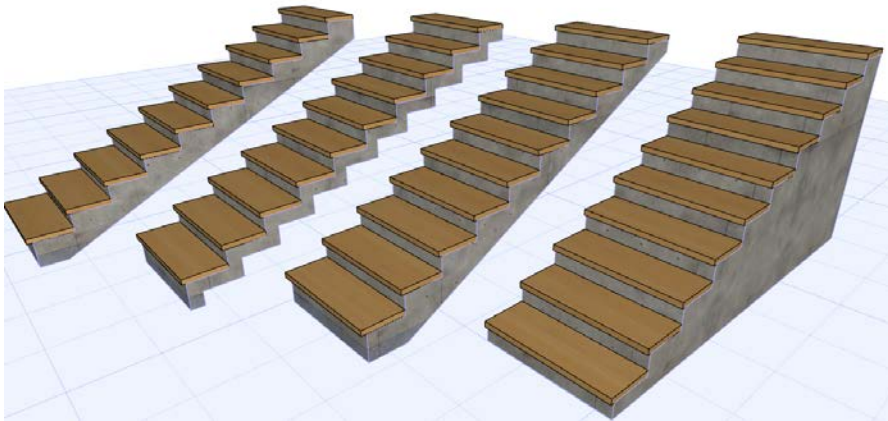
Monolithic Flight Structure: Choose Shape

Use the four buttons to choose a shape for the bottom of a Monolithic flight. From left:

- Flat
- Stepped
- Variable Thickness (at each end)
- Filled

Below, enter values for the thickness/offset variables as applicable. Note the feedback in the Preview window.



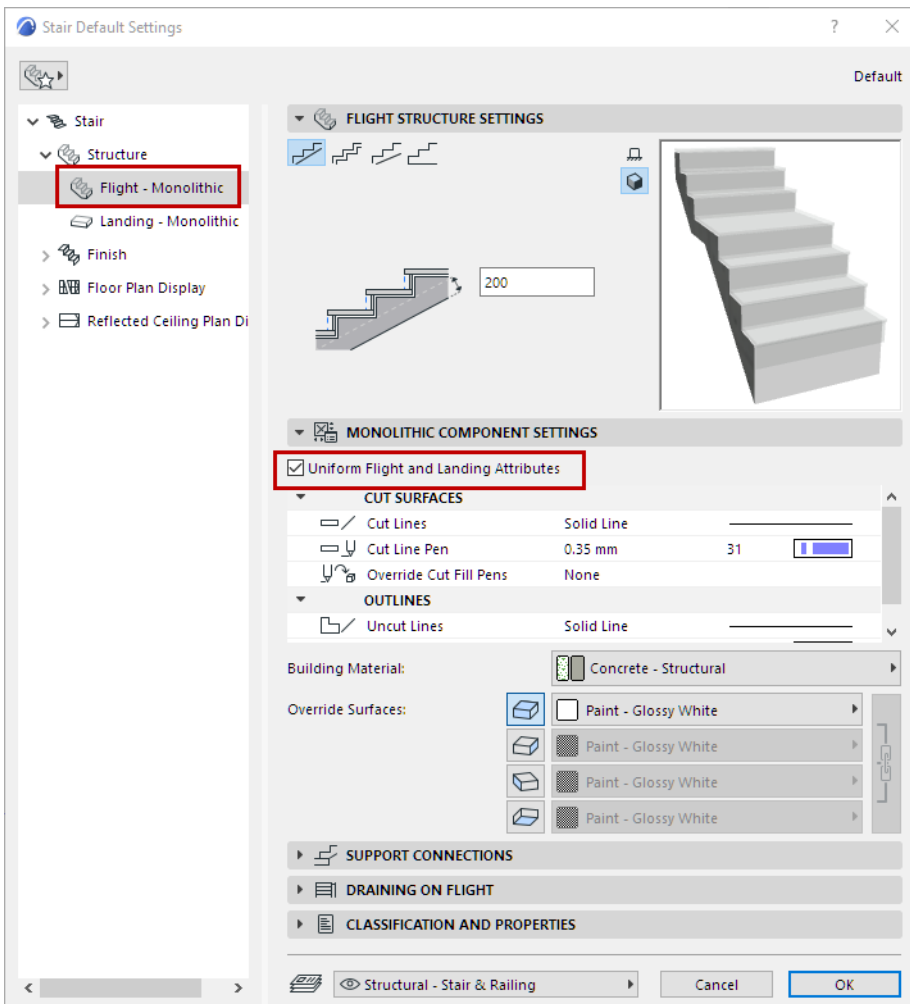


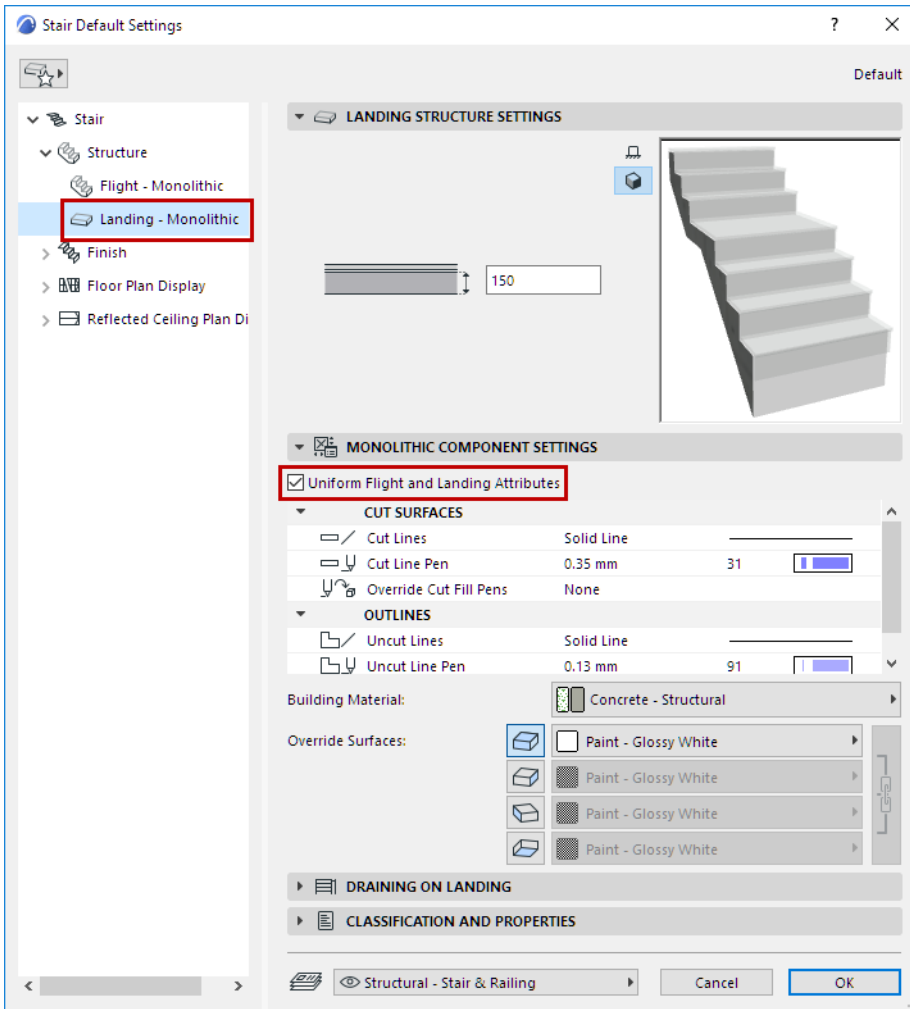
Monolithic Flight Shapes

Monolithic Support Component Settings

Use this panel to set attributes, including Building Material, for the Monolithic Flight/Monolithic Landing. Check **Uniform Flight and Landing Attributes** to apply the same attributes to both Monolithic structures (Flight and Landing).

Note: The “Uniform attributes” checkbox applies only to Structure attributes, not its thickness.

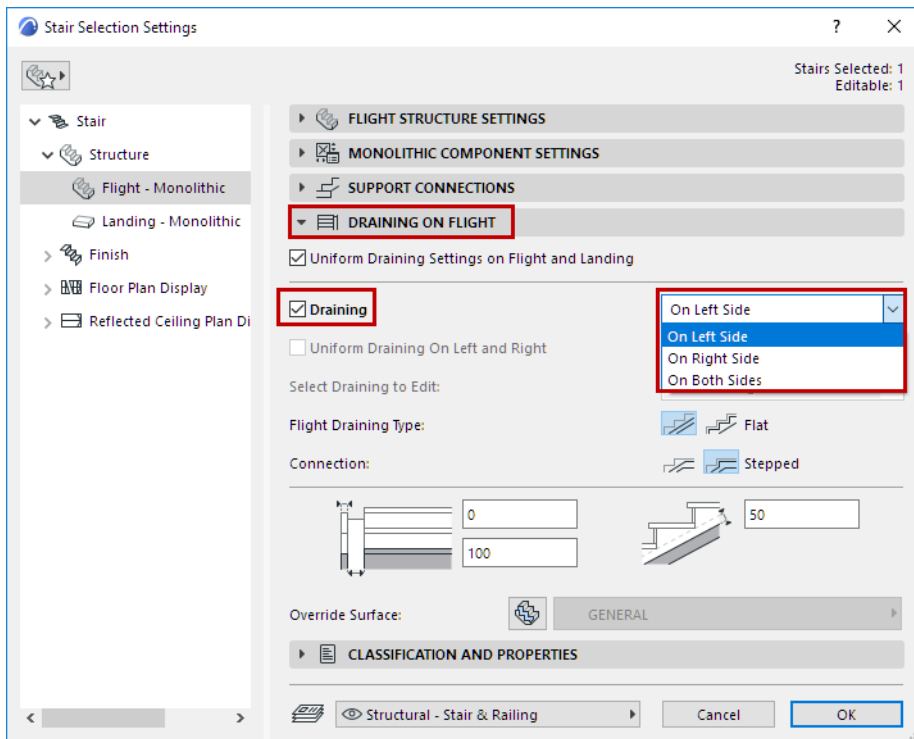




Draining on Flight/Draining on Landing

This panel is available for Monolithic structures only.

Check **Draining** to place a Draining on the left and/or right side of the Stair structure (available for both Flight and Landing).



Check **Uniform Draining Settings on Flight and Landing** to apply settings to both Stair structures.

For Draining on Flight:

- Choose **Draining Type (Flat or Stepped)** for both Flight and Connections. This setting will be used for the Draining on Landing, if any.

Set Draining width/depth values as applicable, and an optional offset.

If needed, apply a Surface override to the Draining.



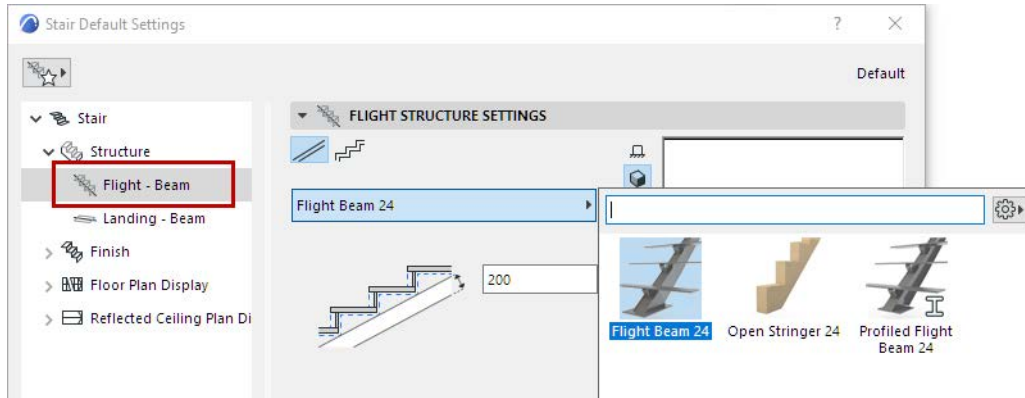
Monolithic Stair with Draining on Both Sides

Beam Flight Structure

These settings are available if you chose a Beam Structure type for the Flight.

[See Choose Structure Type.](#)

Beam Structures are GDL elements: choose a component from the pop-up to serve as the Beam support.

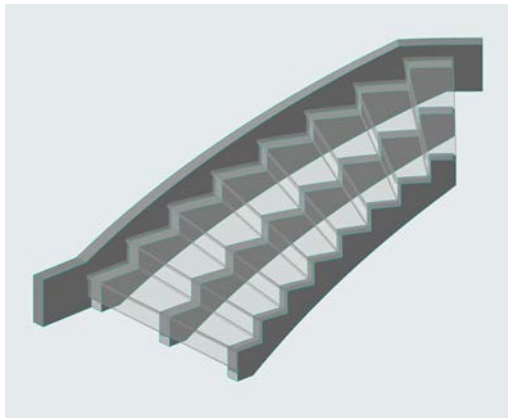
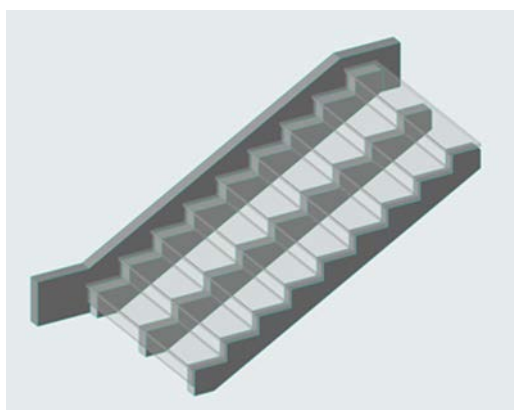
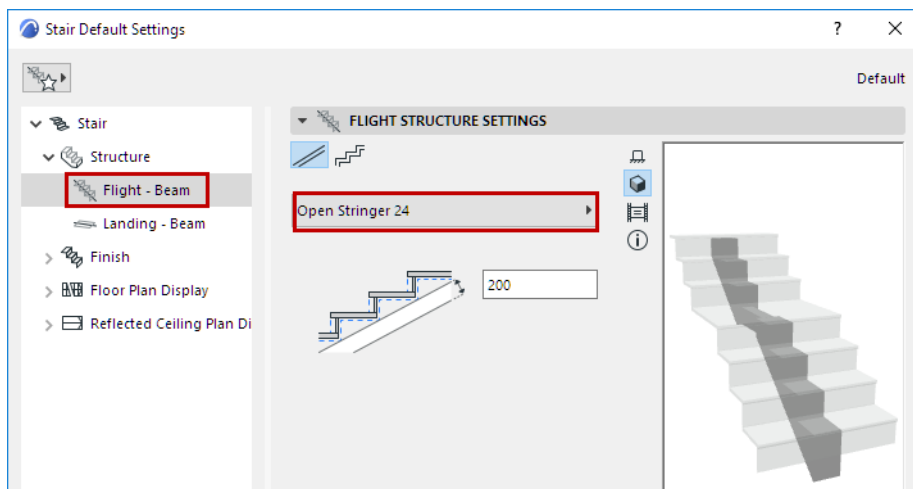


Components include a general Beam; profiled Beam; and Open Stringer (see below).

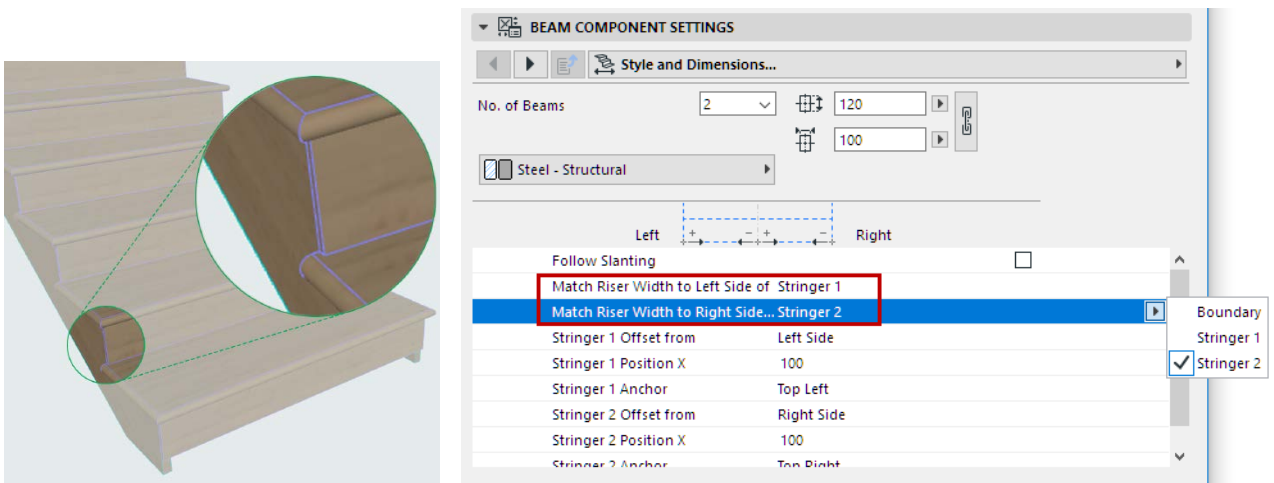
[For details, see Custom Sub-Elements for Stairs and Railings.](#)

Use Open Stringer (Beam Flight Structure)

An “Open Stringer” structure is available from the Beam - Flight component pop-up list.

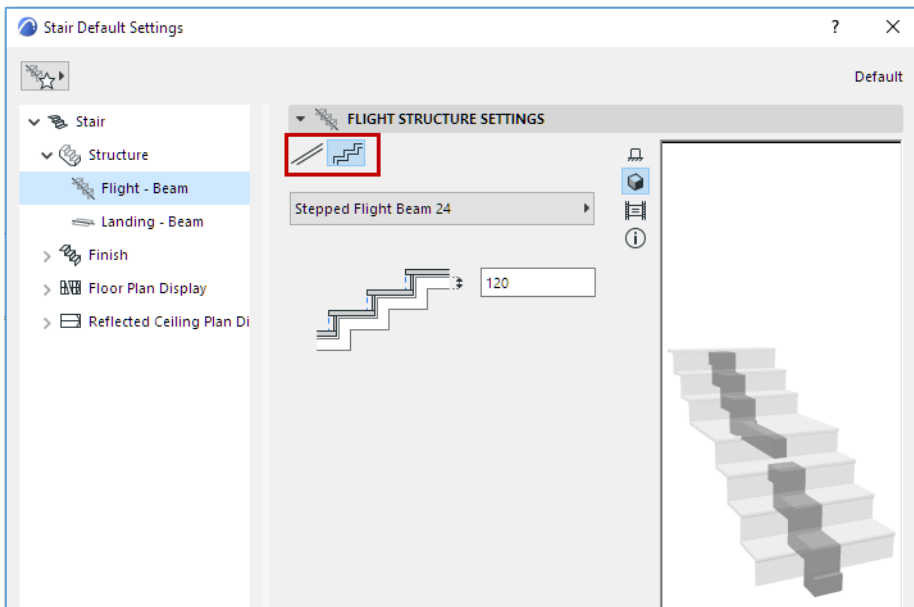


Use Beam Component Settings to position the Stringers. Note the options to match Riser Width to either the Stringers (if they are offset) or the Stair Boundary.

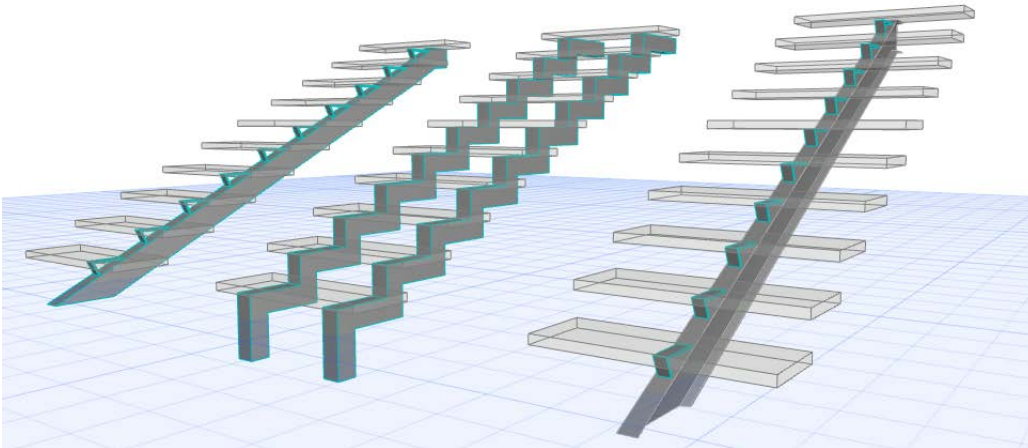


Choose Shape

In Beam Flight Settings, choose a Beam Structure shape: either flat or stepped. Profile components are also available.



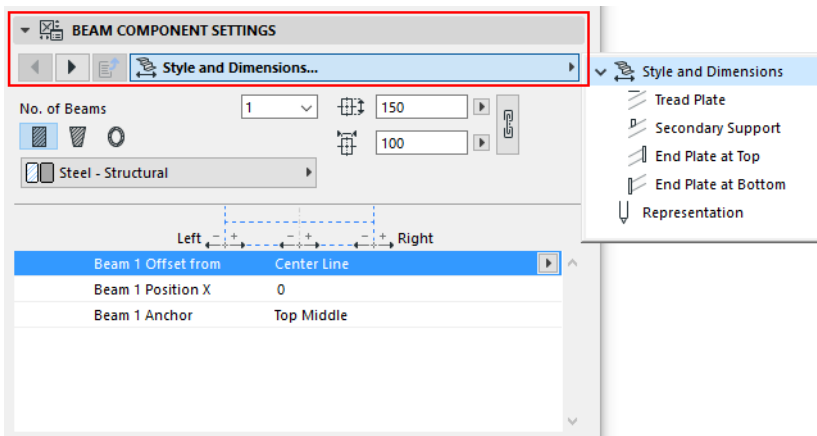
Below, enter a value for the Beam thickness. Note the feedback in the Preview window.



Beam Flight Structures: Flat, Stepped, Profiled

Beam Component Settings

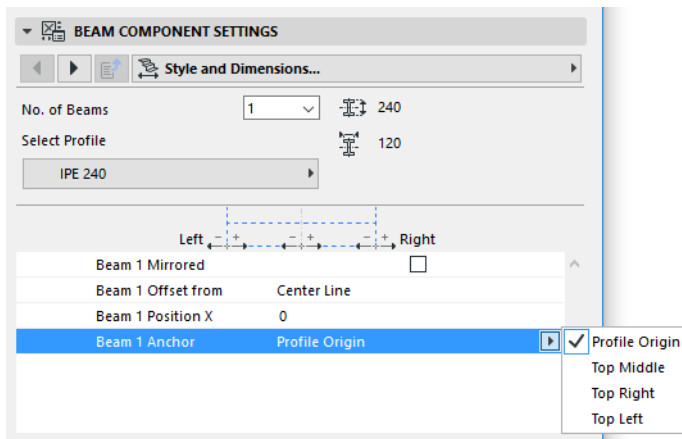
Use this panel with its tab pages to set the Beam component's settings.



Beam Anchor Point

The Anchor Point is at the top of the beam: choose Top Left, Top Middle or Top Right.

Important: For a profiled Beam, the default Anchor is the Profile Origin. For correct connections, make sure the Profile Origin is defined at the *top* of the profile! Alternatively, set the Anchor Point to one of the "Top" options in the pop-up.

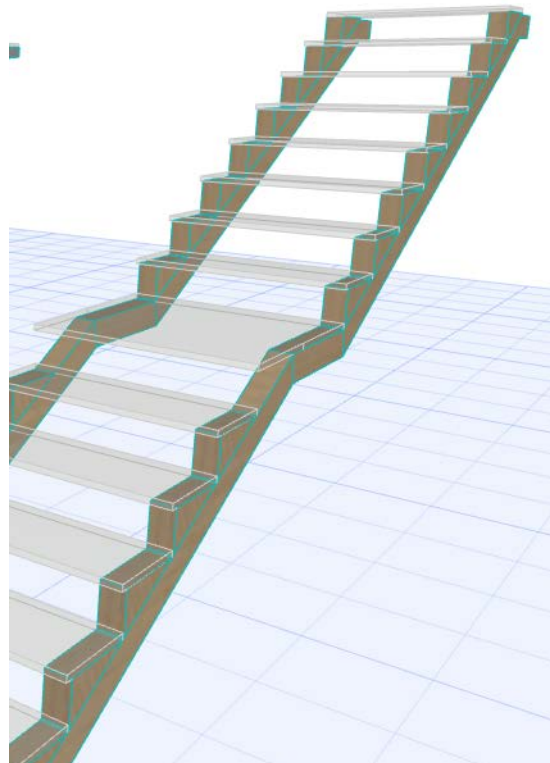
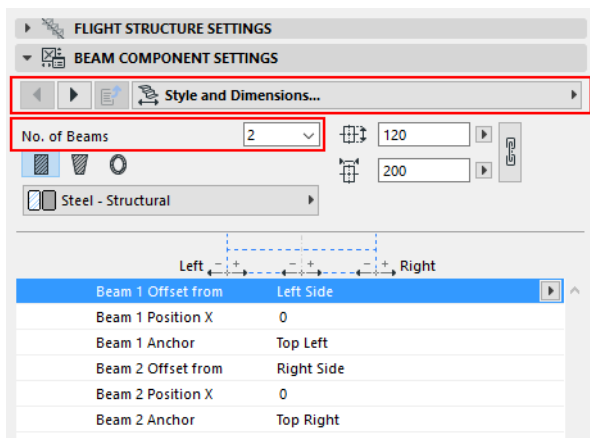


See also [Profiled Stringer: Special Options.](#)

For Profile definition, see [Profile Origin in Profile Editor Window.](#)

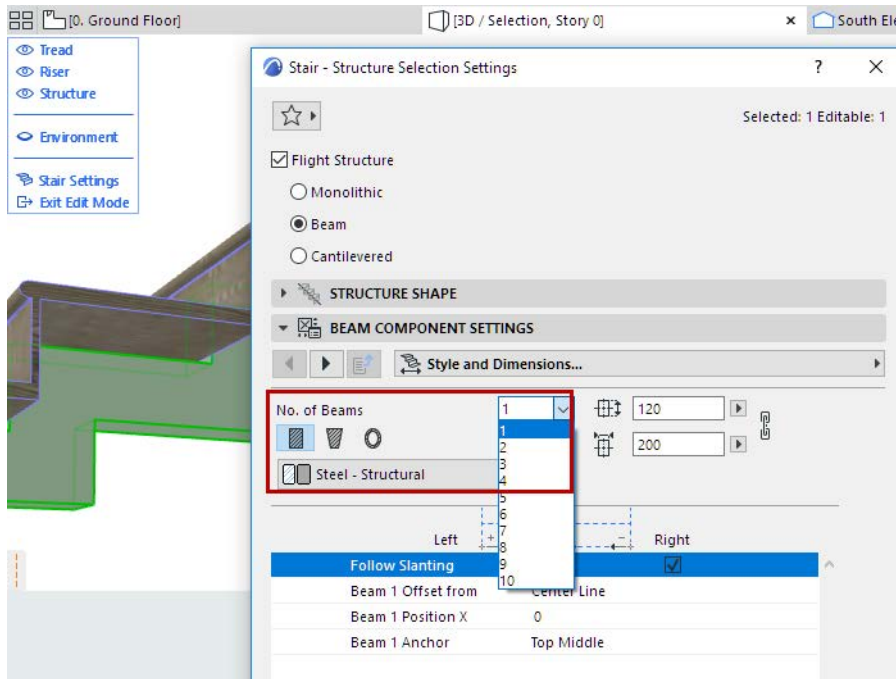
Set Number of Beams

Use this control (**Style and Dimensions** page of Beam Component Settings) to add multiple Beams (up to 10) to serve as the Stair's support structures on all flights.

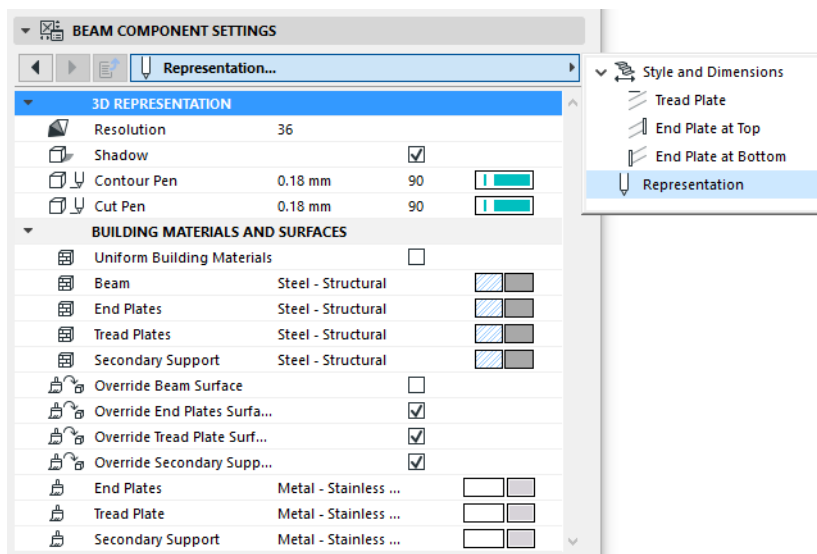


Notes:

- Number of Beams must be set (on separate Settings pages) for both Flight and Landing.
- In Edit mode, you can customize the number of beams per Flight or per Landing, as needed.



Beam Attributes and Representation



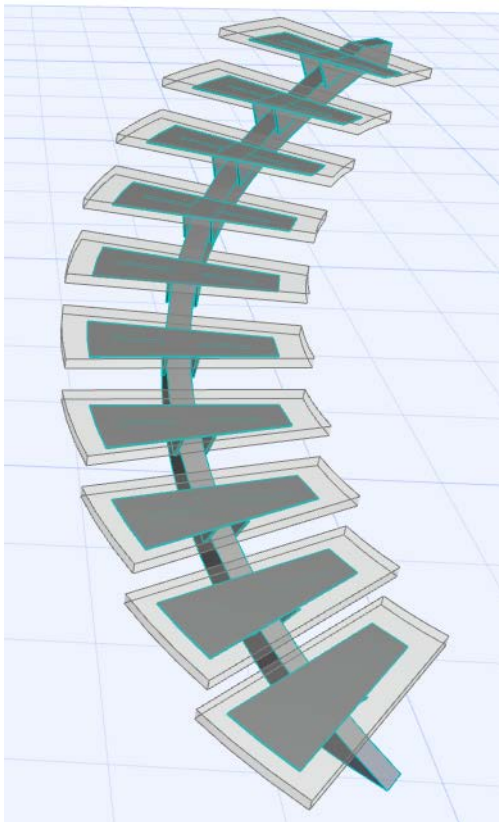
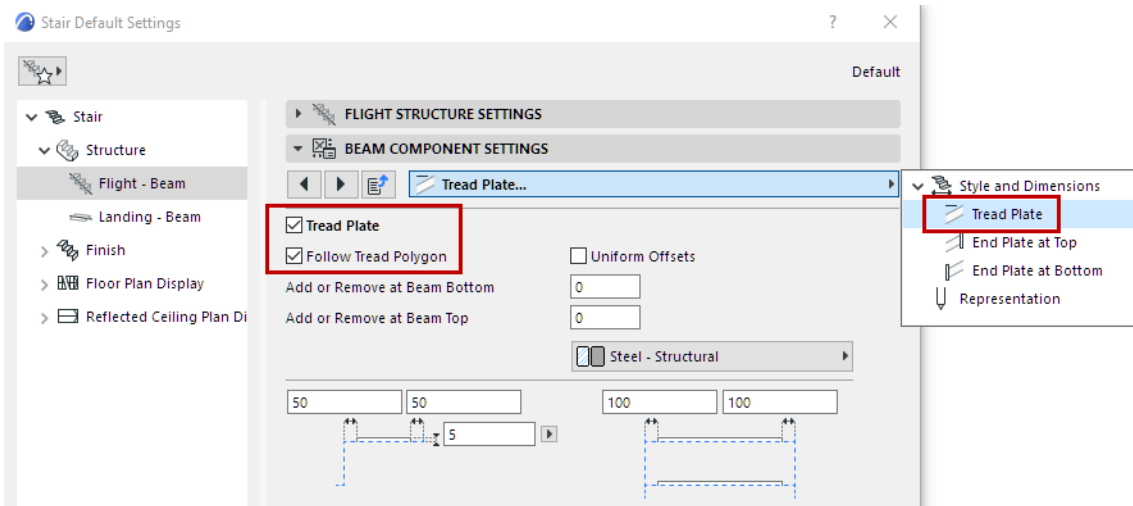
Depending on the Beam component, set its Building Material and/or choose a Profile. Adjust its values as needed.

Note:

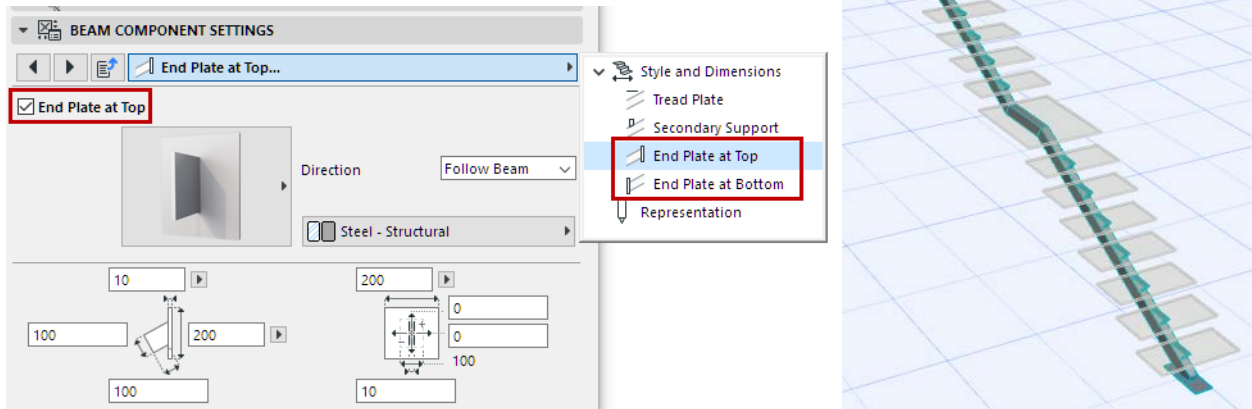
- Beam attributes must be set separately for Flight and Landing.
- The “Uniform Building Materials” checkbox enables a single Building Material for the listed Beam components (e.g. Beam, End Plates, Secondary Support).

Add Tread Plates and/or End Plates to Beams

See also [Add Tread Plates at Beam Landing](#).



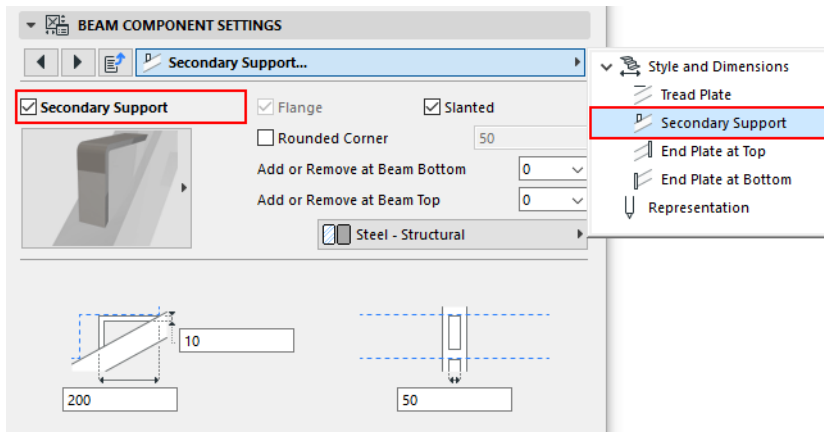
Tread Plate Follows Tread Polygon



End Plates at Top and Bottom

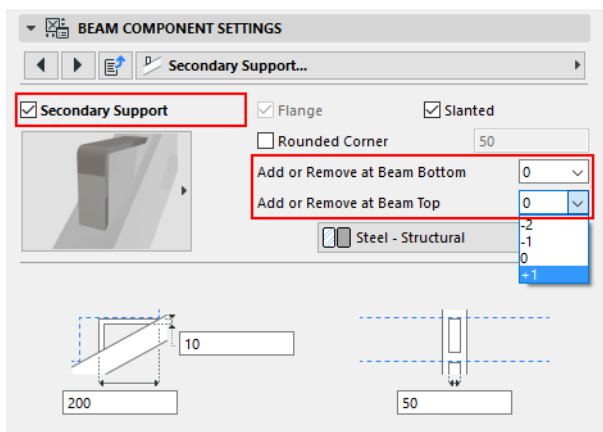
Add Secondary Supports to Beams

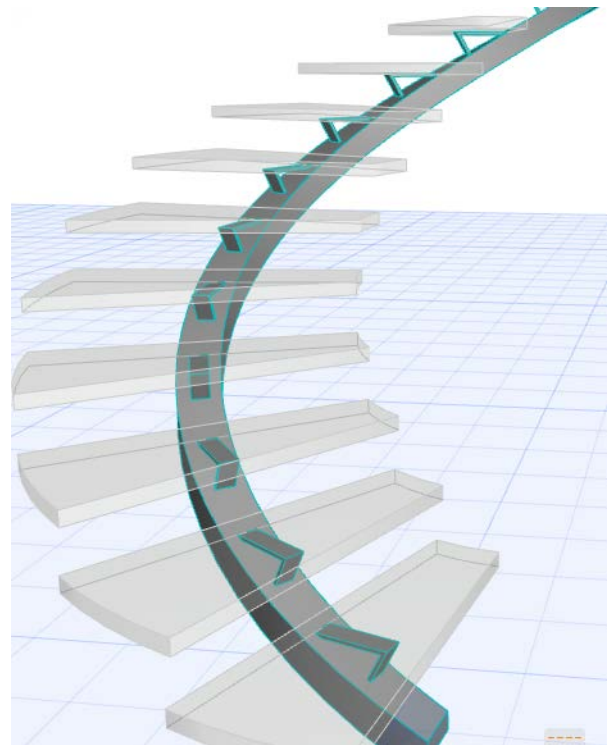
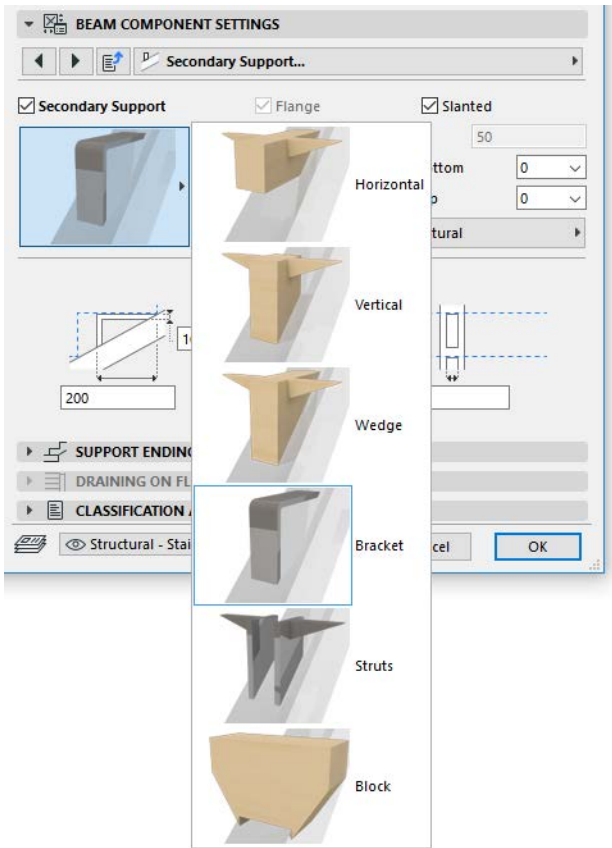
For a Beam Flight Structure, you can add additional components to support the Treads. Go to the Secondary Support tab page, and enable the Secondary Support checkbox.



From the pop-up, choose and set the desired component.

You can control the number of supports at either end of the beam, using the Add or Remove pop-ups:





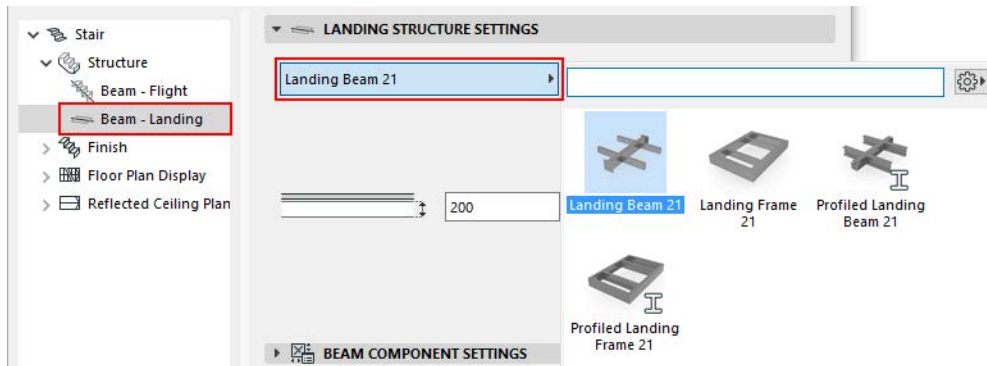
Beam Flight Support, with Secondary Supports

Beam Landing Structure

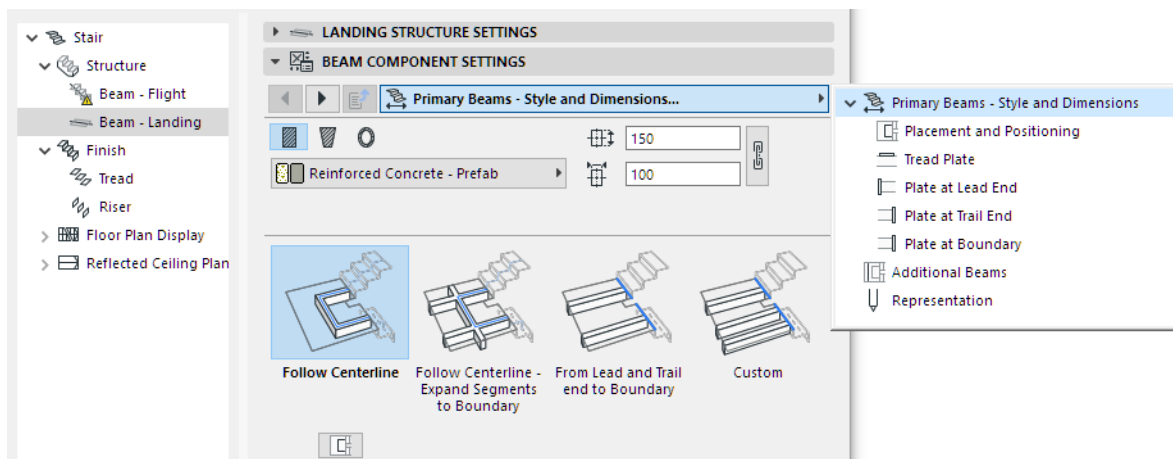
These settings are available if you chose a Beam Structure type for the Landing.

[See Choose Structure Type.](#)

Choose a Landing Support structure for the Stair, which can be Beam-type or Frame-type. Profiled Landing components are also available.



On the panel below, set up the chosen structure.



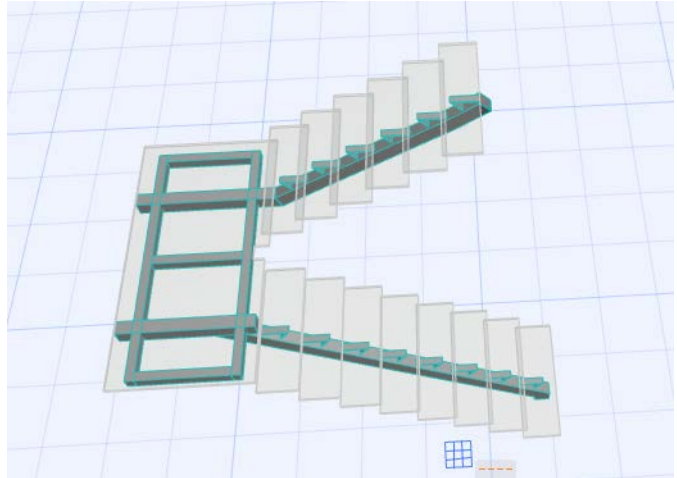
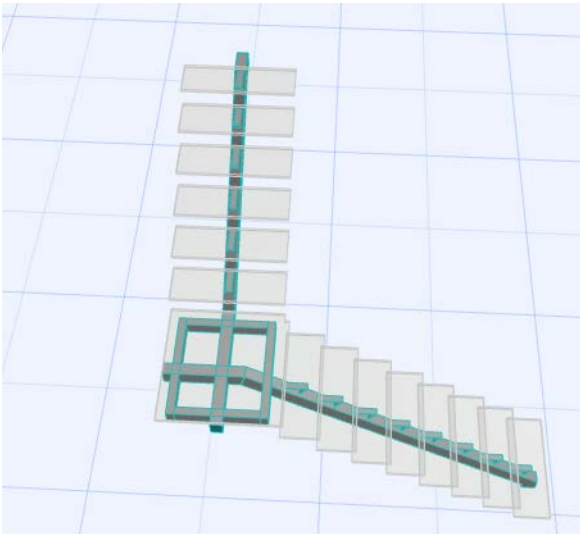
Primary and Secondary Landing Beams

- **Primary Landing Beams** will connect automatically to the Flight beams, as defined at Flight Structure Settings.

[See Stair Endings and Connections.](#)

- **Secondary (Additional) Beams** can be added parallel to the sides of the landing. Secondary Beams do not connect automatically to Flight Beams.

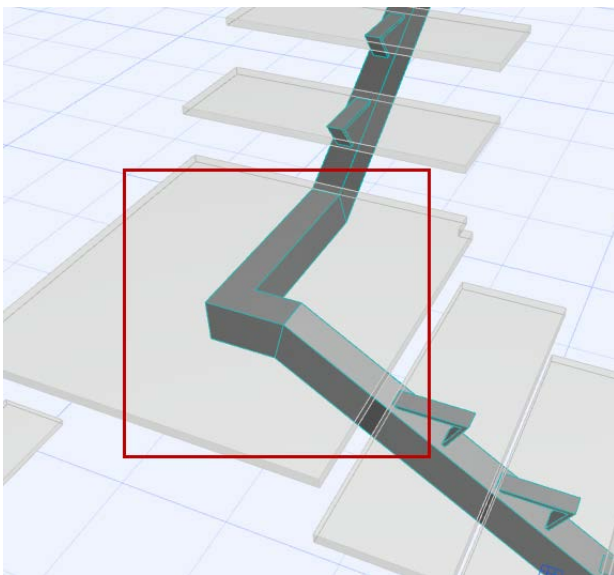
[For more information, see Add/Edit Secondary Beams.](#)



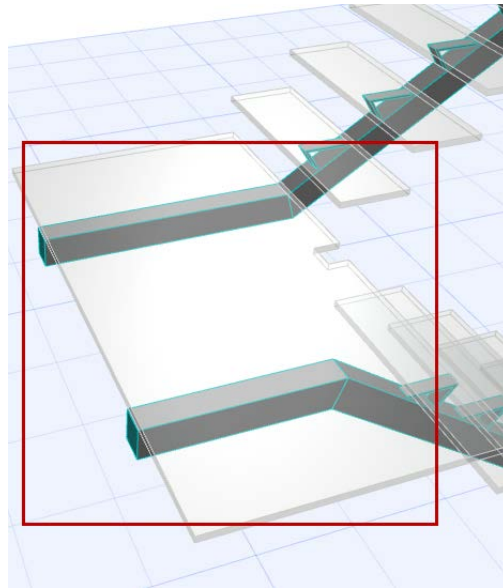
Landings with Primary and Secondary Beams

The **Primary Beam** generally follows the Stair Centerline, but you can also choose a different Beam path (use the **Primary Beams - Style and Dimensions** tab page):

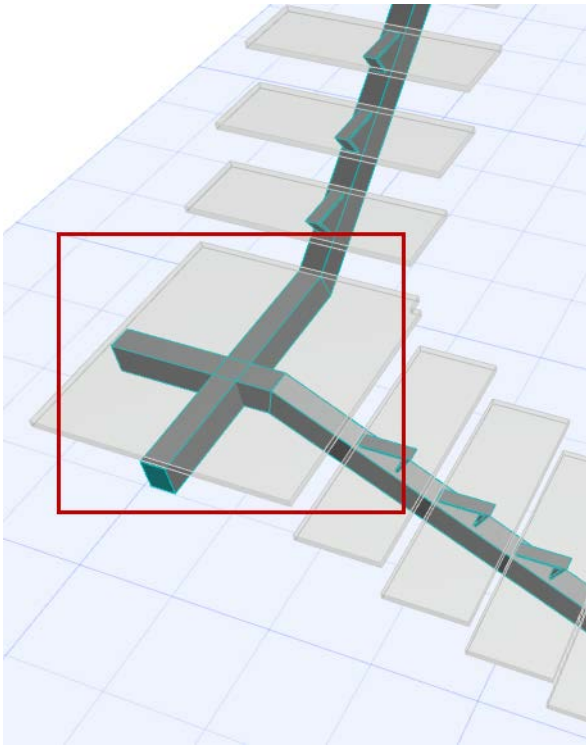
PRIMARY BEAMS POSITION



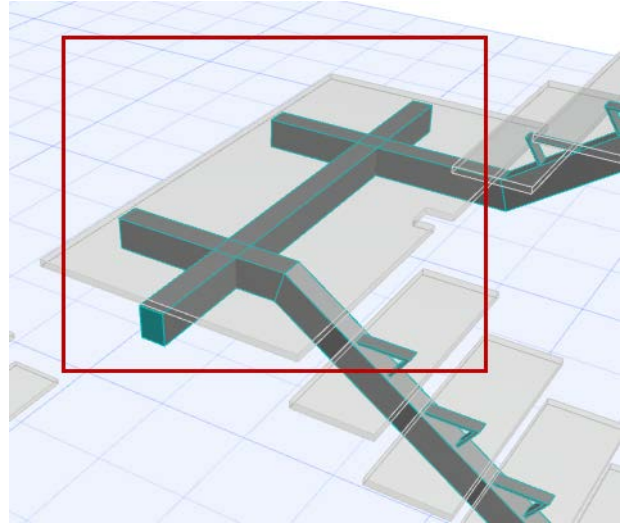
Follow Centerline



Lead/Trail End to Boundary



Follow Centerline - Expand to Boundary

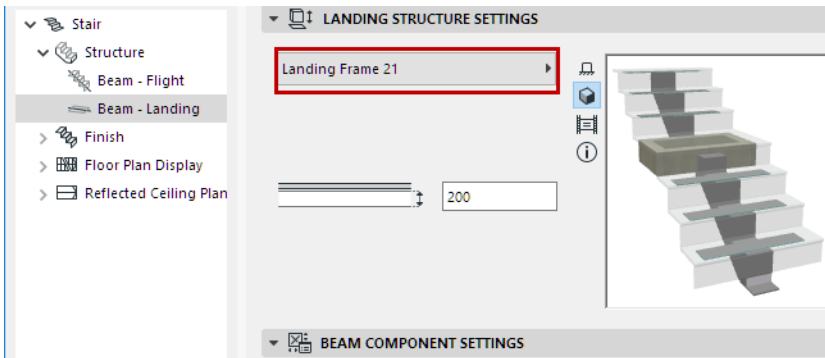


Follow Centerline - Expand to Boundary

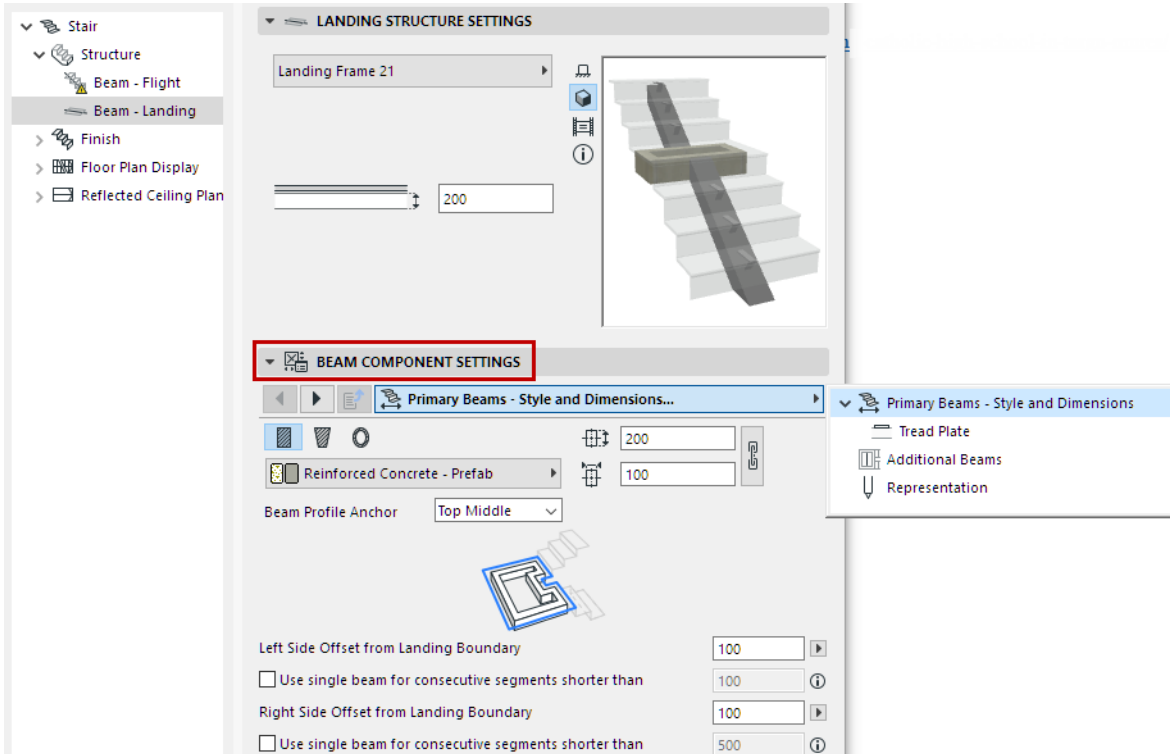
Landing Frame (Primary Beam)

You can choose a Frame to support the Stair Landings. Like other primary beams, the Landing Frame component connects automatically to Flight support, as defined at Flight Structure Settings.

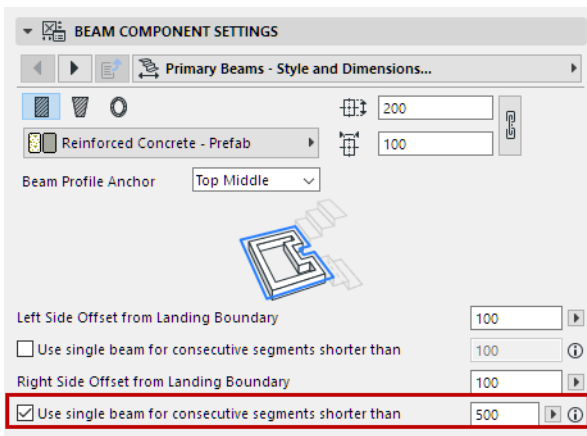
[See Stair Endings and Connections.](#)

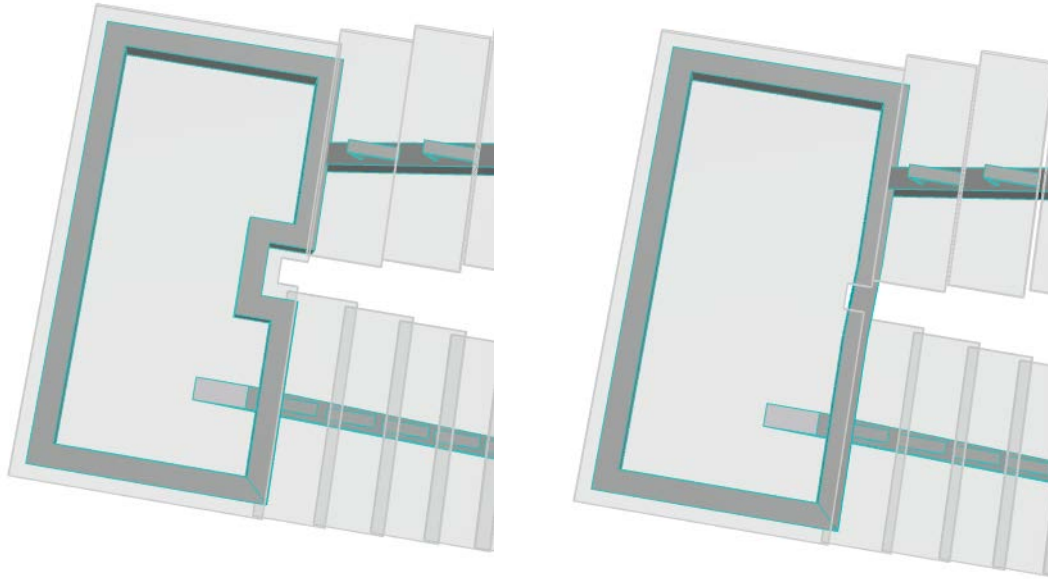


Use Beam Component Settings to configure the chosen Landing Frame:



- By default, the Landing Frame follows the Landing outline.
- However, you can use Beam Settings to adjust the outline so it ignores Landing segments that are shorter than a certain length:



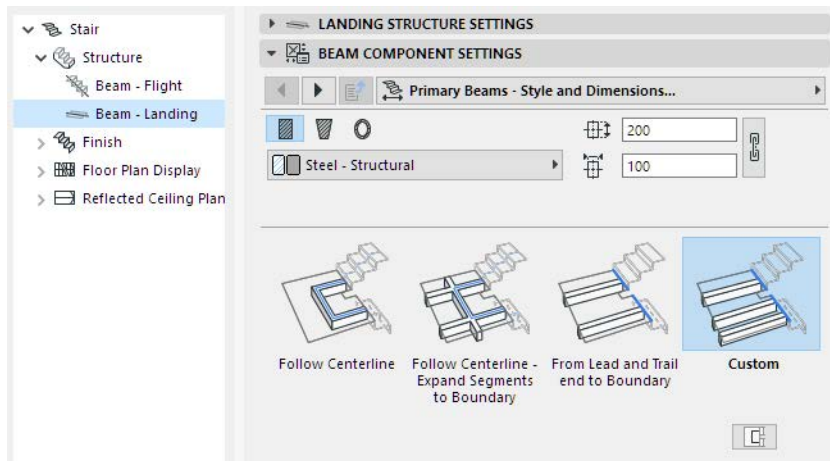


Custom Landing Beam (Primary Beam)

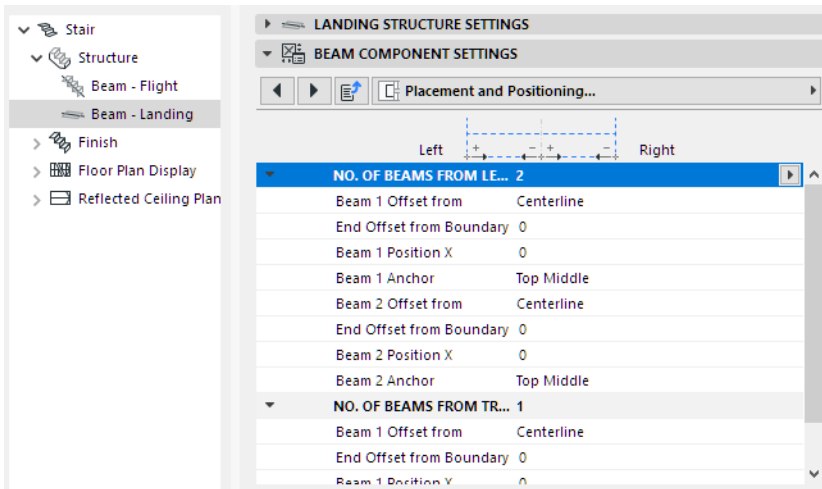
Use this function when the Stair Flight below the Landing uses a different number of Beams than the Flight above the Landing. (You can define beams for each Flight structure separately in Edit mode.)

To place a custom Landing Beam:

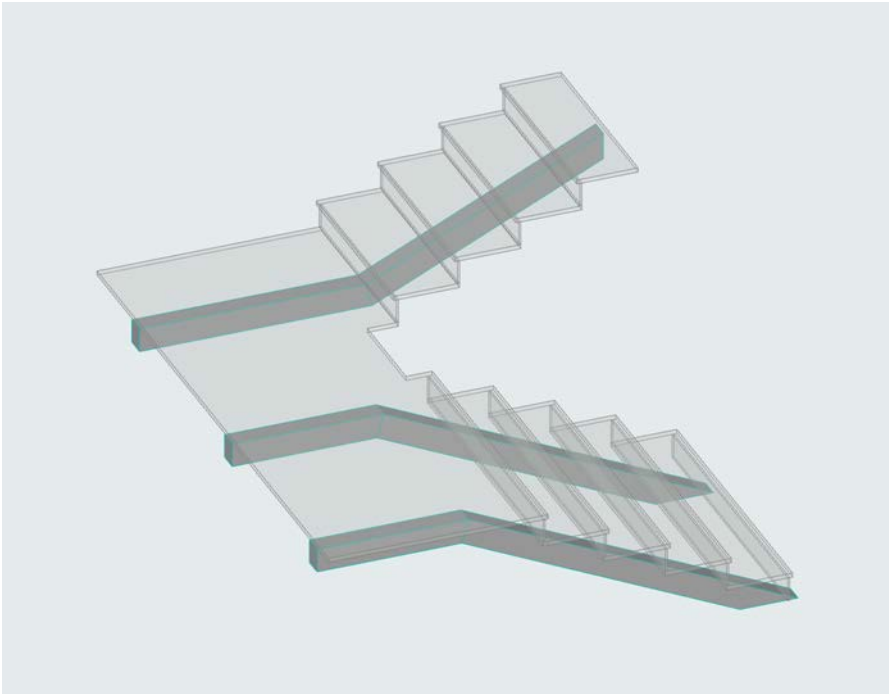
1. Choose Custom from the Beam Settings page (Primary Beams - Style and Dimensions tab page).



2. Switch to the Placement and Positioning tab page.
3. Enter the number of Beams to place.
4. Enter position and offset values for the custom landing Beam(s).



Since these custom Beams are Primary Beams, they can connect to the Flight supports.

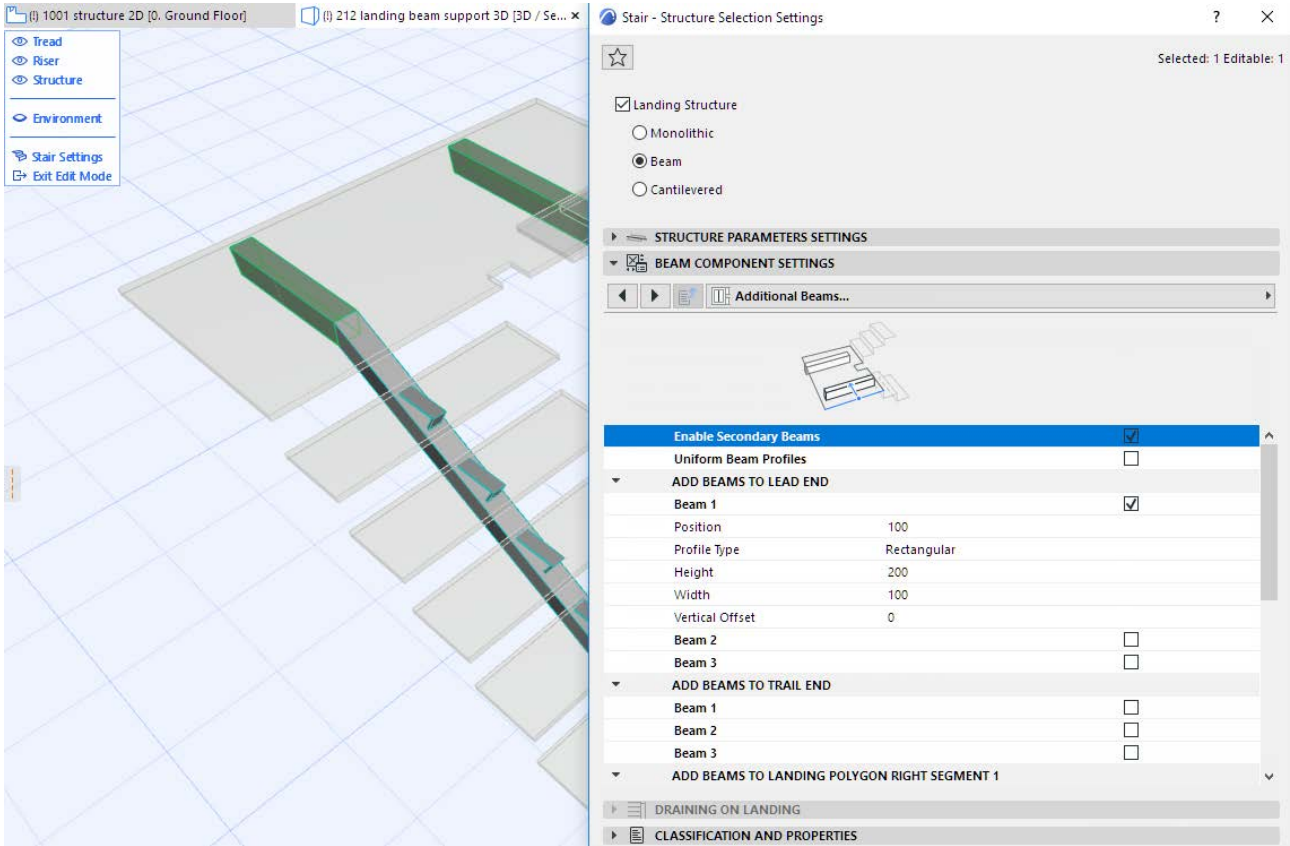


Add/Edit Secondary Beams

To add additional (Secondary) Beams, you must use Edit mode. (Secondary Beams are positioned to an existing Landing's geometry, and thus must be added to a placed, selected Stair.)

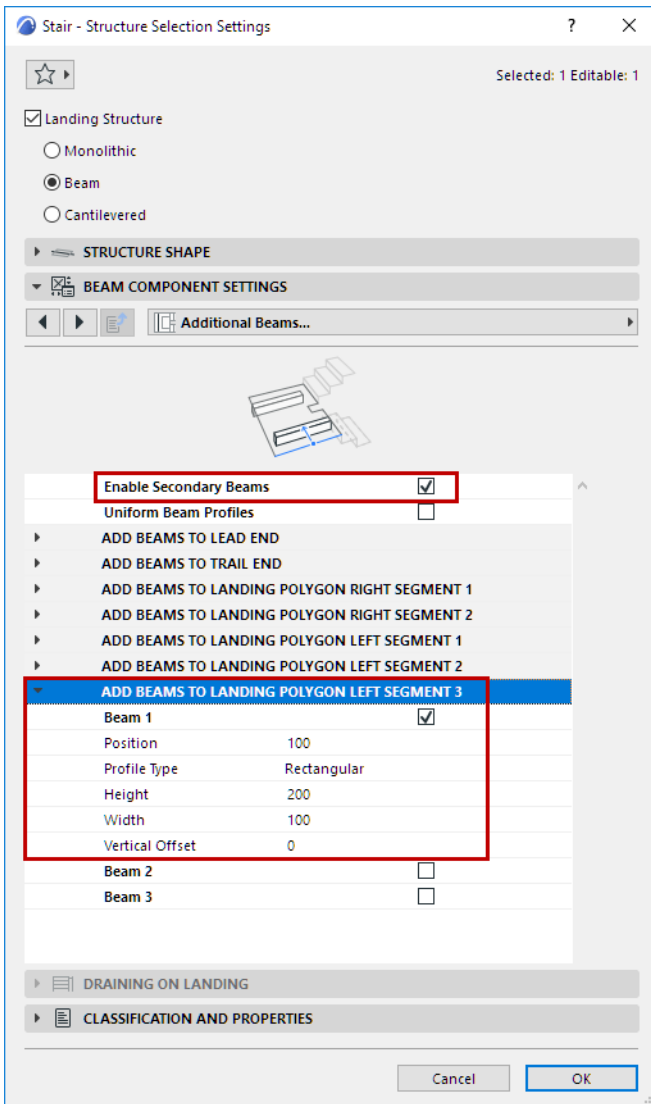
1. Select the Stair and enter Edit mode. ([See Enter Edit Mode.](#))
2. Select the Landing Structure, and open Structure Settings.
3. In the Beam Component Settings panel, open the **Additional Beams** tab.

4. Check **Enable Secondary Beams** (if it is not checked already).

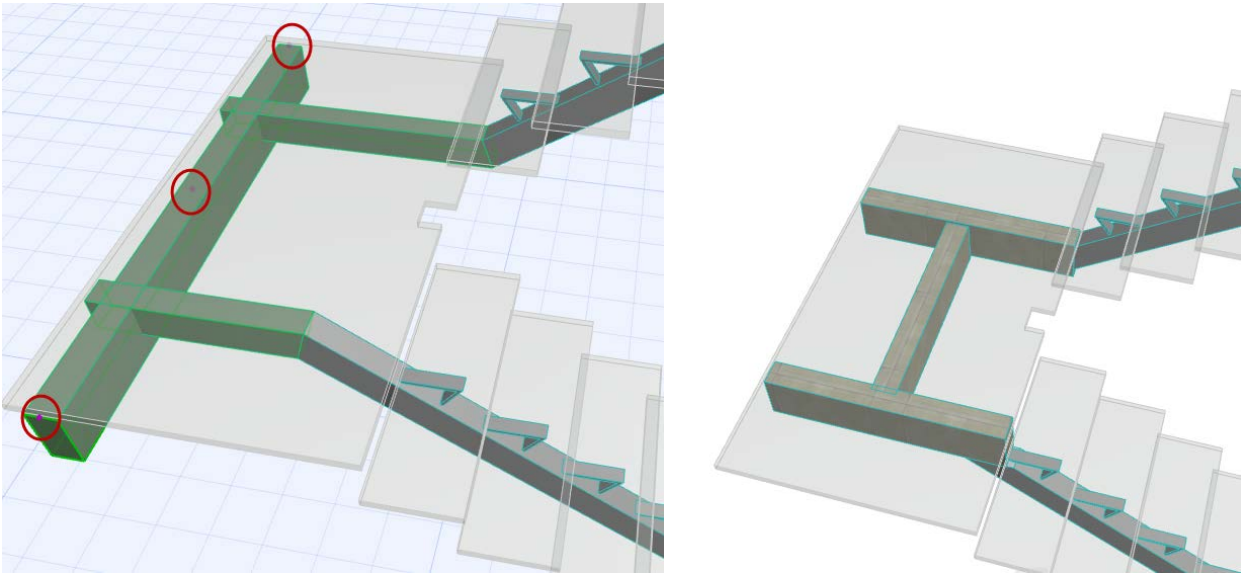


Add Secondary Beams in Edit Mode

- 5. Use the checkboxes and settings to add Secondary Beams to one or more segments of the Landing polygon.

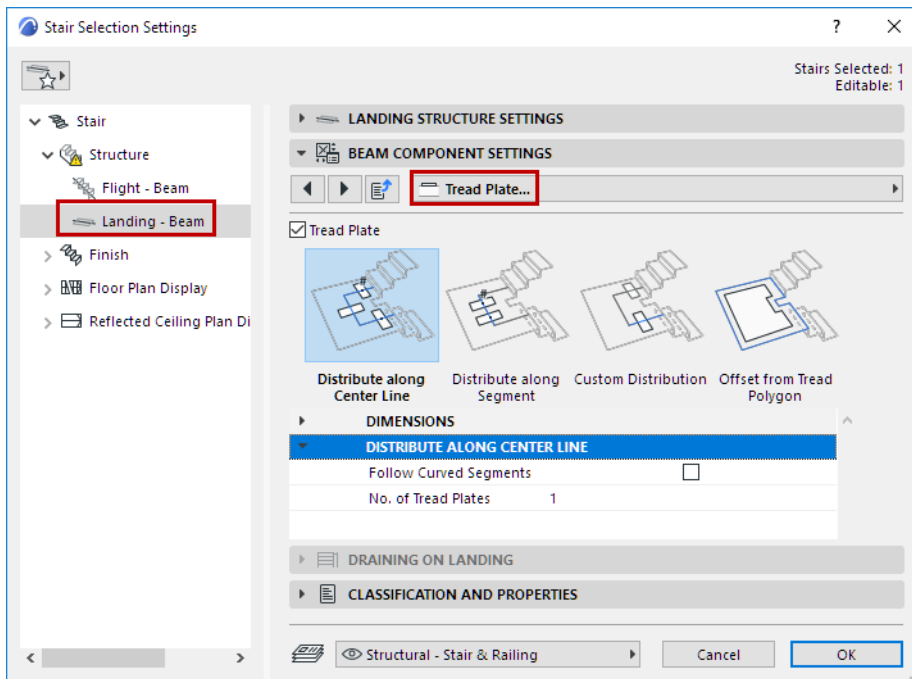


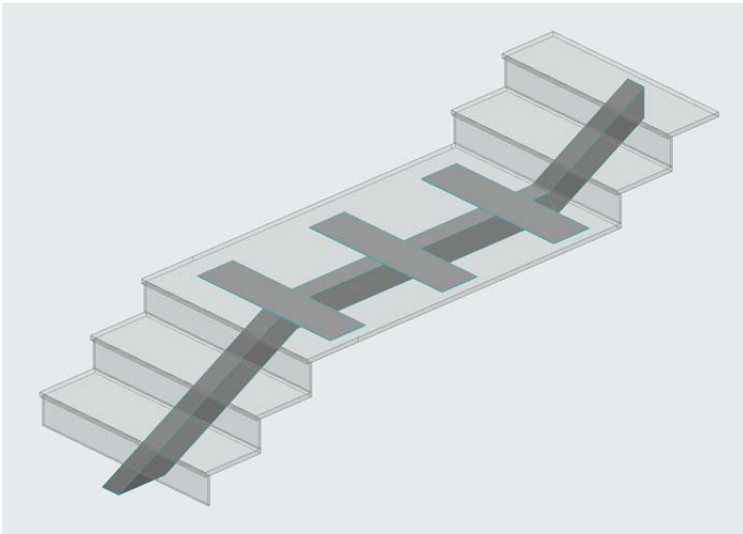
The secondary beam(s) are placed. Use Edit mode to stretch and move them as needed, using the editable hotspots.



Modify Secondary Beam Graphically in Edit Mode

Add Tread Plates at Beam Landing





Tread Plate at Beam Landing

Cantilevered Flight and Landing Structure

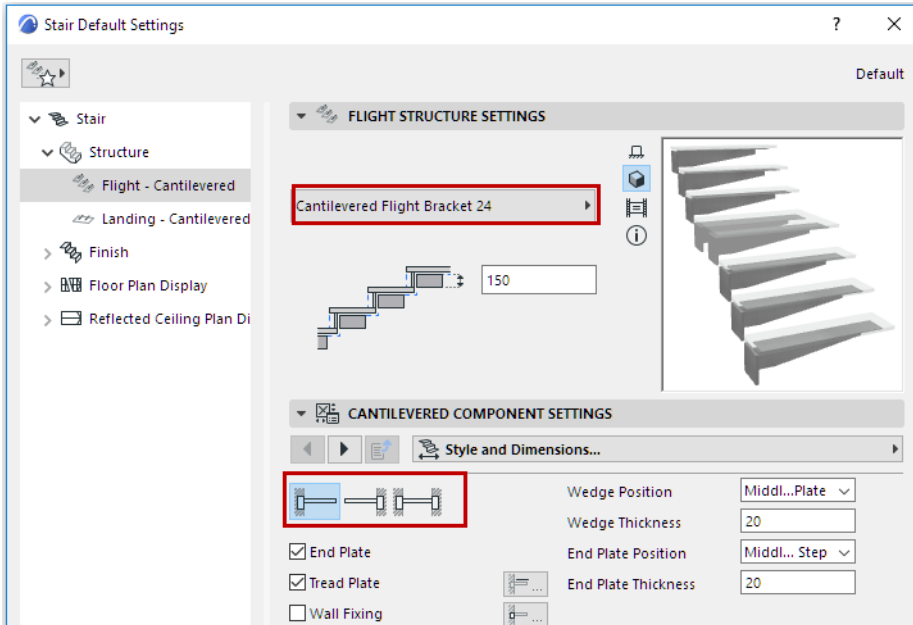
These settings are available if you chose a Cantilever Structure type for the Flight and/or Landing.

[See Choose Structure Type.](#)

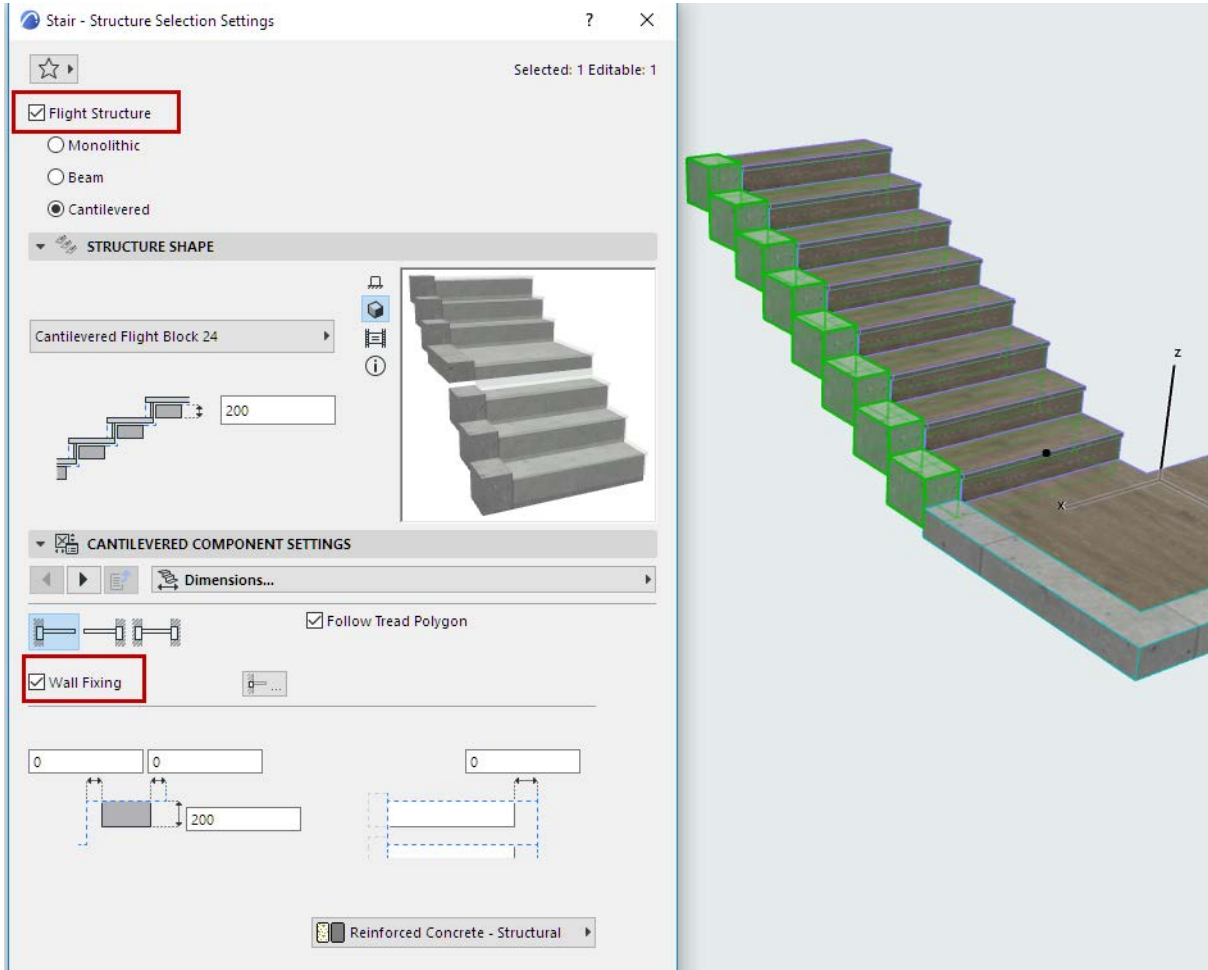
Choose a Cantilever Component type - e.g. Bracket or Block.

Use the Cantilevered Component Panel to configure the chosen structure:

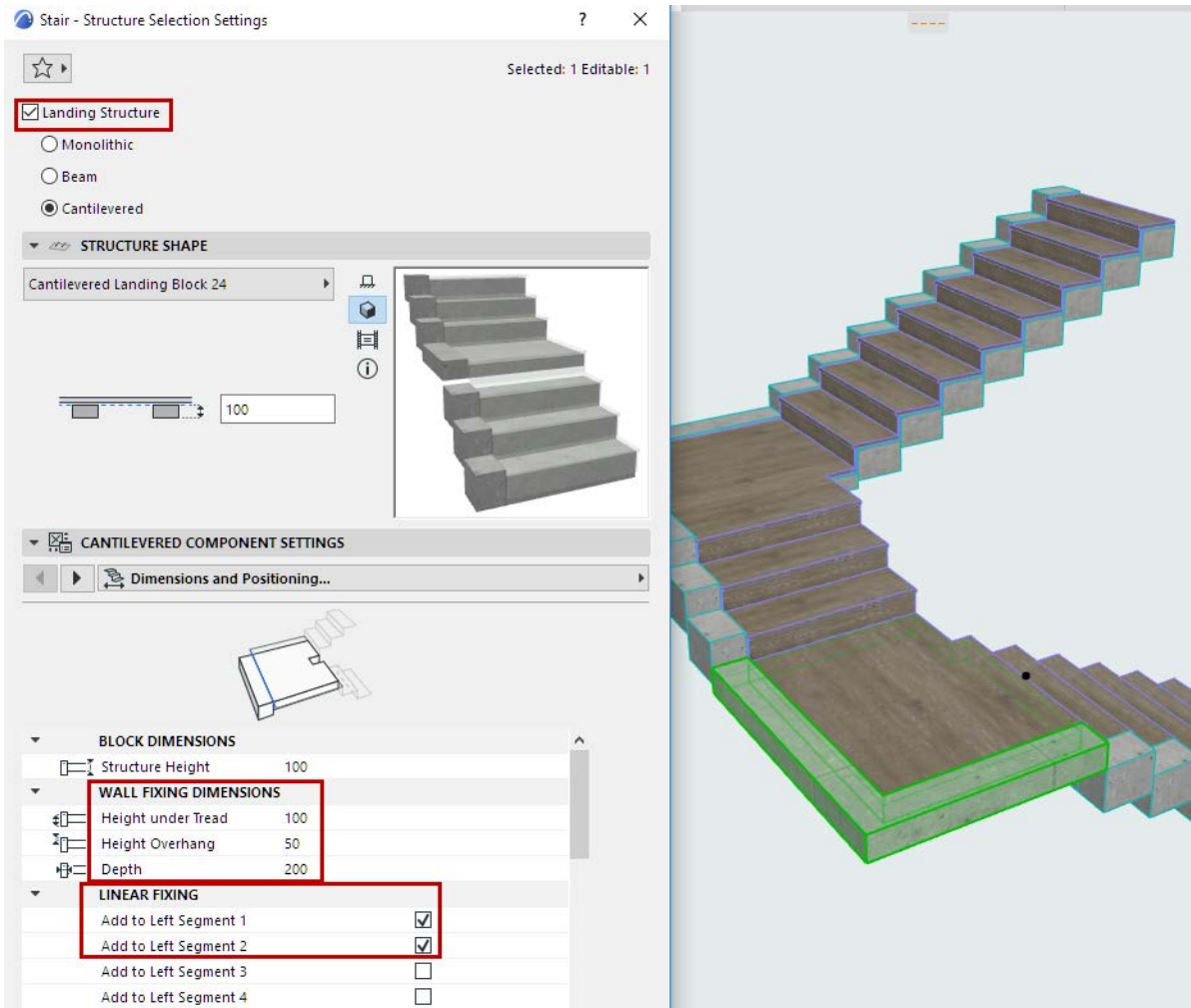
- Set cantilever structure on left, right, or both sides of Stair
- Add an optional Tread Plate



Use Edit Mode to add an optional Wall Fixing to Flight or Landing, as shown below:



Add Wall Fixing to Cantilever Flight (Select and Edit Flight Structure in Edit Mode)



Add Linear Fixings to Cantilever Landing Segments (Select and Edit Landing Structure in Edit Mode)

Stringers Flight and Landing Structure

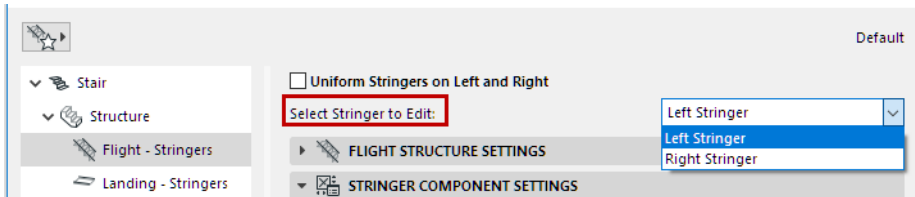
This page is shown in the Stair Settings tree structure

- if the Flight or Landing Structure is set to Stringers Only, or
- if the Stair has any other Flight Structure that uses Stringers

See [Choose Structure Type](#).

Select Stringer to Edit

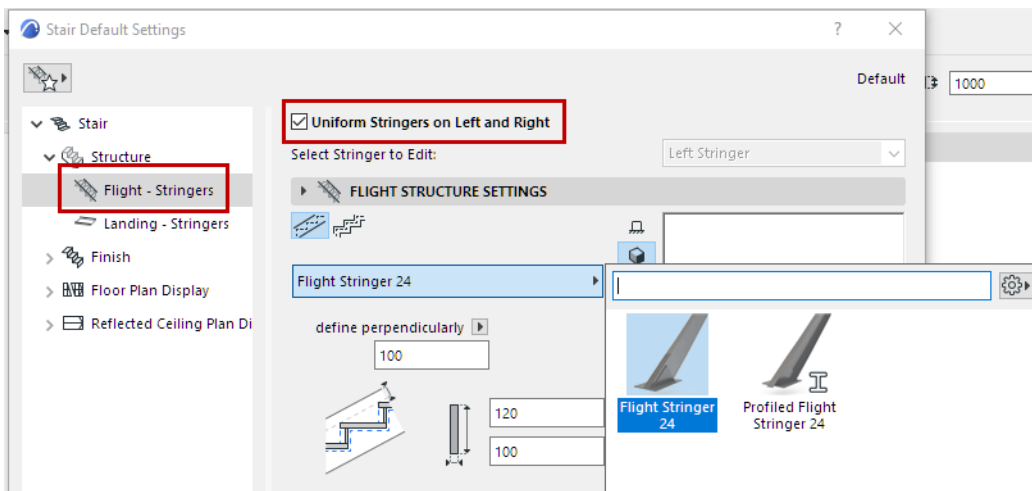
Edit the left and right stringer separately, as needed.



Alternatively, check **Uniform Stringers on Left and Right**, to apply the same settings to both Stringers.

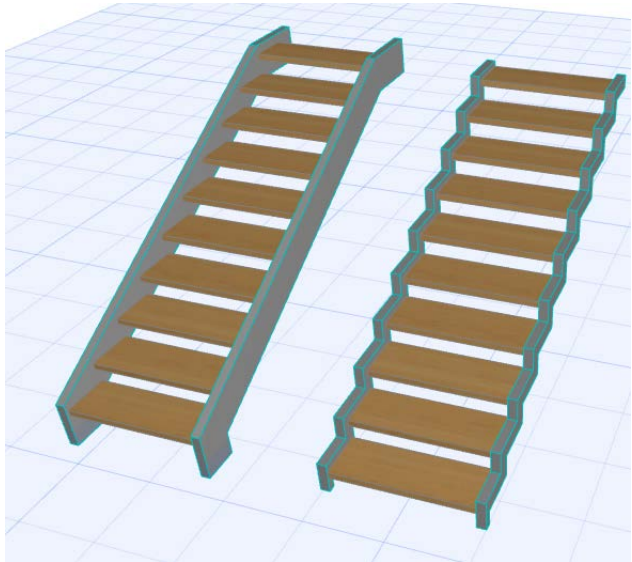
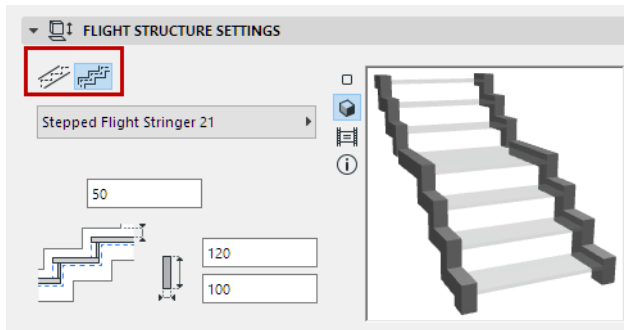
Notes:

- Stringers attributes must be set separately for Flight and Landing.
- The Connection and Ending settings are included in the “uniform” settings. If you want to use identical Stringers but with varying connections (e.g. to the Landing), you must uncheck the Uniform box!



Stringer Structure Settings

For a Flight Stringer: Choose a shape for the Stringer, either flat or stepped.



Stringer Support Component

Choose a GDL component from the pop-up to serve as the Stringer.

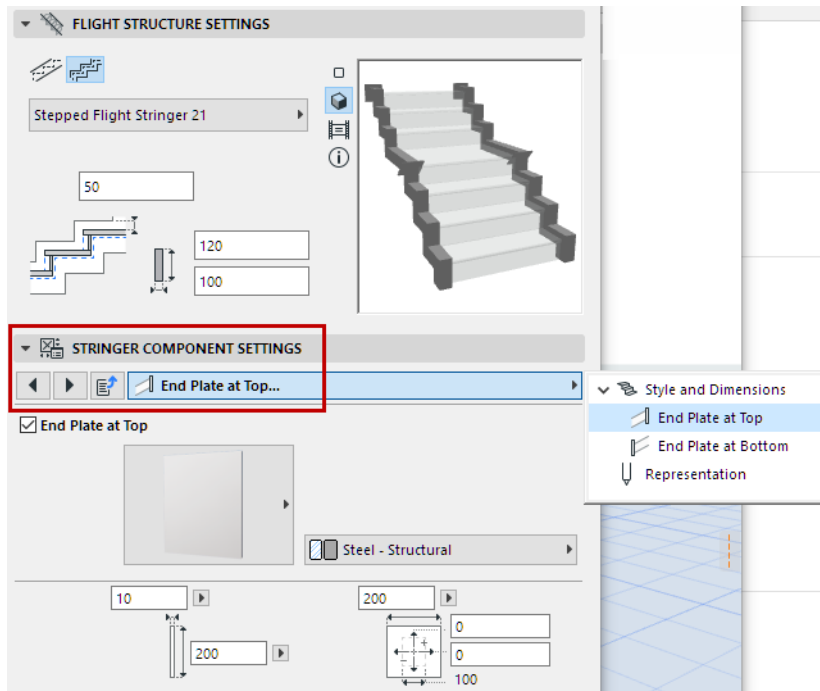
Note: The Open Stringer is available as a Beam. On the Structure page, choose Beam as the Flight Structure, then choose the Open Stringer component from the Beam-Flight node.

See [Use Open Stringer \(Beam Flight Structure\)](#).

Add End Plates

End plates can be placed at Stair top and/or bottom (for a Flight Stringer), or at Lead End and/or Trail End (for a Landing Stringer).

The chosen Stringer's settings are in the panel below.



Profiled Stringer: Special Options

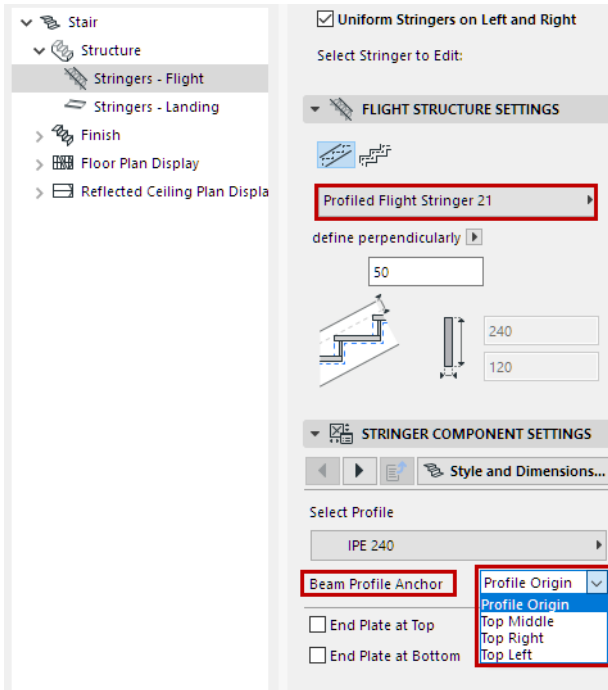
- The profile Stringer can be mirrored: For example, place mirror-image Z-shaped profile stringers on both sides of the Stair.



Z-Profile Stringers - Mirrored

- Adjust the Profile's Anchor point (by default, it is the point defined as the Profile Origin) using the pop-up.

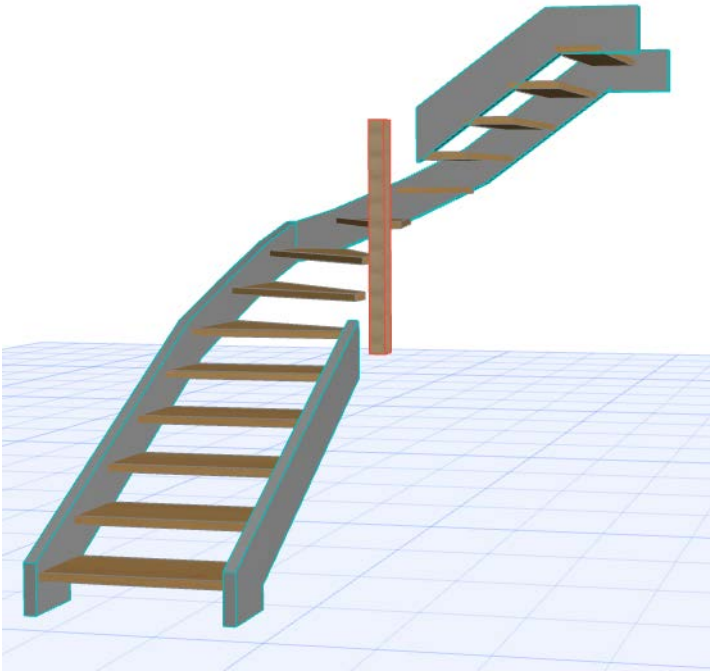
- **Important:** If using the Profile Origin as Anchor Point, make sure that the Profile Origin is defined at the *top* of the profile, to ensure correct connections. (Alternatively, set the Anchor Point to one of the “Top” options in the pop-up.)



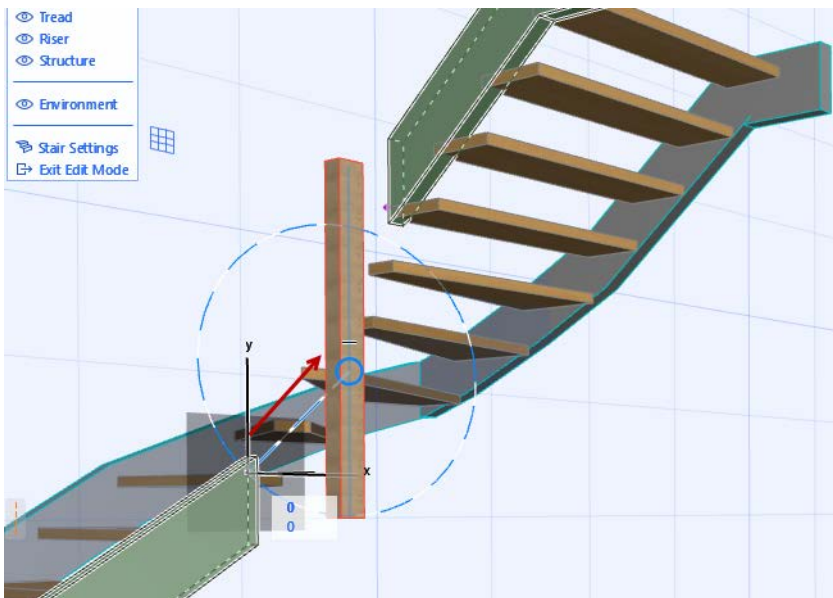
Winder Stair with Stringers

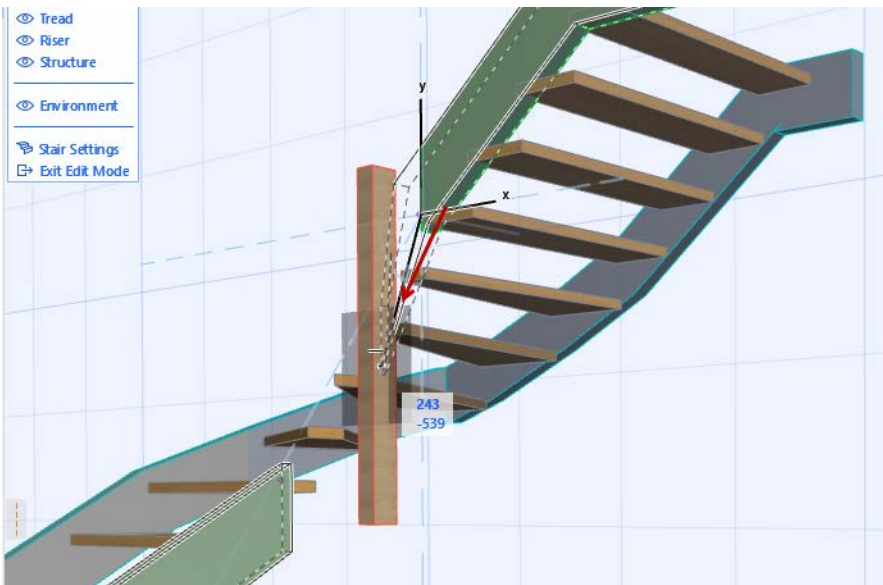
If a winding Stair uses Stringers on both sides, the inside Stringer cannot automatically follow the steeply rising Stair geometry. In this case, Archicad will disconnect the inside Stringer, so that you can adjust it manually as needed.

(The Column at the inside of the Stair is a separate element.)

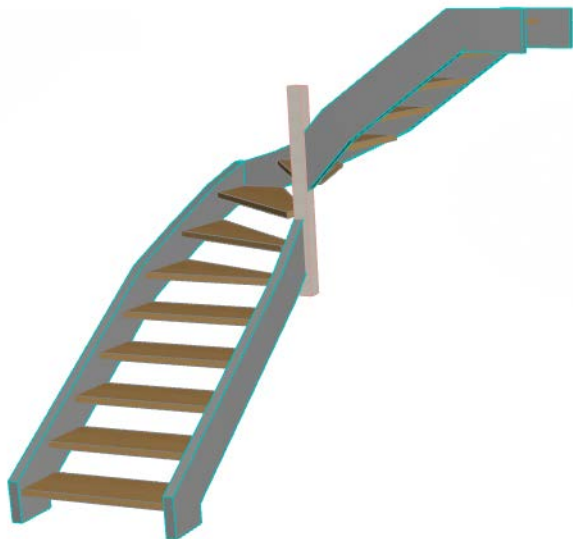


1. Select the Stair in the 3D window and enter Edit mode.
2. Select each end of the disconnected Stringer, and use the editing hotspots to drag them into position.





3. Now the Stringers are in position.

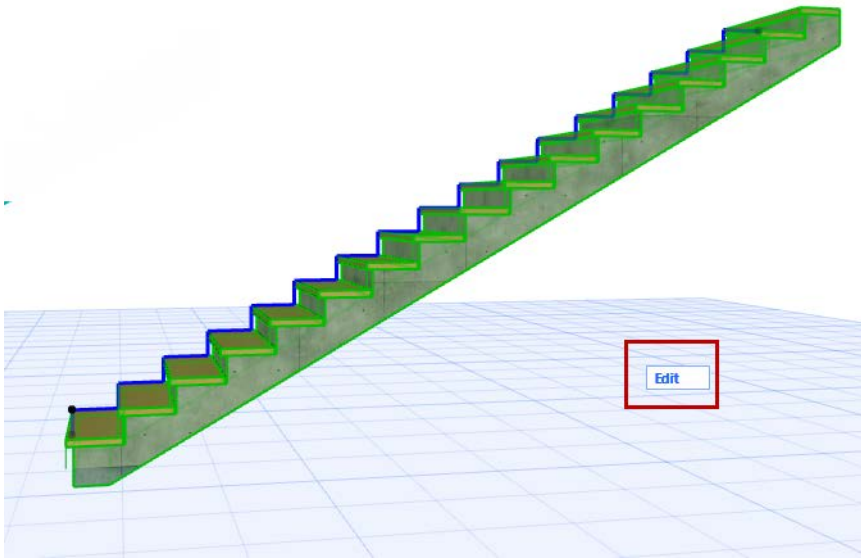


Split Stair Structure

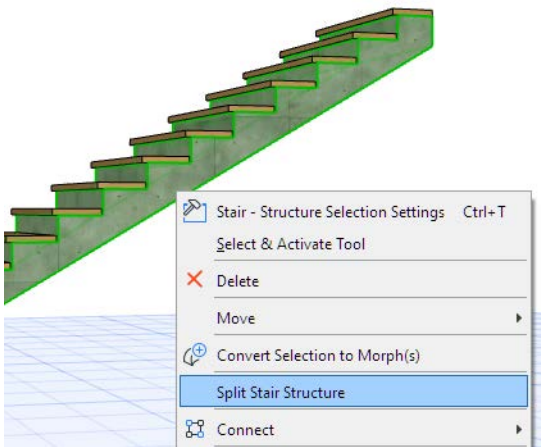
Use this function to obtain multiple Structure types for a single Stair flight.

This command is available in Edit mode, in the 3D window only:

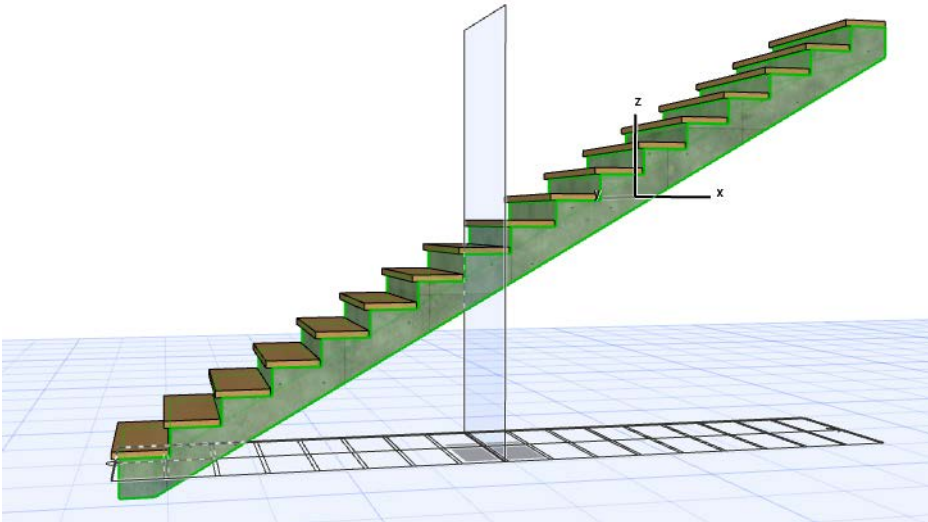
1. Select the Stair in the 3D window.
2. Enter Edit mode.



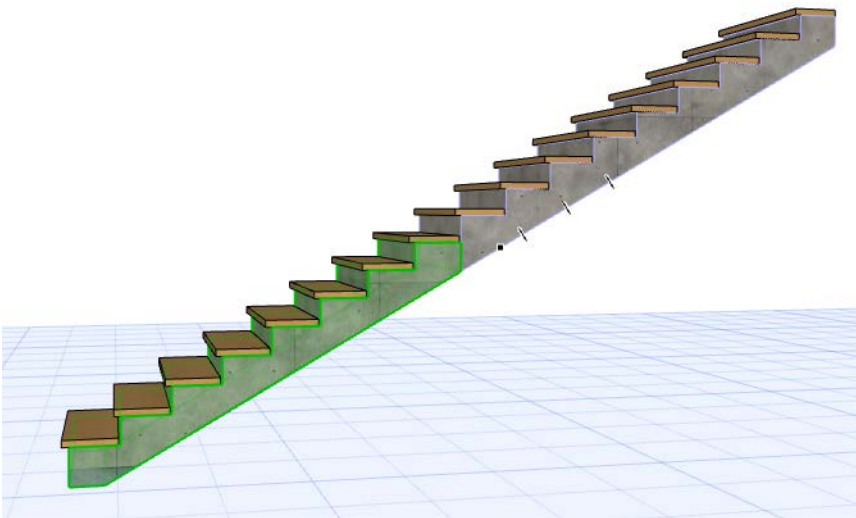
3. Select the Flight structure. ([See Select Stair Component for Editing.](#))
4. From the context menu, choose **Split Stair Structure**.

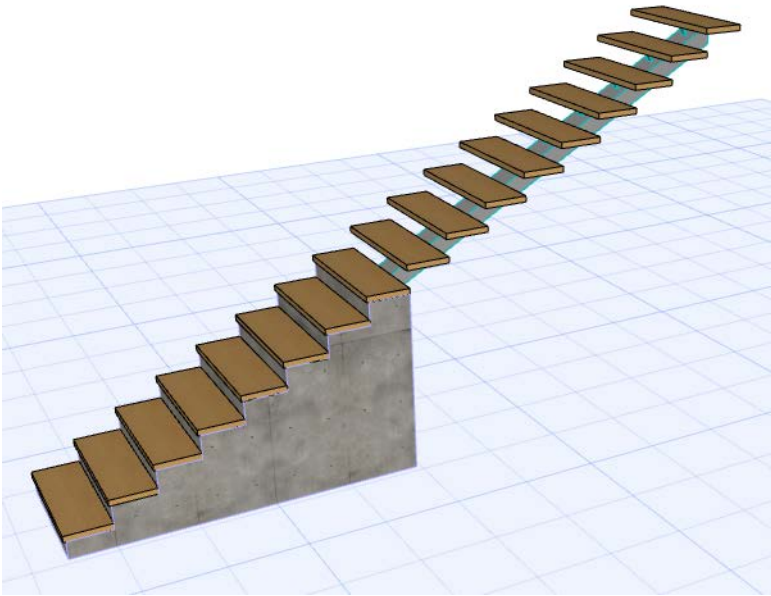


5. Based on the on-screen feedback, drag the cursor and click to split the Structure at the desired point. (The split always occurs at a Riser.)



6. Proceed to select and customize each part of the structure separately, as needed. Split the structure as many times as needed.





Custom Sub-Elements for Stairs and Railings

Stairs and Railings include GDL sub-elements. Many pre-defined sub-elements are available by default, through the element and component settings dialogs. Each such GDL sub-element can be configured with its own Building Material and/or Custom Profile.

To create your own sub-element, you can customize an Archicad element and save it as a specialized Library Part.

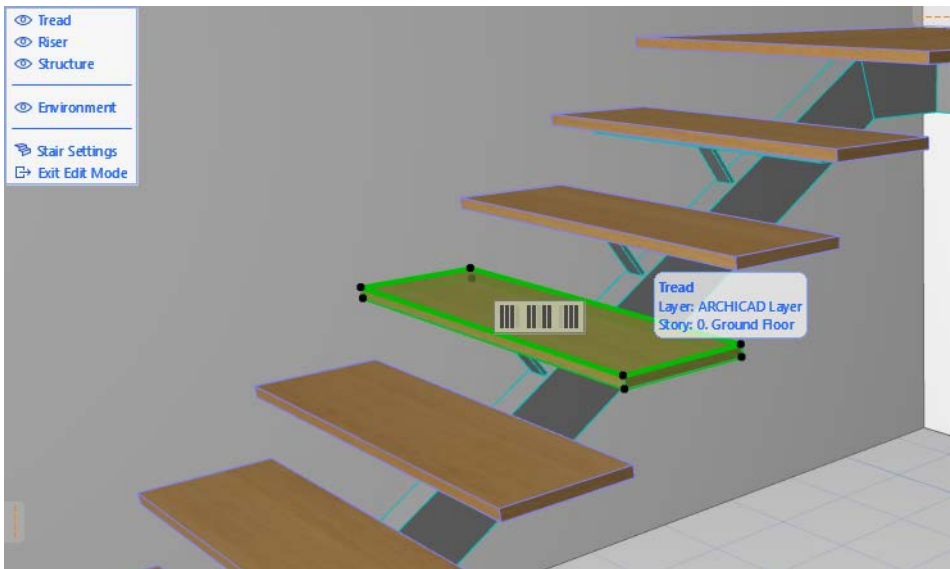
For more information on this function, see [Create Custom Library Parts and Components](#).

For Stairs and Railings, use this function to create custom Treads and Risers (Stairs), and custom Panels and Posts (Railings).

Some examples follow.

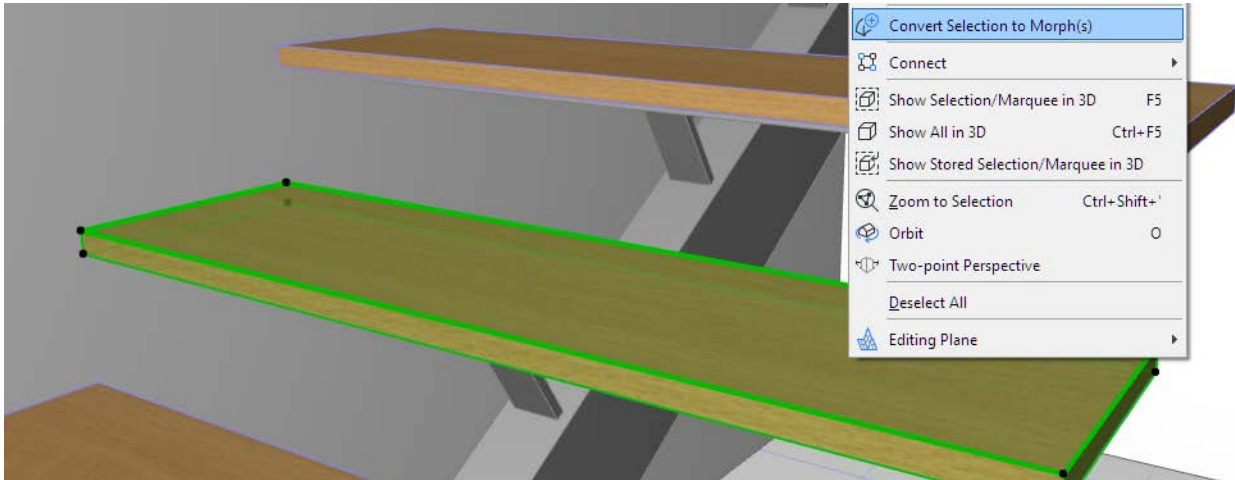
Create Custom Stair Tread

1. Select the Stair and enter Edit mode.
2. Select a Tread.

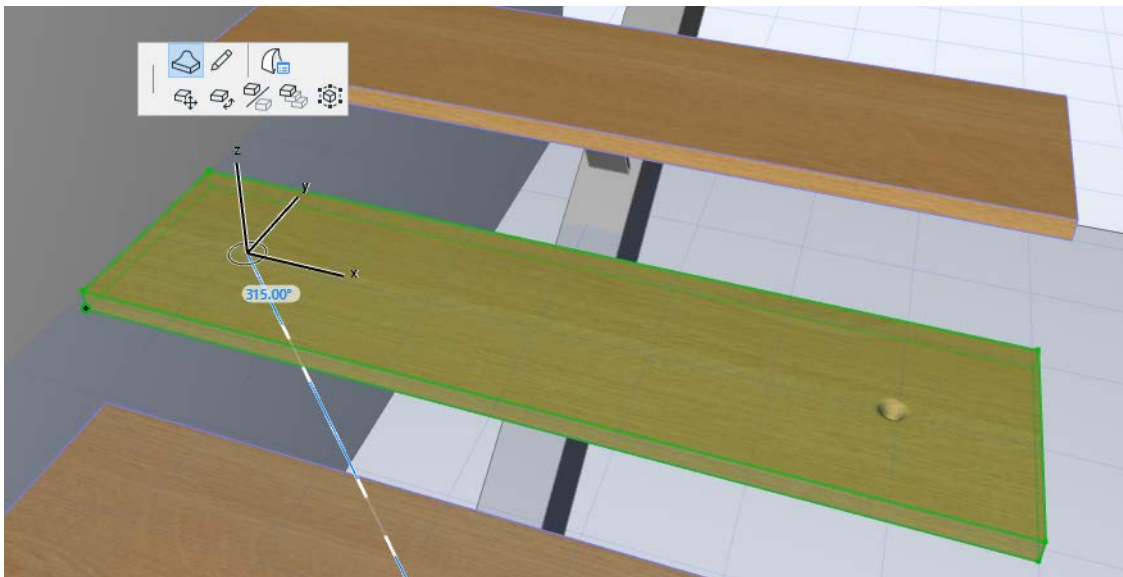


- From the context menu, use **Convert Selection to Morph**. The selected sub-element is now a Morph, and no longer part of your Stair.

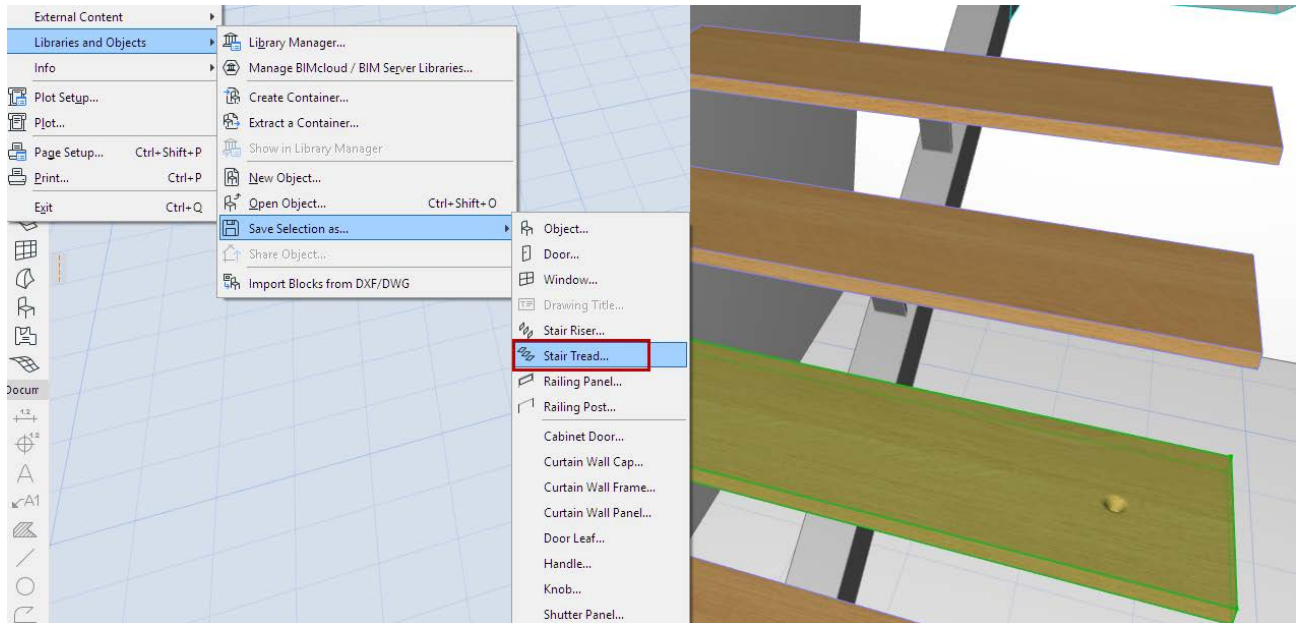
Note: It is also possible to start with any Morph element; it is not necessary to transform an existing Tread to a Morph first.



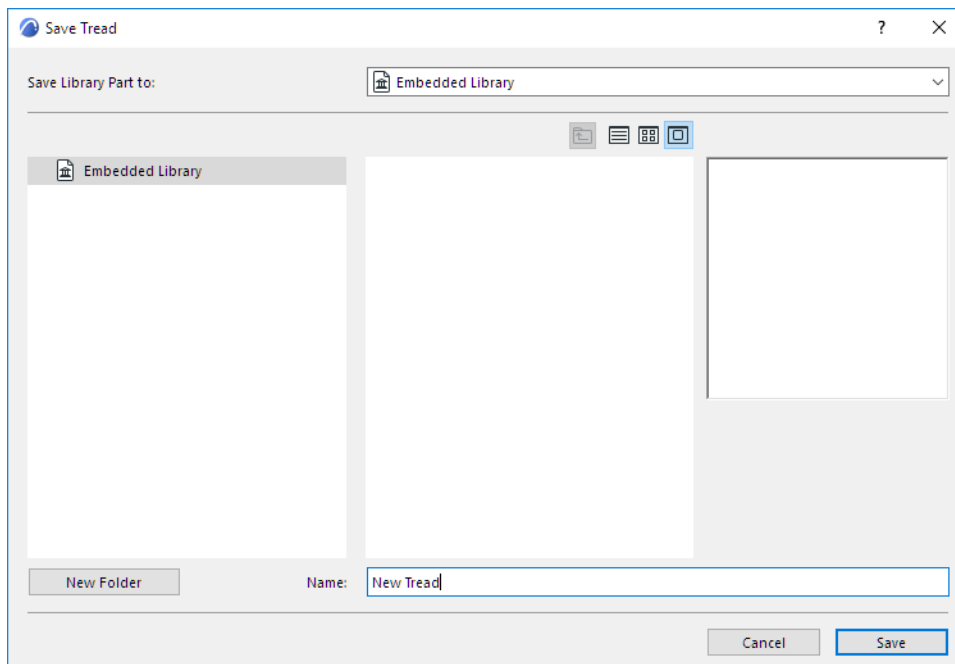
- Exit Edit mode.
- Edit the Morph as needed - here, use the Bulge command to create small recesses on either end.



6. With the Morph selected, use **File > Libraries and Objects > Save Selection as...**

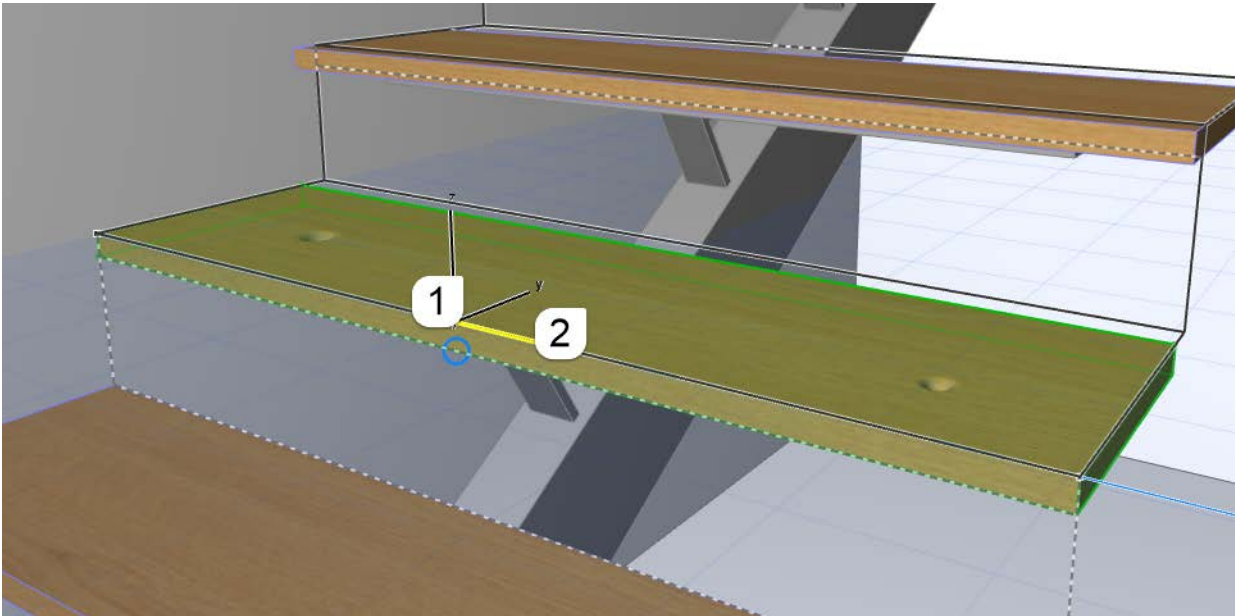


7. Select **Stair Tread** as the predefined sub-type, and save it to the Embedded Library.



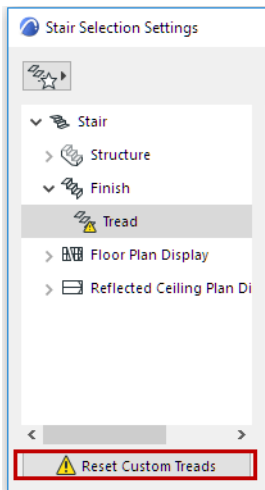
8. A Stair-like ghost feedback appears to help you orient the new Tread object:

- Click once to define the midpoint of the Tread's top edge (without nosing)
- Click a second time, on the edge, to define its horizontal position.

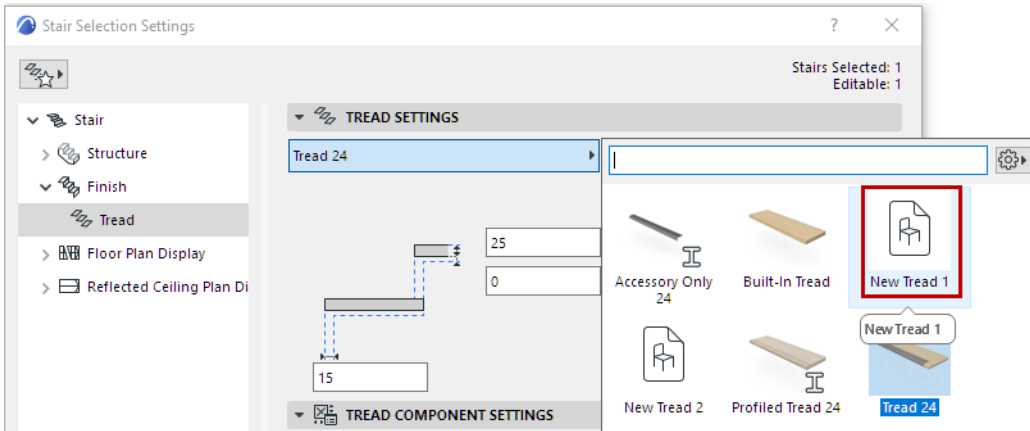


Click at Tread Midpoint, then Tread Edge to Define Position

9. Select the Stair and open Tread Settings.
10. The dialog indicates a custom Tread, because the Tread you converted into a Morph is now missing. Click the Reset Custom Treads button to resolve this.



- 11. The new custom Tread is available as a Component of Tread Settings: Select this Tread to apply it to the entire Stair.



[Watch the Video](#)

Stair Endings and Connections

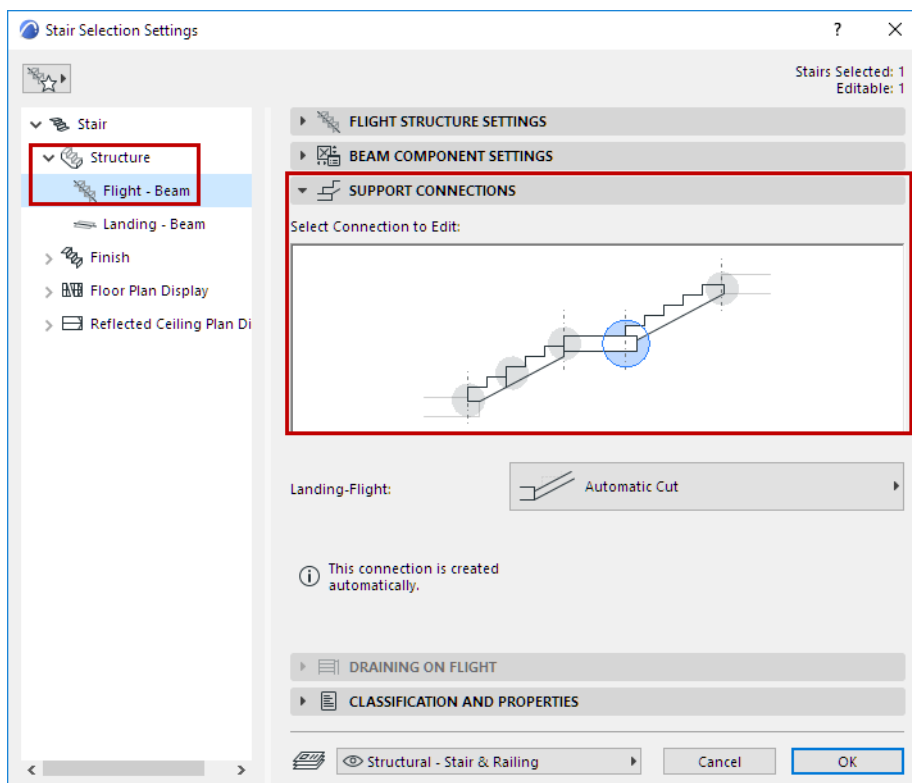
Stairs are not associative to other 3D elements. To join a Stair to a Slab or to a Wall, use Solid Element Operations.

[See example at Solid Element Operations.](#)

However, you can use Stair Settings (Flight Structure):

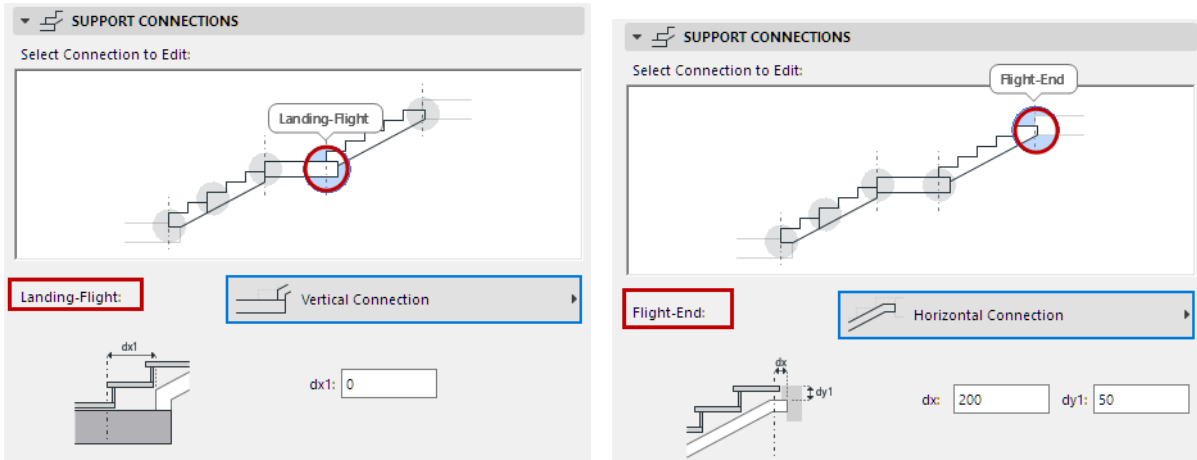
- to define the endings of Stair structures
- to create connections between a Stair's Flights and Landings. You can use different connections within a single Stair.

To set Endings and Connections for the Stair, go to the Flight Structures Settings page, and the **Support Connections Panel**. This panel is available in both Stair Settings (Flight Structure) and for a selected Flight Structure in Edit mode.

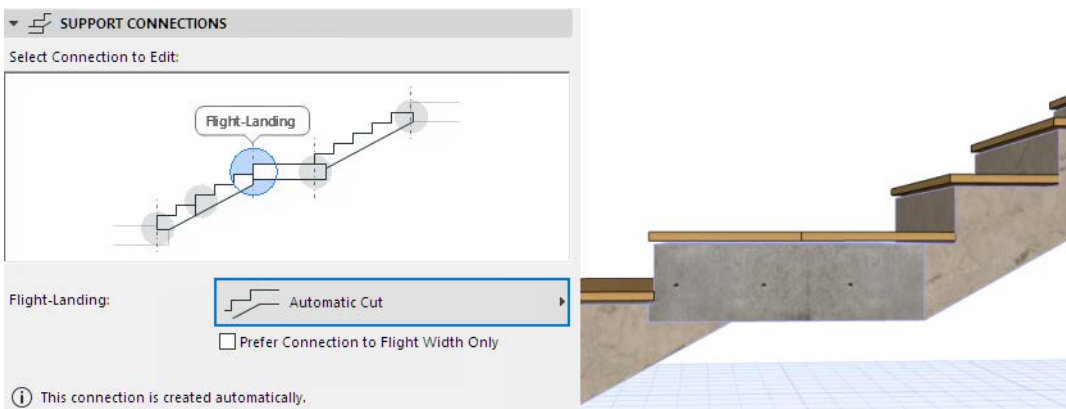


- All the Stair's Flight and Landing connections and endings are handled here:
 - Flight-Start
 - Flight-End
 - Flight-Landing
 - Landing-Flight
 - Flight-Flight

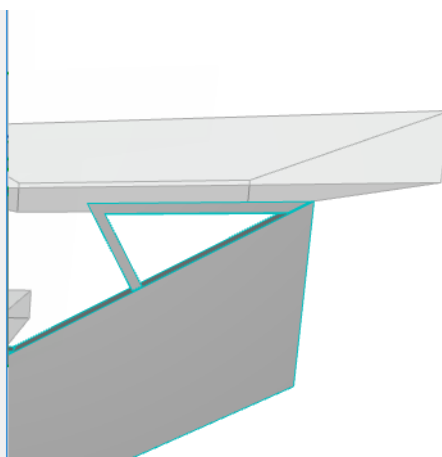
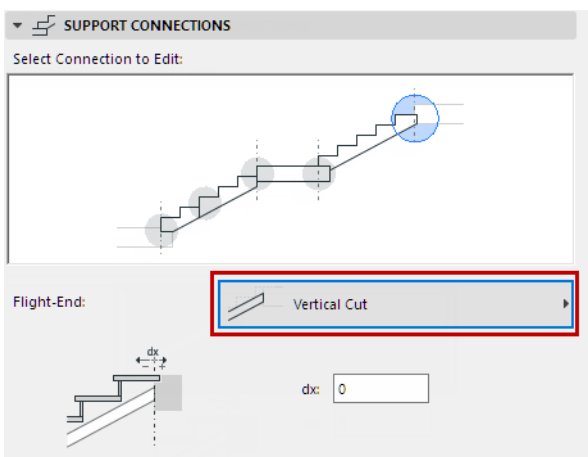
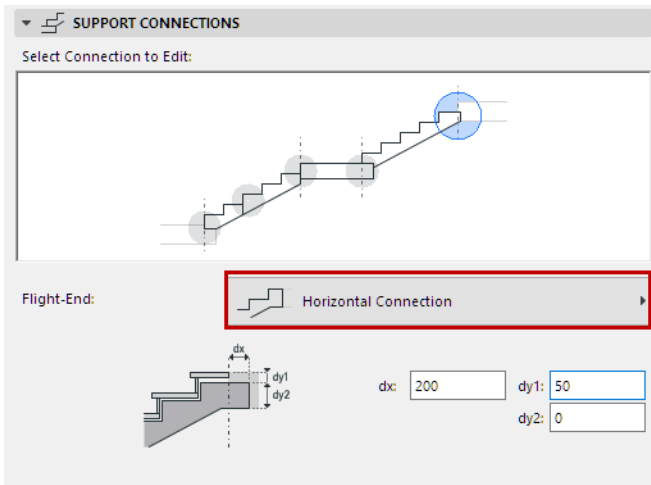
- In the Preview, hover and click to choose the connection or ending you want to edit.



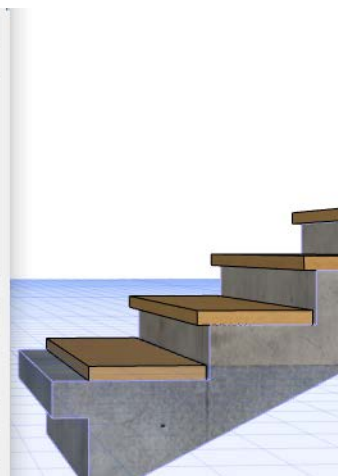
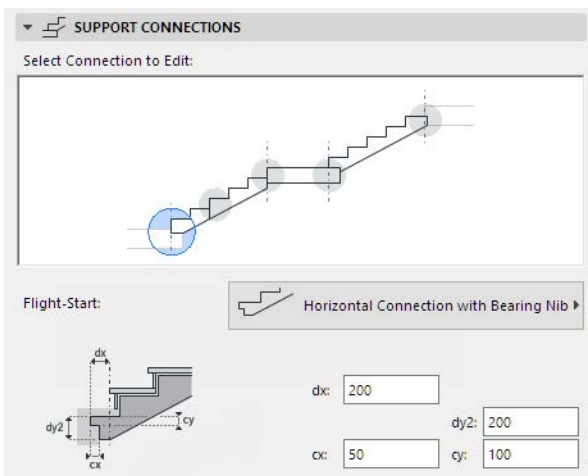
- From the pop-up, choose the desired Connection type. This is a fixed set of options, which vary depending on the structure and connection/type.
- The Automatic options require no additional settings from the user.



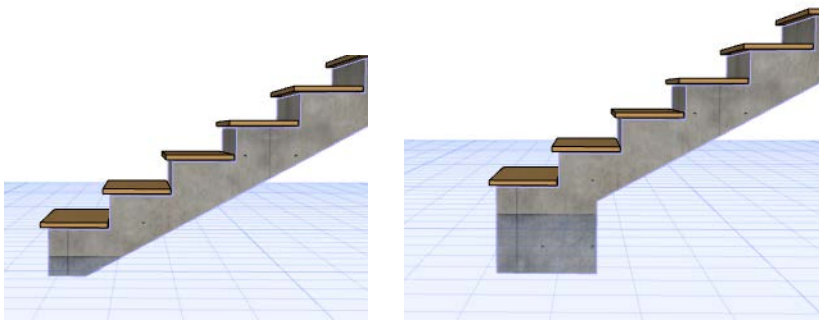
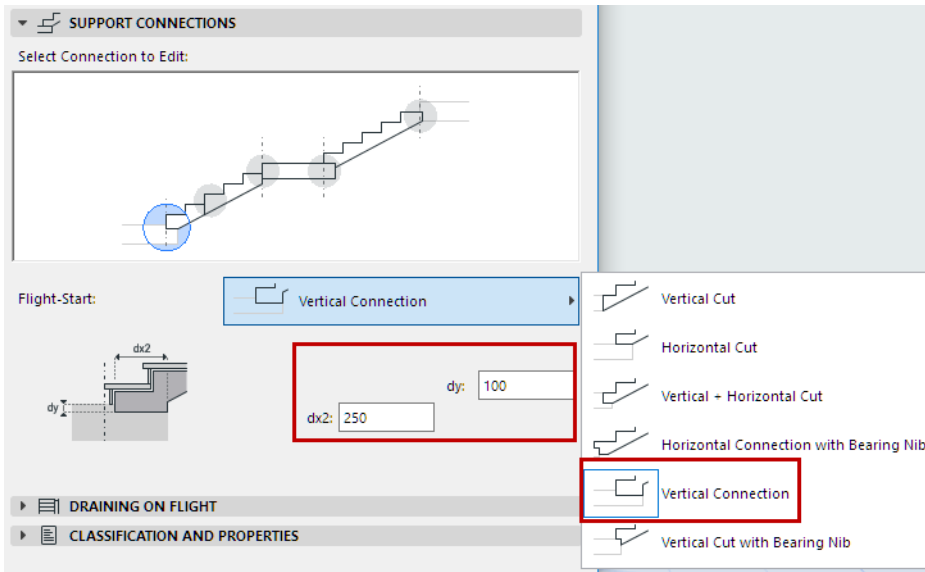
Landing-Flight Connection: Automatic Cut



Beam Flight Ends: Horizontal and Vertical



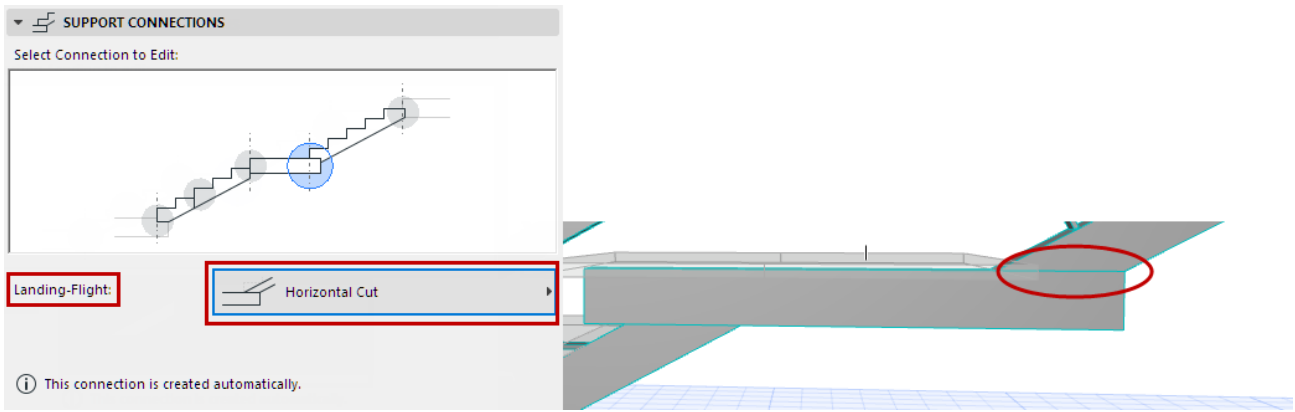
Monolithic Flight End: Horizontal with Bearing Nib



Monolithic Flight End: Adjust Vertical Cut

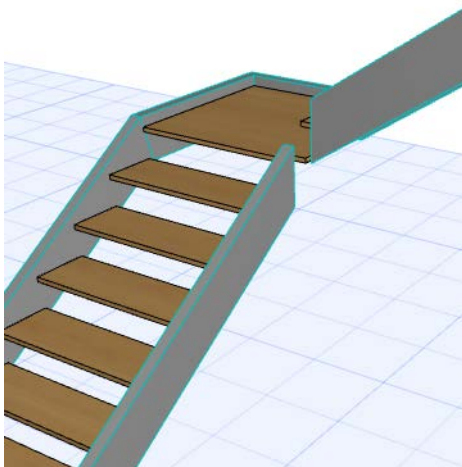
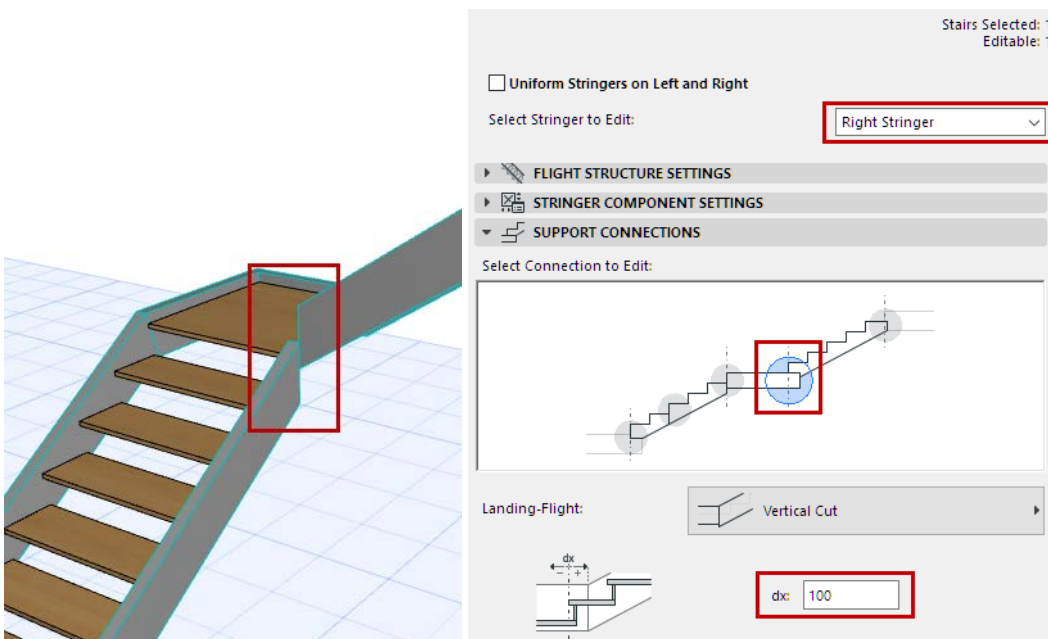


Landing-Flight Connection: Bisector



Landing-Flight Connection: Horizontal

Use the following connection type to make a space for an added element (e.g. Column):



Flight-Landing Stringer Connection: Adjust Vertical Cut to Add Gap

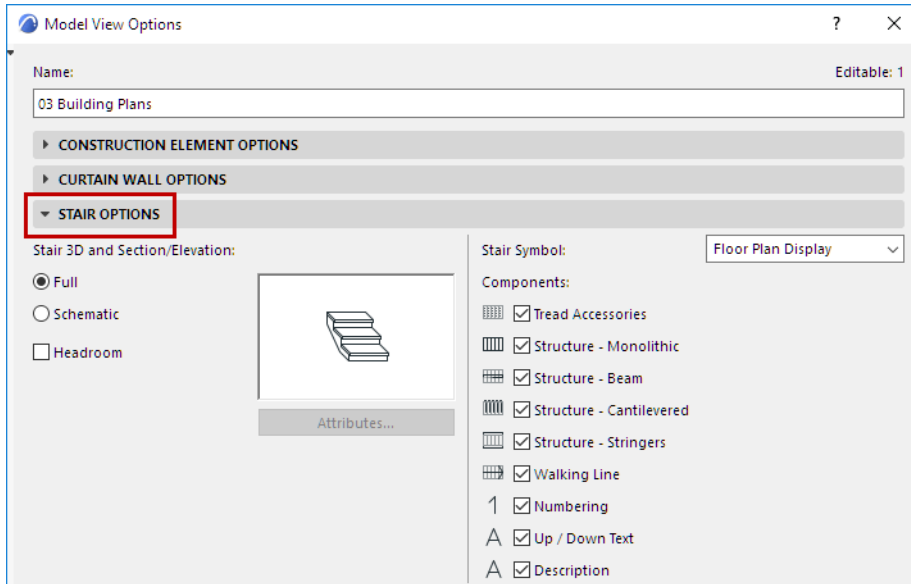
Model View Options for Stairs

Model View Options affect the on-screen and output display of Stairs in 2D and 3D. They set the detail level of Stairs in Sections, Elevations and 3D views, as well as their plan-specific display (Floor Plan or Reflected Ceiling) in 2D.

[Watch the Video](#)

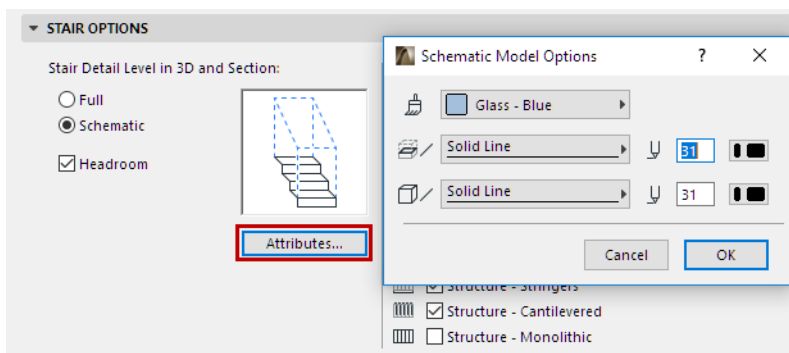
Stair Options

This panel is at **Document > Model View > Model View Options**.



Stair Detail Level in 3D and Section

Choose either **Full** or **Schematic**, to define the display and output of Stairs in Sections/Elevations, and in the 3D and 3D Document windows (including Edit mode).

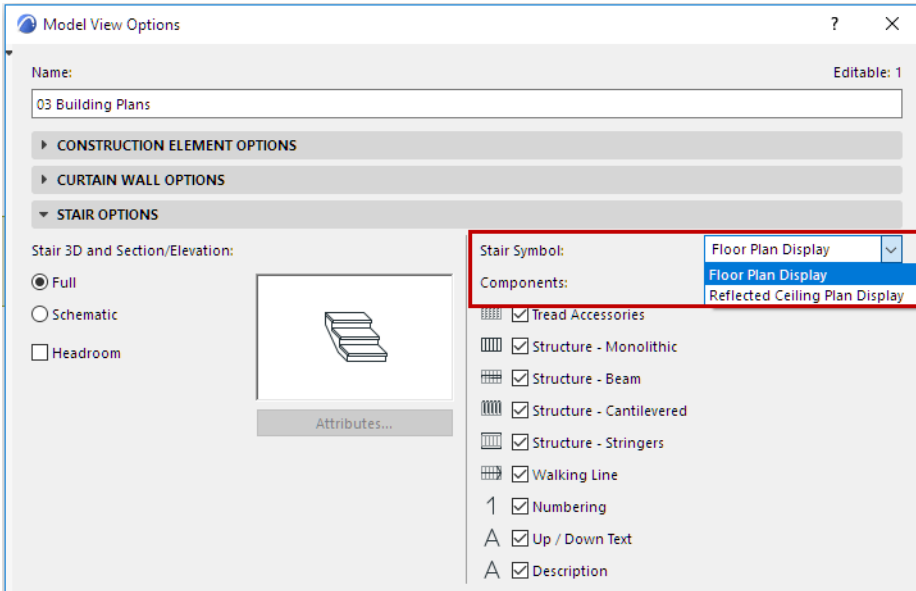


- In Full mode, the complete Stair appears: all 3D sub-elements can be seen in full detail.
- In Schematic mode, a continuous membrane surface is used to indicate the top surface of the steps.
 - For Schematic: Click **Attributes** to customize surface and line options for Stairs displayed in 3D and Section views.
- **Headroom**: Check to show or hide the display of Headroom in 3D and Section.

Note: To define Headroom rules: use the Rules and Standards panel of Stair Settings. (See [Headroom.](#))

Stair Symbol: Choose Floor Plan Display or Reflected Ceiling Plan

Set this control for the **Stair Floor Plan Symbol**: either “**Floor Plan Display**” or “**Reflected Ceiling Plan Display**”.

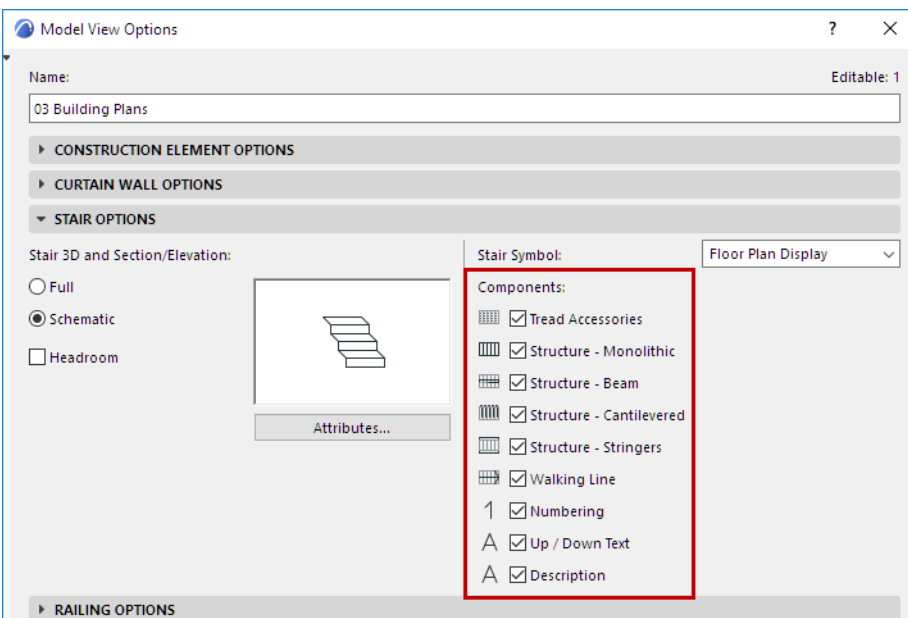


Each Stair (in its own Stair Settings dialog) has two groups of settings, defining display preferences for each of these two symbol types. Switch this Model View Option (Stair Floor Plan Symbol) to control which group of settings to use, by Model View.

[See also Floor Plan/Reflected Ceiling Plan Display \(Stair Settings\).](#)

Show/Hide Stair Symbol Components (by Model View Options)

Use the checkboxes at **Model View Options > Stair Options** to control which items should be used in Stair symbols, by Model View.



You can override these settings for any single Stair, in Stair Settings.

[See Stair Settings: Symbol Components.](#)

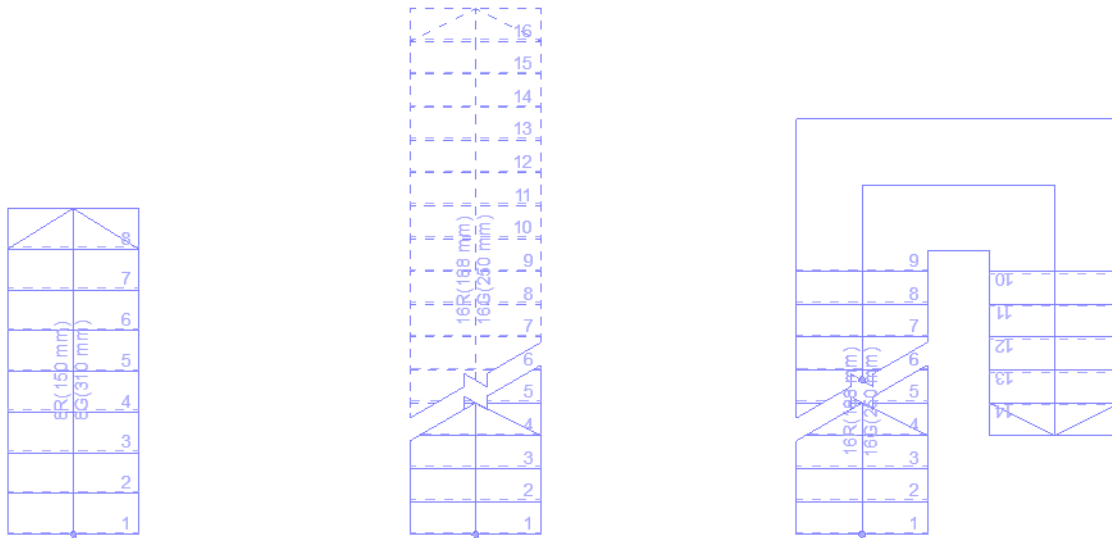
To fine-tune Symbol display (based on a Stair's Visible/Hidden parts), use the "Detail Level" panel of the Model View Options dialog box:

Detail Level of Stair and Railing Symbols

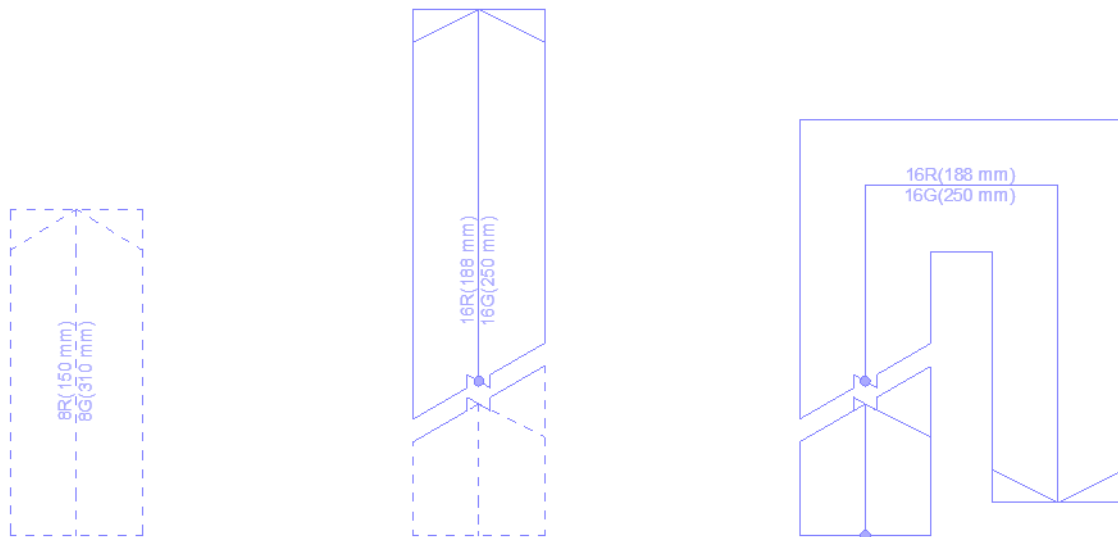
Model View Options for Stairs: Examples

The Model View Options for Stairs provide great flexibility for fine-tuning the 2D display of Stairs according to different Model Views.

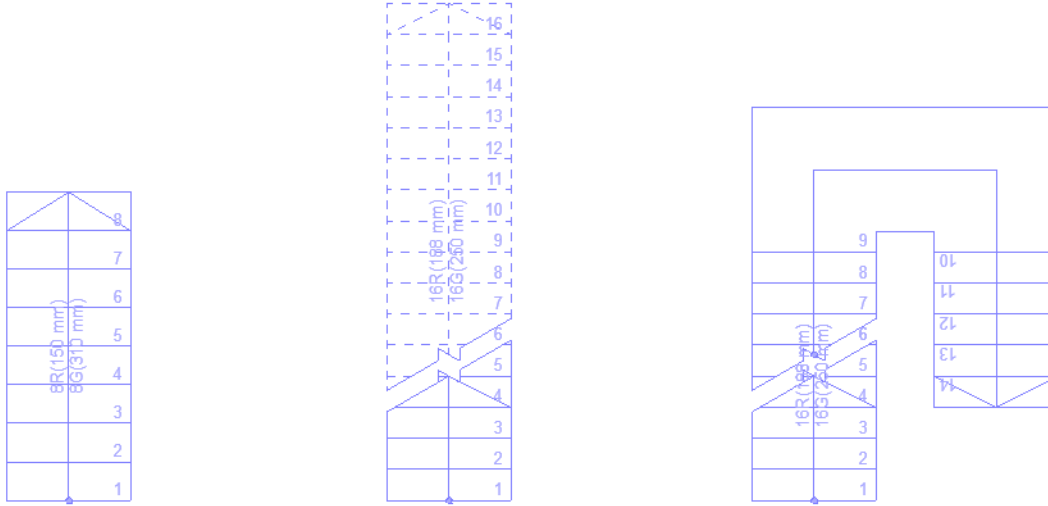
The following three Stairs are shown on the Ground Floor, at various scales. From left: An Entrance Stair; Stairs between Stories; and a Staircase.



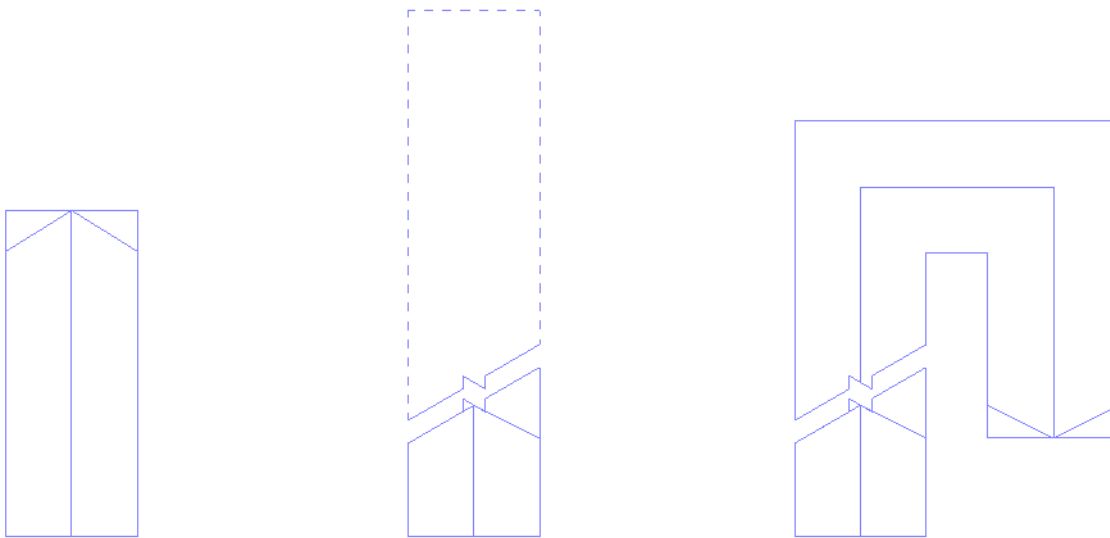
Construction Plan 1:50



Reflected Ceiling Plan 1:50



Building Permit 1:100



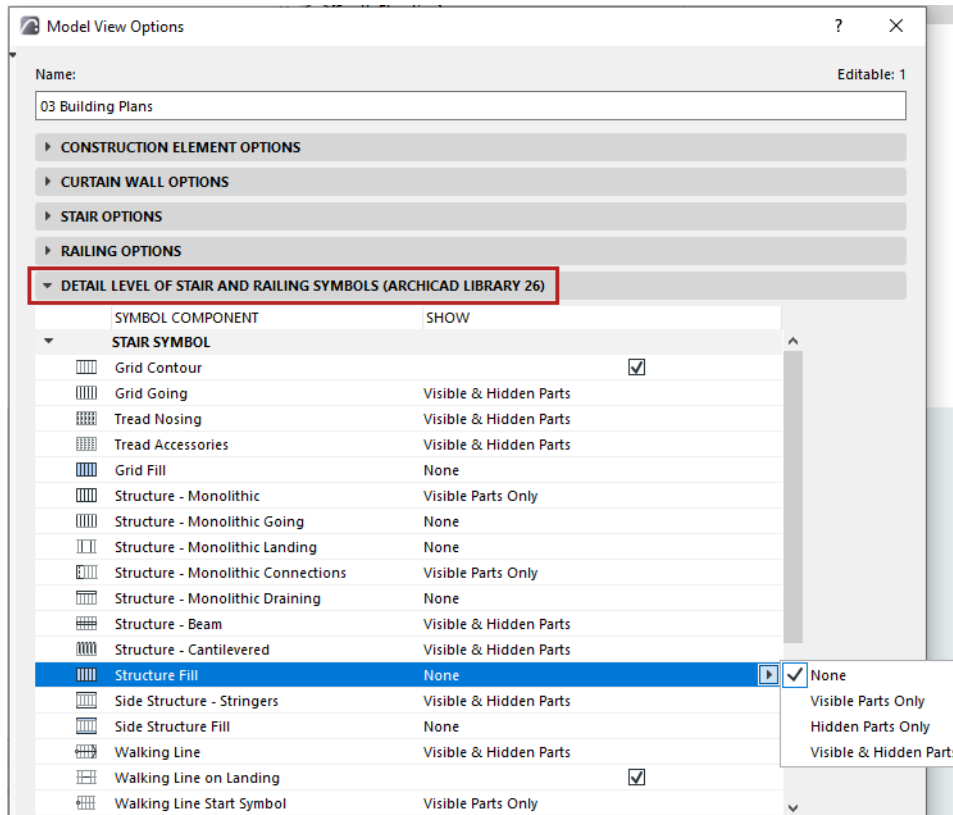
Site Plan 1:500

Detail Level of Stair and Railing Symbols

This panel is at **Document > Model View > Model View Options**.

The list includes **Stair Symbol** Components and **Railing Symbol** Components.

Note: Components are listed here only if they are checked in the “Symbol Components” list, in the Stair Options or Railing Options panels of the same dialog box.



Set the Detail level of each Symbol component, as applicable:

- None
- Visible Parts Only
- Hidden Parts Only
- Visible & Hidden Parts

Visible Parts and Hidden Parts

A Stair’s “Visible Parts” represent the Stair’s primary appearance, and are displayed using the “Visible” line type defined in Stair Settings (usually a solid line).

A Stair’s “Hidden Parts” represent the Stair’s secondary appearance, and are displayed using the “Hidden” line type defined in Stair Settings (usually a dashed line).

[See Set Attributes for Each Stair Symbol Component.](#)

Visible parts are usually under the Break Mark; Hidden parts are usually above the Break mark.

Related Topics:

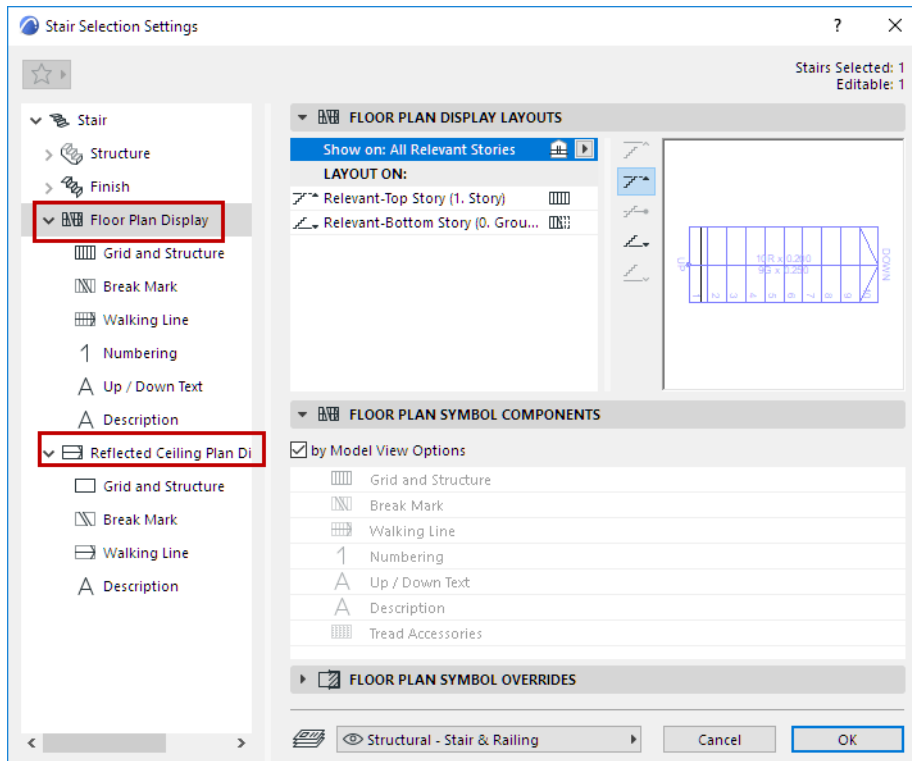
[Model View Options for Stairs](#)

[Model View Options for Railings](#)

Floor Plan/Reflected Ceiling Plan Display (Stair Settings)

Each Stair has two groups of settings, defining display preferences for the Stair Symbol (and its components) on Floor Plan and on Reflected Ceiling Plan.

Note: Switch the Stair's display by view, using Model View Options. [See *Stair Symbol: Choose Floor Plan Display or Reflected Ceiling Plan*](#).



Related Topics:

[Stair Display Layouts \(Show on Stories\)](#)

Choose the stories on which the Stair's 2D symbol should be displayed.

[Layout On \(Stair Display On Each Story\)](#)

Vary a stair's symbol display, story by story.

[Stair Settings: Symbol Components](#)

Show or hide each symbol component of the Stair. (Overrides the show/hide definitions at Model View options.)

[Stair Settings: Symbol Overrides](#)

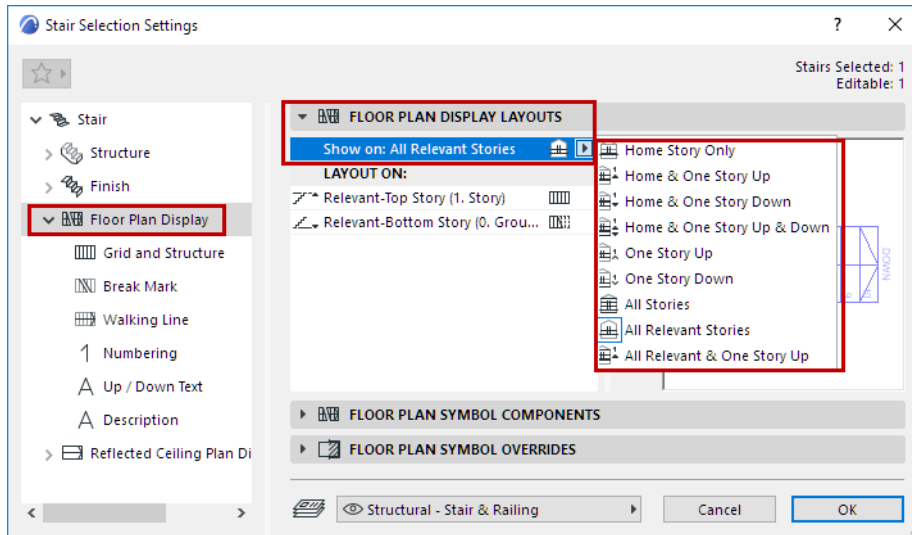
Apply overrides to the display of the Stair's entire 2D symbol (lines, pens, fills).

[Set Attributes for Each Stair Symbol Component](#)

Set attributes and choose symbol type for any Symbol component (e.g. Break Mark, Walking Line, Numbering).

Stair Display Layouts (Show on Stories)

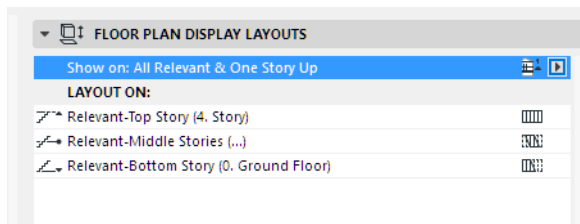
Use the pop-up to define the Stories on which the Stair's 2D symbol should be displayed.



The stories you define (for example, “All Relevant & One Story Up”) are listed in the field below (under “Layout On”).

Relevant Story

A Relevant Story is one which the Stair physically intersects. When defining Stories for Stair display, multistory Stairs can have, for example:

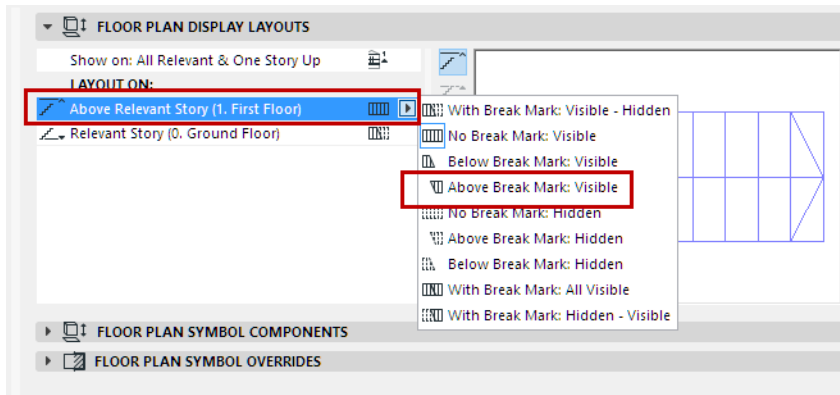


- a Relevant Bottom Story
- one or more Relevant Middle Stories
- a Relevant Top Story

Layout On (Stair Display On Each Story)

Under **Layout On**: Click on each Story name, then choose how to display the Stair on that story:

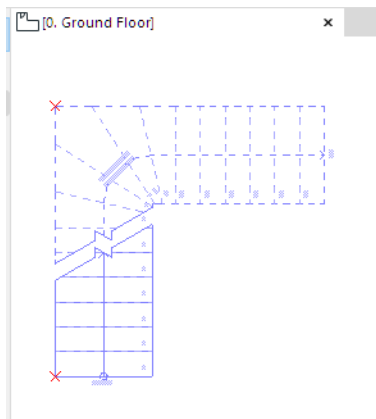
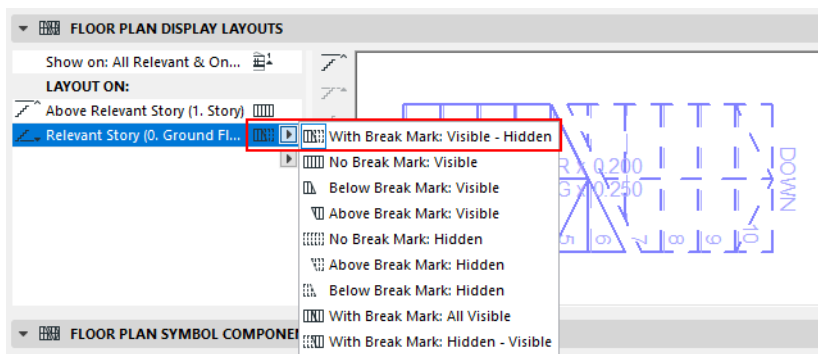
- **Break Mark**: With or without a Break Mark
- **Visible vs. Hidden Attributes**: To use either Visible or Hidden attributes on the displayed parts of the Stair. The set of attributes (for hidden and visible parts) is defined for each symbol component on its own settings page. See below.



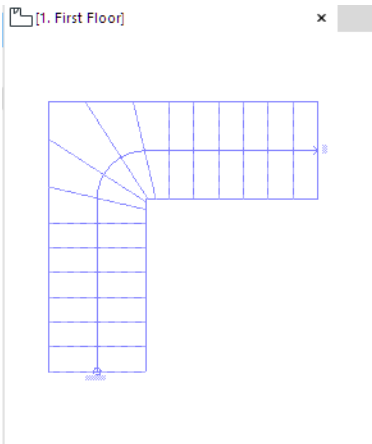
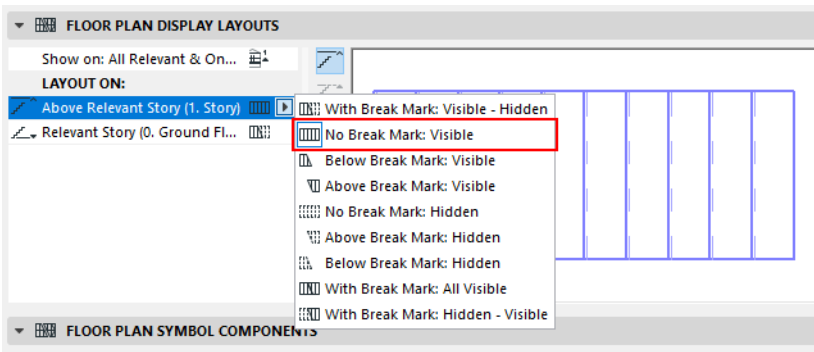
Use the Preview to get an idea of the Stair's 2D display on each story.

Note: If the Stair has an associated Railing, you can apply these same Display settings to the Railing: Use the checkbox in Railing Settings ([see Railing Display Layouts \(Show on Stories\).](#))

Stair Layout Examples



Stair Layout on Ground Floor: With Break Mark, Use Visible Line Type (Below) and Hidden Line Type (Above)

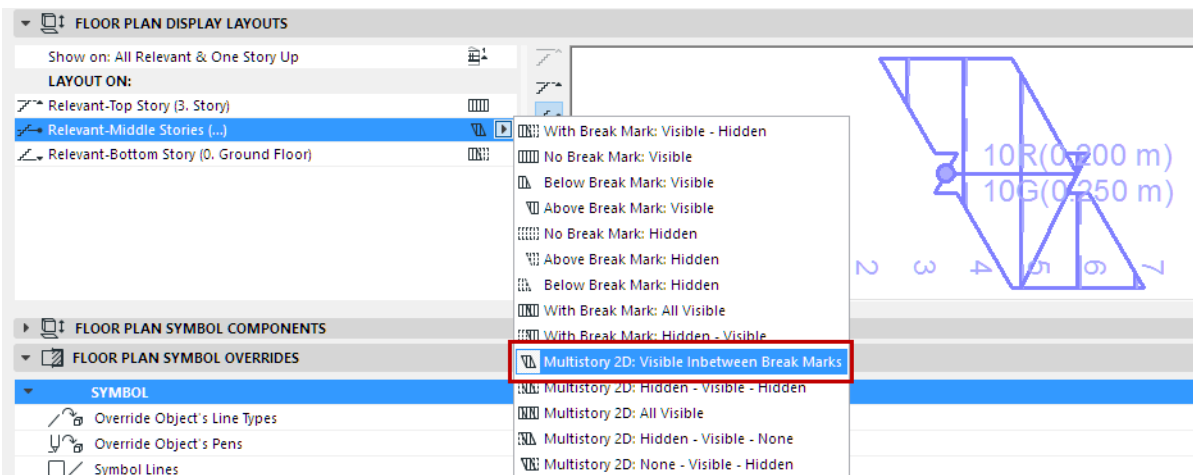


Stair Layout on First Floor: No Break Mark, Use Visible Line Type

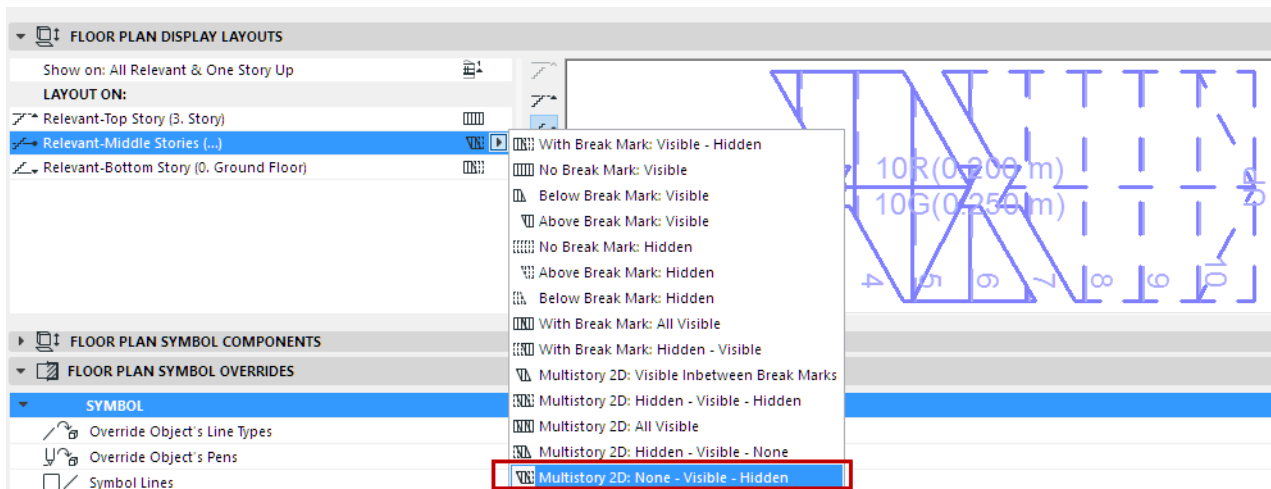
Additional Layouts are available for multistory Stairs, with multiple Break Marks.

[See also Relevant Story.](#)

On a given Story, you can just display the part between Break Marks.



Stair Layout on Middle Stories: Display Between Break Marks Only



Stair Layout on Middle Stories: None (below Break Line1) - Visible (Between Break Lines) - Hidden (Above Break Line 2)

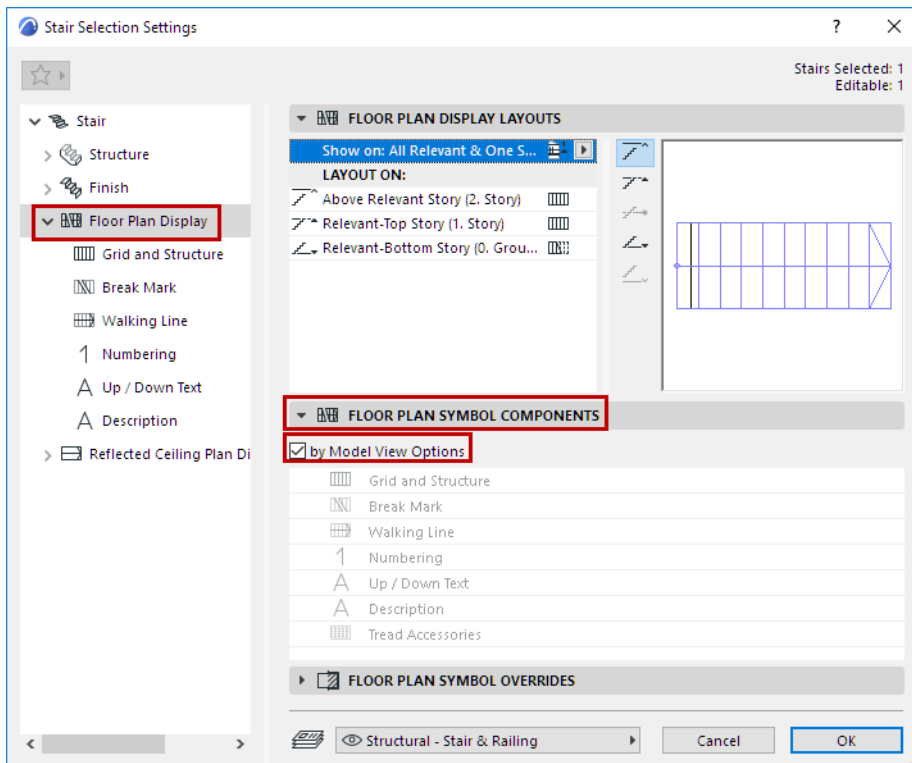
Stair Settings: Symbol Components

These controls are available for both the Floor Plan Display node and the Reflected Ceiling Plan Display node of Stair Settings.

Symbol Components: By Model View Options

Check this to show/hide the Stair's 2D components as defined at **Document > Model View > Model View Options**.

See [Show/Hide Stair Symbol Components \(by Model View Options\)](#).



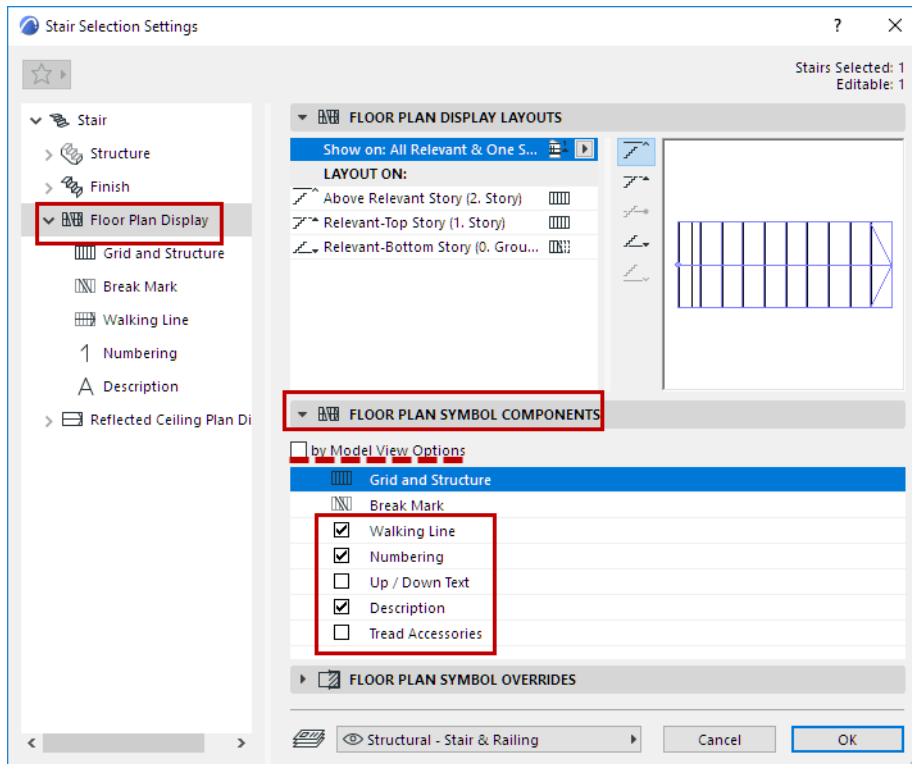
Symbol Components: Customize in Stair Settings

Uncheck “by Model View Options” and use the list below to specify which symbols to show or hide. The Grid and Break Mark cannot be hidden.

For example: If you have both an existing and a new Stair in the same view, you may prefer to display the existing one with less detail.

Select that Stair, open its Settings, and *uncheck* **By Model View Options**.

From the list below, uncheck the items (e.g. Description, Numbering) that you don’t need to see on that Stair.



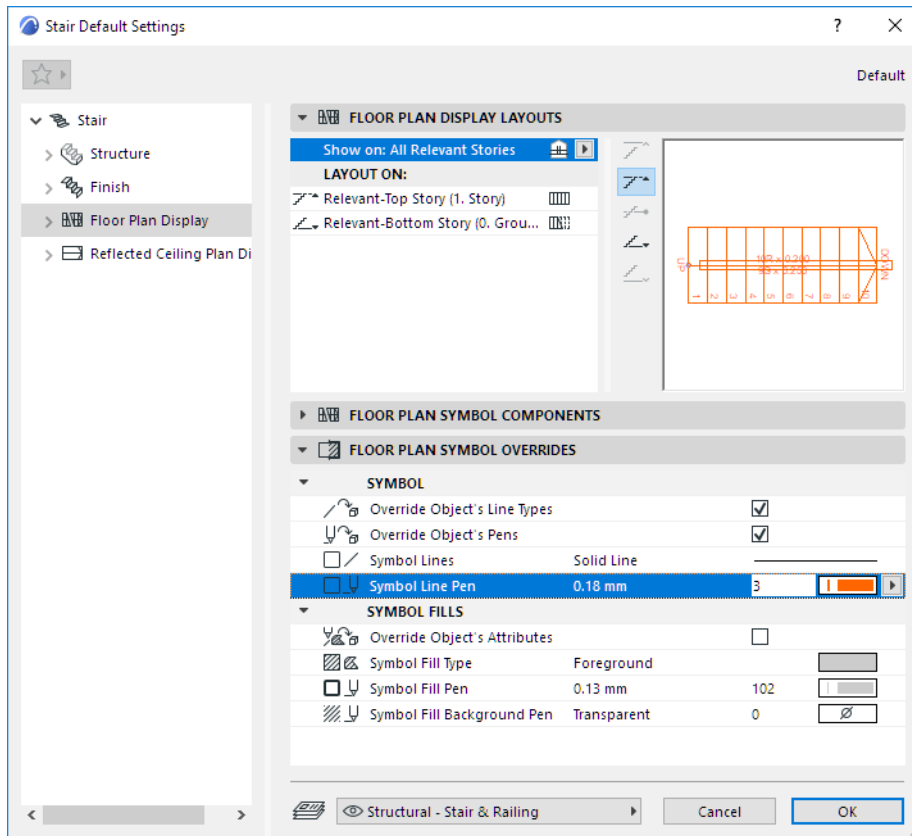
Note: This Stair, for which you customized the visibility of its Symbol components, will always appear as you set it, regardless of the current Model View Options setting!

Stair Settings: Symbol Overrides

Available for both the Floor Plan Display node and the Reflected Ceiling Plan Display node of Stair Settings. These settings are independent of Model View Options.

Use them to optionally apply overrides to the display of the Stair’s entire 2D symbol (lines, pens, fills).

- To apply a Line/Pen/Fill override: Check the respective **Override** box, then select an attribute from the controls below.



Set Attributes for Each Stair Symbol Component

Available for both the Floor Plan Display node and the Reflected Ceiling Plan Display node of Stair Settings. These settings are independent of Model View Options.

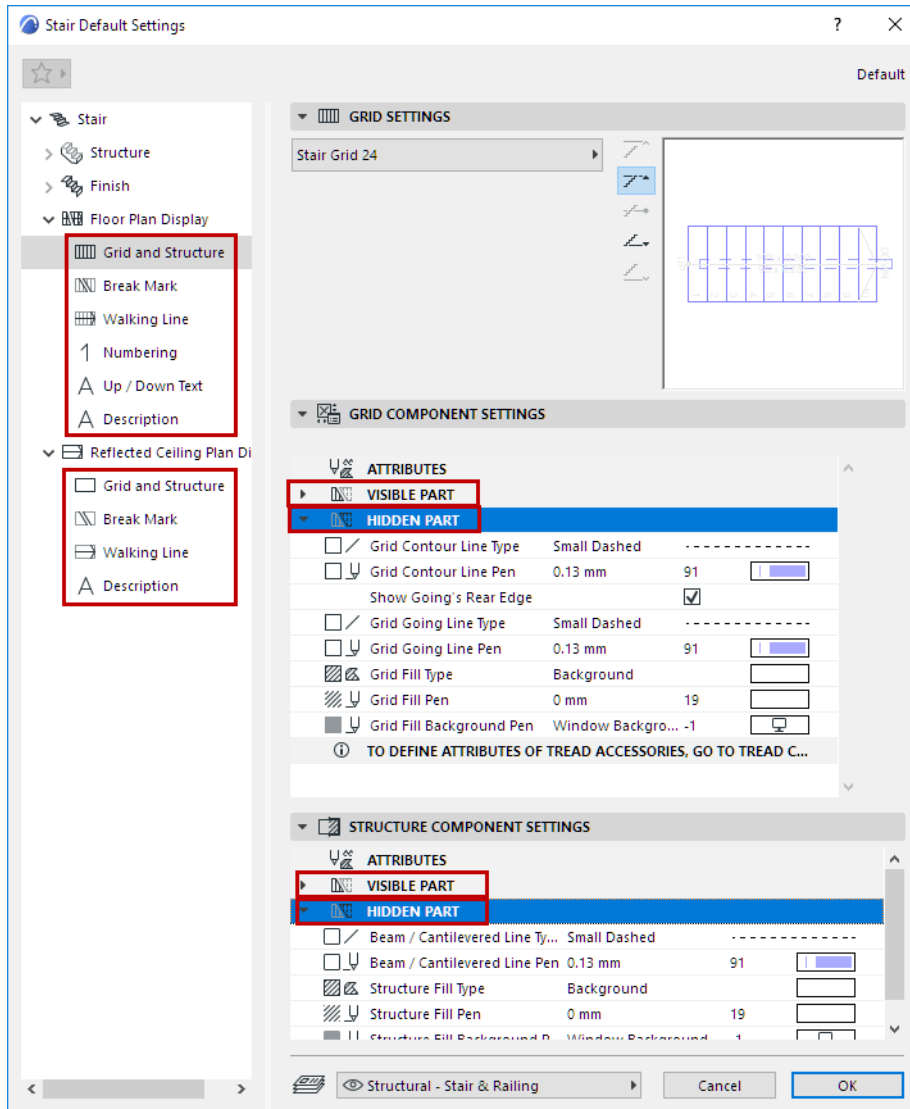
To set attributes, choose a Symbol component (Break Mark, Walking Line, etc.) from the Stair Settings tree structure, to access its settings.

For each Symbol Component:

- Choose a Component type from the pop-up menu
- Set the Component's attributes in the bottom panel - separately for the Visible and Hidden parts of the symbol, as applicable.

See also [Visible Parts and Hidden Parts](#).

- View the Preview to get an idea of how the component will look.



Structure Component Settings

Use these controls to set the 2D display of Stair Structures (Flight and Landing), depending on the Structure type you are using.

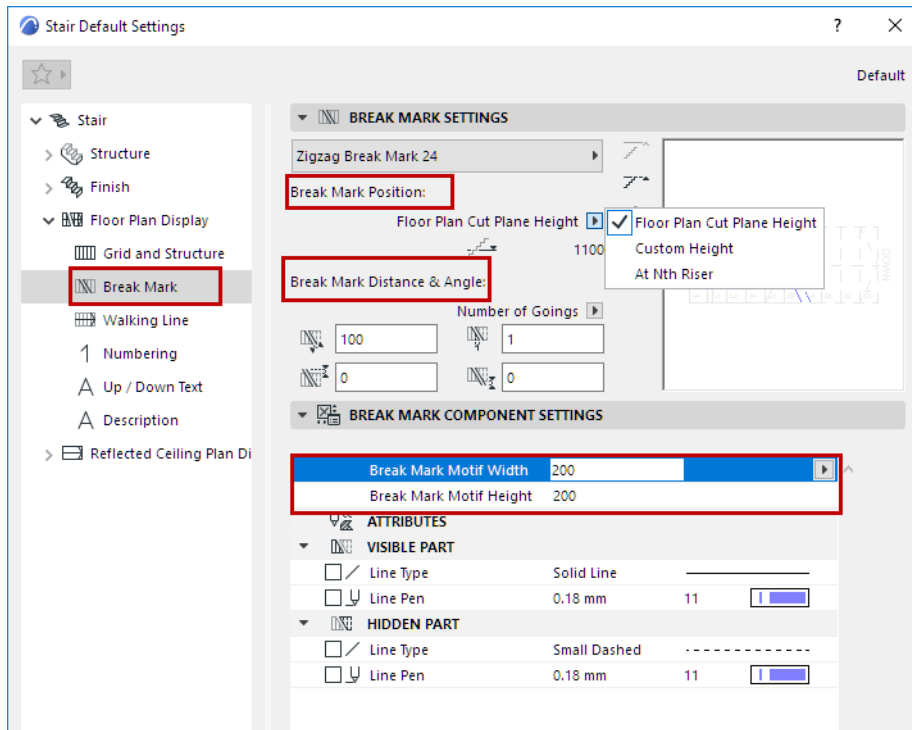
Note: Stair Structure components are selected on the Structure node of Stair Settings. [See Flight and Landing Structures.](#)

Attributes vary for the structure's Visible and Hidden parts.

Break Mark Symbol

- Choose a Break Mark Symbol (e.g. Zigzag)
- Set Break Mark position by Height or at a particular Riser number
- Set Break Mark angle (to Vertical/Horizontal, or define the number of goings that it should span)
- Apply an optional extension to the Break Mark lines

- Set Break Mark attributes (Break Mark Component Settings), including the width of the “Motif” (e.g. zigzag) if any



You can edit the Break Mark symbol graphically, after selecting it in Edit mode (Symbol view).

[See also *Edit Break Mark Symbol*.](#)

Walking Line Symbol: Position and Display Settings

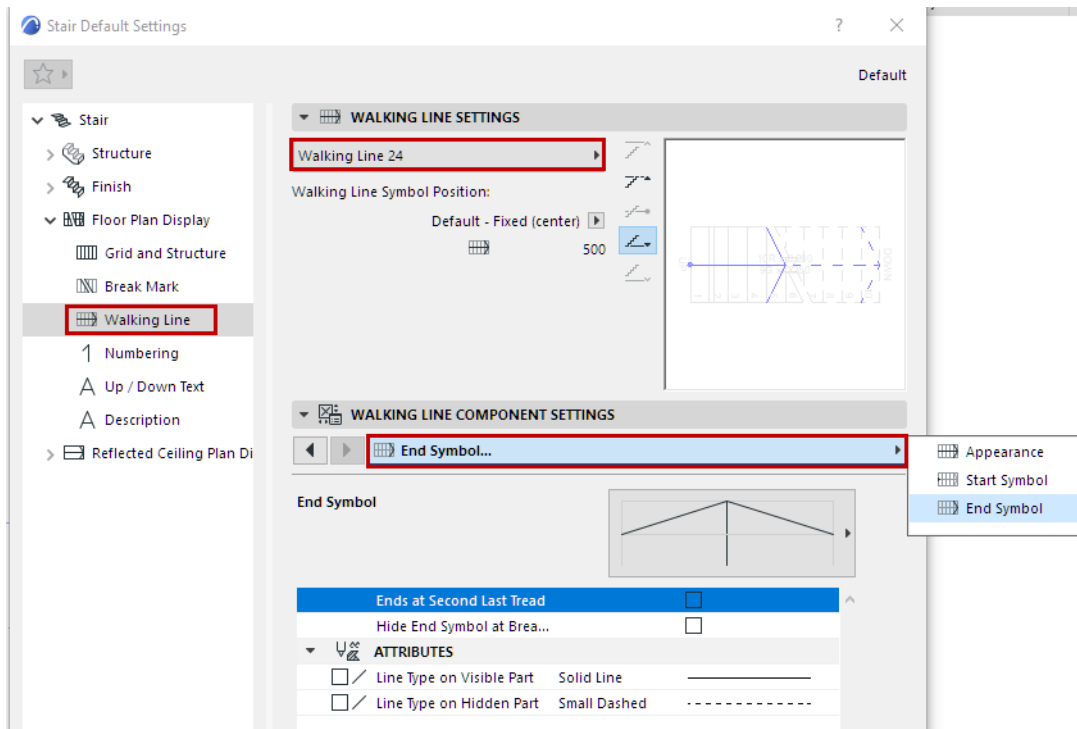
The Walking Line Symbol Position refers to the Symbol, whose position you can edit manually, either here or in Edit mode.

(In contrast, the Calculated Walking Line, defined in the Geometry and Positioning panel of Stair Settings, affects Tread geometry, and can be edited in Stair Settings only. [See *Offset Walking Line on Turning \(Winder with Equal Goings\)*.](#)

In the Display Settings shown here:

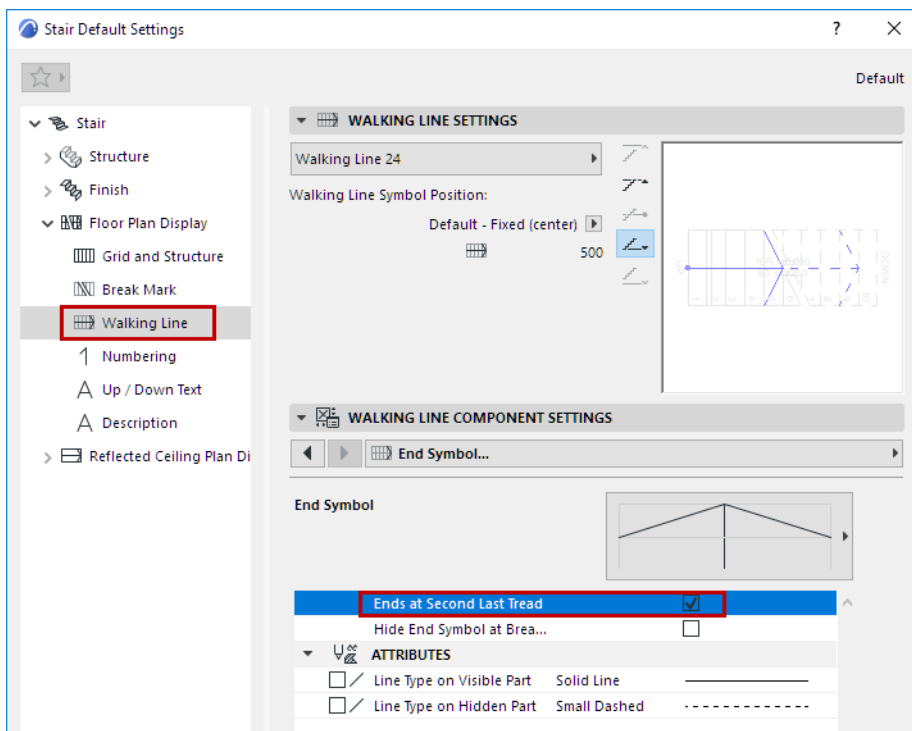
- “Default” setting means that the Walking Line Symbol is at the same location as the Calculated Walking Line, defined in Stair Settings - Geometry and Positioning panel
- “Center” places the Walking Line Symbol at the center of the Stair
- “Custom” means that the Walking Line has been edited graphically to a new position

[For details on Symbol editing, see *Edit Stair Symbol Components Graphically*.](#)



Walking Line End Symbol: End at Second to Last Tread

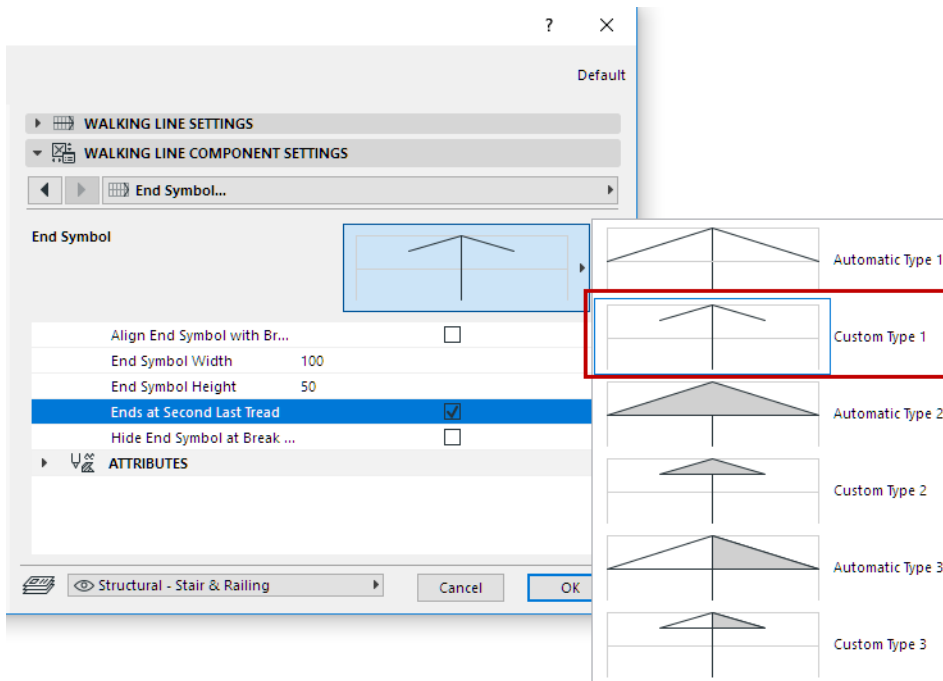
Check this option to end the Walking Line at the tread before the last one.



Walking Line End Symbol: Edit Arrow Width/Height

When choosing an End symbol for the Walking Line, note that the “automatic” options mean that the arrow endpoints are fixed, to exactly one Going width.

To edit arrow endpoints, choose a Custom symbol type. You can edit the arrow graphically, too, in Edit mode.



See also [Edit Walking Line Arrow](#) and [Edit Walking Line Symbol Position](#).

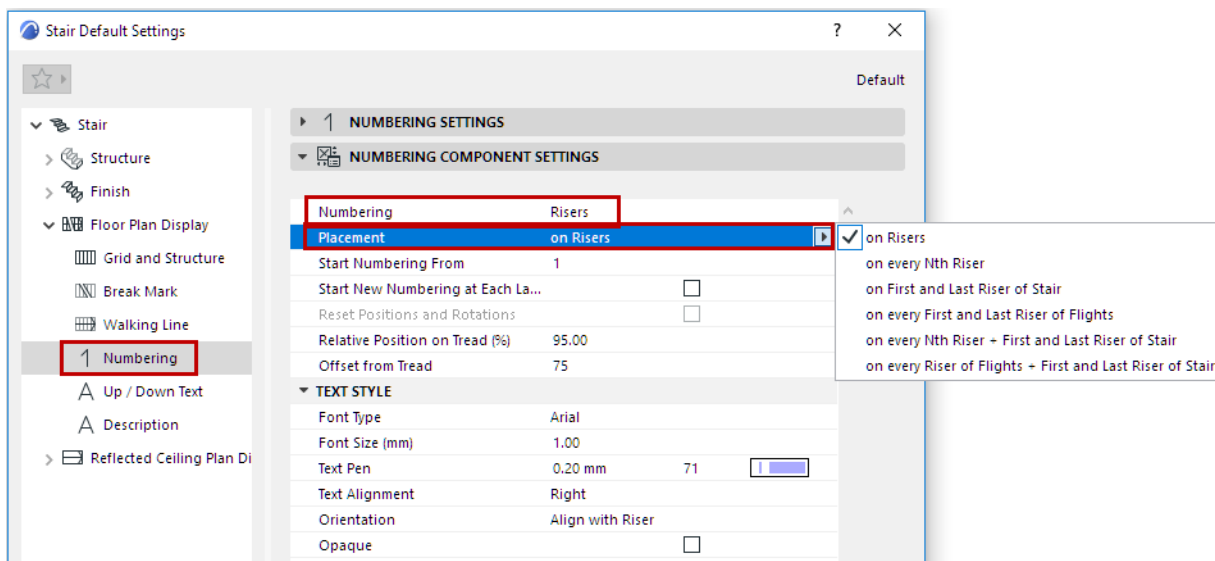
Numbering Symbol: Show Numbers on Stories

Use these controls to set the Numbering preferences for the Stair, including story-dependent display, as needed.

Numbering: Choose Risers or Treads.

Placement: Choose which Treads/Risers should be numbered.

Other options let you define where to begin and/or end the Numbering, or to restart Numbering at Landings.



You can edit (move or rotate) the numbers graphically, after selecting them in Edit mode (Symbol view).

See [Edit Stair Numbering Items](#).

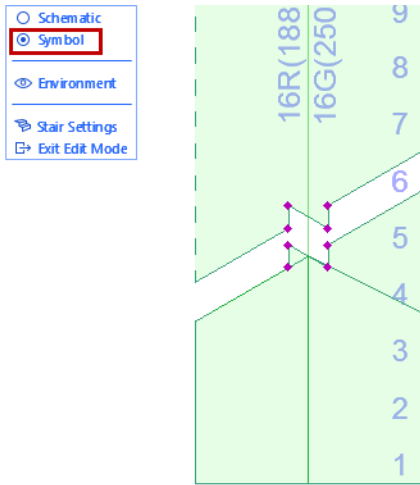
You can also use special Stair Labels to number Treads and Risers: See [Labeling Stairs](#).

Edit Stair Symbol Components Graphically

Use Edit mode to edit the Stair's symbolic components graphically, after selecting the component in Edit mode (Symbol view).

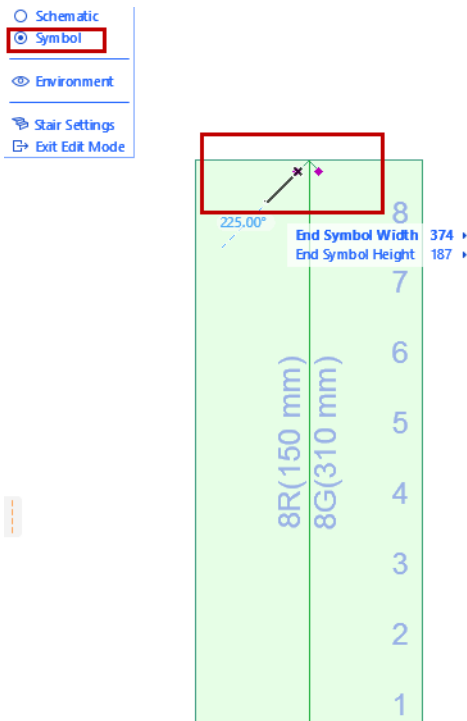
For example:

Edit Break Mark Symbol



Edit Walking Line Arrow

To edit arrow endpoints graphically in Edit mode, you must be using a Custom symbol type.

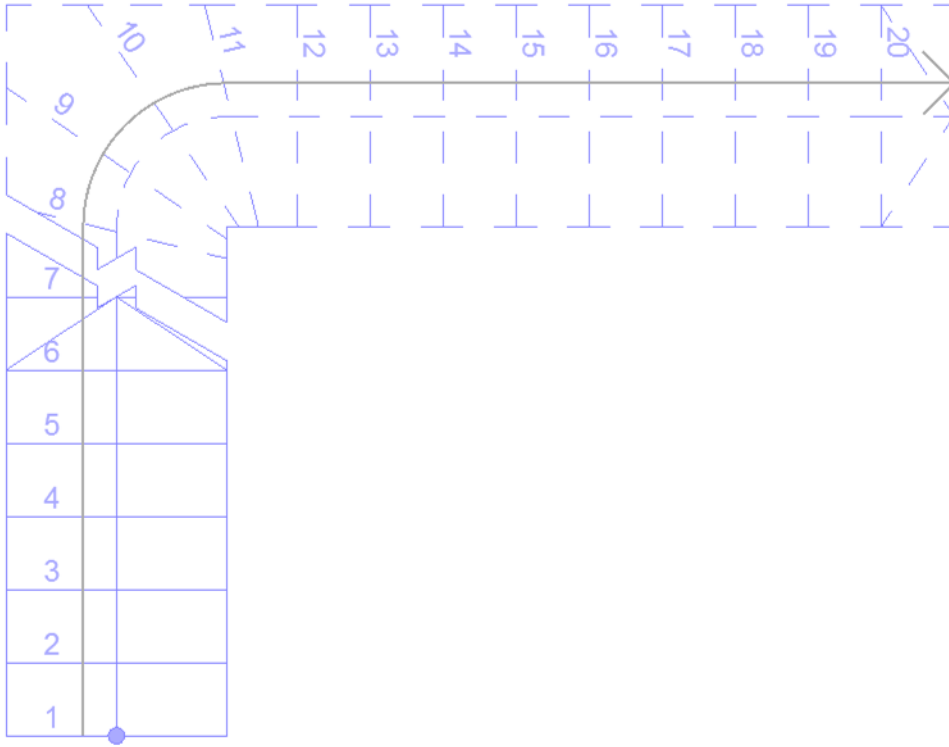


Choose the Walking Line End Symbol in the Floor Plan Display or Reflected Ceiling Plan pages of Stair Settings.

See [Walking Line End Symbol: Edit Arrow Width/Height](#).

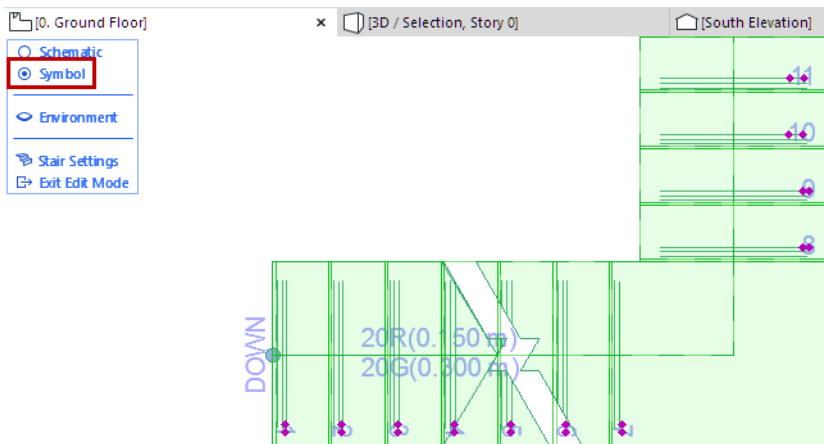
Edit Walking Line Symbol Position

Make sure you are in Symbol mode, then select and graphically edit the position of the Walking Line symbol.



Edit Stair Numbering Items

You can edit (move or rotate) the numbers graphically.



Related Topics:

[Set Attributes for Each Stair Symbol Component](#)

[Walking Line Symbol: Position and Display Settings](#)

[Watch the Video](#)

Railings

[About the Railing Tool](#)

[Place Railing: Basic Polyline Input](#)

[Place Associative Railing](#)

[Working with Railing Segments and Nodes](#)

[Railing Reference Line](#)

[Railing Tool Edit Mode](#)

[Railing Settings: Overview](#)

[Railing Settings: Geometry and Symbol Display](#)

[Segment Settings \(Railing Tool\)](#)

[Toprail Settings \(Railing Tool\)](#)

[Handrail Settings \(Railing Tool\)](#)

[Rail Settings \(Railing Tool\)](#)

[Inner Post Settings \(Railing Tool\)](#)

[Baluster Settings \(Railing Tool\)](#)

[Panel Settings \(Railing Tool\)](#)

[Node Settings \(Railing Tool\)](#)

[Post Settings \(Railing Tool\)](#)

[Connection Settings \(Toprail, Handrail, Rail\)](#)

[Railing Ends \(Toprail, Handrail, Rail\)](#)

[Model View Options for Railings](#)

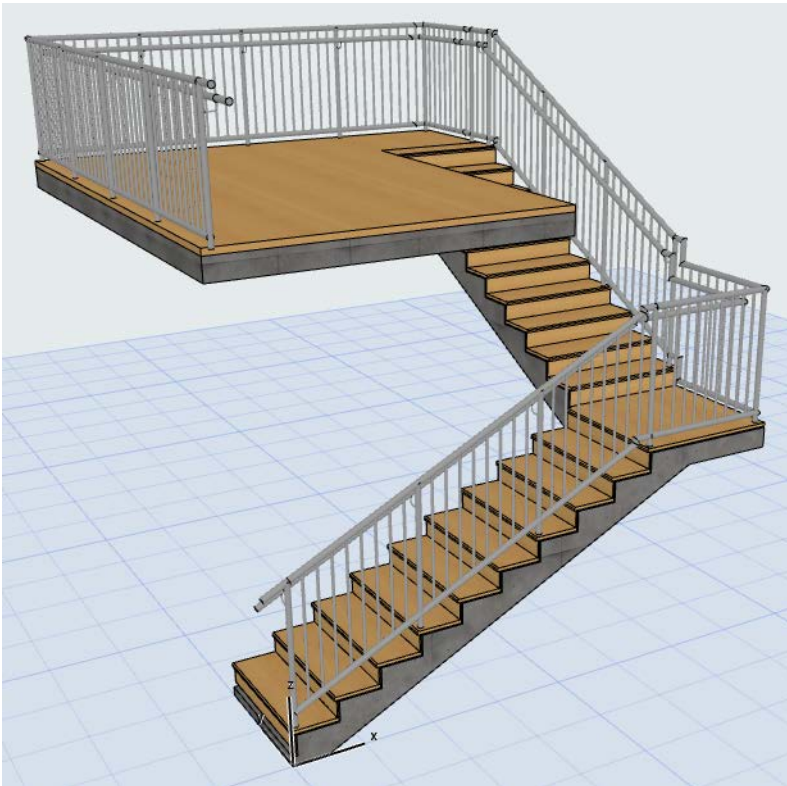
About the Railing Tool

The Railing Tool lets you create any typical or unique Railing, which is a 3D building element. A Railing can be associative to the following 3D elements: Stairs, Slabs, Walls, Roofs and Meshes.

Railing input is similar to polyline input, and consists of **Nodes** and **Segments**. On the Floor Plan, Railing interactions are similar to those of a polyline.

The interactive Pattern Editor helps you assemble a Railing pattern specific to each Segment.

Like the Stair and Curtain Wall, the Railing Tool is a hierarchical element made up of a main scheme plus its components: various posts, rails, and/or panels. These are GDL components that use Building Materials and/or Custom profiles.



[Watch the Video](#)

Railing Components

- The **Toprail** sits on top of the Segment.
- The Railing can contain up to two **Handrails**, each of which can be doubled to the two sides of the Railing. (The two handrails can have different cross sections).
- **Rails** are general horizontal elements. You can offset them perpendicular to the Railing Segment or set them at any height. Rail cross sections can be different for each.
- All **horizontal elements** (Toprail, Handrails, Rails) can have Connections and Endings added to either end.
- You can add multiple **Inner Posts** to the pattern.
- **Balusters** are a pattern within the pattern
- **Panels** can fill the area between the Toprail, Rails, Inner Posts or Segment boundaries.

- For Panels and Posts, you can choose not just built-in GDL components, but also custom ones, saved using a predefined Railing subtype.

Railing Display

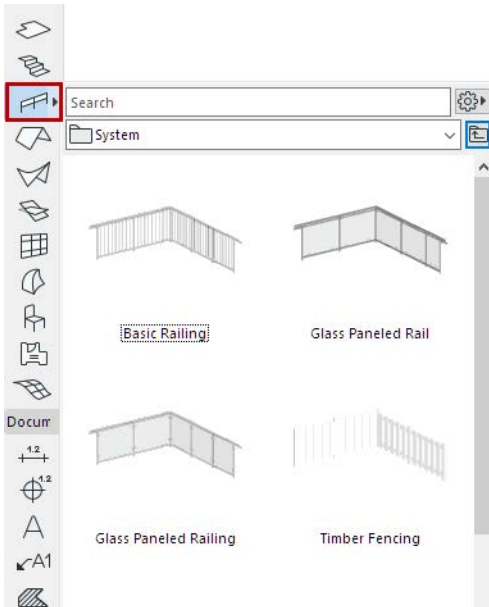
Model View Options provides three Detail Levels (Full, Simplified and Schematic) to display and output Railings in Sections/Elevations, the 3D and 3D Document windows.

See [Railing Options](#).

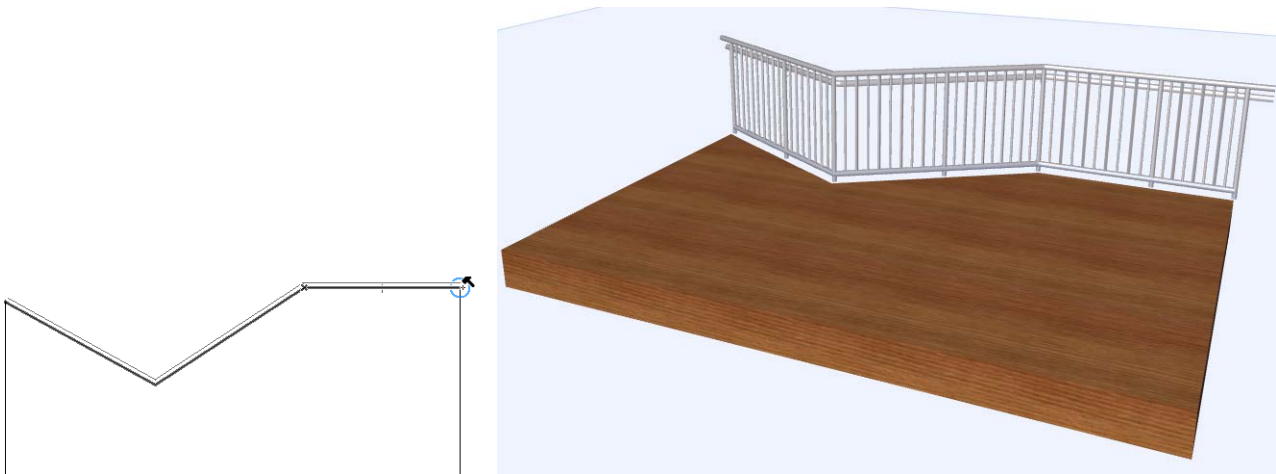
Place Railing: Basic Polyline Input

Railing input is similar to polyline input, and consists of Nodes and Segments along a Railing Reference Line.

1. Activate the Railing Tool from the Toolbox or Info Box. Choose a Favorite, if desired.



2. In the Floor Plan or 3D window, draw a polyline representing the Railing Reference Line. Each click represents a Node.
3. Double-click to complete.

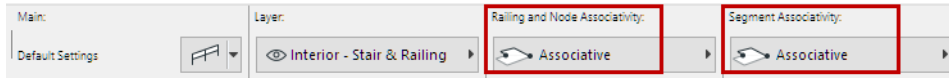


See also [Place Associative Railing](#).

[Watch the Video](#)

Place Associative Railing

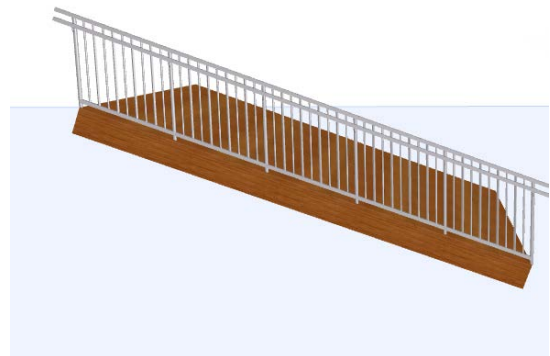
By default, a new Railing is set to be associative to another model element. Archicad Railings can be associative to one or more Stairs, Slabs, Walls, Roofs or Meshes.



This means that if any of these element geometries is modified (such as the Stair on which you placed the Railing), the associated Railing geometry will move with it. If you delete the parent element, the Railing is deleted too.

You can associate a Railing to the following elements:

- Stair (Railing is associated to the Stair Boundary)
- Reference Plane of Slab and Roof
- Reference Line of a Wall
- Nodes and ridges of a Mesh (e.g. fencing)

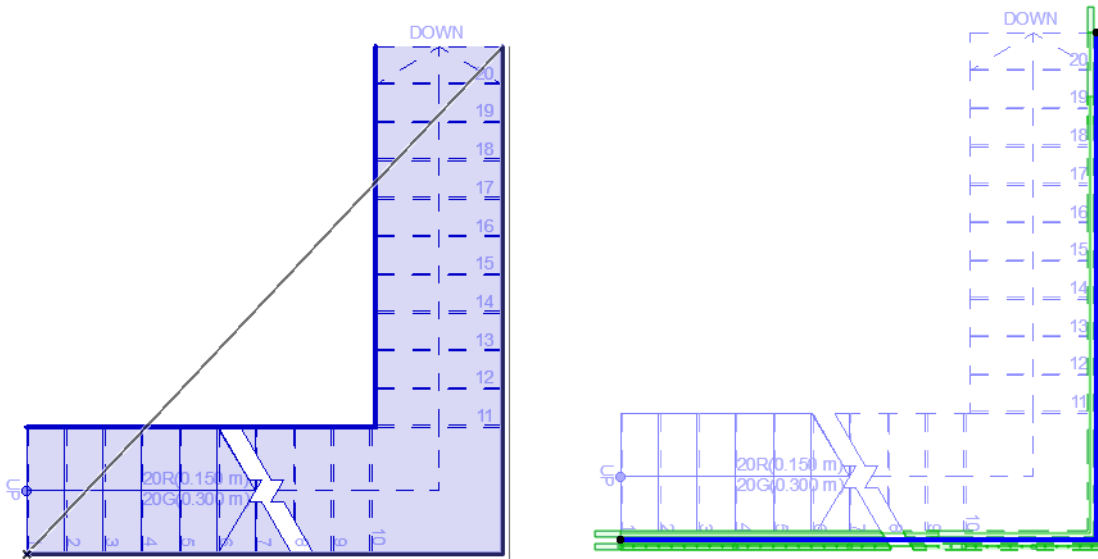


Associate Railing to 3D Model Elements

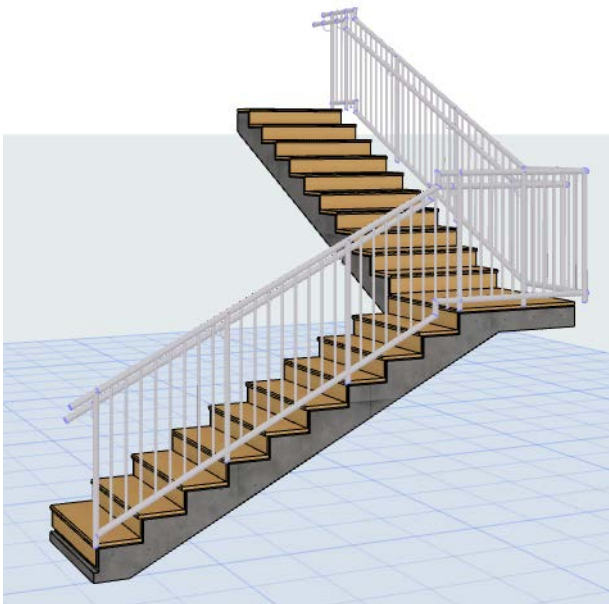
[See the examples below.](#)

Associative Railing on a Stair

1. Activate the Railing Tool from the Toolbox or Info Box.
2. Click at both ends of the Stair Boundary line, which is highlighted during associative Railing input.



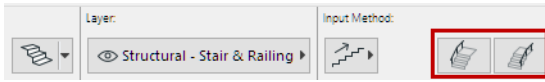
3. Double-click to complete the Railing, which is associated to the Stair. The Railing will automatically follow changes to Stair boundary line (curve/straighten edge, insert/remove Node).



Railing Associative to Stair

Add Railing Automatically to New Stair

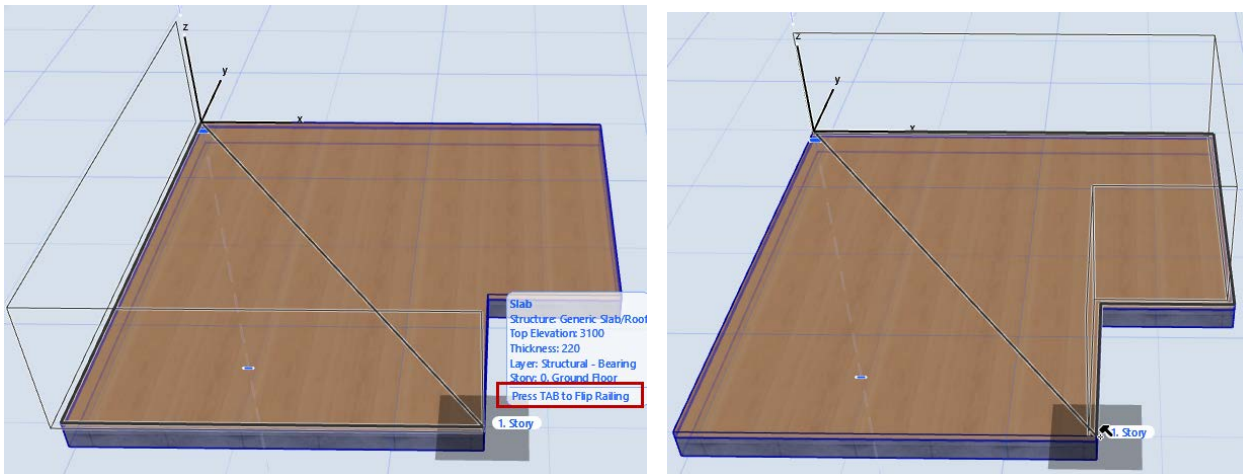
Before placing a Stair, you can choose to add a Railing to it automatically: click one or both “Add Railing” options in the Stair Info Box. This way, you place two elements at once (Stair plus Railing).



Stair Info Box: Add Railing(s) when Placing Stair (Right, Left, Both sides)

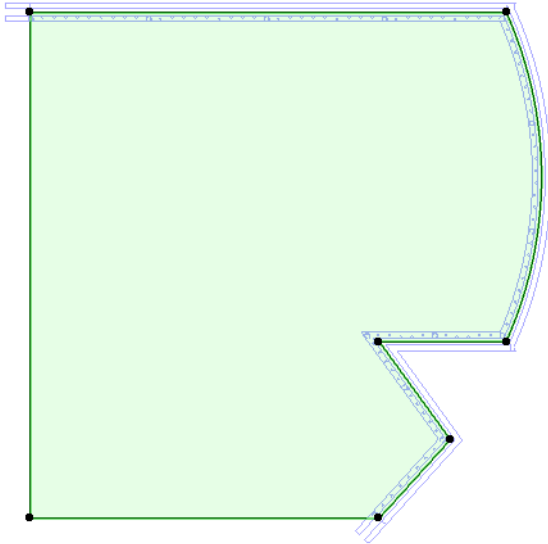
Associative Railing on a Slab

1. Activate the Railing Tool from the Toolbox or Info Box.
2. Click on a Slab node.
3. Move the cursor to another node and hover the cursor.
4. Click Tab to see feedback for the alternative Railing paths:



Press Tab to Choose Railing Path Around Slab

5. Double-click to place the Railing, which is associated to the Slab.



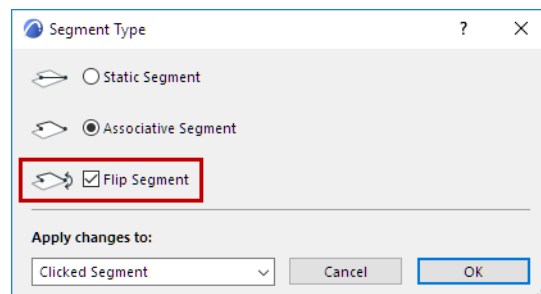
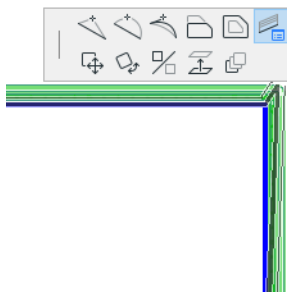
Associated Railing Follows Edits to “Parent” Slab

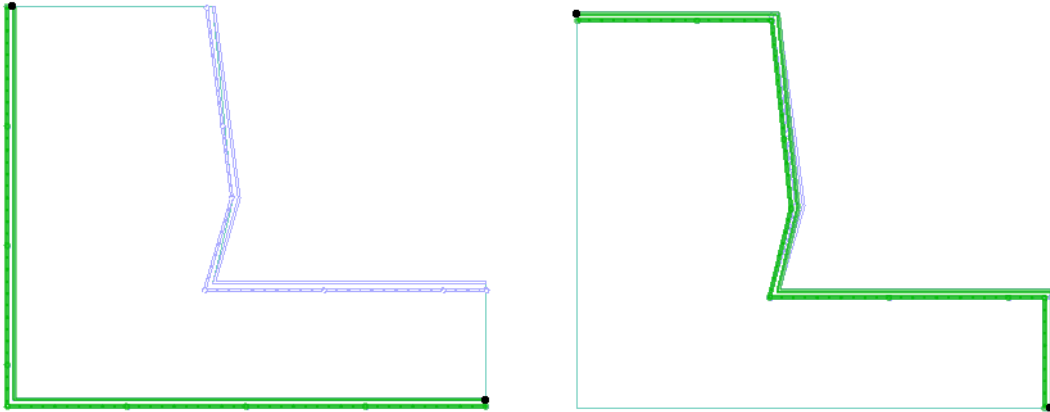
Flip Segment (Redefine Railing Polyline)

If the Railing is associated to one part of a polygon element, you can flip it to other parts of the parent element.

Note: This is only available if the parent element is a polygon (e.g. a Slab). It is not available for Railings on a Stair.

For a selected Railing Segment, use the Segment Type pet palette control. Check the Flip Segment control to flip the Railing to another part of the parent element polygon:



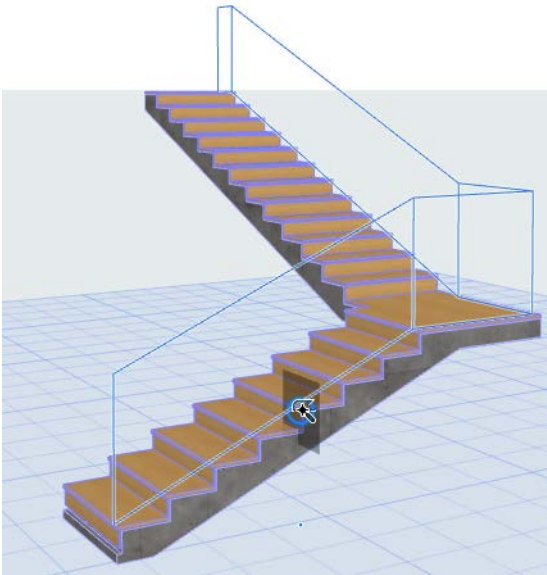


Flip Railing Segment Along Parent Element Polygon

Associative Railing with Magic Wand

Place Railings on associated elements (Stair, Slab, Morph, Wall, Roof) using the Magic Wand.

With Railing tool active, press the Spacebar, then click any element to automatically trace its edge or polygon shape and place an associated Railing.



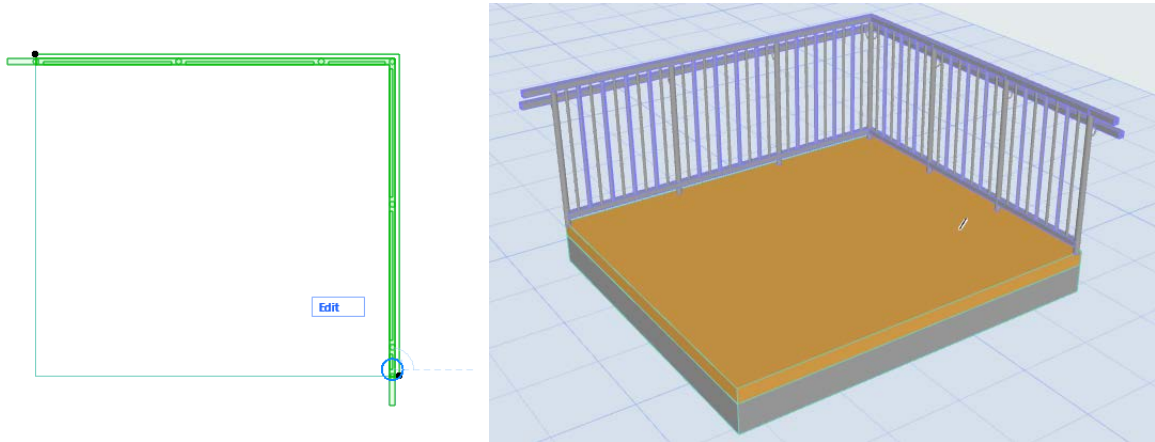
[Watch the Video](#)

[Watch the Video](#)

Working with Railing Segments and Nodes

Segments

A Segment is the part of the Railing between two Nodes (like a segment of a polyline). The Segment has its own settings, which include the Railing Pattern and Segment Height.



One Railing, Two Segments

You can easily see Segments and Nodes in Edit mode: Turn on the Segments and Nodes visibility.

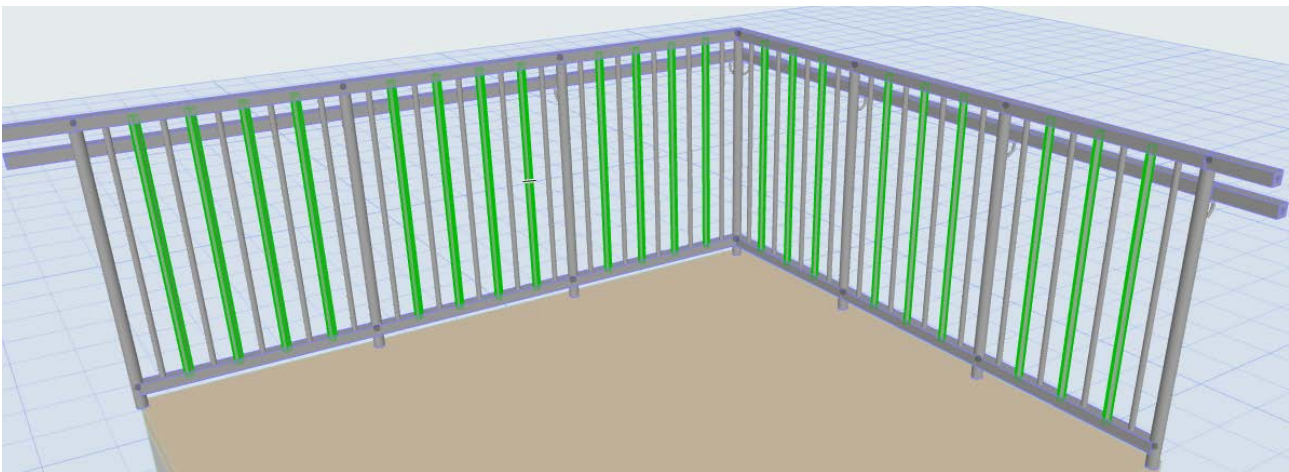


Segments and Nodes in Edit Mode

Within a Segment, all component instances are clones of each other. You can see this in Edit mode: selecting a Baluster or an Inner Post selects all of its instances within the Segment.



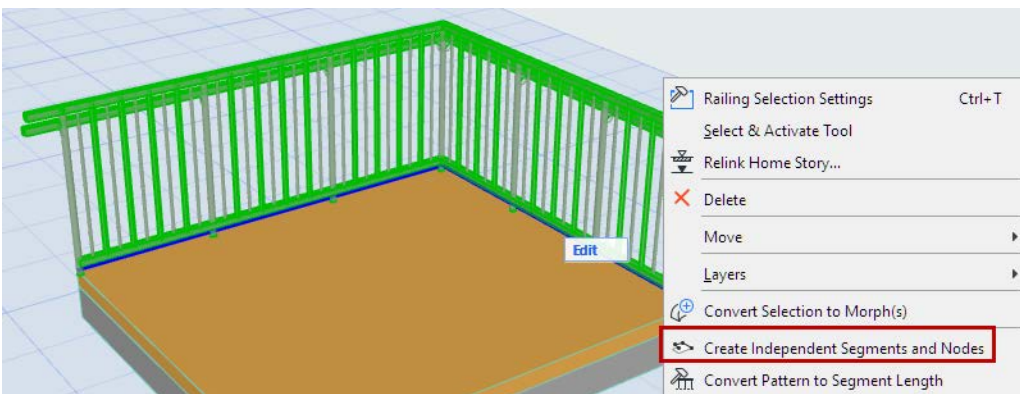
Edit Mode: All Inner Post Instances Are Selected

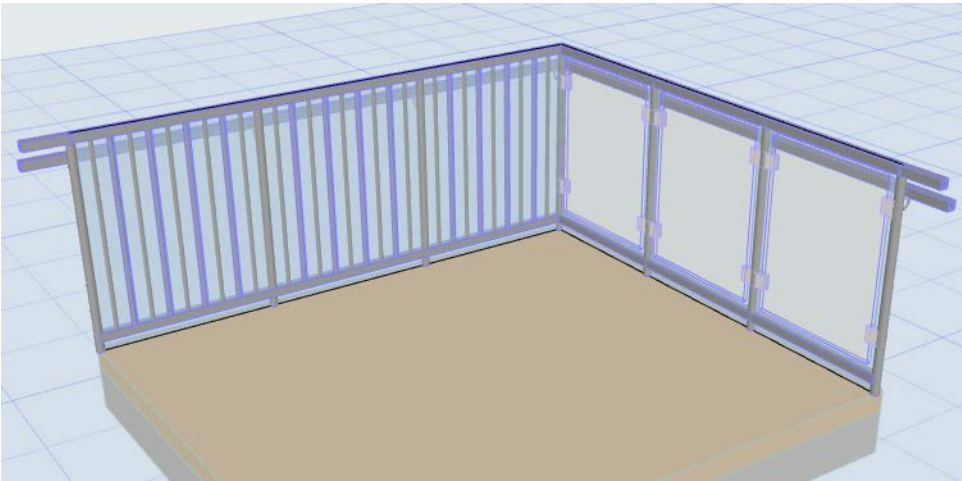


Edit Mode: All Baluster Instances Are Selected

Create Independent Segments and Nodes

You may wish to create independent Segments from the Segments of a selected, associative Railing. Select the Railing and use the pet palette command:





One Railing, Two Independent Segments

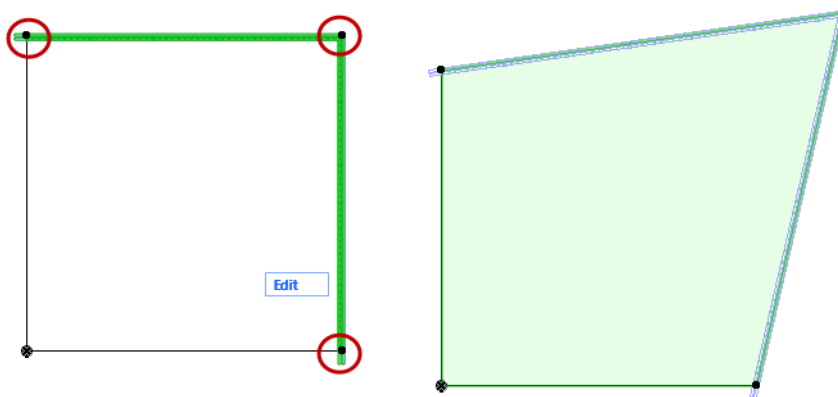
Nodes

Nodes are generated at the points of the Railing polyline. One or more Posts are located at each Node.

Associative Nodes

The visible Nodes of an **associative** Railing are round dots, generated at each point that you click to define the Railing.

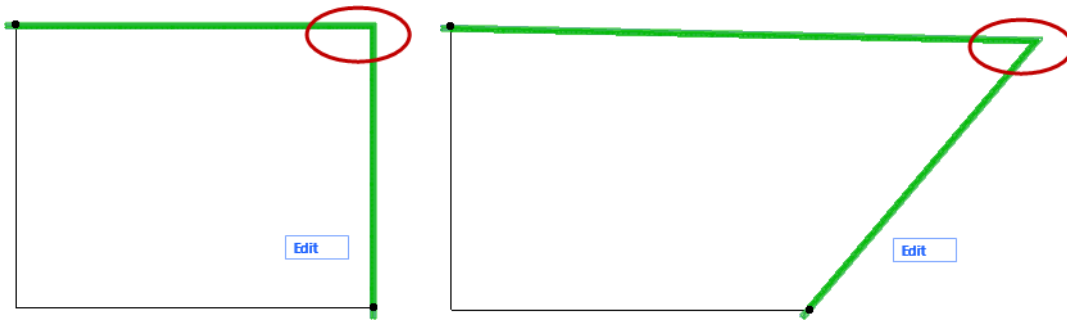
[See also *Place Associative Railing*.](#)



Visible Associative Nodes

A Node may also be invisible: When you place an associated Railing, its Nodes may be generated automatically, even if you didn't explicitly click: for example, when you click on two opposite corners of a polygon.

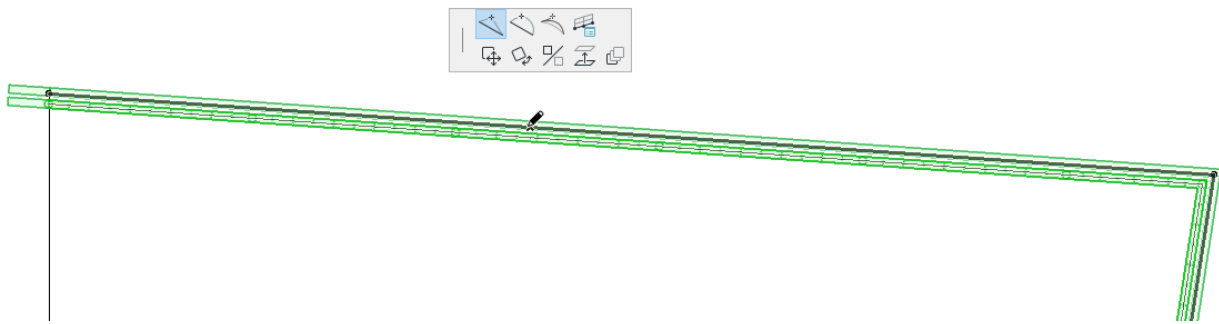
These are also associative Nodes, although they do not appear as dots.



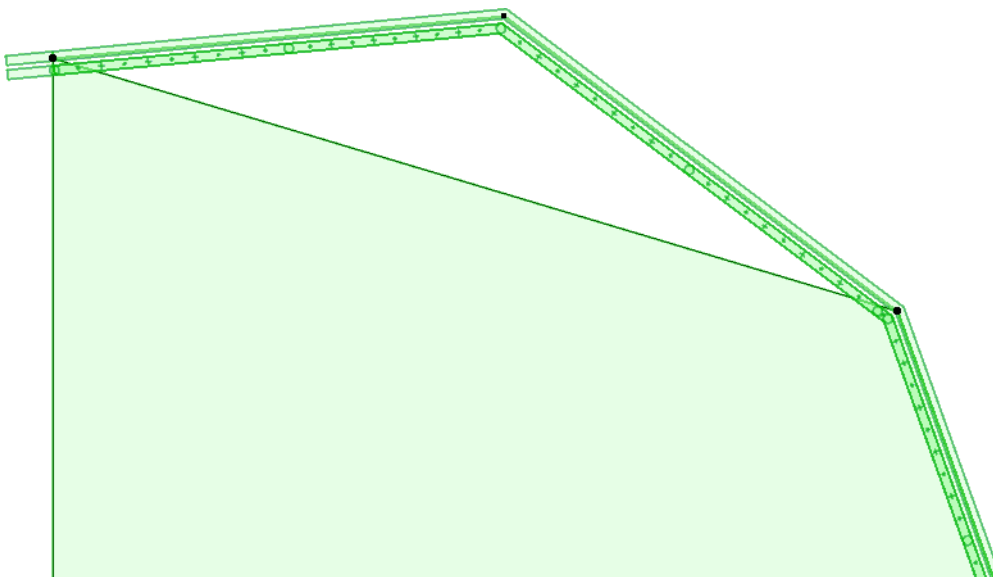
Automatically Generated, Associative Node (Invisible)

Static Node (Square Dot)

A Railing's static Node has a square shape.



Add New Static Node



Associative and Static Nodes of Railing along a Slab

Related Topics

[Node Settings \(Railing Tool\)](#)

[Segment Settings \(Railing Tool\)](#)

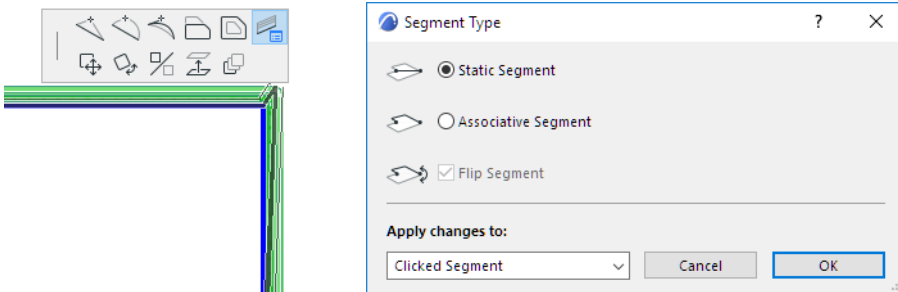
Change Associative Segments/Nodes to Static

Change Associative Segment to Static

1. Select the Railing.
2. Click on a Segment.
3. Use the **Railing Segment Type** command from the pet palette to open the Segment Type dialog box.
4. Choose **Static Segment**.

Now the Segment has been disassociated from the Slab.

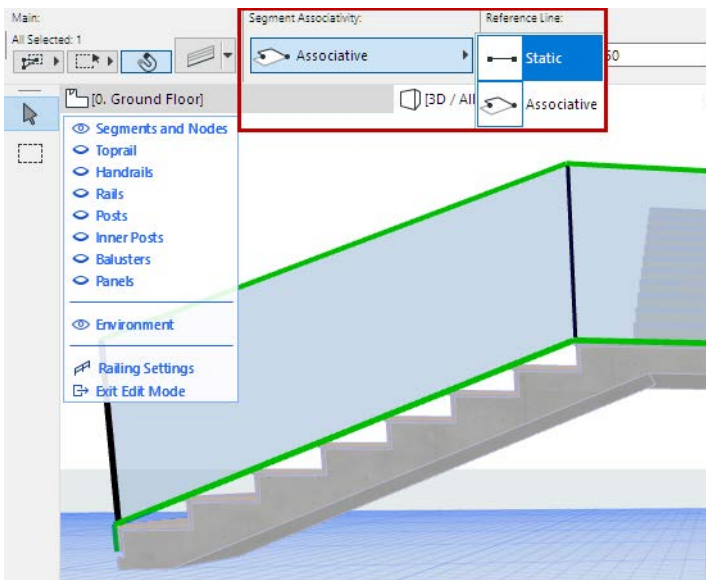
You can now edit the two geometries (e.g. Slab and Segments) independently.



Railing Segment Type: Static or Associative

Another method:

1. Select the Railing.
2. Enter Edit mode.
3. Select the Segment.
4. Use the Info Box or open the Segment Settings dialog to change its Associativity to Static.



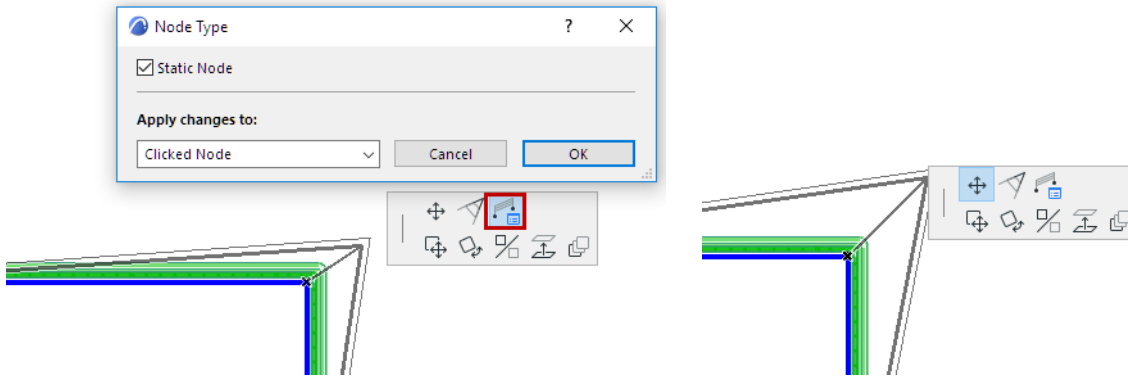
Change Associative Railing Node to Static

1. Select the Railing.
2. Select the associative Node.
3. Do one of the following:

- Drag an associative Node away from its parent element, to a new position.
- Use the **Railing Node Type** command from the pet palette to open the Node Type dialog box, then check **Static Node**.

4. Now the Railing Node has been disassociated from the Slab node. Note the Node shape: it is now rectangular.

You can now edit the two geometries (Slab and Railing Nodes) independently.



Railing Node Type: Change to Static

Another method:

1. Select the Railing.
2. Enter Edit mode.
3. Select the Node.
4. Use the Info Box or open the Node Settings dialog to change its Associativity to Static.

[Watch the Video](#)

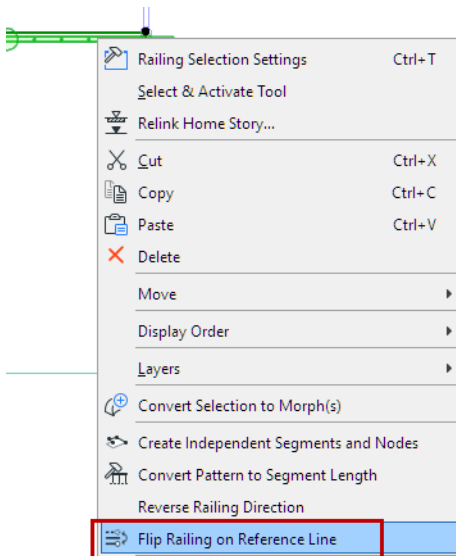
Railing Reference Line

View Railing Reference Line

To see the Reference Line on the Floor Plan: Turn on **View > On-Screen View Options > Railing Reference Lines**. (The arrows indicate the Railing input direction, which determines the Railing's "left" and "right" sides - this is relevant for defining the Handrail position.)

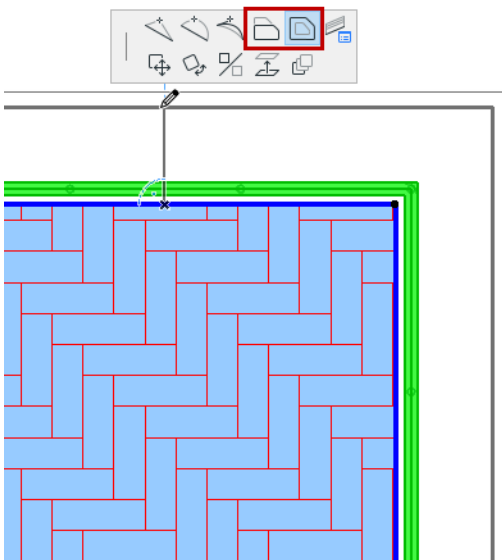
Flip Railing

To flip the Railing to the other side of the Reference Line: Use the **Flip Railing** command from the context menu:



Offset Railing Reference Line

A quick way to edit Railing geometry: Select the Reference Line, then use the offset commands to edit the Reference line position (either one segment, or all segments):



Related Topics:

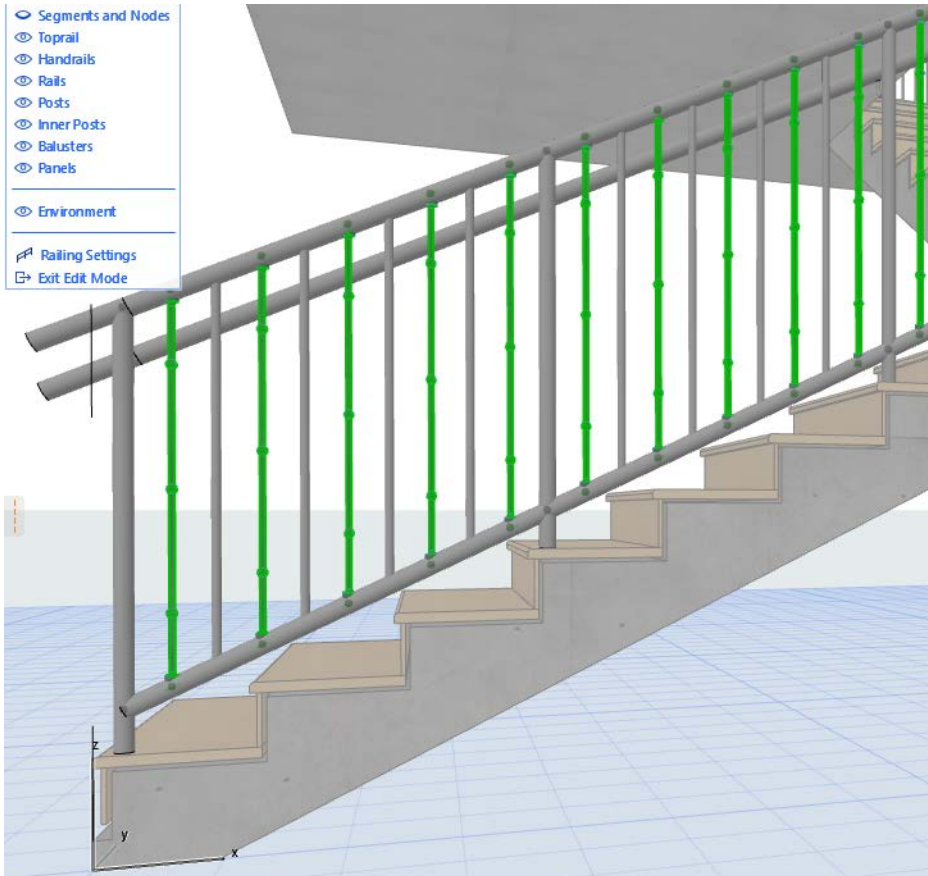
[Offset Railing Segment from Reference Line](#)

[Flip Segment \(Redefine Railing Polyline\)](#)

Railing Tool Edit Mode

Use Edit mode and its dedicated component tools to access and change the individual components of a selected Rail (Posts, Inner Posts, Balusters, Toprail, Handrails, Rails and Panels).

When you select a component (e.g. Inner Post), all its pattern instances are selected, too. When opening the component settings, only the relevant geometry is highlighted and you can edit all instances at once.

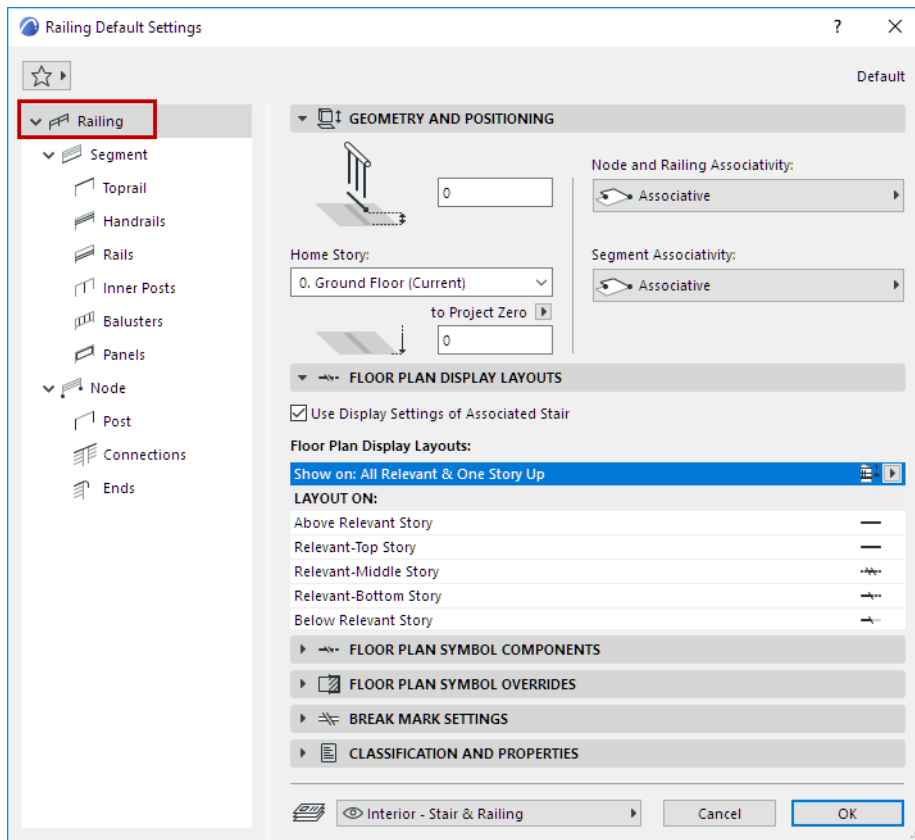


[Watch the Video](#)

[Watch the Video](#)

Railing Settings: Overview

Click the Railing Tool icon to open its main Settings page, whose left side displays a detailed tree.



Each control of this main page is described at [Railing Settings: Geometry and Symbol Display](#).

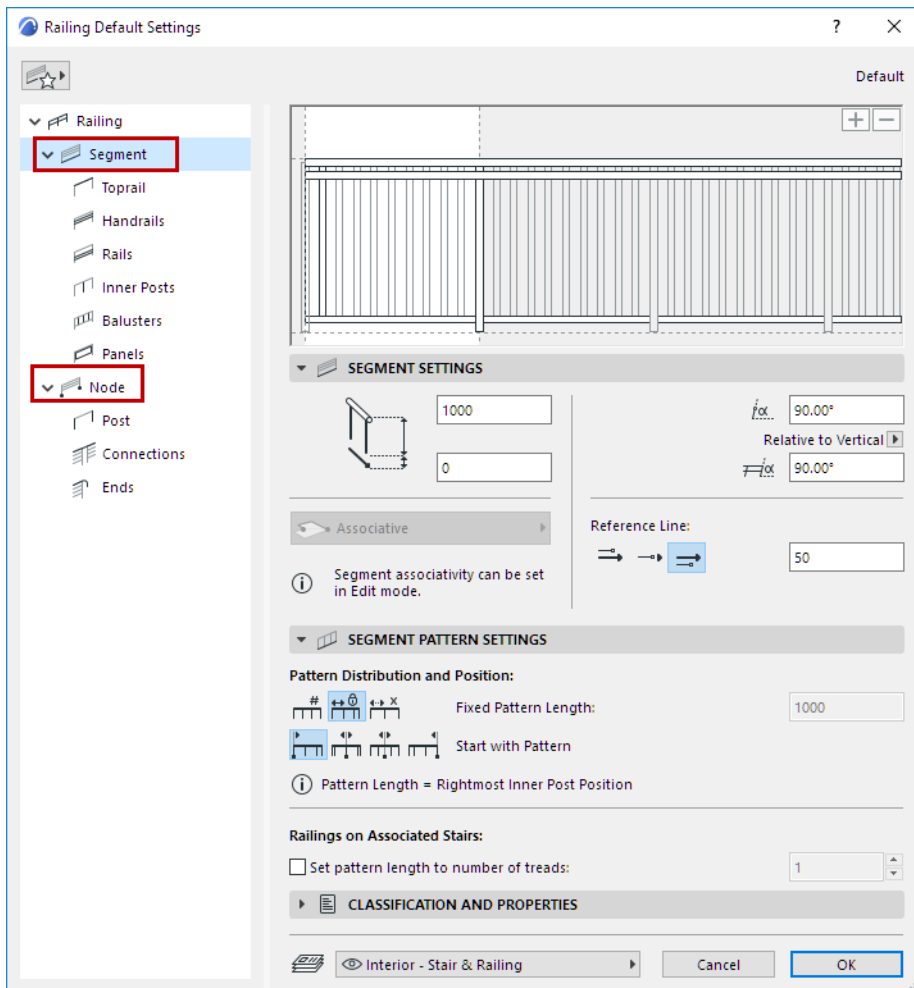
The two main components, listed in the tree, are **Segments** and **Nodes**. Set the parameters specific to Segments and Nodes on their respective pages.

- A **Segment** is the part of the Railing between two Nodes (like a segment of a polyline). A Segment has its own settings, which include the Railing Pattern and Segment Height.

See [Segment Settings \(Railing Tool\)](#).

- **Nodes** are the points that divide the Railing Segments (like points on a polyline). One or more Posts are located at each Node.

See [Node Settings \(Railing Tool\)](#).



The sub-pages under Segment and Node allow you to further specify the Railing components (e.g. Rails, Posts, Panels).

[Watch the Video](#)

Related Topics

[Railing Settings: Geometry and Symbol Display](#)

[Segment Settings \(Railing Tool\)](#)

[Toprail Settings \(Railing Tool\)](#)

[Handrail Settings \(Railing Tool\)](#)

[Rail Settings \(Railing Tool\)](#)

[Inner Post Settings \(Railing Tool\)](#)

[Baluster Settings \(Railing Tool\)](#)

[Panel Settings \(Railing Tool\)](#)

[Node Settings \(Railing Tool\)](#)

[Post Settings \(Railing Tool\)](#)

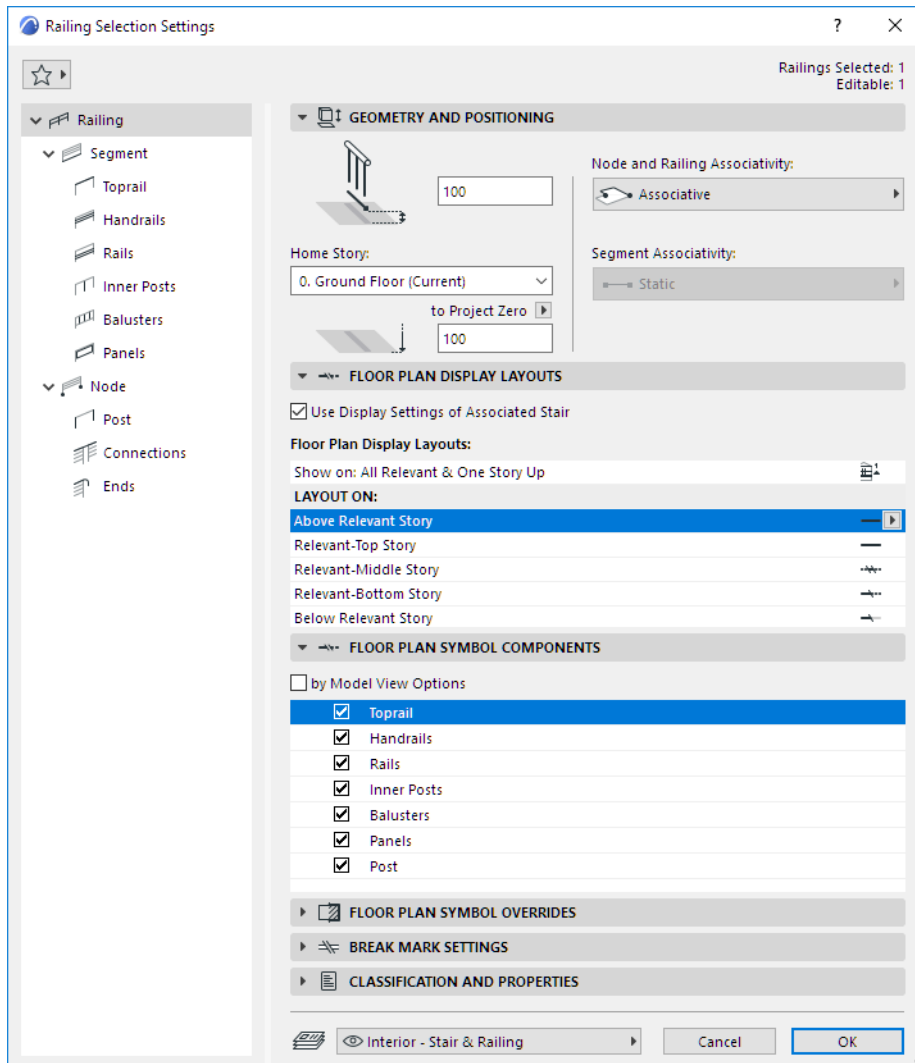
[Connection Settings \(Toprail, Handrail, Rail\)](#)

[Railing Ends \(Toprail, Handrail, Rail\)](#)

Railing Settings: Geometry and Symbol Display

The top level page in the Railing Tool Settings hierarchy controls the **Railing** as a whole.

On this Settings page, you set its Home Story information and any offset, as well as its Floor Plan and Break Mark display.



Related Topics:

[Railing Geometry and Positioning](#)

Choose the stories on which the Railing's 2D symbol should be displayed.

[Railing Display Layouts \(Show on Stories\)](#)

Vary a Railing's symbol display, story by story.

[Railing Floor Plan Symbol Components](#)

Show or hide each symbol component of the Railing (optionally overriding the show/hide definitions of Model View options.)

[Railing Settings: Floor Plan Symbol Overrides](#)

Apply overrides to the display of the Railing's entire 2D symbol (lines, pens, fills).

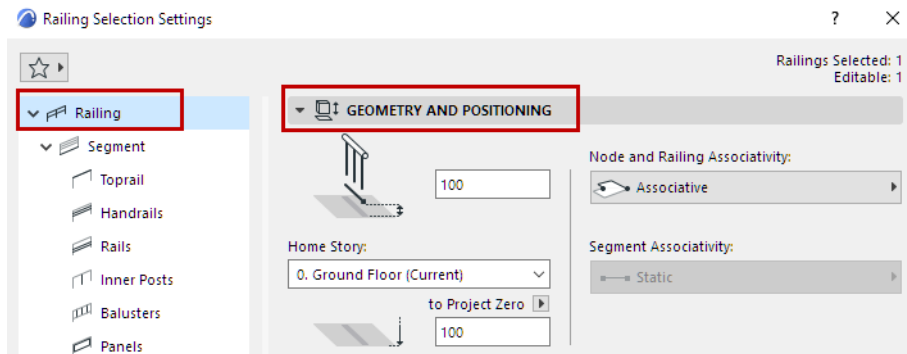
Break Mark Settings

Define Break Mark appearance.

Set Attributes for Each Railing Component

How to set the attributes and choose a symbol type for any Railing component.

Railing Geometry and Positioning



Home Story

Choose one of the following Home Story settings:

- The current story: The Railing's Home Story will be the current story, on which it is placed.
- Choose a story to which to link the bottom of the Railing. Click **Select Story** to bring up the full list of stories in the project, if they are not all shown in the list.

Offset Railing from Home Story

Optionally, define an offset for the Railing bottom from its Home Story. (For associative Railings, the offset field is grey, because the Railing follows the Home Story of its parent element.)

Bottom Elevation [to Reference level]

Note: For associative Railings, this field is grey, because the Railing follows its parent element.

Calculates the current elevation of the Railing's reference line, as measured from the Reference level (by default, this Reference level is Project Zero).

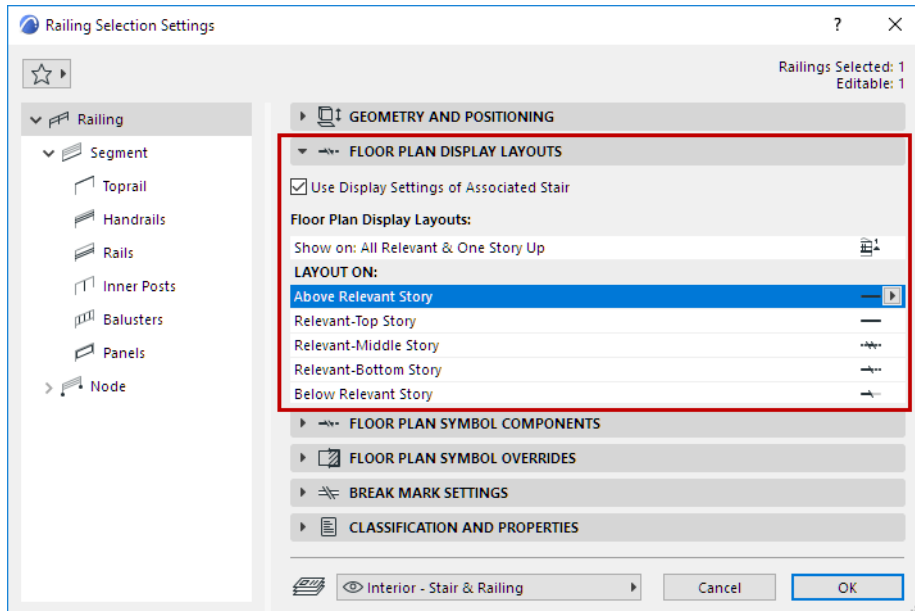
As with Morphs, the Railing's Bottom Elevation value shows the current elevation of the Railing's lowest point. The Railing's height is measured from this value.

Click the pop-up arrow to change the Reference level, if needed.

Note: Reference levels are defined at **Options > Project Preferences > Reference Levels**.

Railing Display Layouts (Show on Stories)

Use this panel to define how to display the Visible/Hidden parts of the Railing symbol on each story.



Use Break Mark Settings of Associated Stairs

This box is checked by default. If the Railing is associated to a Stair, then it will be displayed on the same stories as the Stair, using the same Layout options, and the same Break Mark settings.

These options are defined for the Stair at [Stair Display Layouts \(Show on Stories\)](#).

Show on Stories

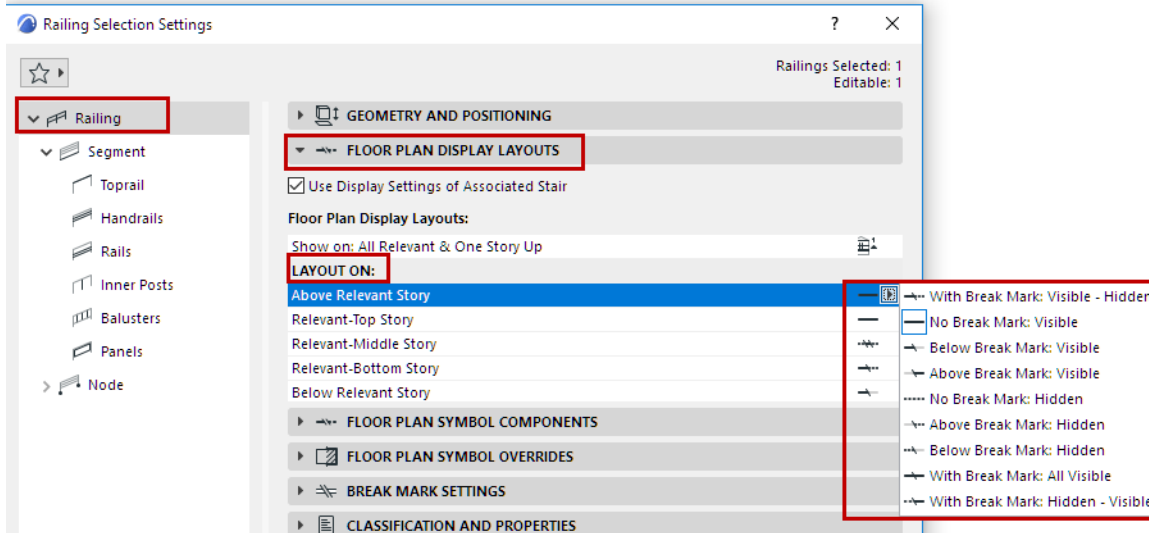
Use the pop-up to define the Stories on which the Railing's 2D symbol should be displayed.

The stories you define (for example, "All Relevant & One Story Up") are listed in the field below (under "Layout On").

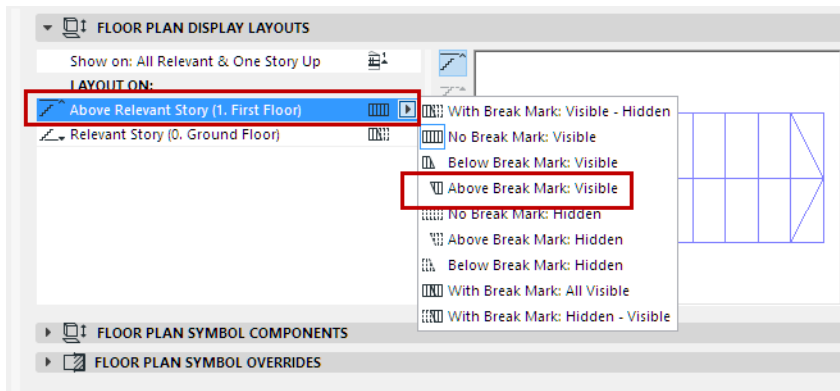
A Relevant Story is one which the Railing physically intersects.

Layout On (Railing Display On Each Story)

Under **Layout On**: Click on each Story name, then choose how to display the Railing on that story:



- **Break Mark:** With or without a Break Mark
- **Visible vs. Hidden Attributes:** To use either Visible or Hidden attributes on the displayed parts of the Railing. The set of attributes (for hidden and visible parts) is defined for each symbol component on the Floor Plan Symbol Components panel (see below).



Visible Parts and Hidden Parts

A Railing’s “Visible Parts” represent the Railing’s primary appearance, and are displayed using the “Visible” line type (usually a solid line).

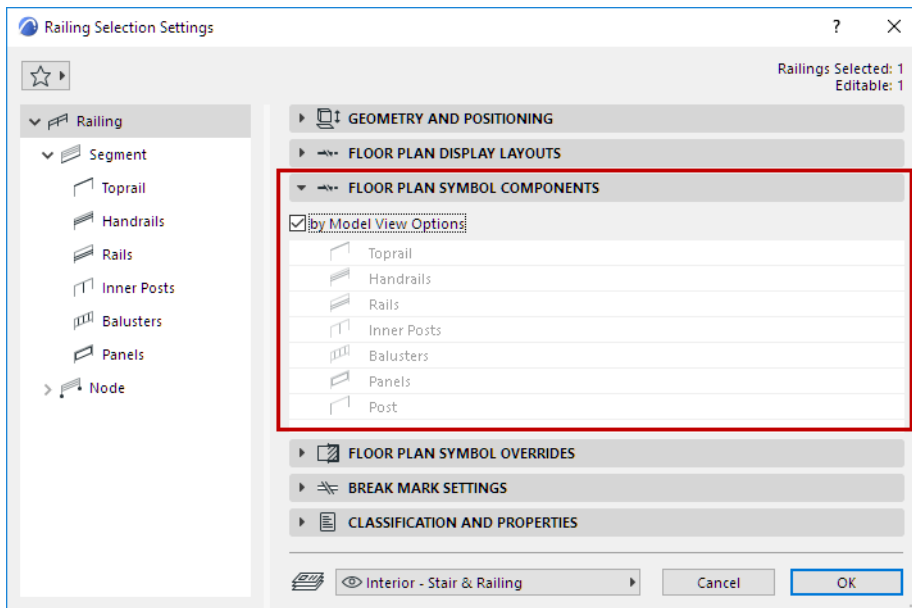
A Railing’s “Hidden Parts” represent the Railing’s secondary appearance, and are displayed using the “Hidden” line type (usually a dashed line).

Visible parts are usually under the Break Mark; Hidden parts are usually above the Break mark.

Railing Floor Plan Symbol Components

Use this panel to control the display (on or off) of the Railing's 2D symbol components.

Show/Hide Symbol Components: By Model View Options

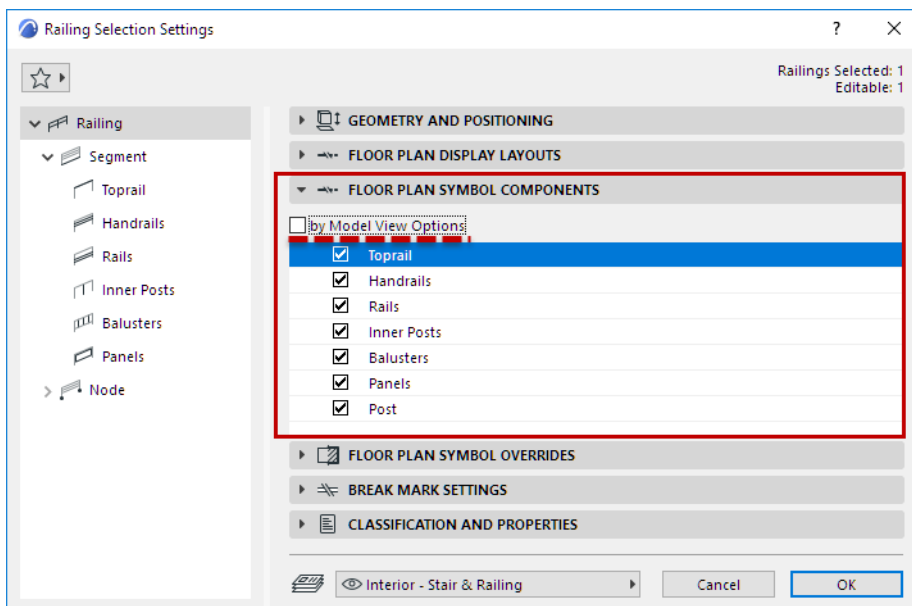


Check this to show/hide the Railing's 2D components, as defined at **Document > Model View > Model View Options (Railing Options)**.

[See Model View Options for Railings.](#)

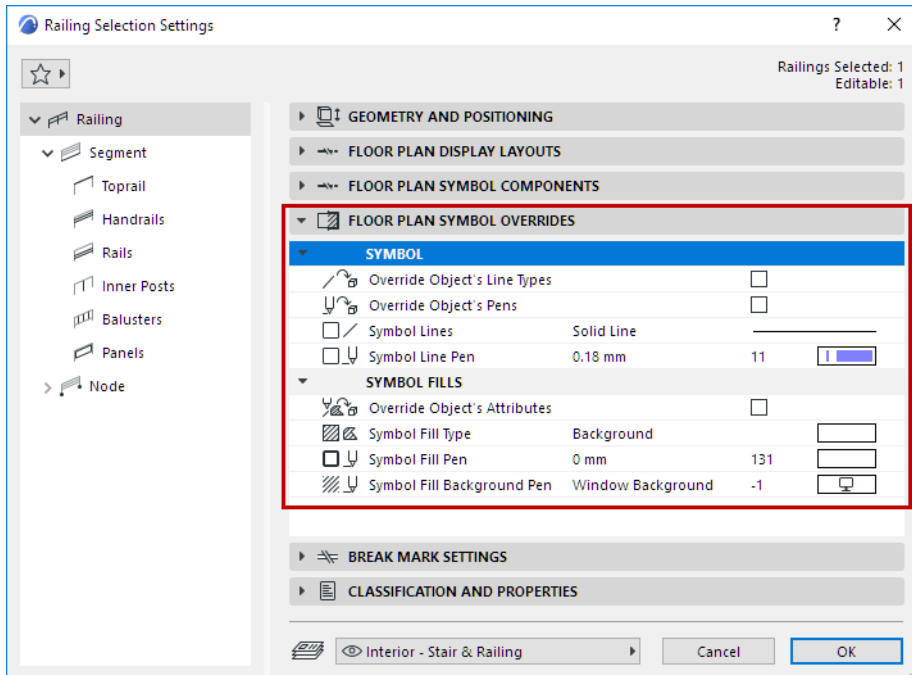
Show/Hide Symbol Components: Customize in Railing Settings

Uncheck "by Model View Options" and use the list below to specify which symbols to show or hide for this Railing.



Railing Settings: Floor Plan Symbol Overrides

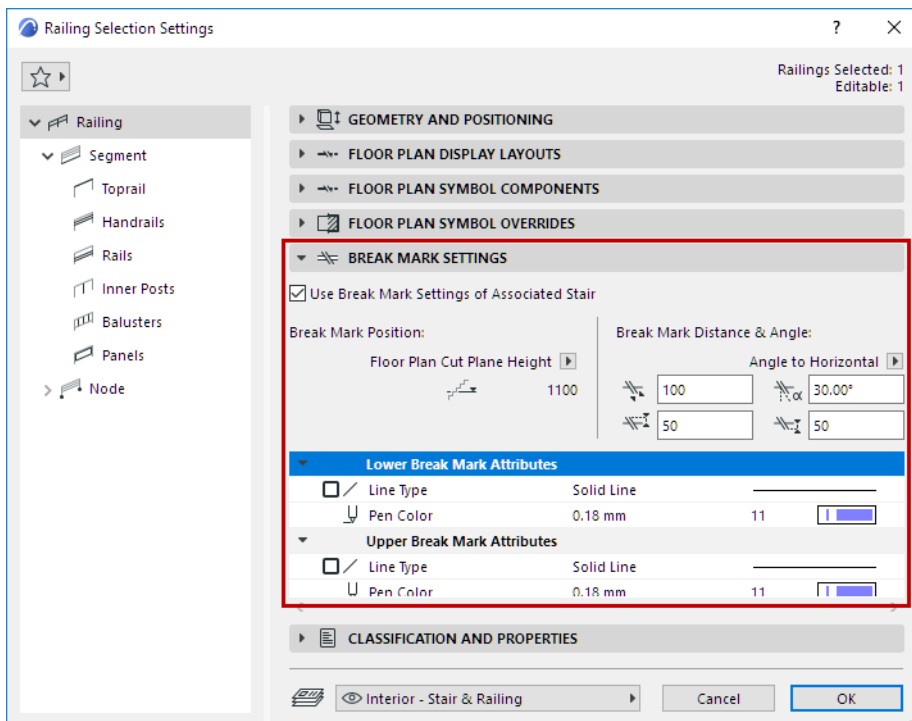
Use this panel to optionally apply overrides to the display of the Railing's entire 2D symbol (lines, pens, fills).



- To apply a Line/Pen/Fill override: Check the respective **Override** box, then select an attribute from the controls below.

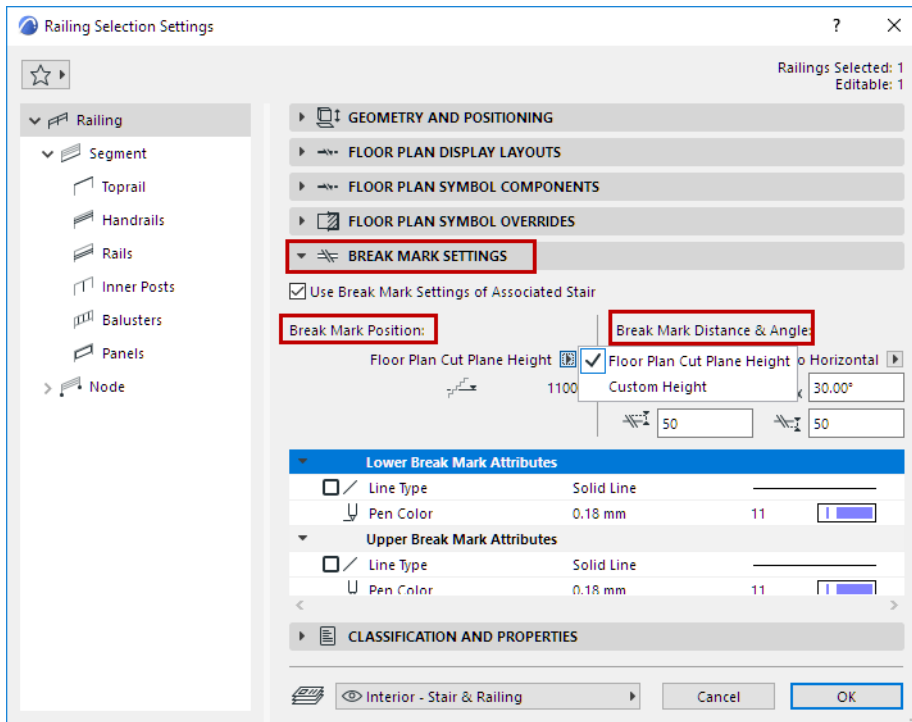
Break Mark Settings

Use these controls to define Break Mark display for the Railing.



Note: For a Railing associated to a Stair: Check **Use Display and Break Mark Settings of Associated Stairs** to apply the applicable settings to this Railing.

- **Position:** Place the Break Mark at the Floor Plan Cut Plane, or else define a Custom Height using the two fields.
- **Distance** between the two Break Mark segments; and the segment Angle (defined to Horizontal or to Vertical)
- Extensions to the Break Mark segments
- **Attributes** (Line Type, Pen Color) of the Break Mark’s upper and lower line.

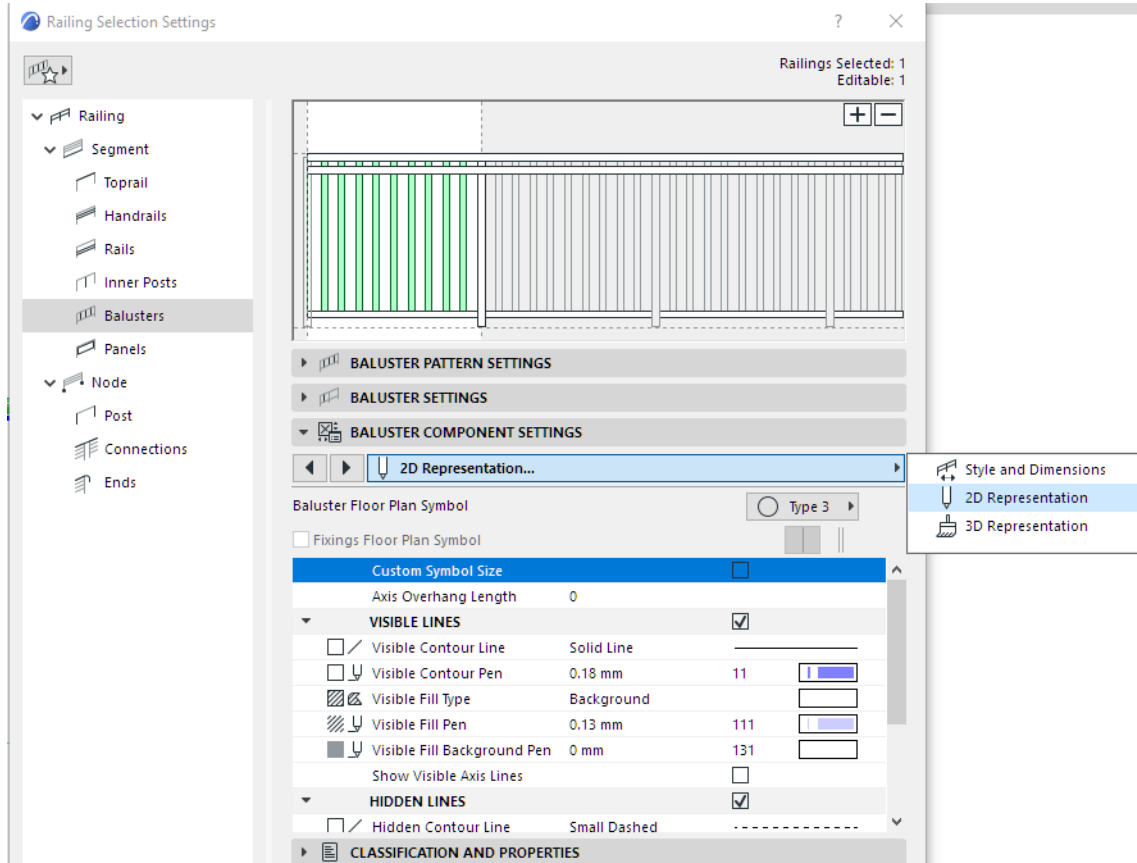


Set Attributes for Each Railing Component

To set attributes, choose a Symbol component (e.g. Balusters) from the Rail Settings tree structure.

For each Component:

Use the 2D Representation and 3D Representation pages of the Component Settings Panel to set its attributes - separately for the Visible and Hidden parts of the symbol, as applicable.



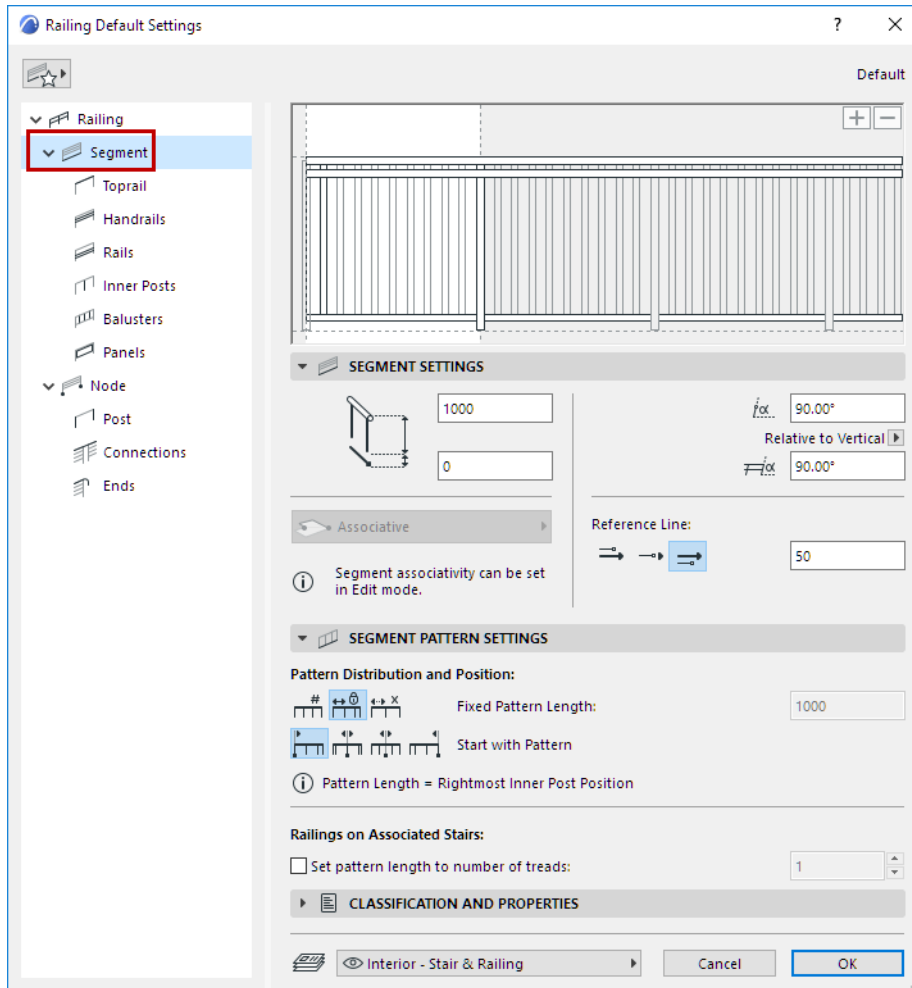
See also [Visible Parts and Hidden Parts](#).

Segment Settings (Railing Tool)

A **Segment** is the part of the Railing between two Nodes (like a segment of a polyline).

A Segment has its own settings, which include the Railing Pattern and Segment Height.

Use this page to configure the overall Segment parameters (including Segment height) and to set up a Pattern, consisting of various Rails, Posts, and Panels.



Related Topics:

[The Pattern Editor \(Railing Tool Settings\)](#)

[Segment Settings \(Page of Railing Tool Settings\)](#)

[Segment Pattern Settings \(Panel of Railing Tool\)](#)

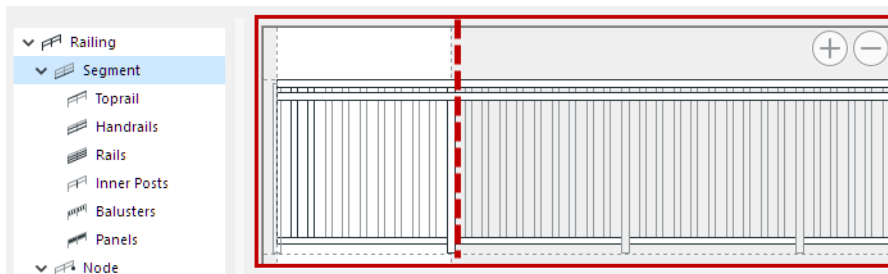
The Pattern Editor (Railing Tool Settings)

The Pattern Editor is the graphical interface at the top of the Segment settings page and all of the Railing component settings pages.

Use the Pattern Editor to assemble a unique Railing pattern specific to each Segment. The pattern can be repeated within each Segment, according to a distribution logic defined by you.

The Pattern Editor is a symbolic representation of this pattern. It may consist of Toprail, Handrails, Rails, Inner Posts, Balusters and Panels.

The white part of Pattern Editor interface represents the pattern itself, which is placed (repeatedly, as needed/ depending on distribution settings) over the whole length of the Segment. This white section - that is, the Pattern Length - extends to the last (rightmost) Inner Post within the pattern (marked with a dotted line in the image below):



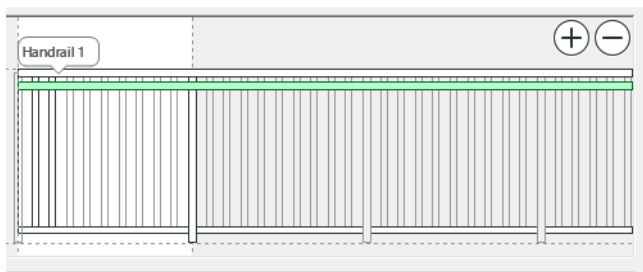
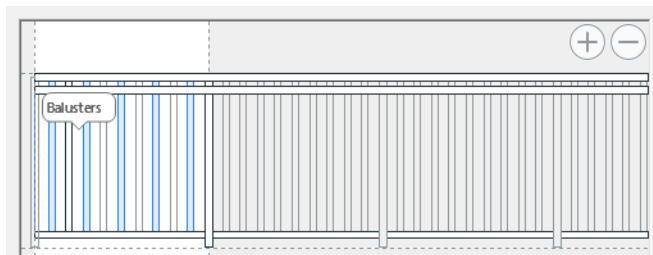
Pattern Length

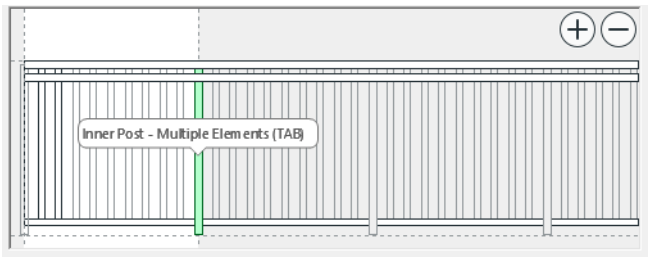
See also [Pattern Length](#), below.

[Watch the Video](#)

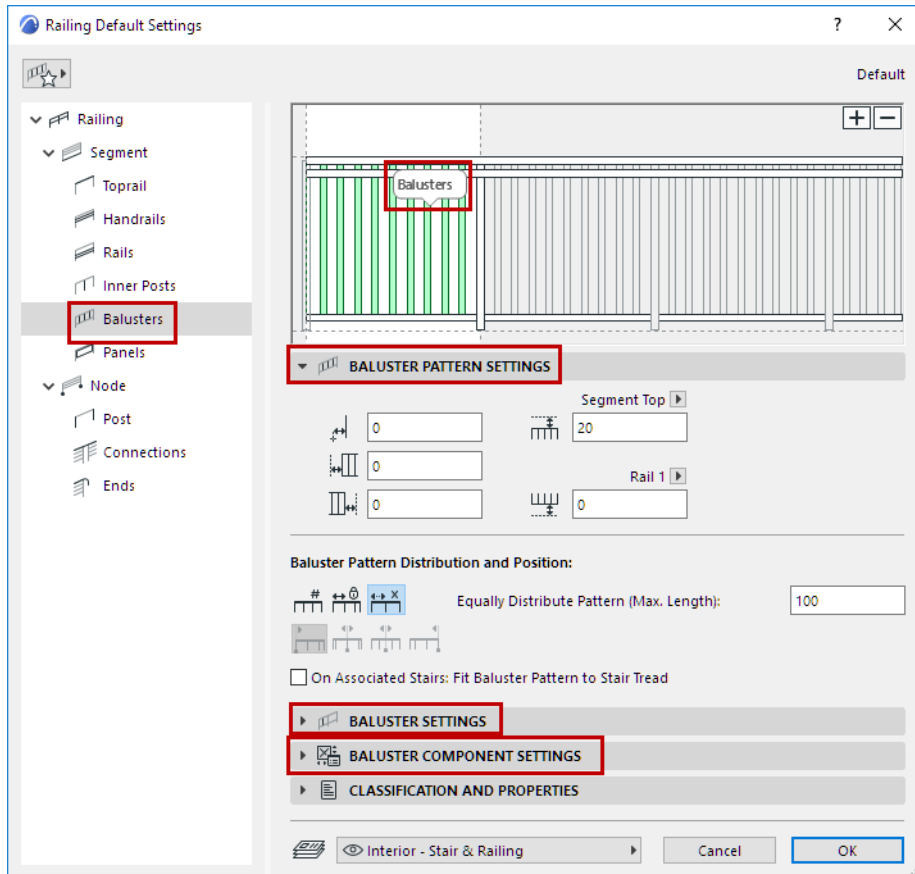
Selection in Pattern Editor

To select a component from the pattern, hover over it to see the feedback:





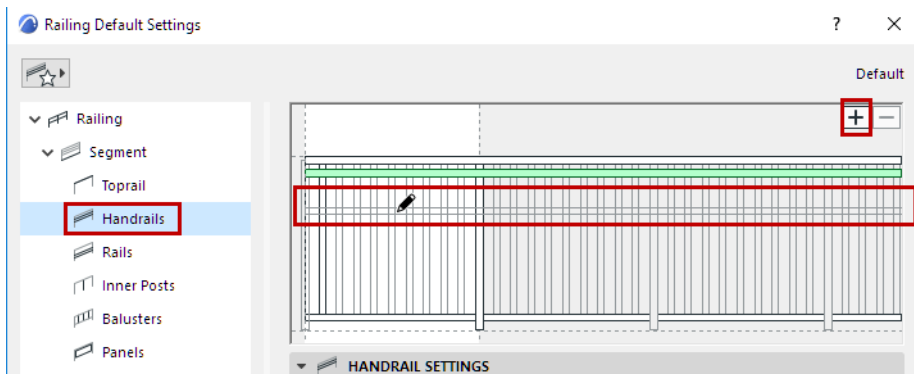
Click on a component to select it. Notice that the component type is also selected in the tree structure, and its settings are available in the dialog box.



Add Component in Pattern Editor

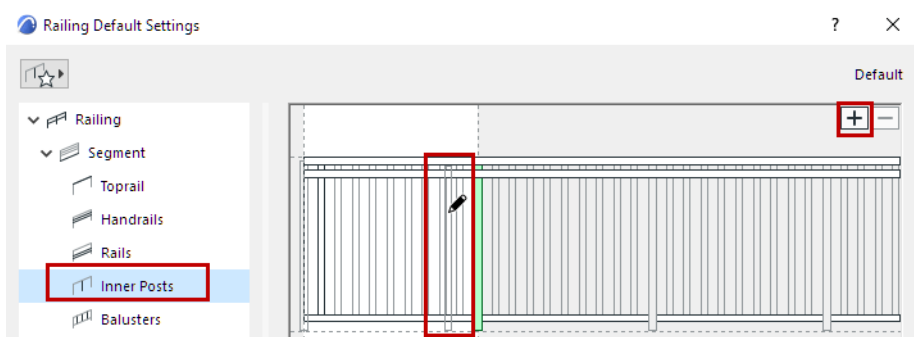
To add a Railing component to the Pattern Editor:

1. Select the component (e.g. Handrails) from the tree structure
 - Note:** You must select a specific Railing component, not the Segment page.
2. Click the **Plus** sign.
3. Drag the appearing symbol into place on the Pattern Editor.
4. Click to place.



Example: Add Inner Post in Pattern Editor

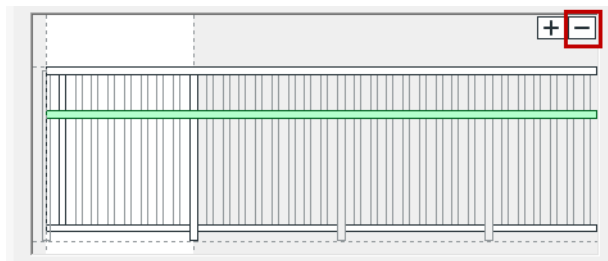
1. Select Inner Posts from the tree structure.
2. Click the Plus sign, then drag the appearing post symbol into place on the Pattern Editor.



Delete Component from Pattern Editor

To delete a component from Pattern Editor:

Select it in the pattern, then do one of the following:

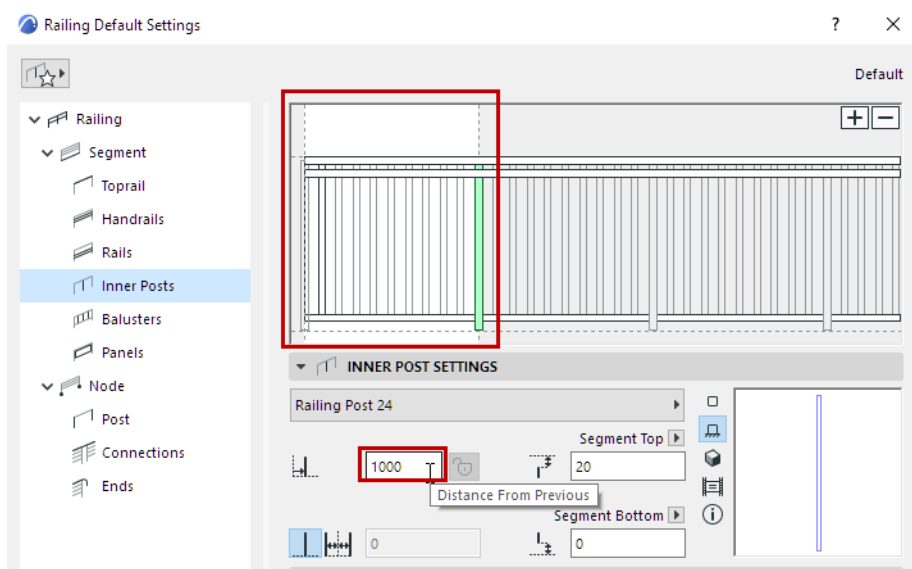


- Click the **Minus** sign
- Click **Delete**
- Click **Backspace**

Pattern Length

The pattern extends to the rightmost Inner Post within the white area of the Pattern Editor.

To change Pattern Length, select the Inner Post and adjust the **Distance from Previous** value in the Settings below.



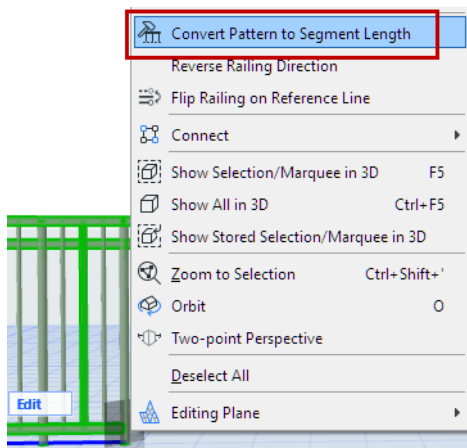
Pattern Length

For more information on Inner Posts, see [Inner Post Settings \(Railing Tool\)](#).

Convert Pattern to Segment Length

For a placed Railing:

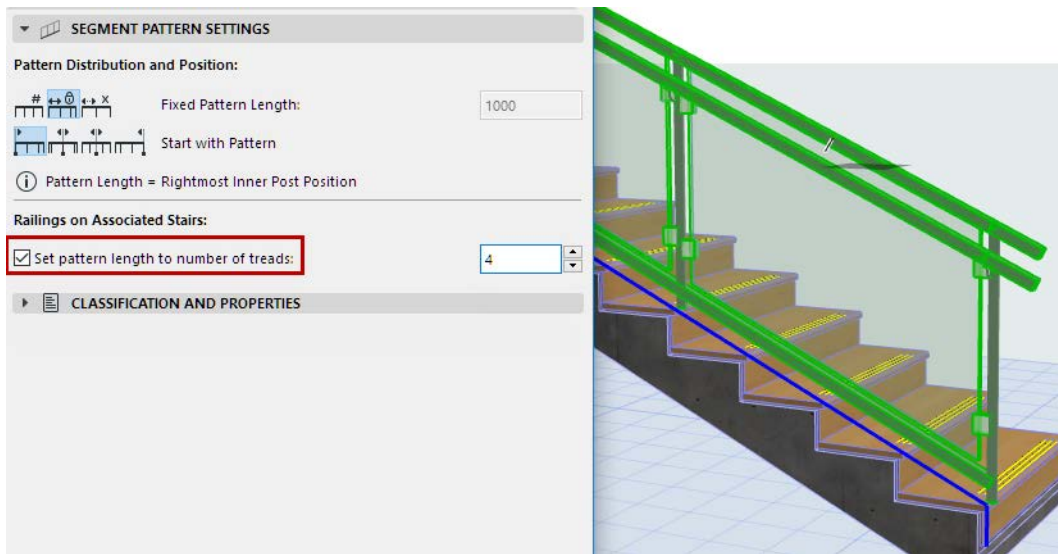
Select the Railing, and use **Convert Pattern to Segment Length** from the context menu.



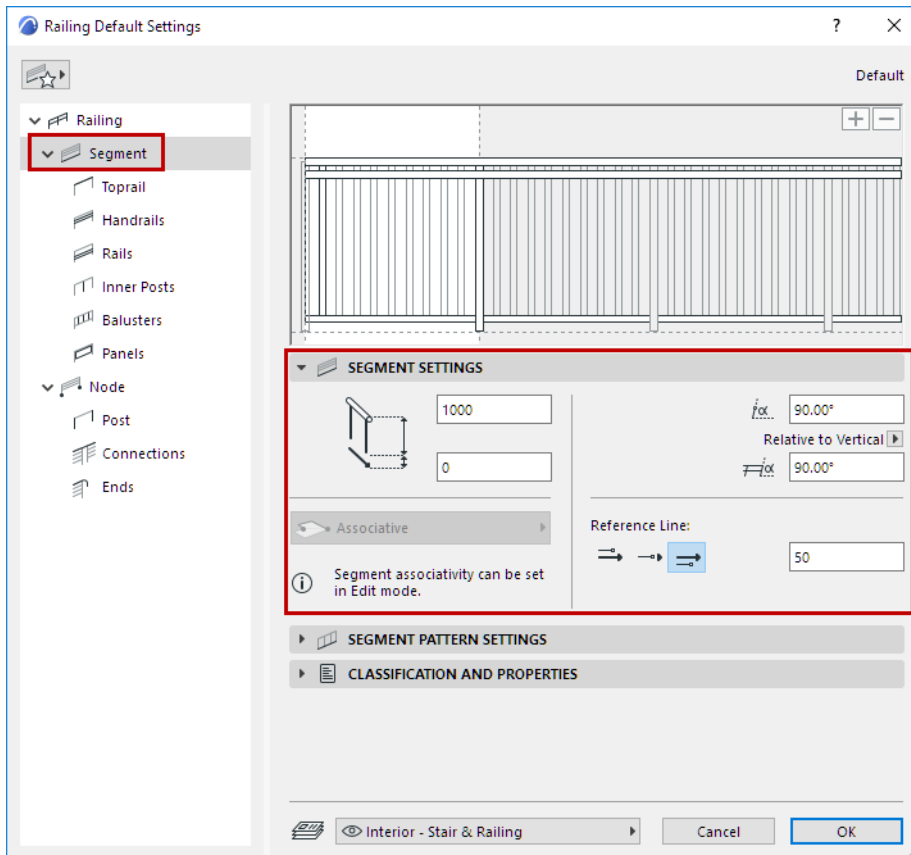
You can also fit the Pattern Length to Stair Tread width:

Railings on Associated Stairs

- Set pattern length to a specific number of treads:



Segment Settings (Page of Railing Tool Settings)



Segment Height and Vertical Offset

Enter the Segment height.

The position of other Railing components can depend on the Segment height:

- The height of the Toprail equals the Segment height (this is not true for Handrails or Rails).
- The tops of Railing Posts, Inner Posts, Balusters and Panels can be defined as offset from the Segment top.

Optionally, add a vertical offset (positive or negative) to the Segment height.

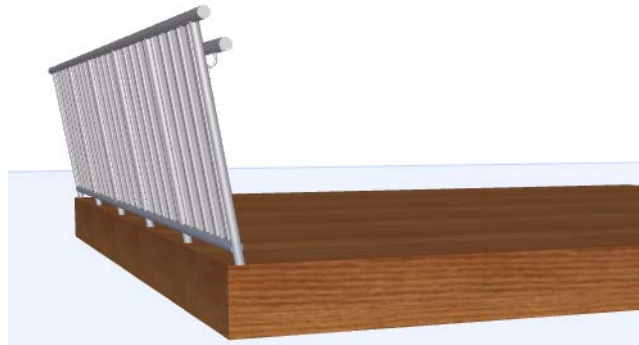
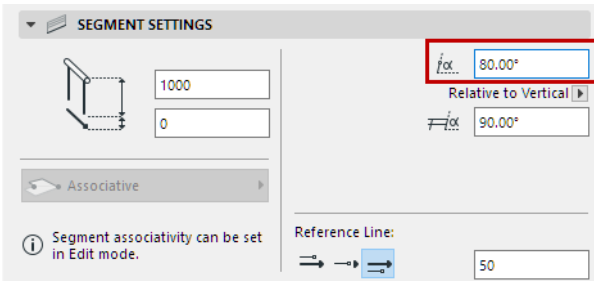
Segment Associativity

A Railing Segment is associative by default, but you can change any Segment to Static.

See [Working with Railing Segments and Nodes](#).

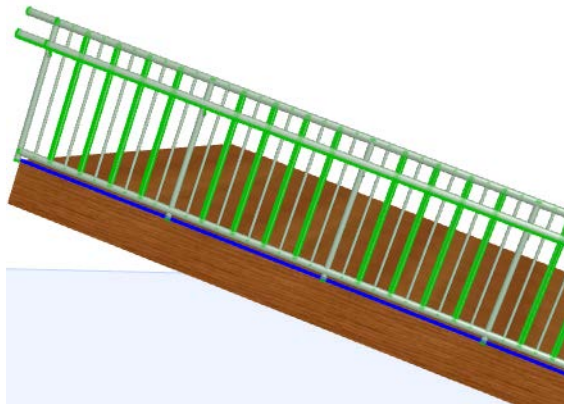
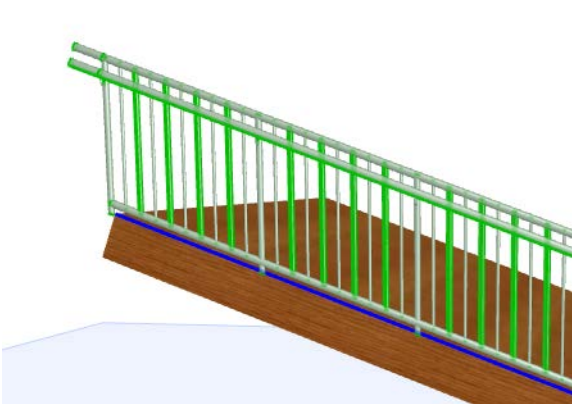
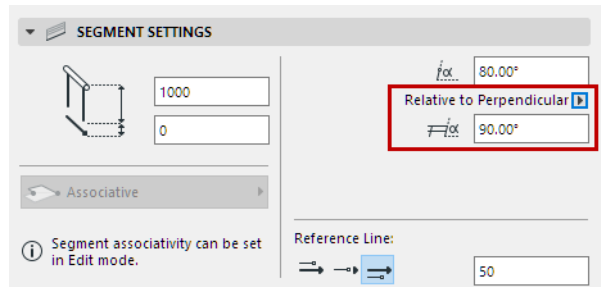
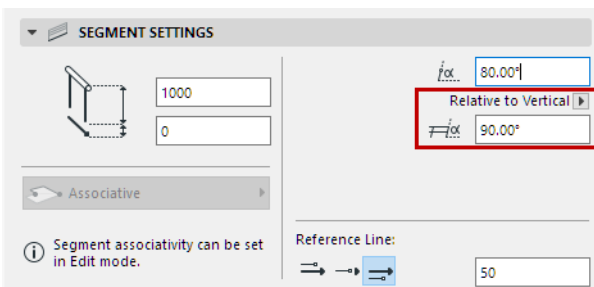
Slant and Skew Railing Segment

Slant the Segment by entering a slant degree, measured perpendicularly to the Segment plane.



Slanted Railing Segment

Optionally, skew the Segment. Set the skewing angle: either relative to vertical, or perpendicular to the Railing reference line.



Skew Railing (to Vertical or Perpendicular Plane)

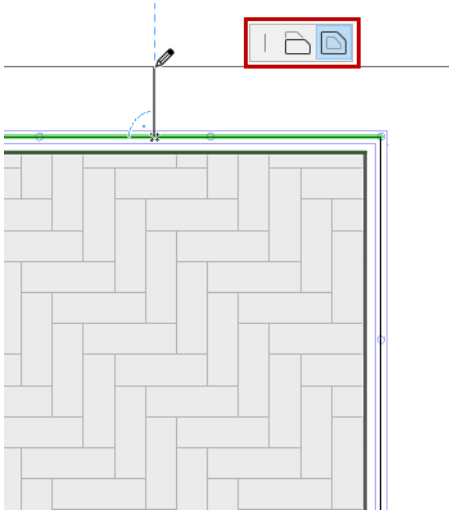
You can also skew or slant the Segment graphically, by modifying the Toprail angle in Edit mode.

[See Slant or Skew Railing in Edit Mode.](#)

Offset Railing Segment from Reference Line

As needed, offset the Segment from either side of the Railing Reference Line. Enter the offset value in the Reference Line field of Segment Settings.

In Edit mode, you can offset any or all Railing segments graphically (**Reference Line remains in place**, e.g. with associated element)



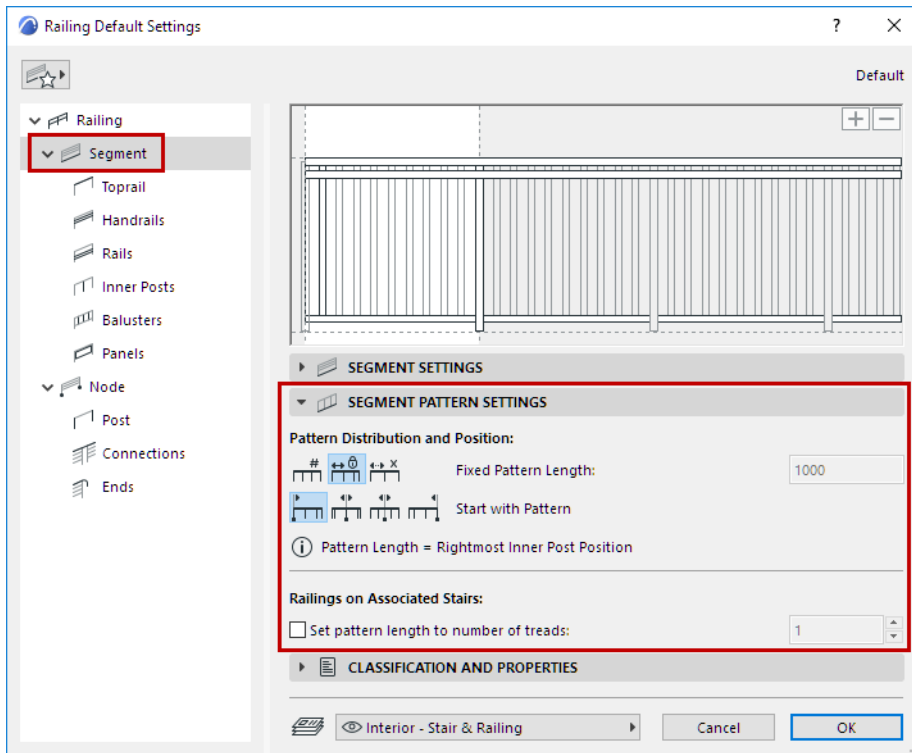
For other Reference Line uses, see [Railing Reference Line](#).

Segment Pattern Settings (Panel of Railing Tool)

Use these controls to define how to distribute the Pattern within the Railing Segment.

For information on setting up the pattern, see [The Pattern Editor \(Railing Tool Settings\)](#), above.

Note: Pattern distribution settings affect the pattern within a Segment. To continue the pattern over multiple Segments (without stopping at Nodes), see: [Pattern Continuity](#) at Node Settings.

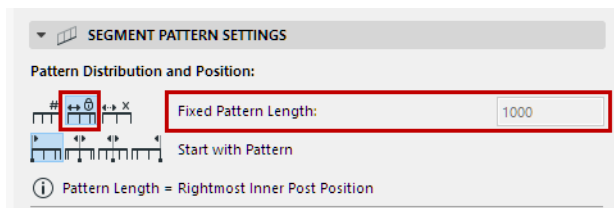


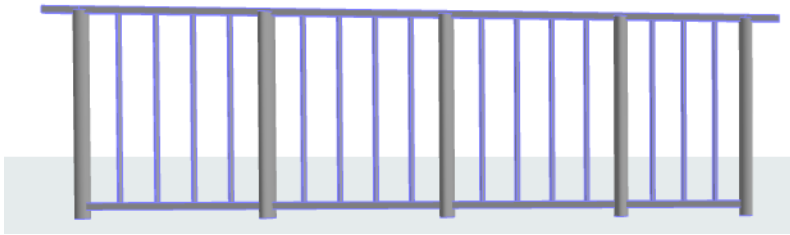
Pattern Distribution and Position

Choose one of the three Pattern Distribution options:

- **Fixed Pattern Length:** The pattern will be distributed, with its length fixed and unchanged, over the length of the Segment. The pattern will be cut off at the end of the Segment.

With this option, the pattern is not stretched.





Fixed Pattern Length (Cut off Pattern As Needed)

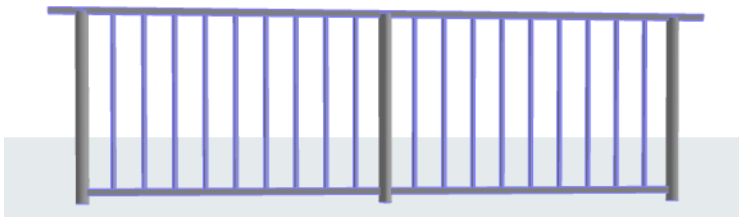
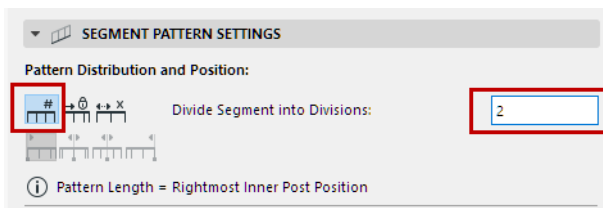
Pattern length is not editable here. Pattern Length extends to the rightmost Inner Post within the white area of the Pattern Editor.

See also [Pattern Length](#).

- **Divide Segment Into Divisions:** Enter the number of Segment divisions (e.g. 2). The pattern will be stretched so as to fit exactly into each division.

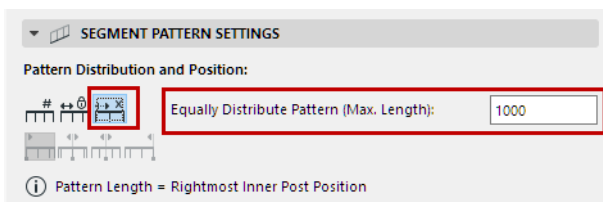
With this option, the pattern is stretched, and the distance between Balusters may change.

Inner Posts may change position. To fix an Inner Post (so that it doesn't move as a result of the Pattern stretch), lock its **Distance from Previous** value at Inner Post Settings. (See [Inner Post Settings \(Railing Tool\)](#).)



Segment With Two Divisions (Stretch Pattern to Fit)

- **Equally Distribute Pattern:** The pattern is distributed equally over the length of the Segment. The Pattern Length is adjusted (stretched) as needed so that the pattern is not cut off at the end of the Segment. Enter a maximum Pattern Length.





Distribute Pattern Equally Over Full Segment (Stretch Pattern to Fit)

Note: With this option, even if the Pattern is stretched, the gap between Balusters is unaffected. The Baluster gap is set in Baluster Settings ([see *Baluster Settings Panel*](#)).

Related Topic

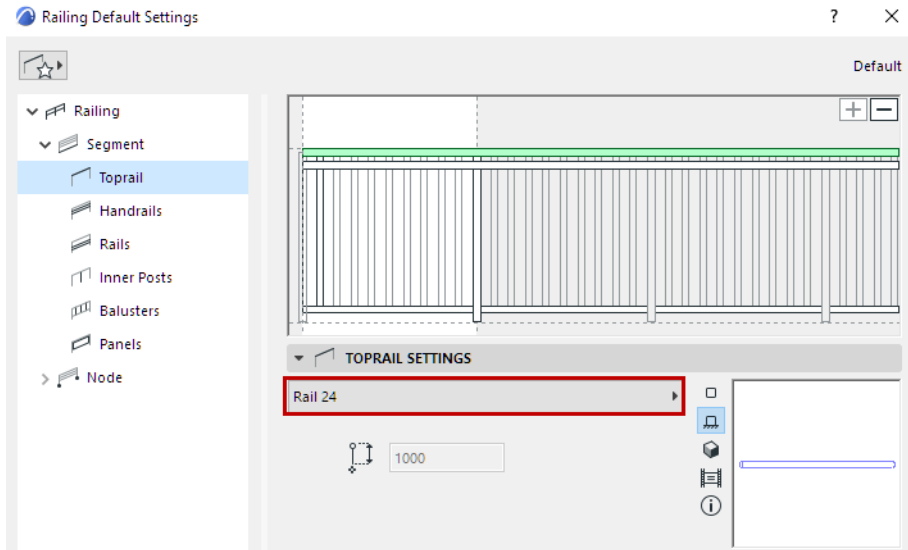
[Pattern Continuity](#)

Toprail Settings (Railing Tool)

Each Segment has a single Toprail which sits atop the Segment: Toprail height is therefore equal to the Segment height (defined in Segment Settings. [See Segment Height and Vertical Offset.](#))

You cannot edit Segment height in Toprail Settings.

You add a total of one Toprail to a Segment. (Click the Plus sign to place it.)

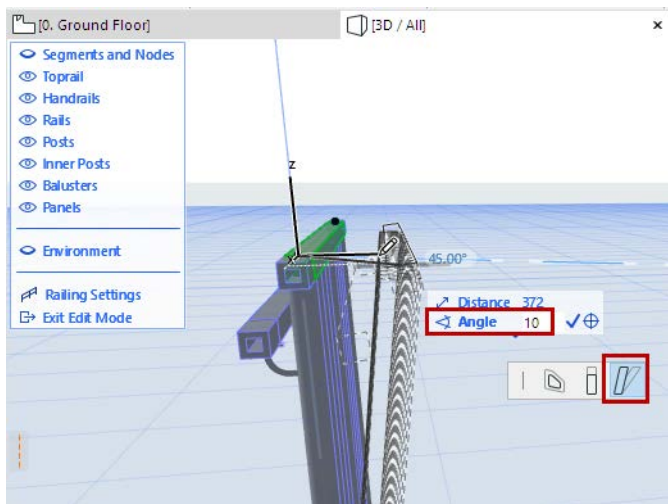


Choose a GDL component as the Toprail, or choose **Invisible** from the pop-up (if you do not want to place a physical Toprail).

To define settings for the GDL component, use the panel below. [See Rail Component Settings \(Toprail/Handrail/Rail\).](#)

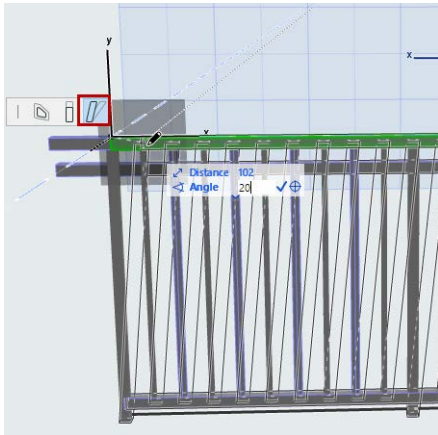
Slant or Skew Railing in Edit Mode

1. In Edit Mode, select the Toprail.
2. From the pet palette, use the Modify Angle command to adjust the Slant angle graphically.

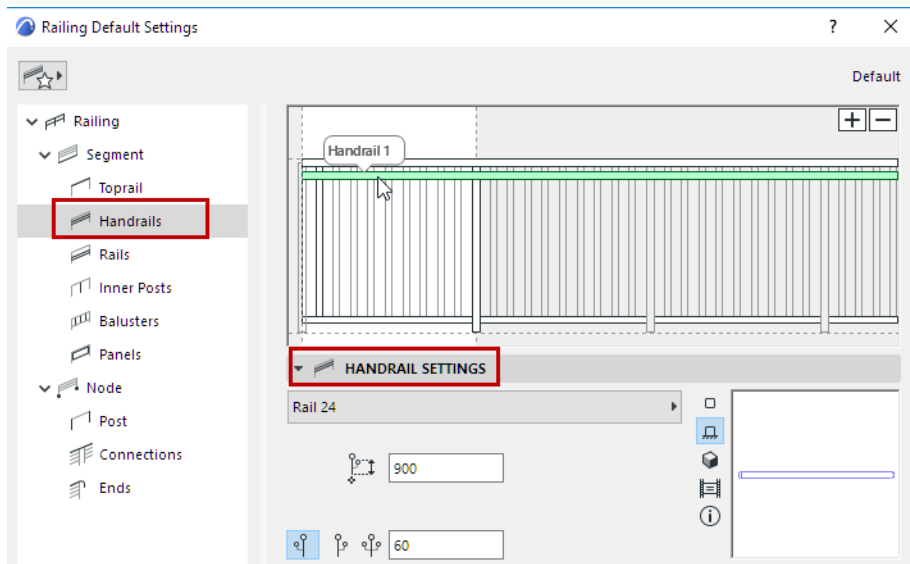


The entire Railing Segment follows - this is equivalent to changing the Segment's Slant angle.

Similarly, you can skew the Railing graphically: select the Toprail in Edit mode and use Modify Angle along the vertical editing plane.



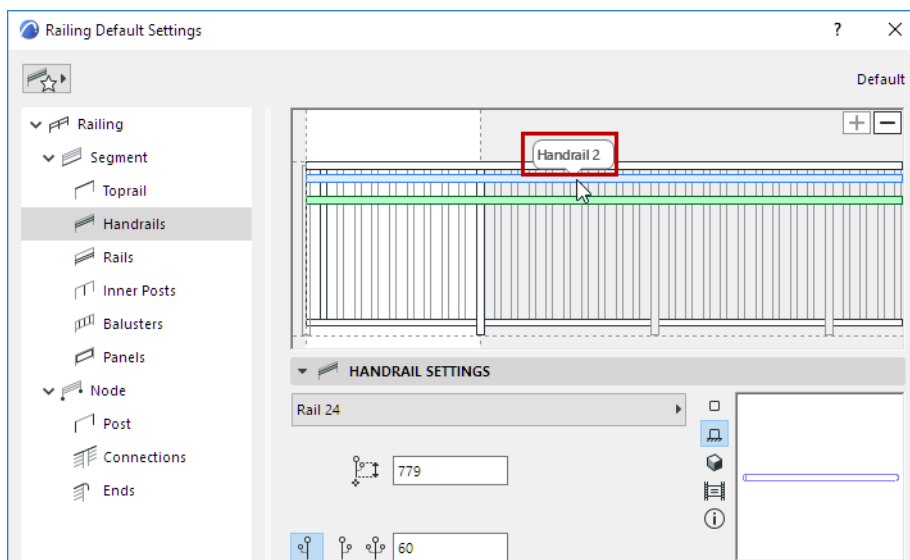
Handrail Settings (Railing Tool)



The Segment pattern can contain up to two **Handrails**. (Each can be doubled - see [Use Double Handrails](#), below.)

To add or delete Handrails, use the Plus or Minus sign in the Pattern Editor.

[See also Add Component in Pattern Editor.](#)



For each Handrail selected in the Pattern Editor, define its parameters in Handrail Settings:

- Choose a GDL component as the Handrail.

[To define settings for the GDL component, use the panel below. See Rail Component Settings \(Toprail/Handrail/Rail\).](#)

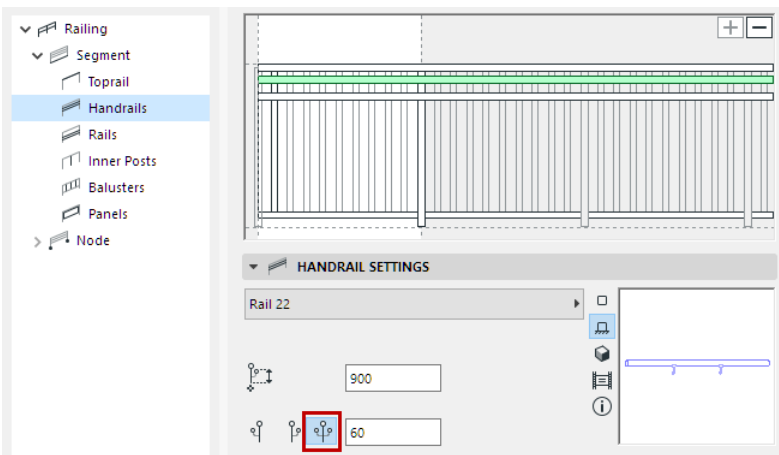
- Choose to place the Handrail on the left or right side of the Railing (“left” and “right” are based on the Railing input direction), or on both sides (see below).
- Horizontal Offset: Define the lateral offset of the Handrail(s).



Two Handrails

Use Double Handrails

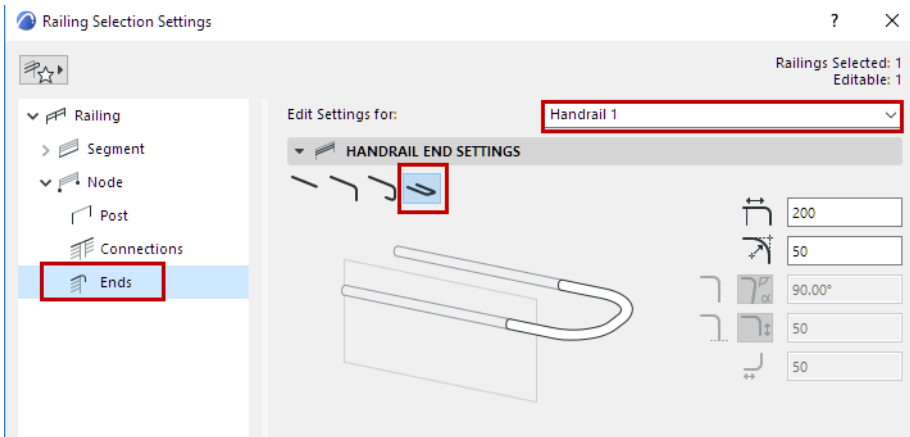
Each of the two Handrails can be doubled to the other side of the Railing, for a total of four physical Handrails: choose the “Both Sides” option from the Handrail Settings:



Two Handrails, Doubled

Connect Ends of Double Handrails

To connect the ends of the doubled Handrails, use the Ends page of Railing Settings:



See also [Railing Ends \(Toprail, Handrail, Rail\)](#).

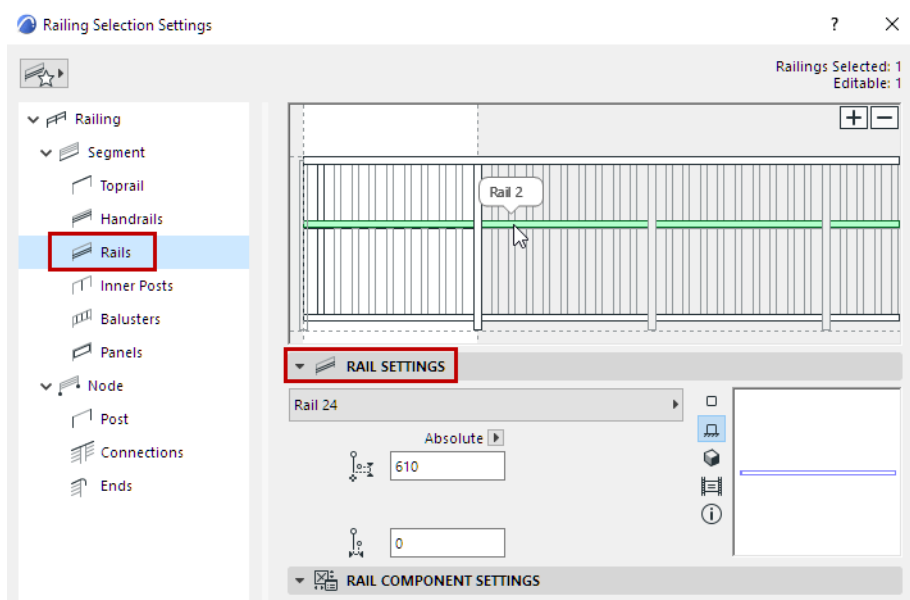
Rail Settings (Railing Tool)

Rails are general horizontal elements. You can offset them from the Railing Segment, and set them at any height. Rail cross sections can be different for each.

Rails can also be used as a reference point when positioning Panels and Balusters.

To add or delete Rails, use the Plus or Minus sign in the Pattern Editor. There is no limit to how many Rails you can add to the pattern.

For more information, see [The Pattern Editor \(Railing Tool Settings\)](#).

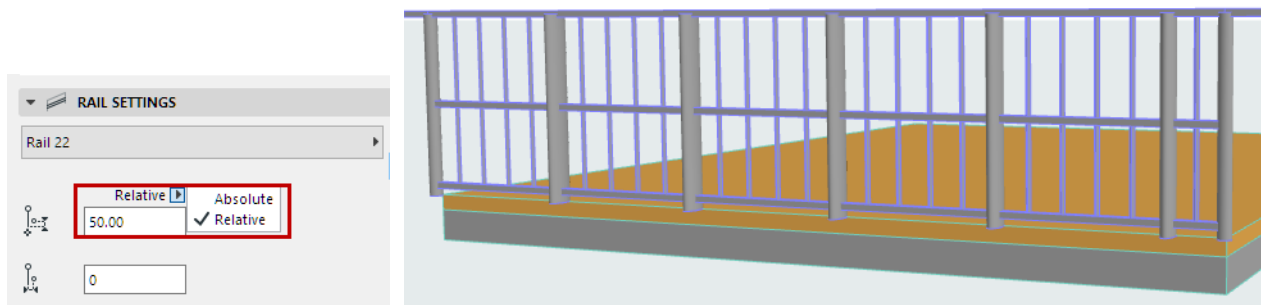


For each Rail selected in the Pattern Editor, define its parameters in Rail Settings:

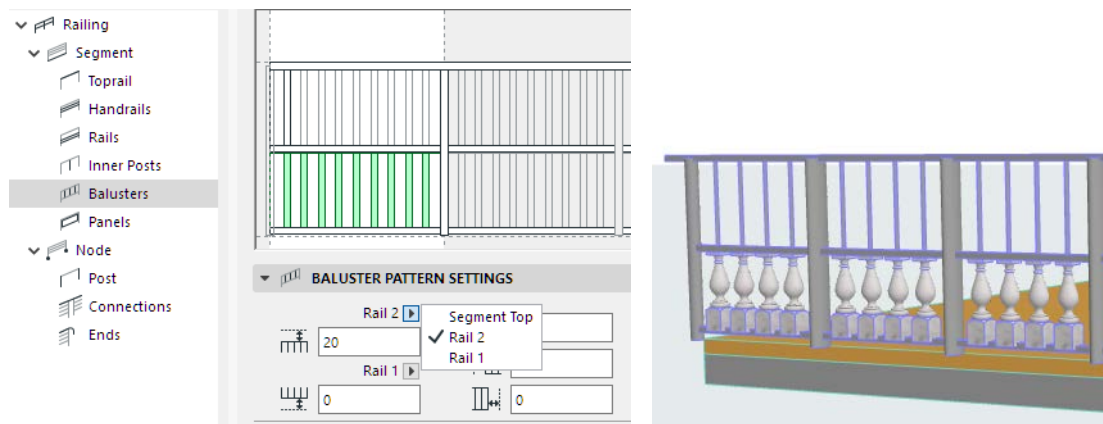
- Choose a GDL component as the Rail, or choose **None** from the pop-up (if you do not want to place a physical Rail).

To define settings for the GDL component, use the panel below. See [Rail Component Settings \(Toprail/Handrail/Rail\)](#).

- Set the **Rail Height**, defined as Absolute (from the Segment Bottom), or as a Percentage of the total Segment height (e.g. 50%).



- Horizontal Offset: Define the lateral offset of the Rail.
- Note that a newly placed Rail will intersect any Balusters, and can divide these Balusters into separately editable Baluster components, if Baluster heights are defined with reference to the Rail height.



Horizontal Rail Intersects and Splits Balusters

Of course, you can define the Baluster top/bottom to extend to the Segment top/bottom, instead of the Rails.

Note: Segment height is defined in Segment Settings. [See Segment Height and Vertical Offset.](#)

Rail Component Settings (Toprail/Handrail/Rail)

This panel is available in the Settings dialogs of all Rail-type Railing components: Toprail, Handrails and Rails. Available controls depend on the selected Rail component (e.g. Built-in, Profiled).

You must set up Component Settings for each horizontal Rail type separately (Toprail, Handrail, or Rail).

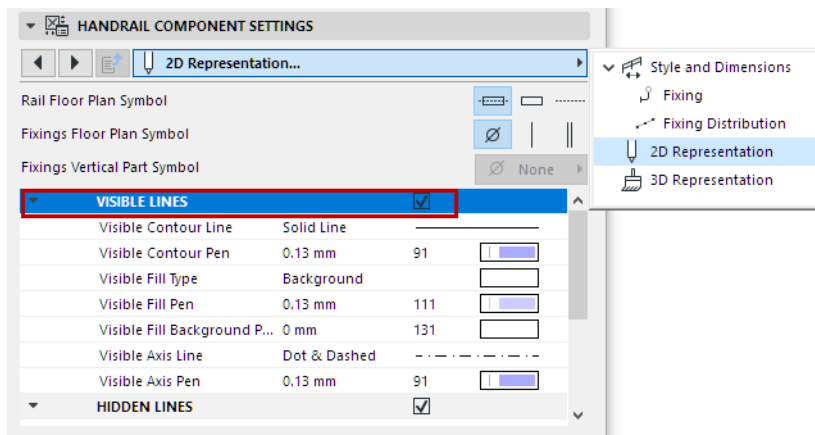
Make sure you are in the particular dialog box of the Rail component you need (e.g. to edit the Handrail, use Rail Component Settings from the Handrail Settings dialog).

Rail 2D Symbol Type

Select a **Symbol Type** on the 2D Representation tab page of Rail Component Settings.

Rail 2D Symbol Display

Set 2D display preferences for the Rail component, using the controls below (for Visible and Hidden parts of the Railing).



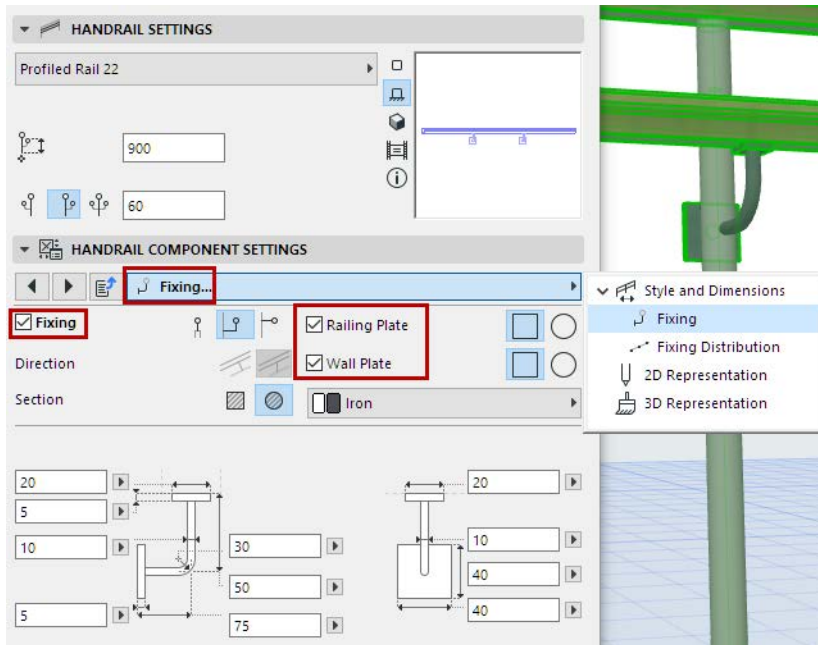
Show/Hide Rail's 2D Symbol

By default, the visibility of Railing components is defined at **Document > Model View > Model View Options > Railing Options**.

Customize Rail Symbol visibility in Railing Settings. ([See Railing Floor Plan Symbol Components.](#))

Fixing Settings

Use the Fixing tab page of Rail Component Settings to access the settings, including size and Building Material. The Profiled Rail Fixing, for example, can consist of a Wall Plate, Railing Plate or both.

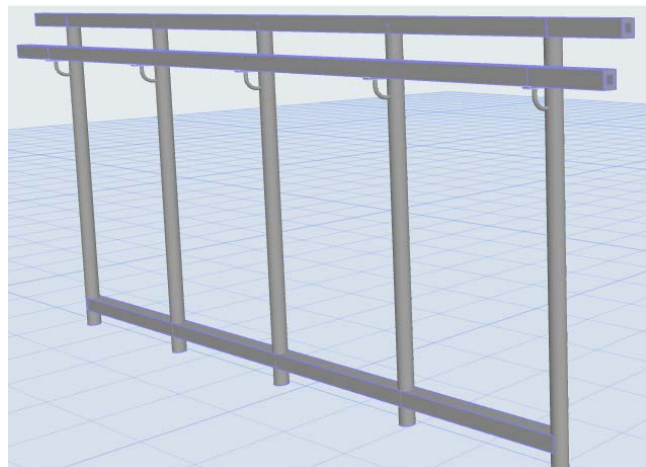
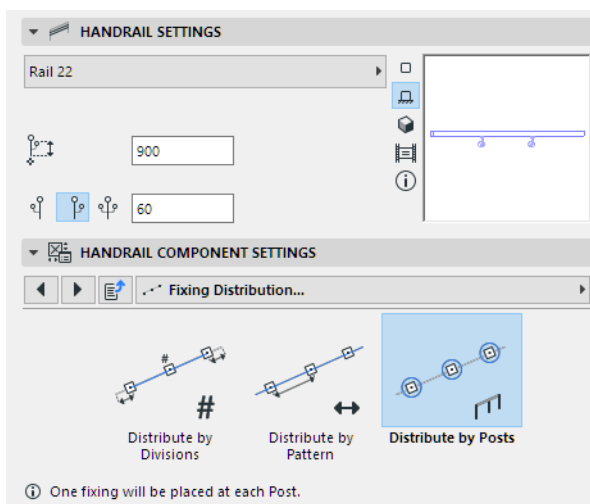


Fixing Distribution

Use the Fixing Distribution tab page of Rail Component Settings to define how to place the Fixings along the Rail.

Choose a Distribution method:

- **Distribute by Posts:** Place a Fixing at each Post and Inner Post.

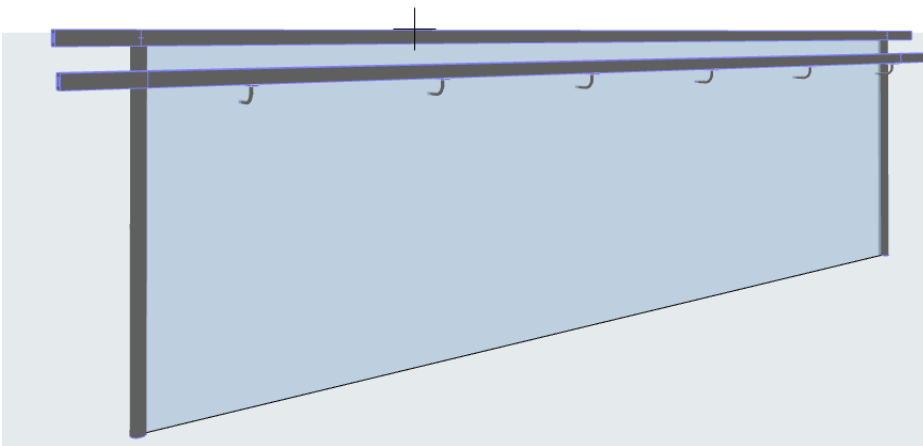
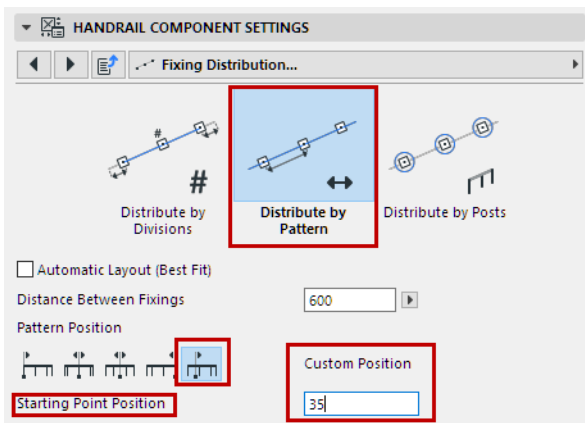


Handrail Fixings at Posts and Inner Posts

- **Distribute by Divisions**
 - In the **Number of Divisions** field, enter the total number of Fixings to place along the Segment length.
 - **Distance from Ends** refers to the gap between the first/last Fixing and the end of the Rail.

- **Distribute by Pattern**

- Distribute the fixings evenly within the Pattern Length you specify
- Check Automatic Layout (best fit), or
- Choose a startpoint for distributing the Fixings.
 - Segment Start or Segment End
 - Center of Segment: One Fixing is placed in the center, and then rest distributed, at set intervals, on each side.
 - Center of Segment with Best Fit: Fixings are distributed at the set interval, so as to be symmetrical over the length of the Segment.
 - Custom Position: Enter the startpoint (distance from Segment start)



Evenly Spaced Handrail Fixings: Start from Custom Position

Inner Post Settings (Railing Tool)

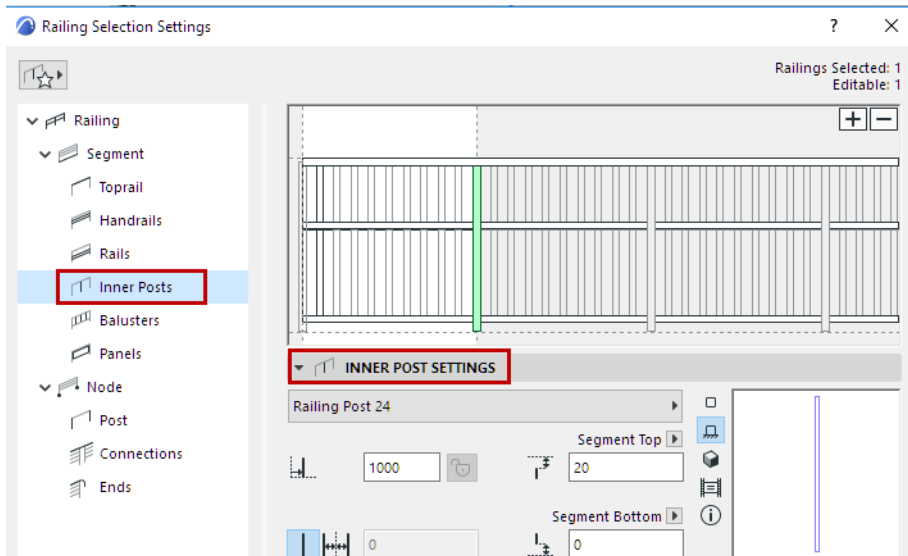
Use these Settings to add and define any number of Inner Posts in the Pattern Editor.

Inner Posts are located between Segment Nodes (as opposed to Posts, which are located at the Nodes.)

Inner Posts can also be used as a reference point when positioning Panels or Rails.

To add or delete Inner Posts, use the Plus or Minus sign in the Pattern Editor.

[See also Add Component in Pattern Editor.](#)



Choose a GDL component as the Inner Post, or choose **None** from the pop-up (if you do not want to place a physical Inner Post).

[To define settings for the GDL component, use the panel below. See Post Component Settings \(Posts/Inner Posts/Balusters\).](#)

In addition to predefined content, you can save a custom component as a Post.

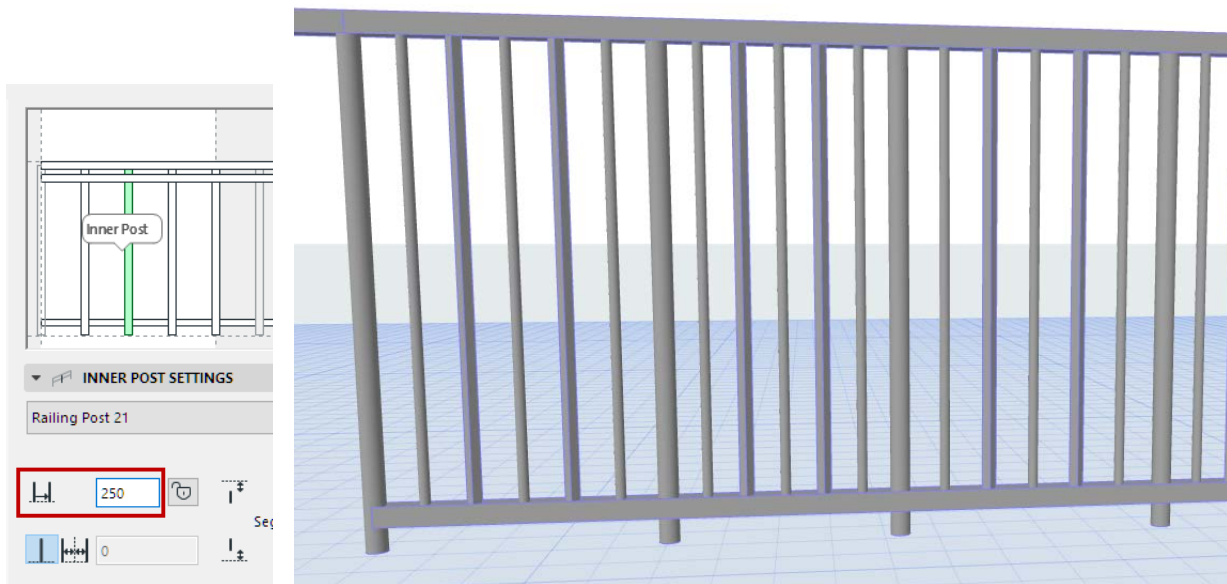
[For details, see Custom Sub-Elements for Stairs and Railings.](#)

Distance from Previous

For the Pattern's first Inner Post: Enter a value for the Inner Post's distance from the left edge of the Pattern.



For each additional Inner Post: the offset is defined as the distance from Inner Post to its left.

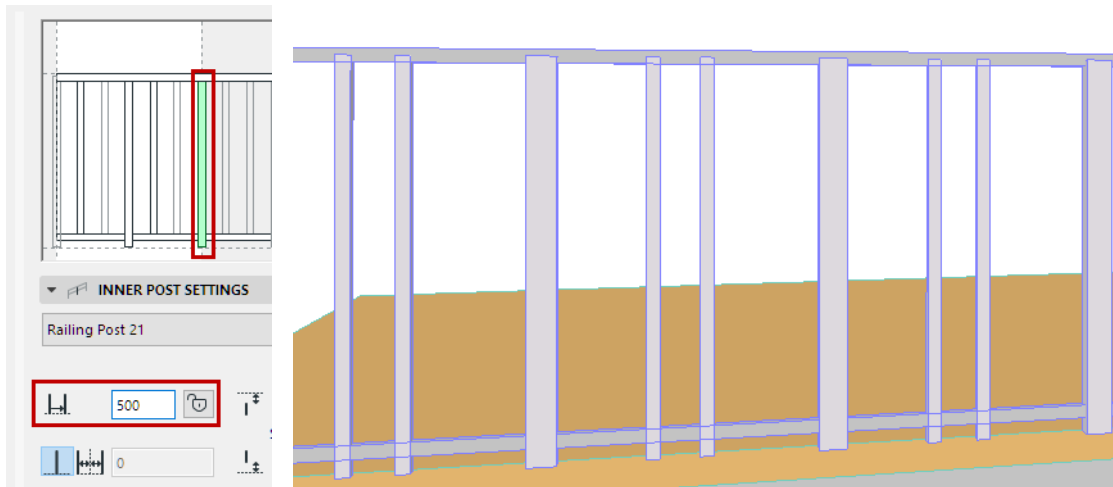


Click the lock icon to fix the Inner Post at the defined position. This is relevant if the Pattern Distribution would enable the pattern to be stretched. Inner Posts that are locked will not move.

[See *Pattern Distribution and Position*.](#)

At least one Inner Post has to be flexible (remain unlocked). Use this option if your pattern contains panels of fixed size.

For the Inner Post that is at the right edge of the white part of the Pattern Editor: this Distance also represents the Pattern Length, as shown here:

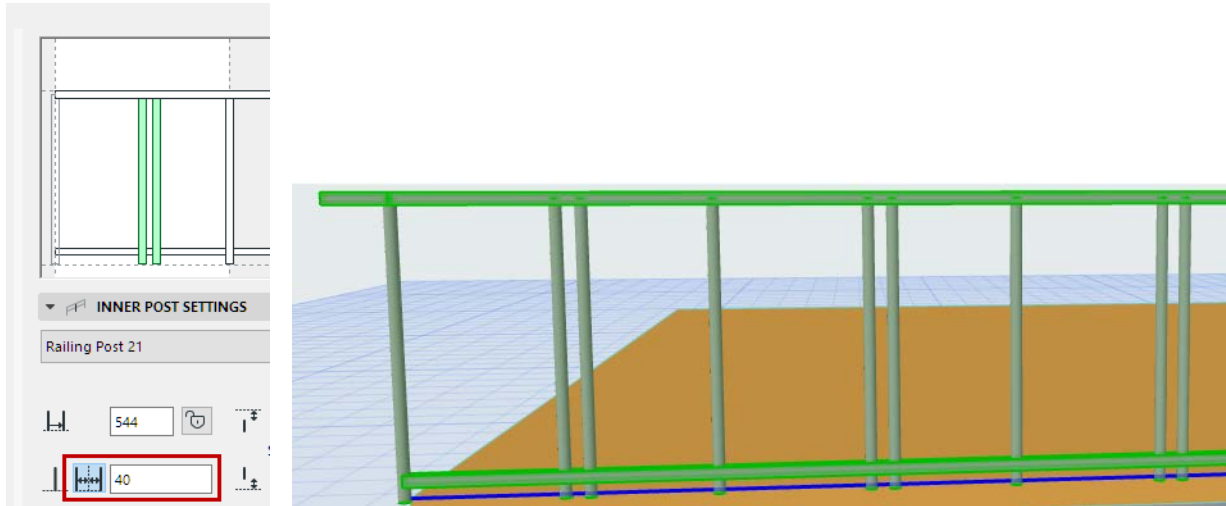


Inner Post at Right Edge of Pattern Editor = Pattern Length

[See also *Pattern Length*.](#)

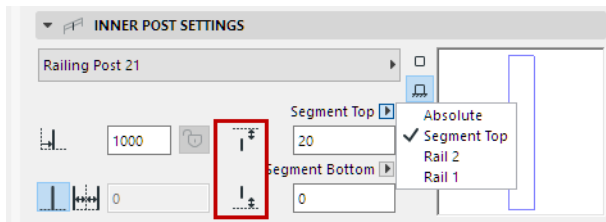
Duplicate Inner Posts

Use this control to double the selected Inner Post, with a defined offset between them:



Railings with Glass Panelings, Supported by Duplicated Inner Posts

Inner Post Top/Bottom Offset



Enter a bottom and top offset value for the Inner Post.

Use the pop-up to define how to calculate the offset:

- **Segment Top:** Set the offset value relative to the Segment height.
Note: Segment height is defined in Segment Settings. [See Segment Height and Vertical Offset.](#)
- **Absolute:** Enter the Inner Post's absolute height. The Inner Post is then offset from the Segment top accordingly.
- **Rail 1/2...:** If you have any Rails in the pattern, you can define the top or bottom offset from the height of a particular Rail.

Baluster Settings (Railing Tool)

Balusters are a pattern within the pattern. By default, the pattern contains two sets of Balusters; selecting one Baluster selects all its instance copies in the Pattern Editor.

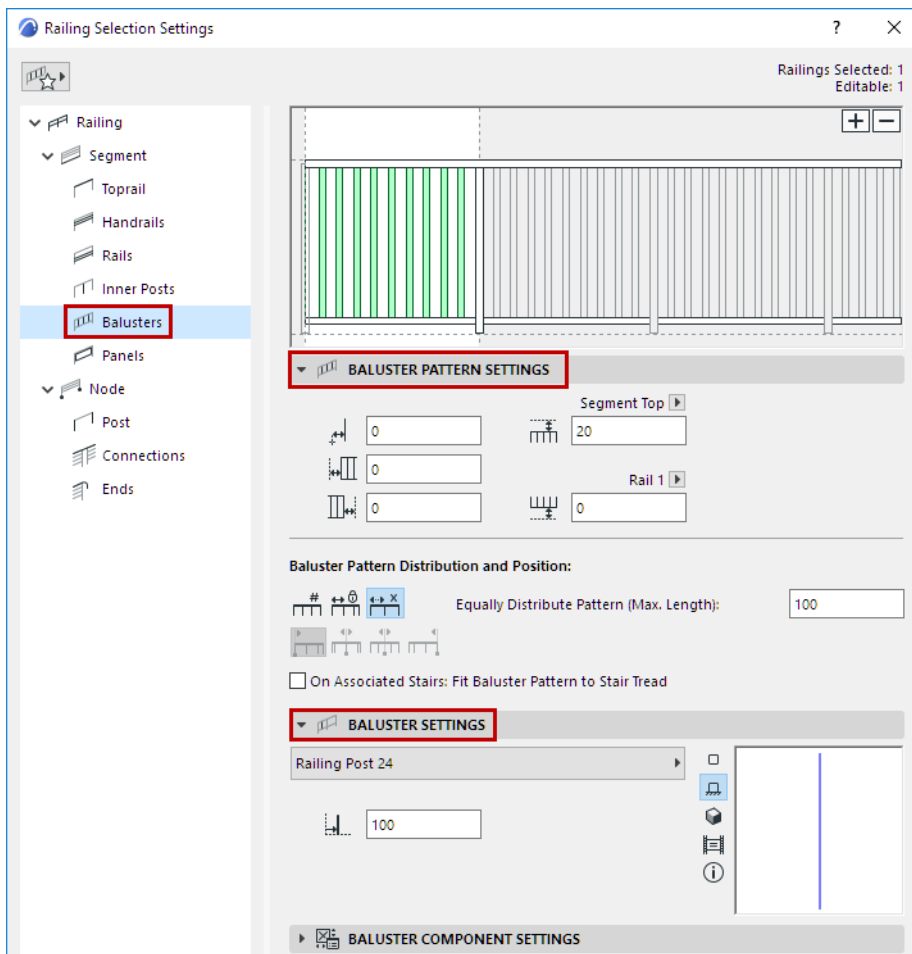
Balusters do not have fixings; they can be placed in the spaces between Rails/Toprails, or the Segment top/ bottom.

To add or delete Balusters, use the Plus or Minus sign in the Pattern Editor.

[See also Add Component in Pattern Editor.](#)

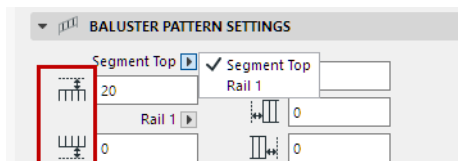
The number of Balusters you can add to the Pattern may be constrained by:

- the Baluster **Distance from Previous** value, which is set on the Baluster Settings Panel (see below), and
- the Baluster Pattern Distribution, as when the Pattern is constrained to a maximum Pattern Length.



Baluster Pattern Settings

Baluster Top/Bottom Offset



Enter a bottom and top offset value for the set of Balusters.

Use the pop-up to define how to calculate the offset:

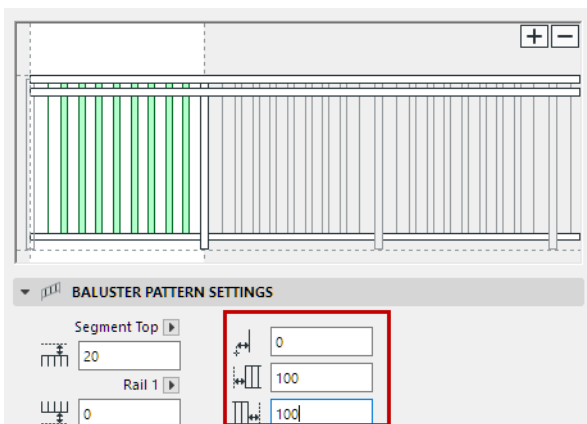
- **Segment Top:** Set the offset value relative to the Segment height.
Note: Segment height is defined in Segment Settings. [See Segment Height and Vertical Offset.](#)
- **Rail 1/2...:** If you have any Rails in the pattern, you can define the top or bottom offset from the height of a particular Rail.

Horizontal Offset

Define a lateral offset for the Balusters.

Left/Right Offset

Enter optional left/right offsets for the Baluster Pattern. Offsets are measured from the left and right ends of the pattern.

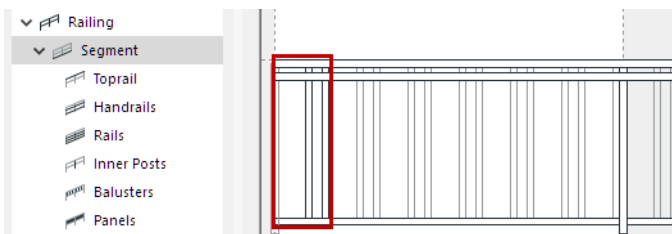


Baluster Pattern Offsets: Horizontal, Left, Right

Pattern Distribution and Position (Balusters)

These options define the Baluster Pattern distribution.

Note: The Baluster Pattern Length can be seen on the Segment page of Railing Settings: the black Balusters represent the Baluster Pattern. (In contrast, the Segment Pattern Length runs to the first Inner Post.)

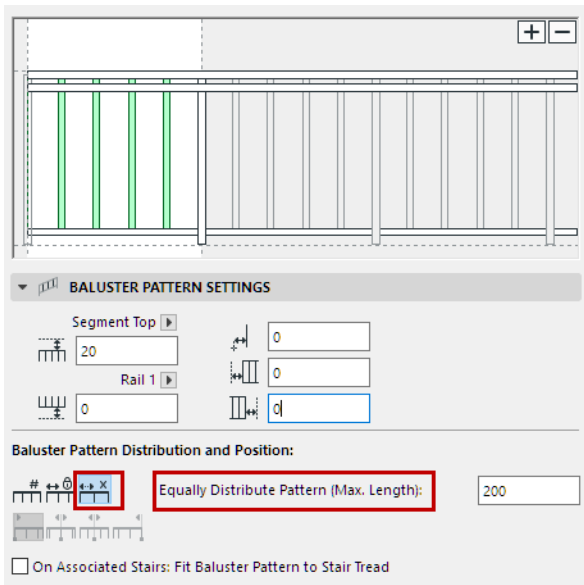


Baluster Pattern Length (shown in Pattern Editor, at Segment Settings)

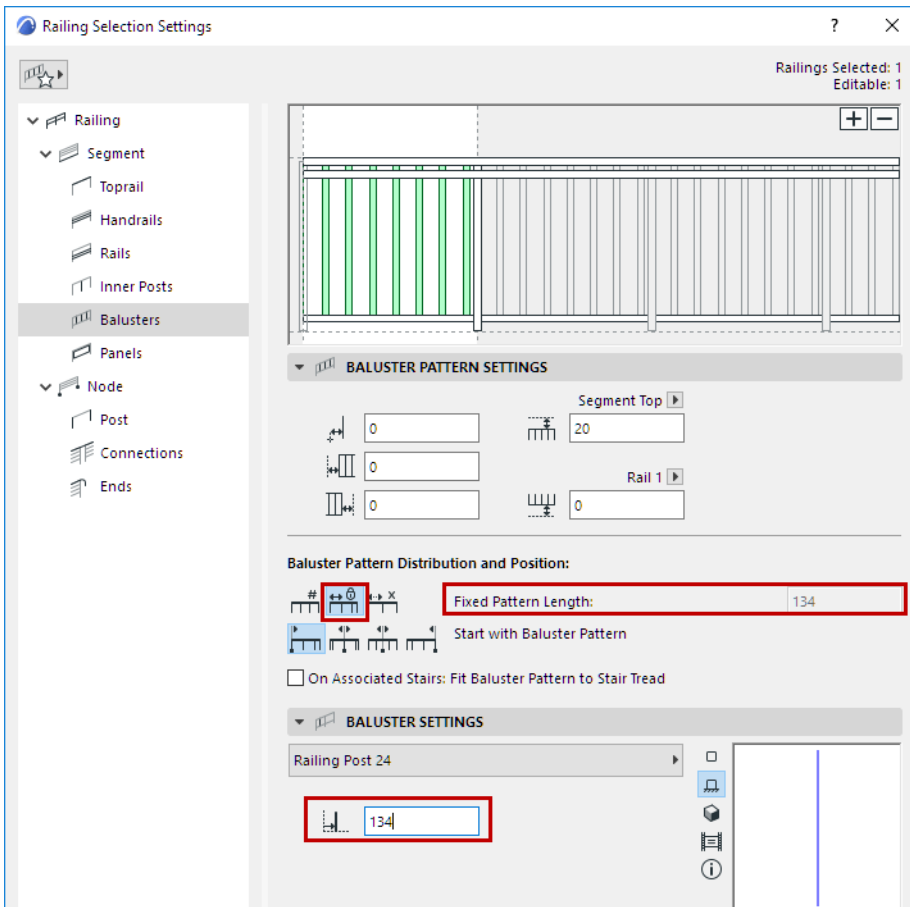
Choose one of the four Pattern Distribution options:

- **Equally Distribute Pattern:** The Baluster pattern is repeated over the Segment Pattern. The Pattern Length is adjusted (stretched) as needed.

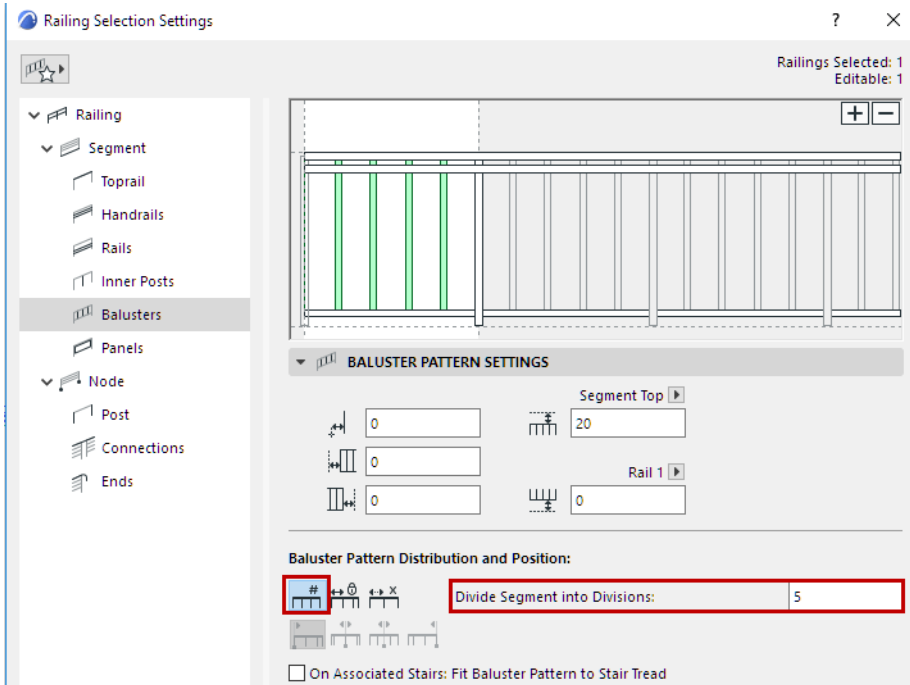
Enter a Maximum Length for the Baluster Pattern:



- Fixed Pattern Length:** The Baluster Pattern Length is fixed and unchanged. The Baluster Pattern Length is the sum of the Baluster sets' Position values, defined in Baluster Settings below.



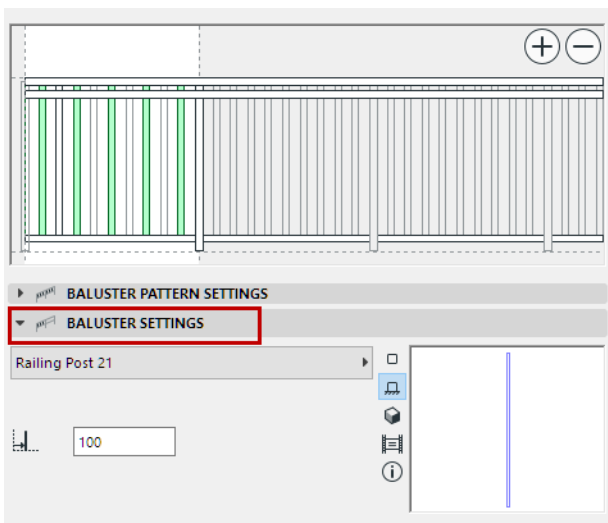
- Divide Segment Into Divisions:** Divide the Baluster Pattern Length into divisions (enter the number of divisions in the field, e.g. 5). The Baluster Pattern will be stretched so as to fit exactly into each division.



- Baluster Pattern Length = Tread Width:** For Railings associated to a Stair. (Choosing this option overrides any other Baluster Pattern distribution option.)

[See also Railings on Associated Stairs in Segment Settings.](#)

Baluster Settings Panel



Choose a GDL component as the Baluster.

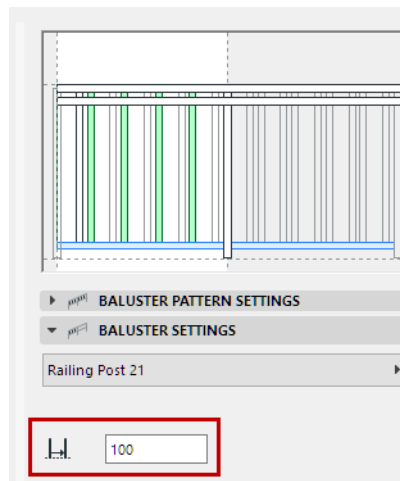
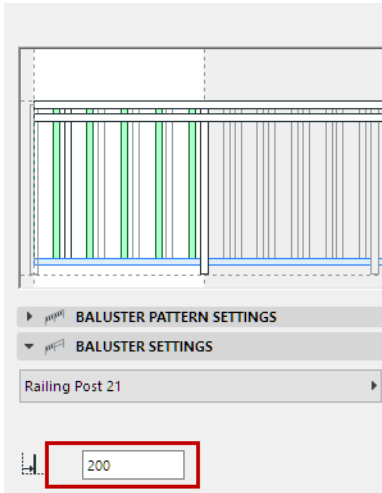
[To define settings for the GDL component, use the panel below. See Post Component Settings \(Posts/Inner Posts/Balusters\).](#)

In addition to predefined content, you can save a custom component as a Post, and use it as a Baluster.

[For details, see Custom Sub-Elements for Stairs and Railings.](#)

Baluster Position

For each Baluster set selected in the Pattern Editor: define the Baluster's position as the distance from the previous. (If you have more than one Baluster Set, the sum of the two values is the Baluster Pattern Length).



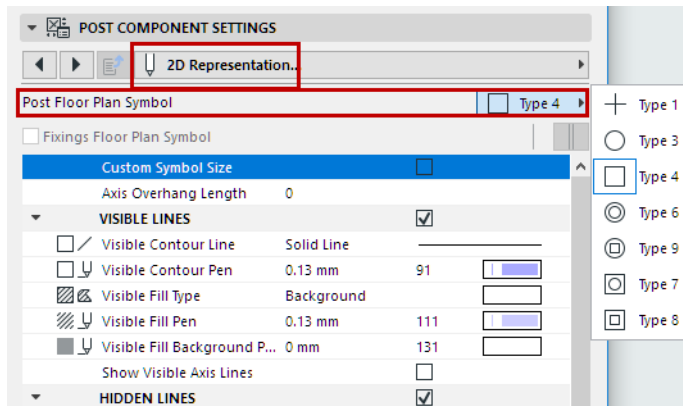
Post Component Settings (Posts/Inner Posts/Balusters)

This panel is available in the Settings dialogs of all Post-type Railing components: Posts, Inner Posts and Balusters.

Available controls depend on the selected Post component (e.g. Built-in, Profiled, Revolved). Some of the key controls are described here.

Post 2D Symbol Type

Select a **Symbol Type** on the 2D Representation tab page of Post Component Settings.



Post 2D Symbol Display

Set 2D display preferences for the Post component, using the controls below (for Visible and Hidden parts of the Railing).

[See Railing Settings: Geometry and Symbol Display](#)

Show/Hide Post's 2D Symbol

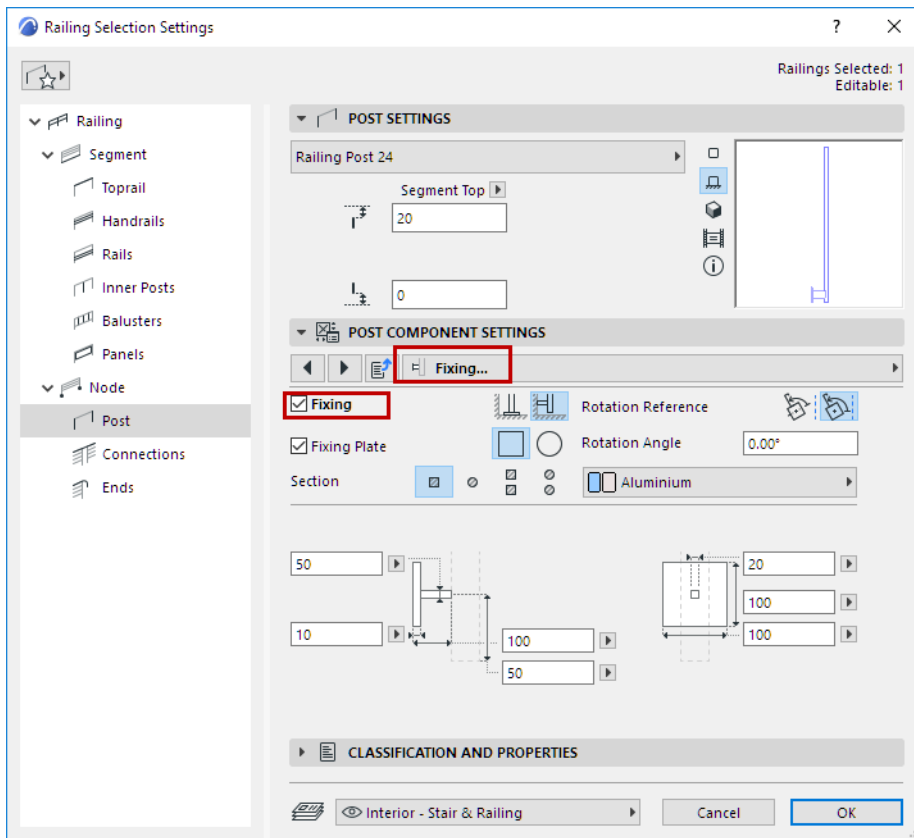
By default, the visibility of Railing components in 2D views is defined at **Document > Model View > Model View Options**.

Customize Post Symbol visibility in Railing Settings. ([See Railing Floor Plan Symbol Components.](#))

Show Fixing

Note: Not available for Balusters.

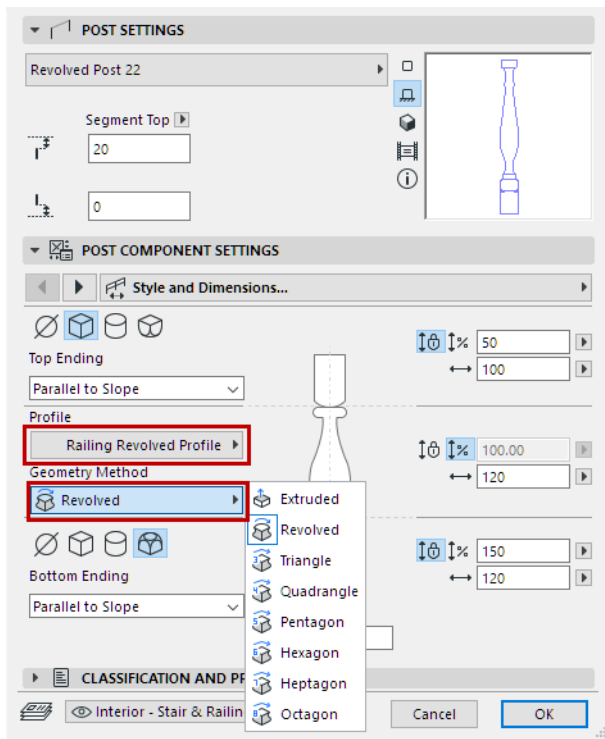
Use a fixing to attach the Post with a simple console. Click **Fixing** on the Fixing Settings tab page of Post Component Settings to access the settings, including size, profile and Building Material. (You can also open Fixing Settings from the Style and Dimensions tab page.)



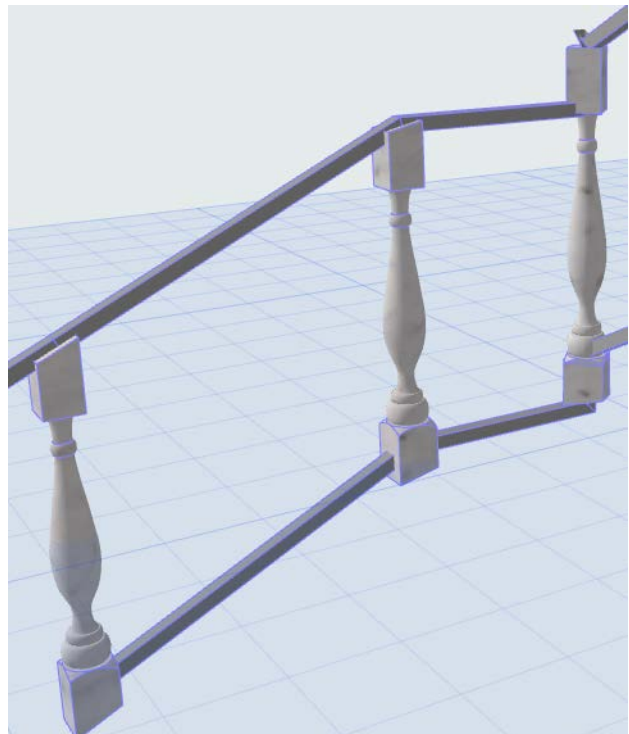
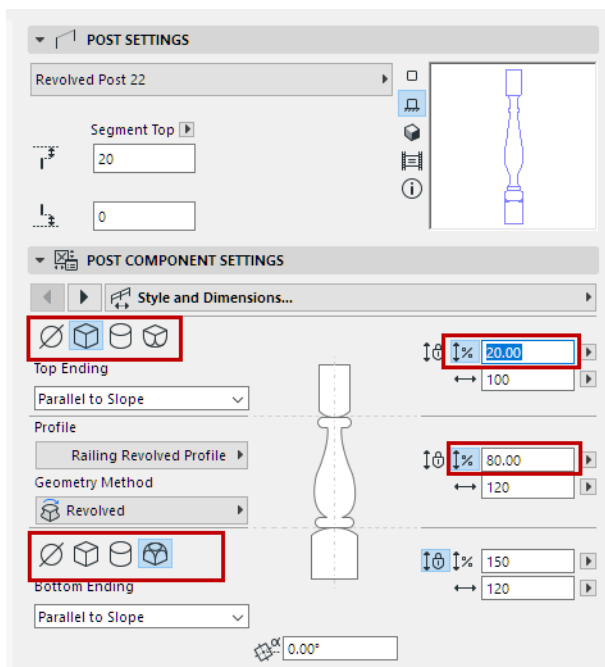
Revolved Post Options

For a Revolved Post, you have many customization options. View the Preview to get the idea.

- Choose a Profile (Middle Profile) for the center part of the Revolved Post
- Choose a Geometry method



- Add top and bottom parts to the post in a chosen Style
- For each part of the post (Top, Middle, Base):
 - Set an absolute height for each post part, or
 - Click the **Stretch** icon for one or more of the post parts (Top, Middle, and/or Base) to stretch it by a certain percentage of the whole. (The total stretching will equal 100 percent.)



Panel Settings (Railing Tool)

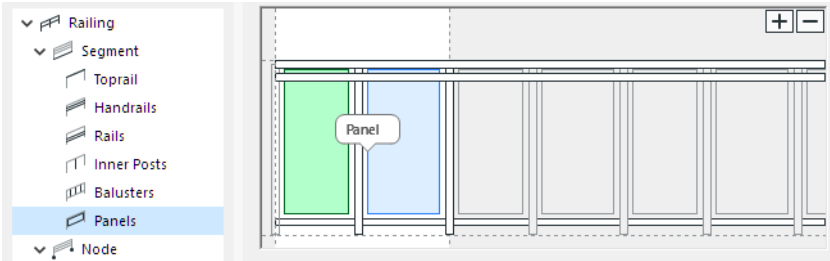
Use these Settings to add/remove and define Railing Panels in the Pattern Editor.

Add Panels to Pattern Editor

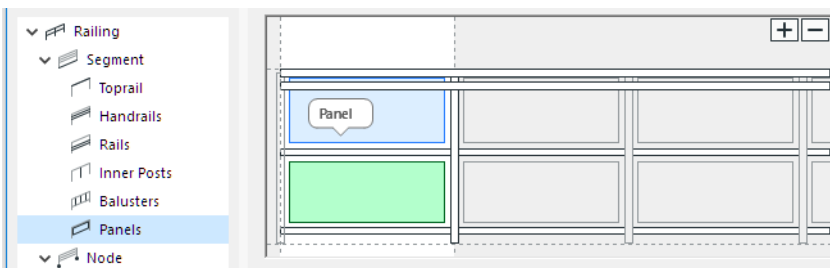
To add or delete Panels, use the Plus or Minus sign in the Pattern Editor.

See also [Add Component in Pattern Editor](#).

You can add a Panel into the empty space between two Inner posts or Rails. If the field already contains Balusters, you cannot also add a Panel.

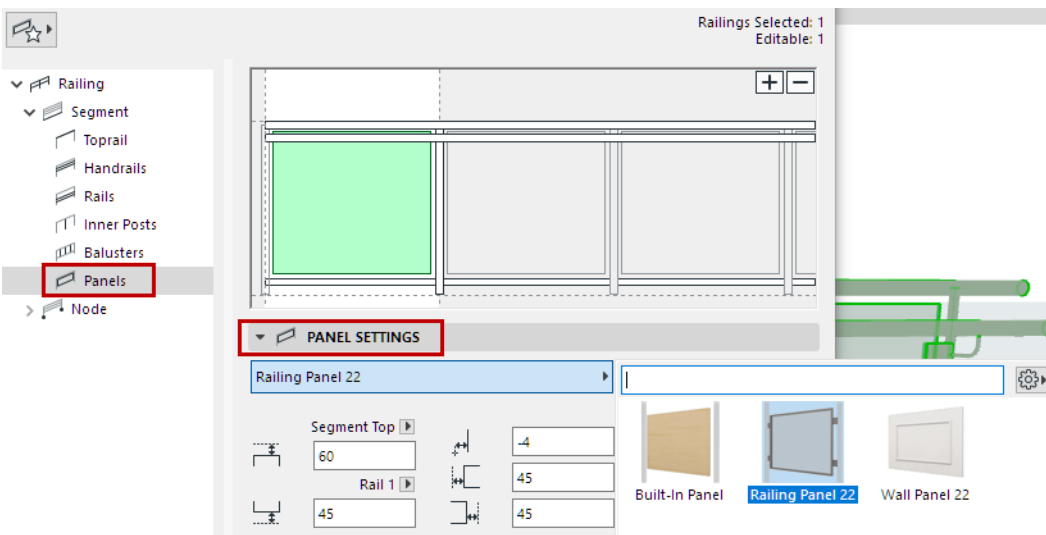


Panels Divided by Inner Post



Panels Divided by a Rail

Panel Settings



From the pop-up, choose a GDL component as the Panel.

To define settings for the GDL component, use the settings in the panel below. See [Railing Panel Component Settings](#).

In addition to predefined content, you can save a custom component as a Panel.

For details, see [Custom Sub-Elements for Stairs and Railings](#).

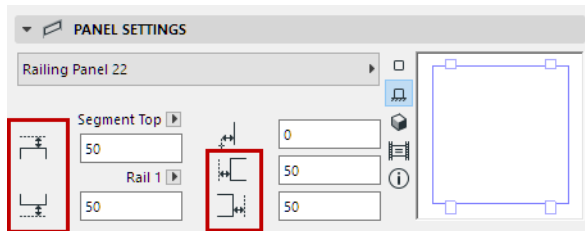
Define Panel Position

- Define the top and bottom offset of the Panel with reference to the Segment top/bottom, or to a Rail, if the Pattern contains them.

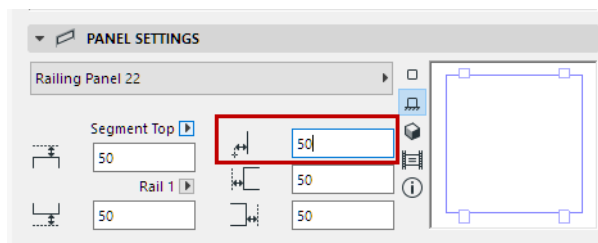
Note: Segment height is defined in Segment Settings. See [Segment Height and Vertical Offset](#).

- Define left and right offsets from the Inner Posts.

Note: Positive offset values mean that the panel size shrinks.



- Horizontal Offset: Define the lateral offset of the Panel (if any)



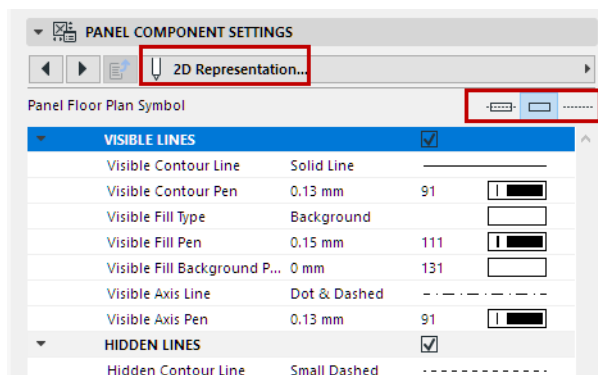
Railing Panel Component Settings

Panel 2D Symbol Type

Select a **Symbol Type** on the 2D Representation tab page of Panel Component Settings.

Panel 2D Symbol Display

Set 2D display preferences for the Panel component, using the controls below (for Visible and Hidden parts of the Railing).



Show/Hide Panel's 2D Symbol

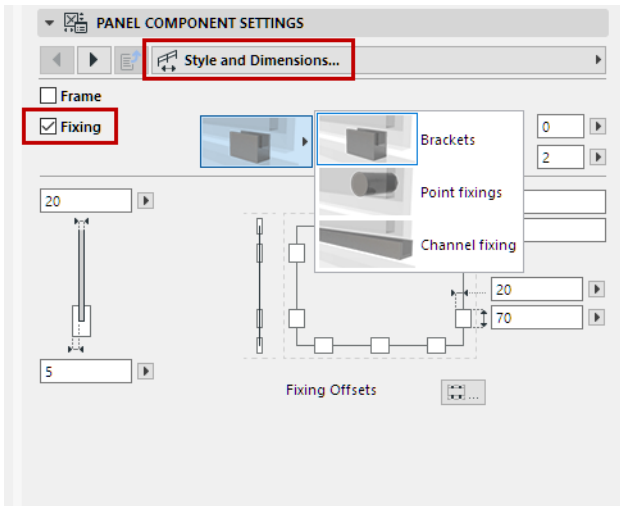
By default, the visibility of Panel components in 2D views is defined at **Document > Model View > Model View Options**.

Customize Panel visibility in Railing Settings. (See [Railing Floor Plan Symbol Components.](#))

Show Fixing

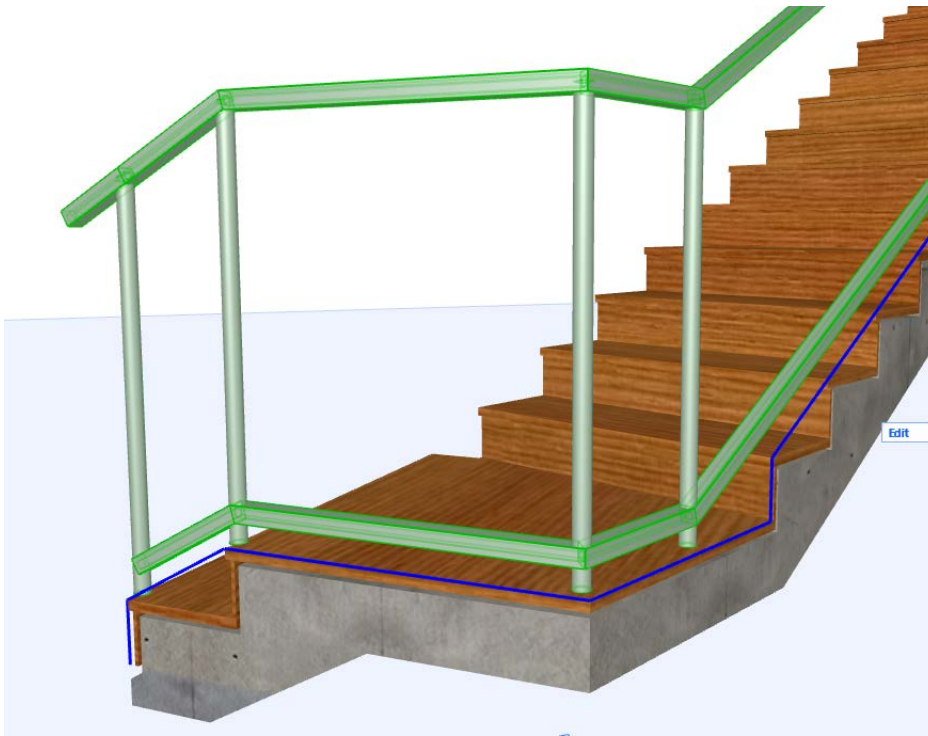
Click **Fixing** on the Style and Dimensions tab page of Railing Panel Component Settings to access the settings, including type, size, profile and Building Material, and number.

The controls depend on the type of Fixing you choose (Brackets, Point fixings, or Channel fixing).

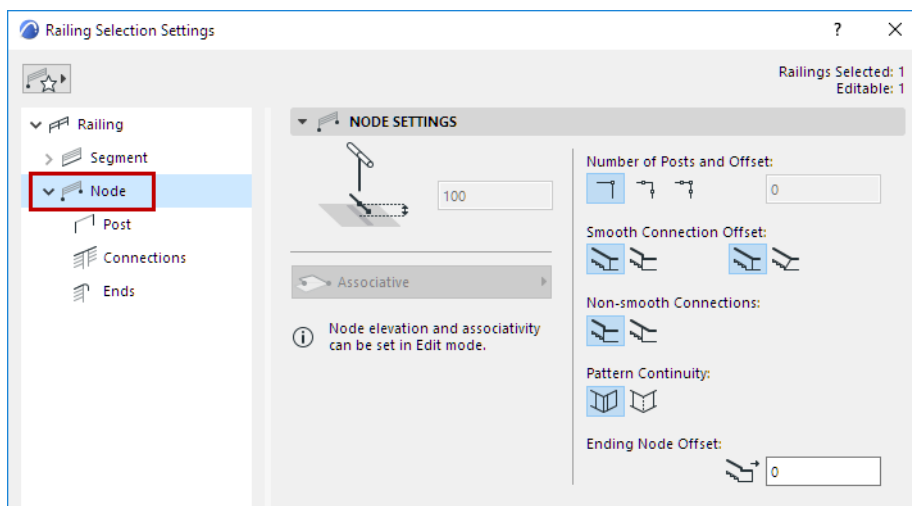


Node Settings (Railing Tool)

Nodes are the points that divide the Railing Segments (like points on a polyline).
One or more Posts are located at each Node.



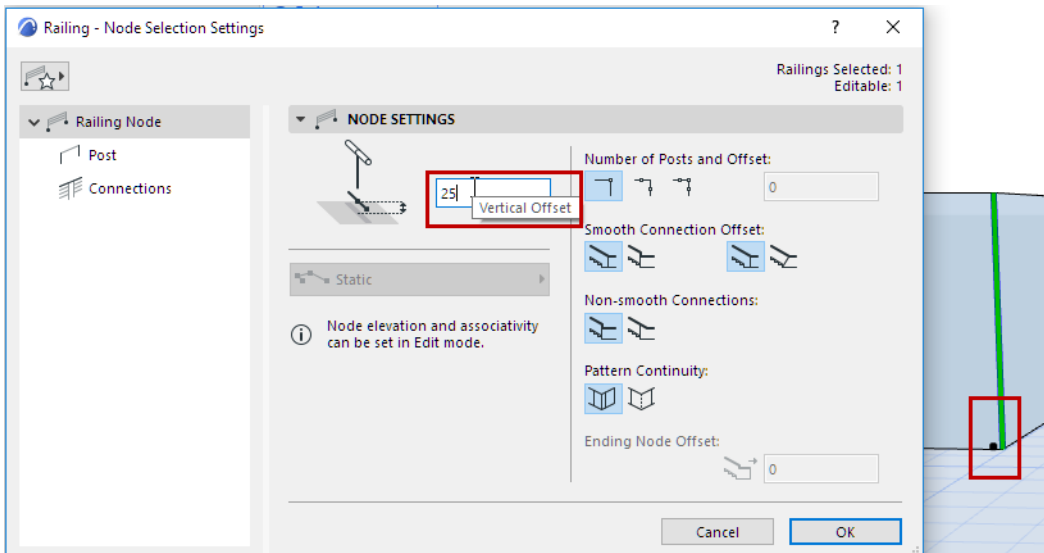
Use the Node page of Railing Settings to define the number and position of Posts at the Nodes, and how to handle the connection of two Segments when they meet at a Post.



Vertical Offset

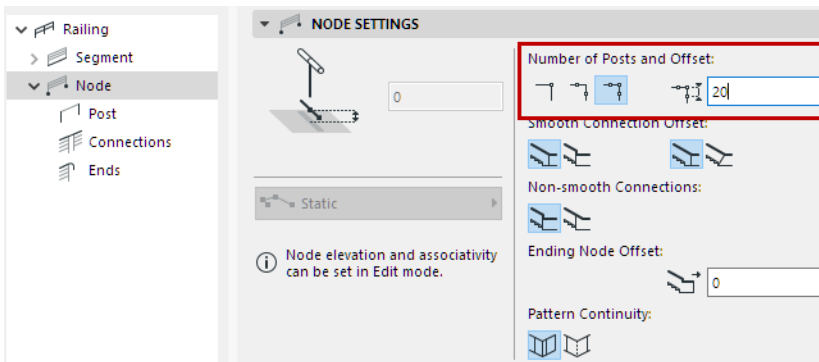
Available in Edit Mode only.

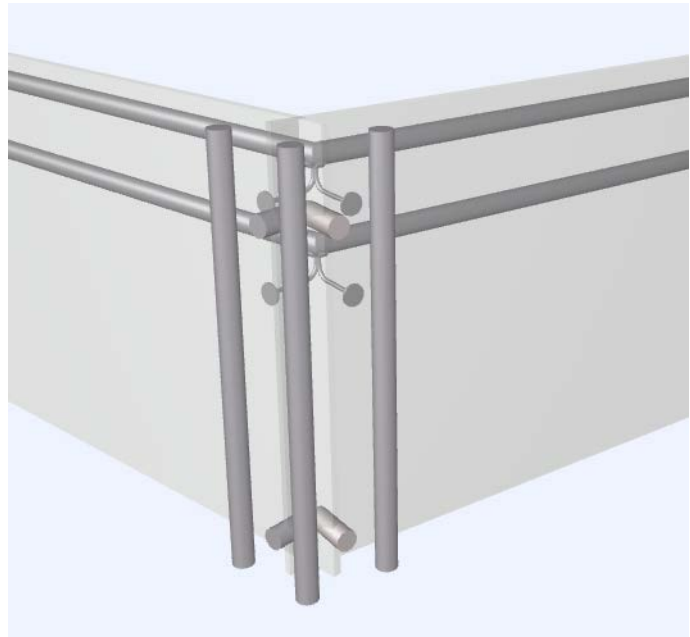
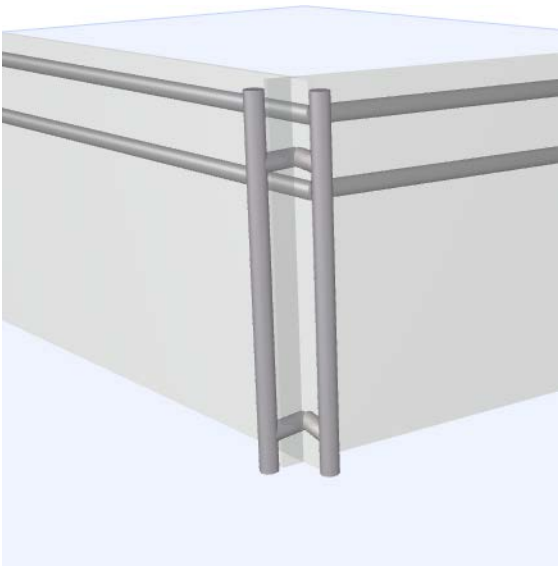
Set an optional elevation (vertical offset) for the Nodes (this is relevant for static Nodes).



Number of Posts per Node; Offset from Node

Set the number of posts per Node (one post at the Node, or two posts on either side of the Node, or all three posts).





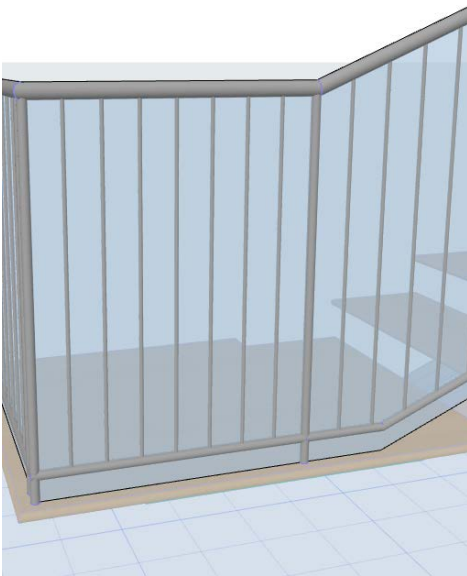
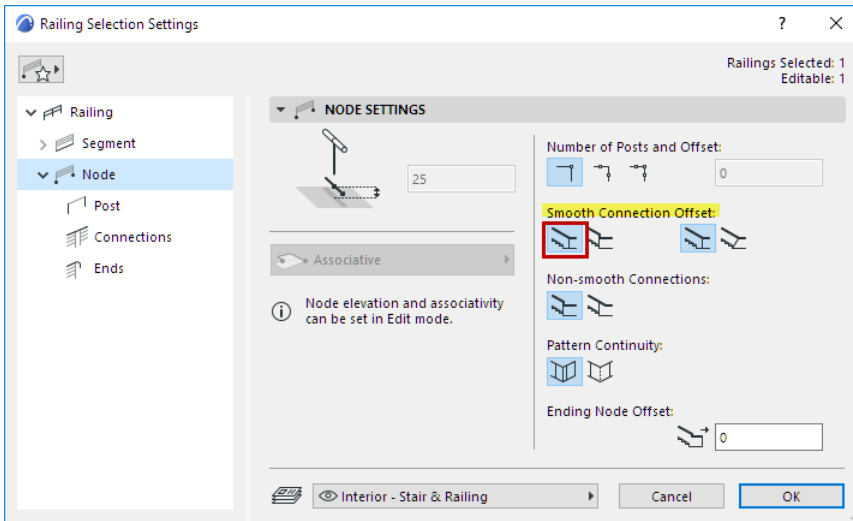
Node with Two Posts and Three Posts

If the Node has more than one post, set the offset distance of the posts on either side of the Node.

Smooth (Tangential) Connections - Post Offset

This is relevant for Posts located where two Railing Segments connect tangentially.

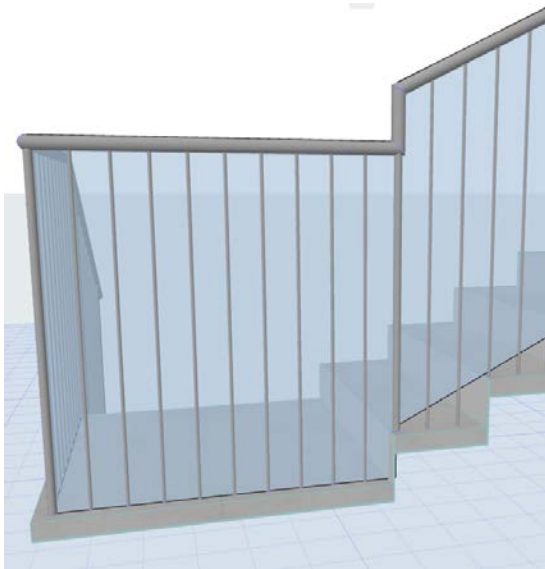
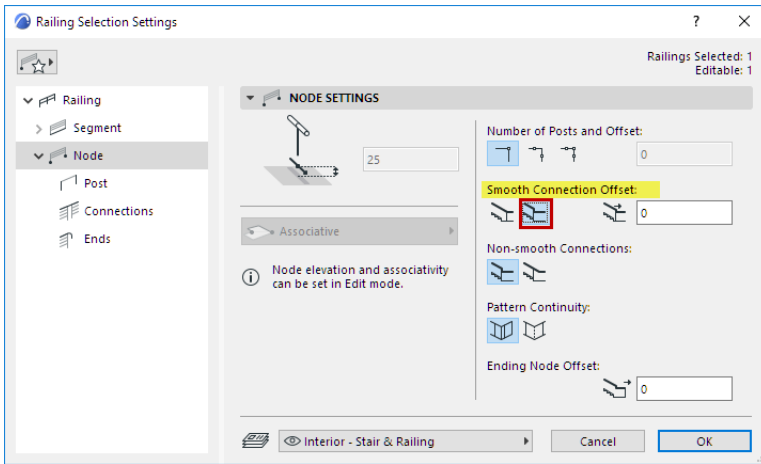
- To adjust the post position, choose either **Automatic** or **Manual** (use the first pair of icons)
 - **Automatic**: The post is moved to its new position automatically.



At Smooth Segment Connection: Post Moves Automatically

You can further define the connecting Post as **Vertical** or **Tilted**. (use the second pair of icons)

- **Manual:** You define the Post position, entering the desired offset

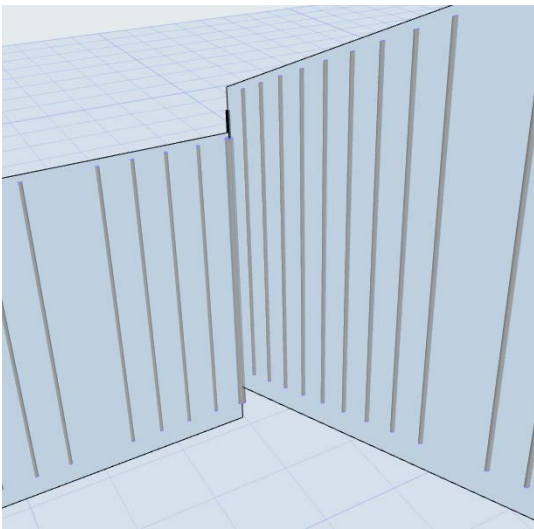
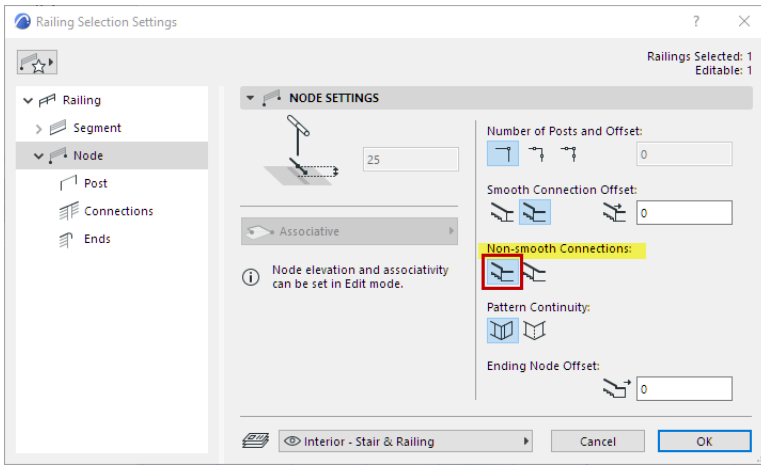


At Smooth Segment Connection: Offset Post Manually

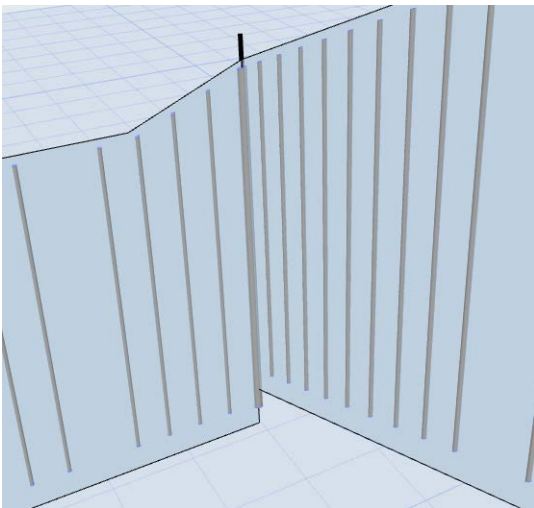
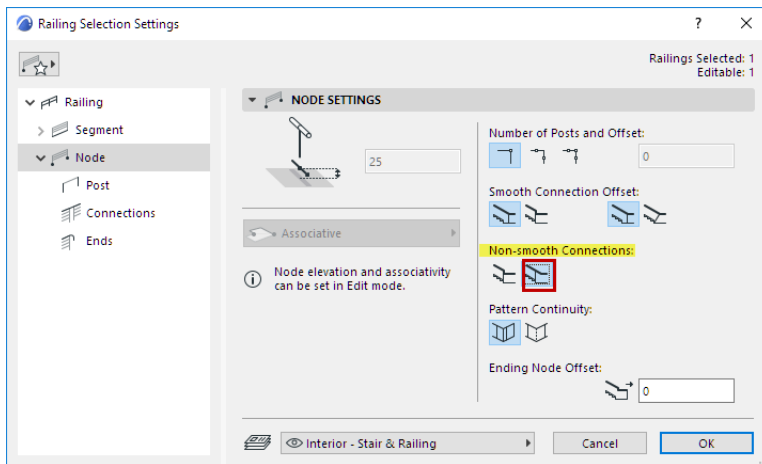
Segments at Non-Smooth Connections

This option is relevant for two Railing Segments of differing heights, which connect at an angle (non-smooth) at a node.

Use the icon pair to choose how to join the two segments: either leave them “stepped” at the node, or else adjust their geometry to connect at a point past the node.



Stepped Segments at Non-Smooth Connection



Adjusted Segments at Non-Smooth Connection

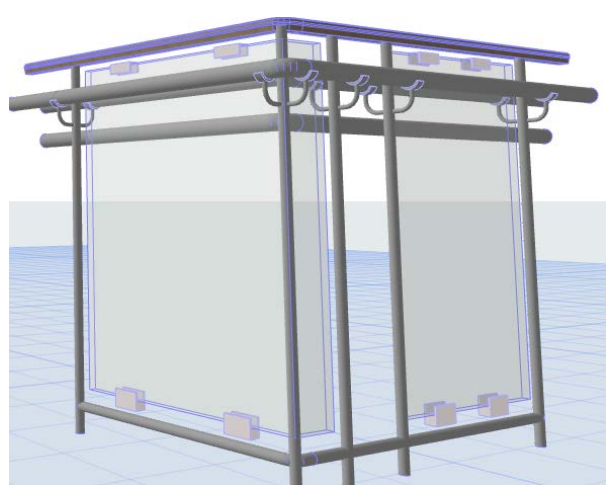
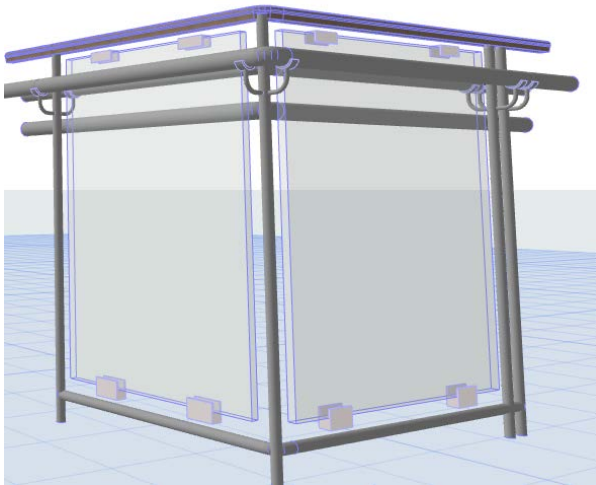
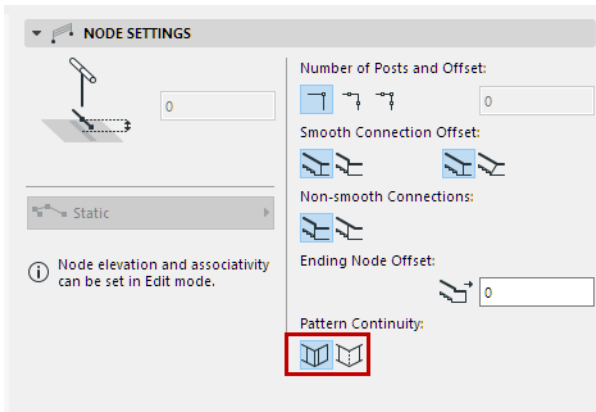
Ending Node Offset

Optionally, enter an offset for the Railing's Ending nodes. (This will offset the Railing from both ends of its Reference Line.)

Pattern Continuity

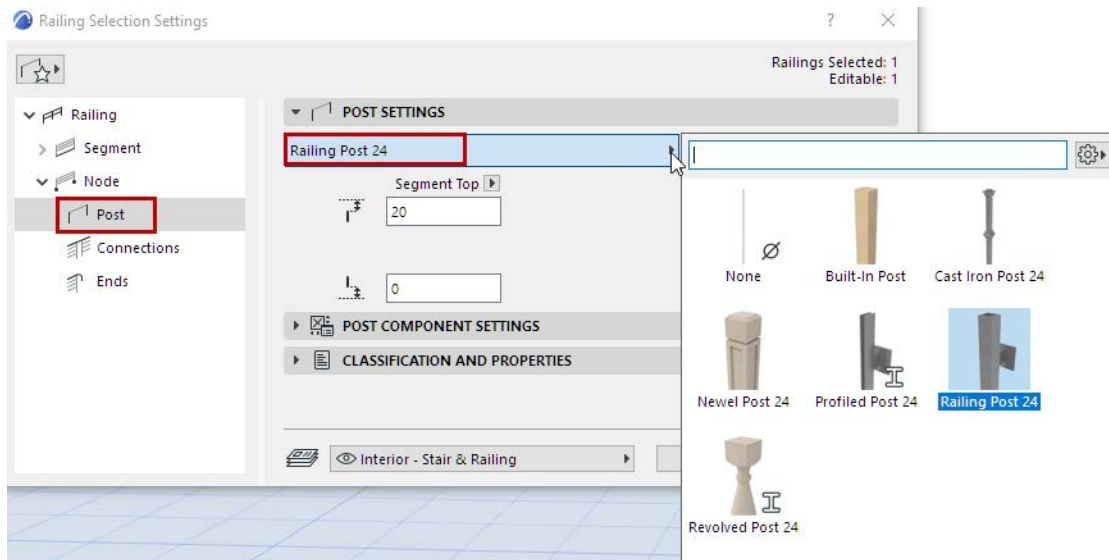
If two neighboring Segments use the same pattern: choose whether the pattern should end at the Node and start over, or else continue without interruption.

For information on setting the Railing Pattern, see [The Pattern Editor \(Railing Tool Settings\)](#).



At Nodes: Crop Segment, or Continue Pattern

Post Settings (Railing Tool)



From the pop-up, choose a GDL component to be placed as a Post at each Node of the Railing, or choose **None** from the pop-up (if you do not want to place a physical Post).

To define settings for the GDL component, use the panel below. See [Post Component Settings \(Posts/Inner Posts/Balusters\)](#).

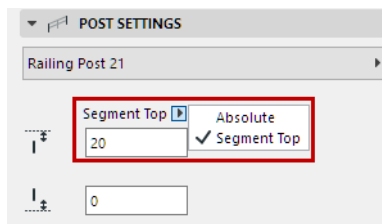
In addition to predefined content, you can save a custom component as a Post.

For details, see [Custom Sub-Elements for Stairs and Railings](#).

Post Top/Bottom Offset

Enter a bottom and top offset value for the Post.

Use the pop-up to define how to calculate the offset:

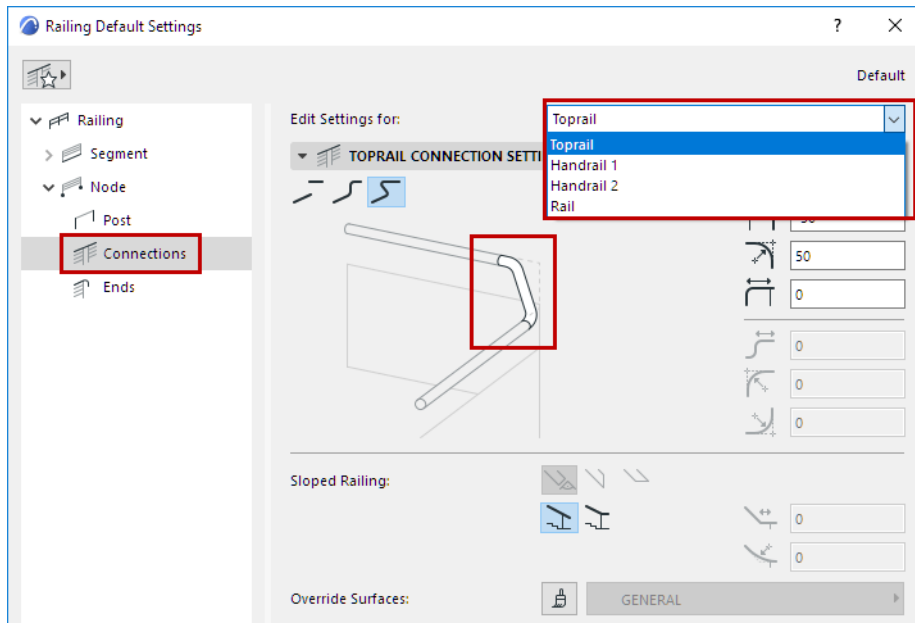


- **Segment Top:** Set the offset value relative to the Segment height.
Note: Segment height is defined in Segment Settings. [See Segment Height and Vertical Offset.](#)
- **Absolute:** Enter the Post's absolute height. The Post is then offset from the Segment top accordingly.

Connection Settings (Toprail, Handrail, Rail)

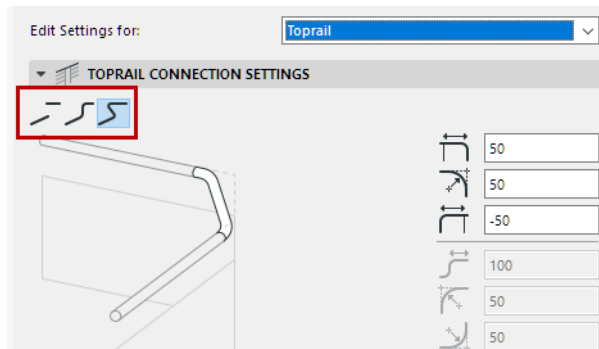
For each type of Rail (Toprail, Handrails, regular Rails), use these settings pages to define the geometry at connecting Segments, for each rail type.

You can display the connection segment using a different surface from the connecting rails (use the Override Surfaces control).



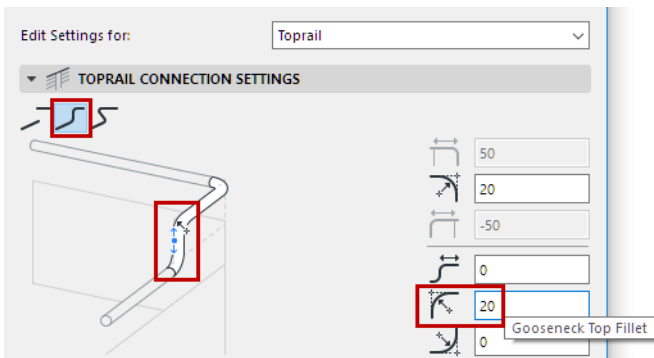
Rail Connection Types

Choose how to connect the selected Rail type between segments, using one of the three connection types:

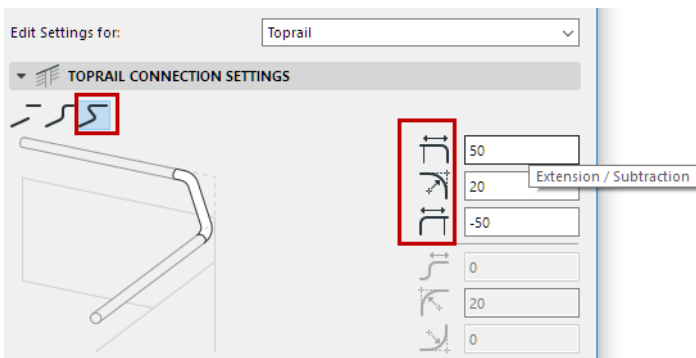


- **Disconnected:** Rails of the adjoining segments will not connect. Add an optional extension (positive or negative) to each.
- **Gooseneck:** Adjoining Rails are connected with a gooseneck consisting of horizontal and vertical parts. In the editable fields on the right, set the extension (positive or negative) and fillet values for each part of the gooseneck connection.

Note that the preview highlights the part you are editing.

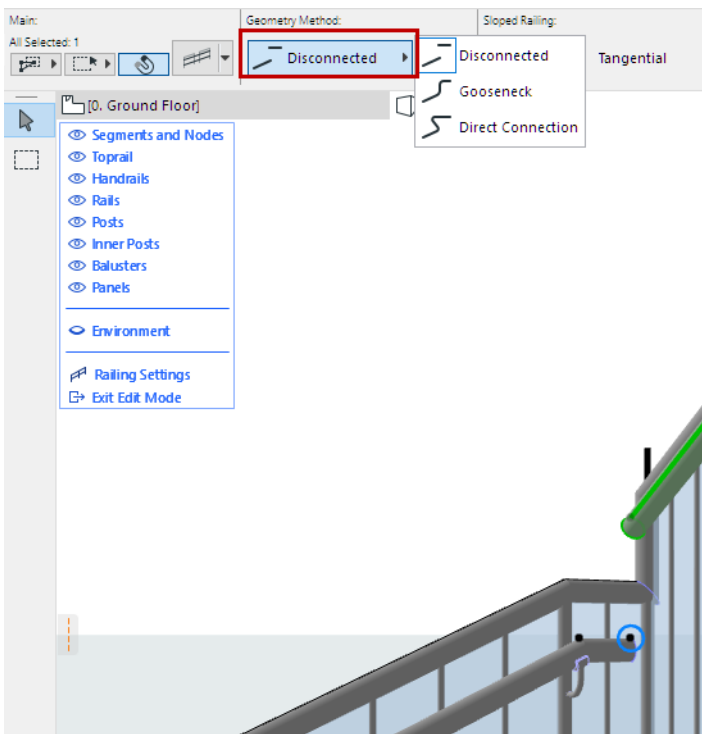


- **Direct Connection:** Connect the two adjoining rails by extending each one, then joining them together. In the editable fields on the right, set the extension (positive or negative) values, and a fillet value for the curved parts that curve off the horizontal parts.



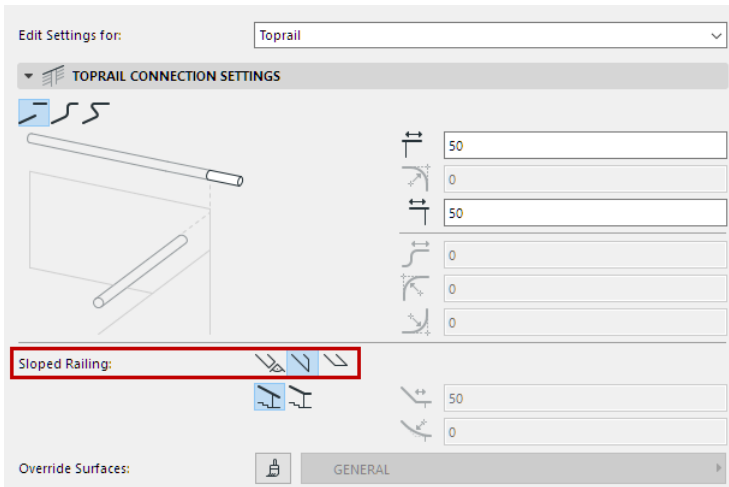
Edit Connections

- In Edit Mode, use the Info Box or End Settings to adjust settings of any selected Connection.



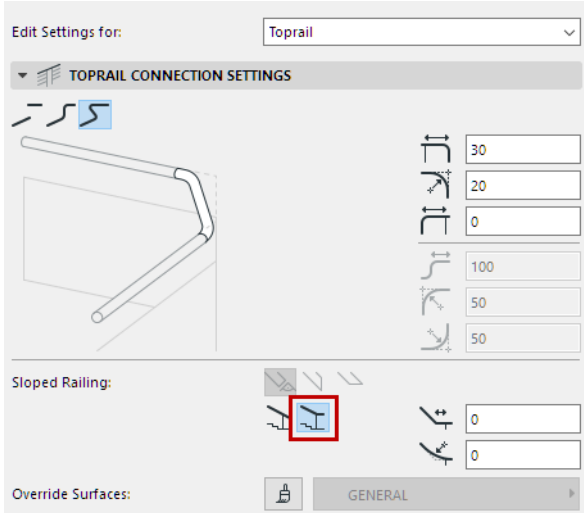
Sloped Railing (Connection) - Ending Geometry

- Choose the geometry for the ends of a connecting segment for a Sloped Railing: perpendicular, vertical, or horizontal.

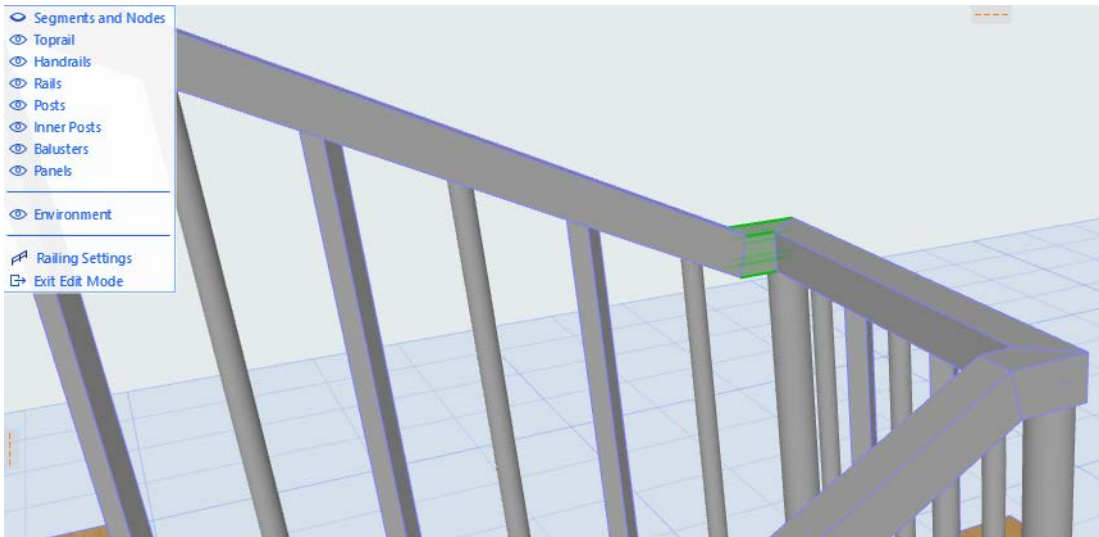


Sloped Railing (Connection) - Tangential to Segment or Horizontal

- Switch this option to add a horizontal segment to the Connection, where a sloped railing meets a horizontal turn.
- Enter the length of the added segment and an optional fillet.

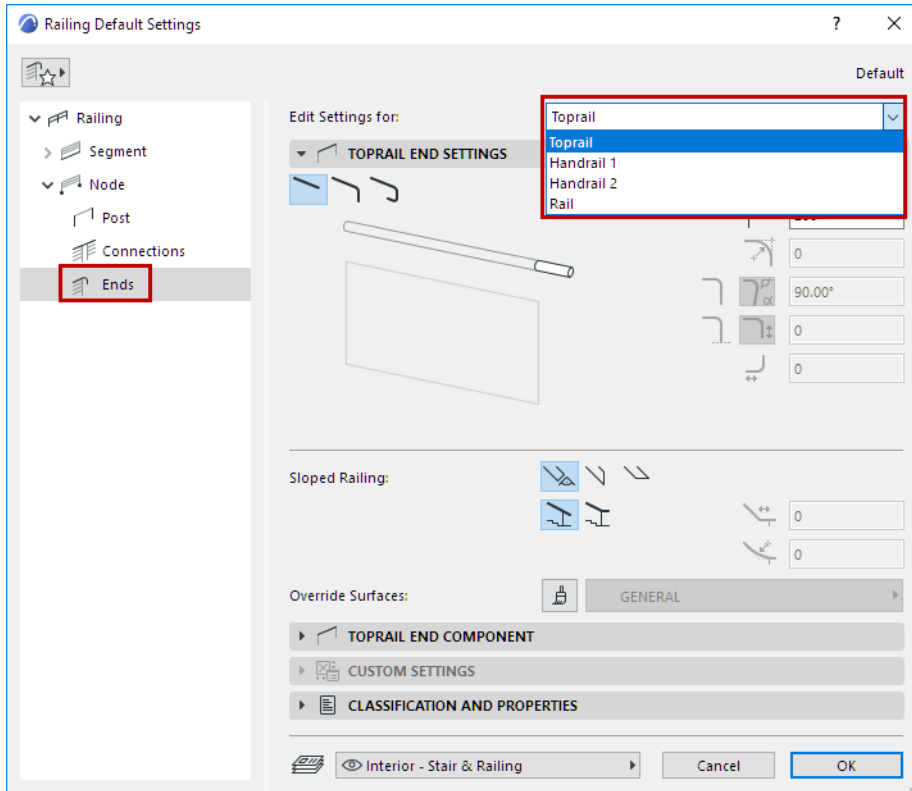


The resulting segment in Edit mode:



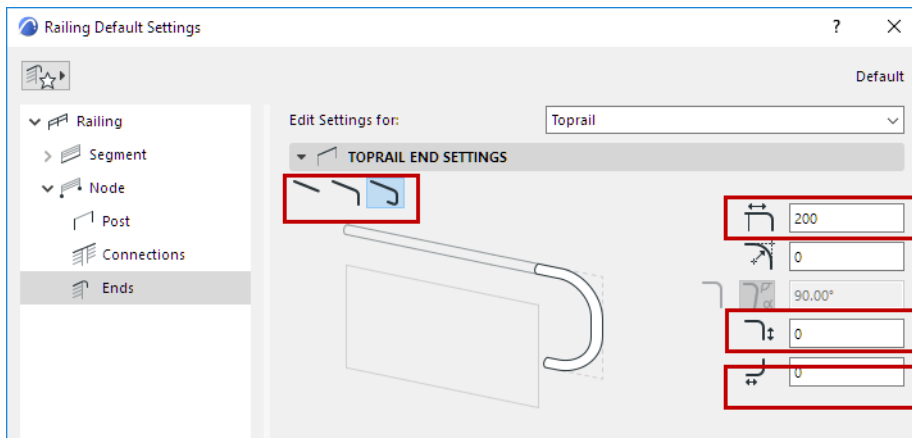
Railing Ends (Toprail, Handrail, Rail)

Each type of Rail (Toprail, Handrails, regular Rails) can have one or more Extension segments at their ends. In Railing Settings (Ends page), set ending parameters separately for each Rail type by choosing one from the pop-up list:



Define Rail End Geometry

The Rail Ending consists an Extension that can take the form of a single straight Extension, a Curve, or Full Return.



Choose the ending you prefer from End Settings. The Extension lengths are set in the fields at right.

Notes:

- If you do not want to use the ending, set the Extension Length to 0.

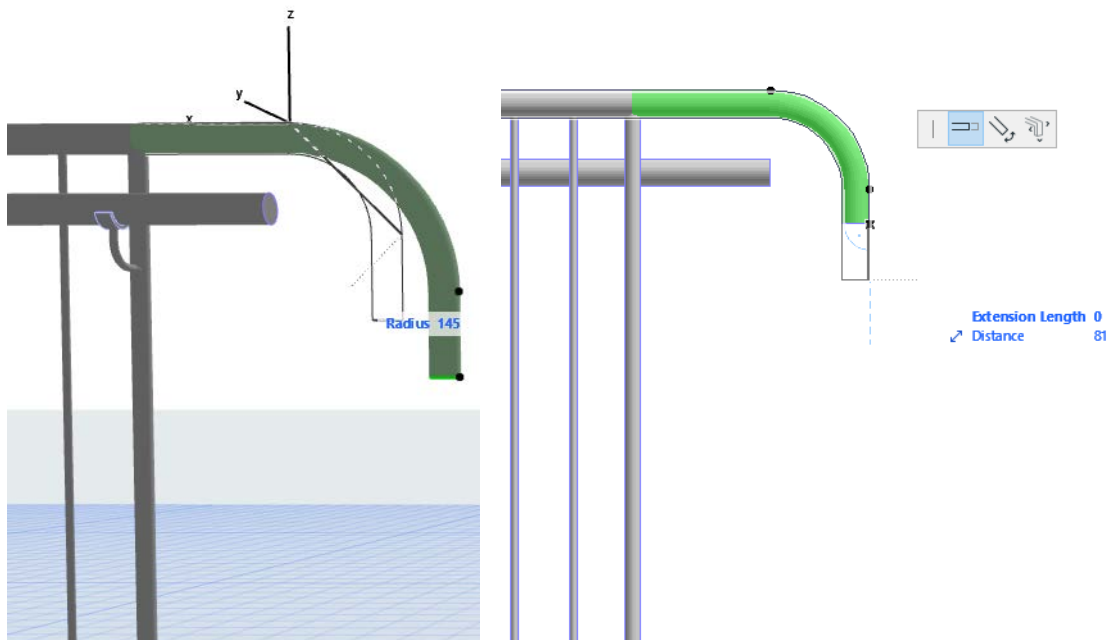
- Only a single extension is available for plain Rails.
- For doubled handrails: Use a fourth option to connect the handrail with its twin on the opposite side.

[See *Connect Ends of Double Handrails*.](#)

Edit Rail Ends

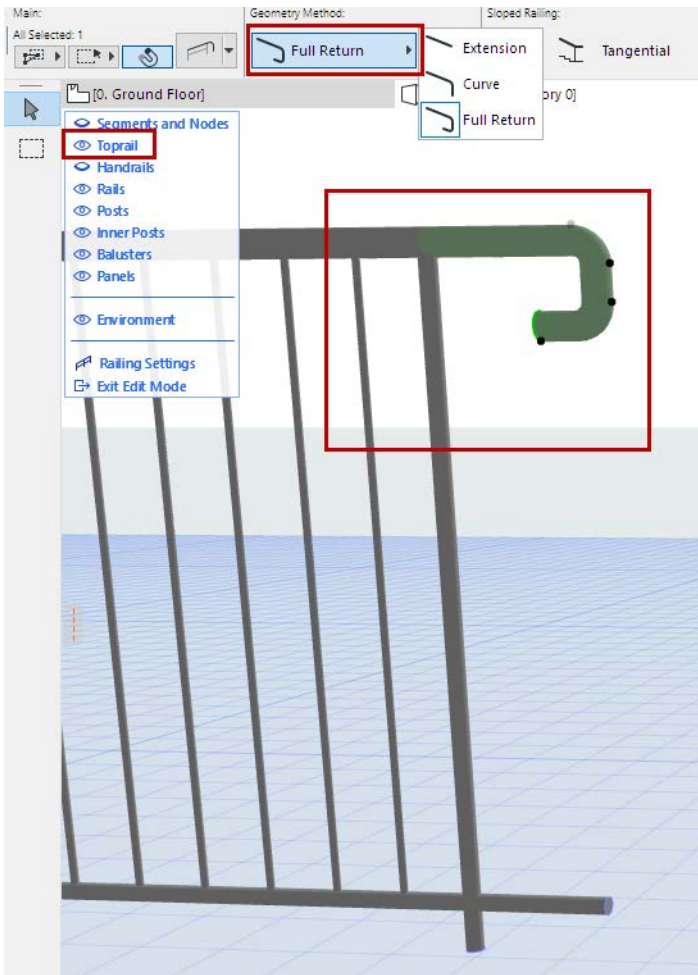
The End Settings defined in Railing Settings are applied to both ends of the Rail (Toprail/Handrail), but you can customize either end in Edit mode.

- Edit the endings graphically in Edit mode, using the hotspots to edit Extension length, Radius, and/or angle of the selected Ending. Where a node has multiple editing possibilities, the pet palette appears with the options:



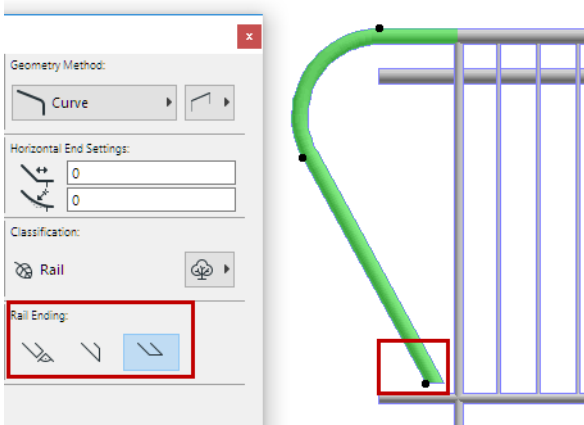
Edit Rail Endings Graphically in 3D and Section

- Use the Info Box or End Settings to adjust settings of any selected Rail ending.

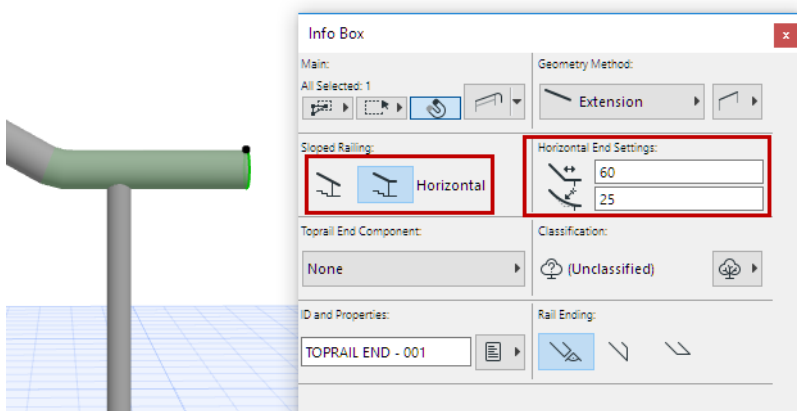


Sloped Railing: Ending Geometry

In Info Box or Railing Settings, choose the ending geometry of a Sloped Railing: perpendicular, vertical, or horizontal.



Sloped Railing End: Tangential to Segment or Horizontal



- Switch this option to end a sloped Railing with a horizontal segment.
- Enter the length of the added segment and an optional fillet.

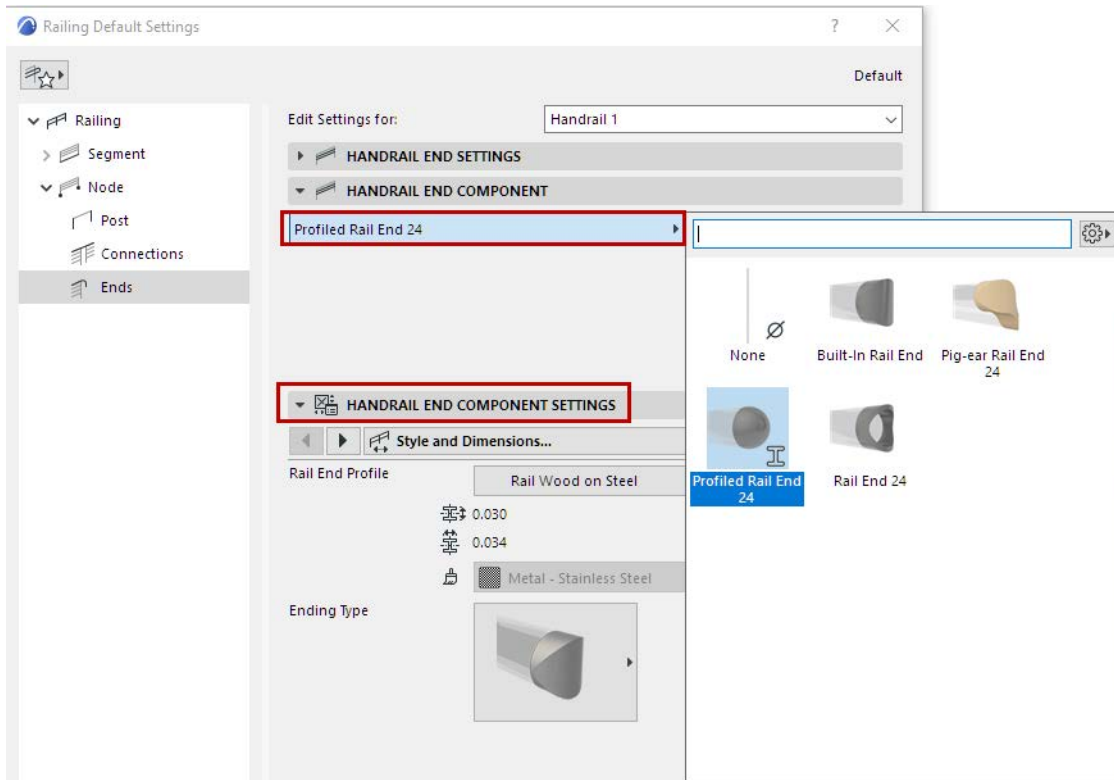
Override Surfaces

Display the end component using a different surface.

Add Ending Component to Rail End

The End component is added to the rail's Extension segment.

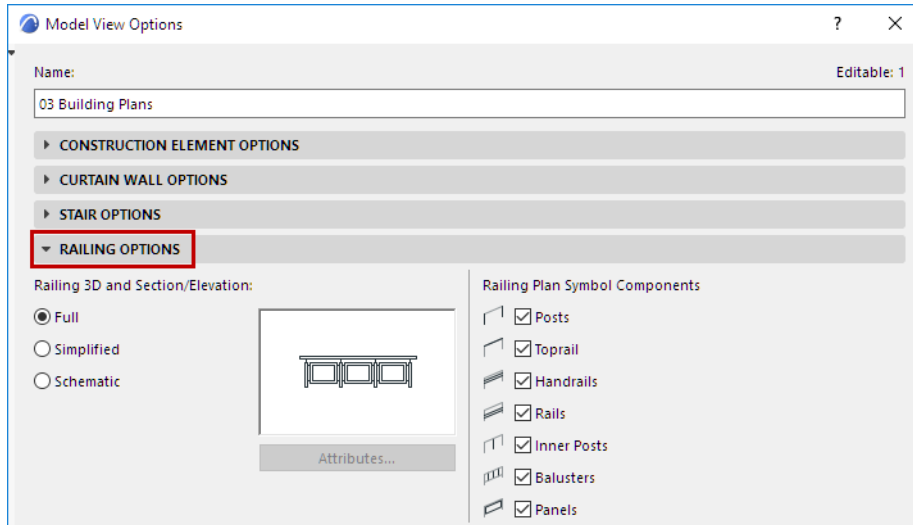
1. From the **End Component Panel**, choose the desired Rail End object if needed.
2. Use **Rail End Component Settings** panel to set its parameters.



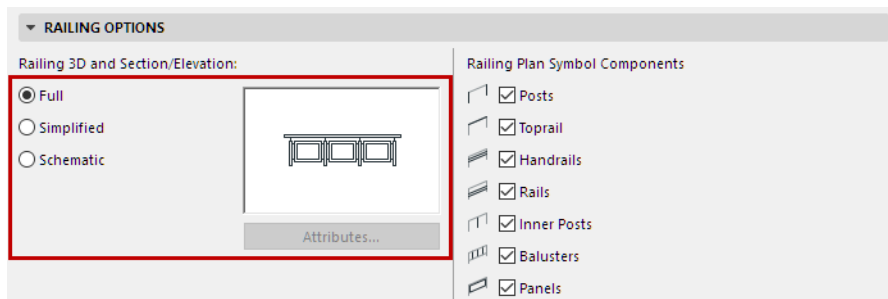
Model View Options for Railings

Railing Options

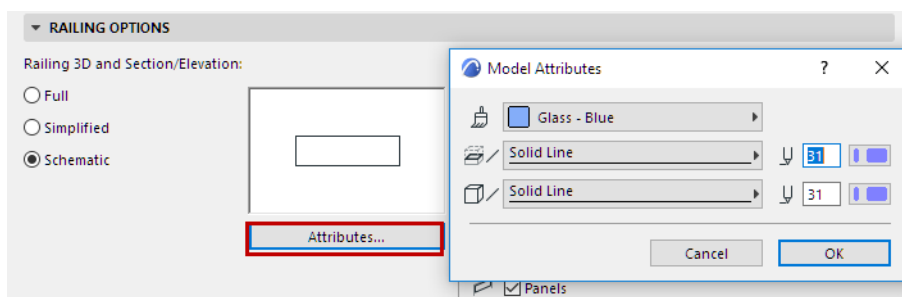
This panel is at **Document > Model View > Model View Options**.



Railing Detail Level in 3D and Section

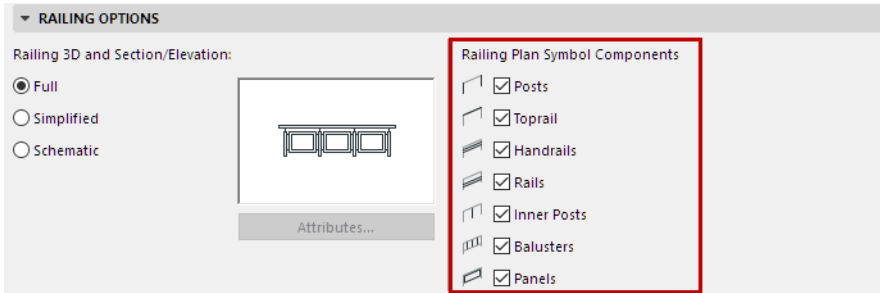


- In Full mode, the complete Railing appears: all 3D elements can be seen in full detail.
- In Simplified mode, the complete Railing appears: all 3D elements can be seen, but in a simplified view: bounding box of sub-elements are shown.
- In Schematic mode, a continuous membrane surface is shown to indicate the path of the Railing.
 - For Simplified or Schematic: Click **Attributes** to customize surface and line options for the Railings displayed in 3D and Section views.



Show/Hide Railing Floor Plan Symbol Components

Use the checkboxes to show/hide the Railing's Floor Plan symbol components.



To show/hide individual Railing Symbols, use Railing Settings.

[See Railing Floor Plan Symbol Components.](#)

To fine-tune Symbol display (based on a Railing's Visible/Hidden parts), use the "Detail Level" panel of the Model View Options dialog box:

[See Detail Level of Stair and Railing Symbols.](#)

Morphs

Use the Morph tool to create freeform elements within Archicad, so you don't have to import special shapes from other programs.

The Morph tool is fully integrated into Archicad, and works with a familiar logic and interface. Like other construction elements, a Morph's essential structure derives from its Building Material.

You can enable priority-based connections between Morphs and other elements.

[Watch the Video](#)

Morph Geometry and Sub-Elements

- Compared to traditional construction elements, the Morph has practically no geometric limits: every edge and every surface can be moved and shaped in any direction, so you can create exactly the element you need to fit project-specific situations.
- Morphs consist of one or more sub-elements: these can include faces and/or edges. For any Morph, you can edit the entire element as a whole, or any of its sub-elements separately, or in combination. A new selection logic is available to enable you to edit sub-elements.
- A selected Morph has only a single hotspot, which represents the point of its elevation value.

Morph Creation

- In addition to familiar input methods, the Convert Selection to Morph command enables the quick creation of a new single Morph out of existing elements.
- The Magic Wand is especially powerful with Morph creation: tap any surface (e.g. a Mesh) to create a Morph.
- Unlike other construction elements, 2D Morph shapes can be created in the 3D window (lines and planes). These can be used for sketching or documenting purposes.

Morph Settings

- In Morph Settings, you will notice that there are no numerical geometric parameters, apart from its default elevation.
- The Floor Plan and Section controls are similar to those of the Shell Tool. By default, Morphs (as well as Shells) are displayed on the Floor Plan in true 3D projection mode.
- Morph Selection Settings varies depending on whether you have selected an entire Morph, or any combination of its sub-elements (e.g. faces, edges). In sub-element selection, only the Model panel is available, and only the relevant controls are available, depending on whether you have selected edges, faces or a combination thereof.

Graphical Editing

- Morph creation and editing is most intuitive in the 3D window. Graphical interface uses familiar methods and interactions: select any edge, grab any point or surface, or any combination of their sub-elements, to modify the Morph with the help of the pet palette and real-time feedback. Notice that Fillet and Chamfer are available for Morphs in the 3D window, applied to any or all edges or corners.
- Morphs (like Shells) can be input onto any plane of the 3D window, and with many editing interactions, you can also switch the editing plane as you go, so that there are few constraints.

- There are separate Solid Operation commands for Morphs. Unlike Solid Element Operations, which create connections, Solid Morph Operations are not associative - the final result of the operation is permanent.
- Morphs can participate in priority-based connections with other elements, provided that you merge them using the Design > Connect > Merge Elements command.

Morph Smoothing

Morph surfaces are defined by faces, and as you edit the Morph, its many segments are automatically smoothed to create curved surfaces. In addition, a number of commands allow you to transform the Morph with a single click, by smoothing or subdividing its sub-elements, and also gives you some control over the smoothing mechanism. These commands and controls include:

- Smooth & Merge Faces
- Curve & Merge Edges
- Modify Segmentation
- Hard or Soft Edges: in the Info Box or Morph Settings, any edges can be individually re-set to be “hard” or “soft”. By default, they are hard; if you reset an edge to be “soft,” then its adjacent faces will behave as though merged into a single, flexible face. A soft edge is an internal sub-element that divides segmented faces. Such faces are displayed (in Hardware acceleration and rendering) as smooth and curved.

Texture Mapping for Morph Faces

As with other 3D elements, if you have applied a surface that includes a texture, you can use the Align 3D Texture command to fine-tune the origin and rotation of the texture pattern. For the Morph, these options are also available individually and separately for each face.

Related Topics:

[Morph Selection](#)

[Create a 2D Morph](#)

[Create a Revolved Morph](#)

[Create a Box Morph](#)

[Create Morph from Existing Elements](#)

[Basic Morph Editing](#)

[Useful Commands for Transforming Morphs](#)

[Morph Edge Type: Hard, Hidden, Soft](#)

[Boolean Operations with Morphs](#)

[Morph Texture Mapping and Alignment](#)

[Fix Modeling Errors: Check and Solidify Morphs](#)

[Pick Up/Inject Morph Parameters](#)

[Merge Elements: Roofs, Shells, Morphs](#)

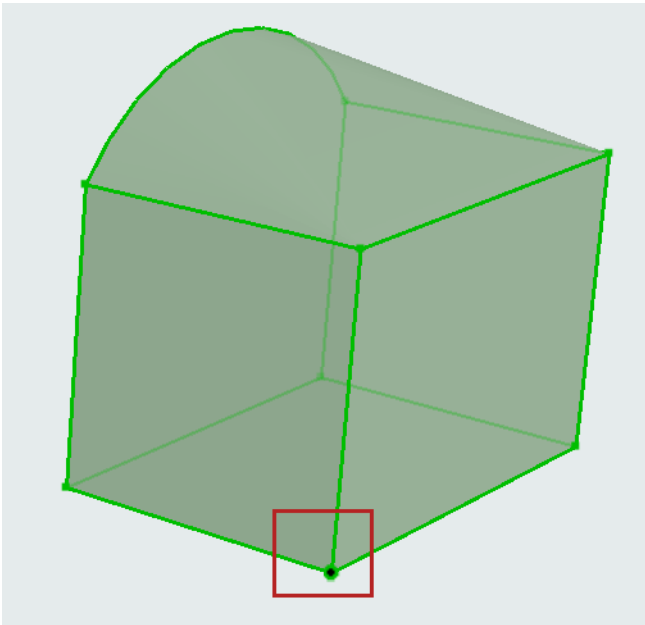
Morph Selection

Morphs consist of multiple sub-elements (faces and edges).

Select Entire Morph

- Click on any part of the Morph to select the entire Morph element (including its sub-elements).

A selected Morph has only a single hotspot, which represents the point of its elevation value; but all other nodes/corners are also editable.

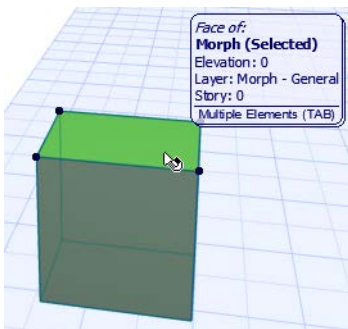


- Check the Info Tag to see the sub-elements that make up the selected Morph.

Area selection will select the entire Morph.

Select Morph Sub-Element

Use Ctrl+Shift to select just a sub-element (an edge or a face) of the Morph. (The cursor changes to a white arrow to indicate that you are selecting a sub-element.)



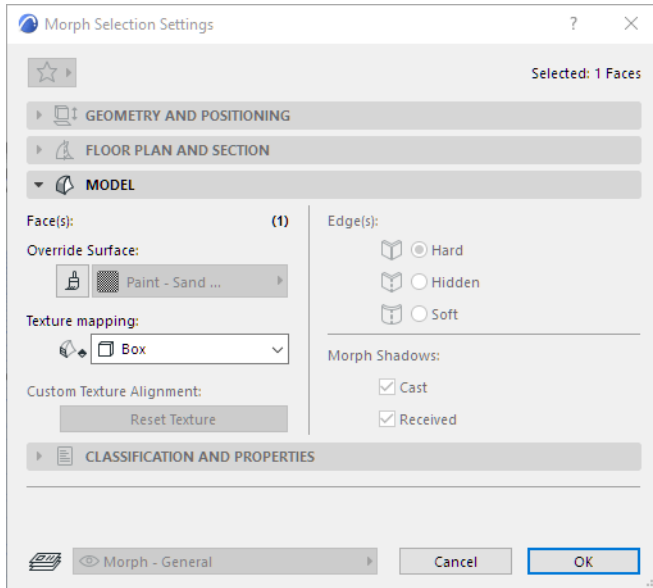
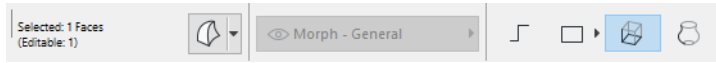
With Ctrl + Shift pressed:

- Pre-selection highlight shows you exactly what will be selected when you click
- Use the tab key to cycle among the pre-selected sub-elements

When a sub-element is selected, all the pet palette commands will work on that sub-element only.

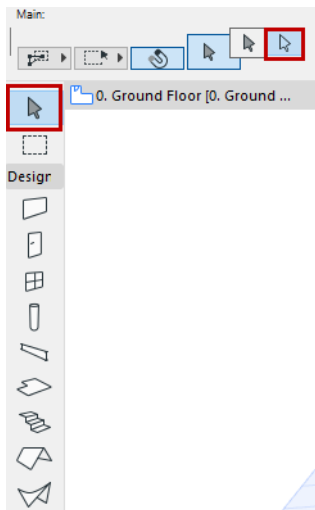
For example, the Move command on a selected face will move the face in space (and the connected parts of the rest of the Morph along with it) - it does not move the whole Morph element.

Sub-element selection is also reflected in the Info Box and Morph Selection Settings.

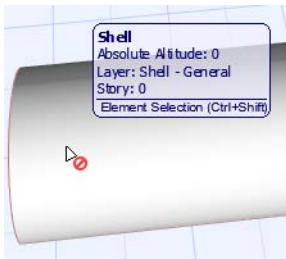


Sub-Element Selection by Default

If you are editing Morphs, you may wish to select only Morph sub-elements (faces and edges) by default, rather than the entire Morph. To do this, activate the Sub-Element selection method for the Arrow tool.



This only works for Morph selection. If Sub-Element selection is on, and you try to select other element types, you get a "Do Not Enter" cursor, and selection is not possible unless you use the modifier key (Ctrl + Shift).



If you have activated Sub-Element Arrow selection by default, then the usual selection logic is reversed: sub-elements are selected by just plain clicking, while to select an entire Morph, you have to use the modifier key (Ctrl + Shift).

Change the Sub-Element Selection Modifier Key

Use the Work Environment's Shortcut page to assign a different modifier key for the "Sub-Element Selection Modifier" command, which toggles between entire element/sub-element selection of Morphs.

Create a 2D Morph

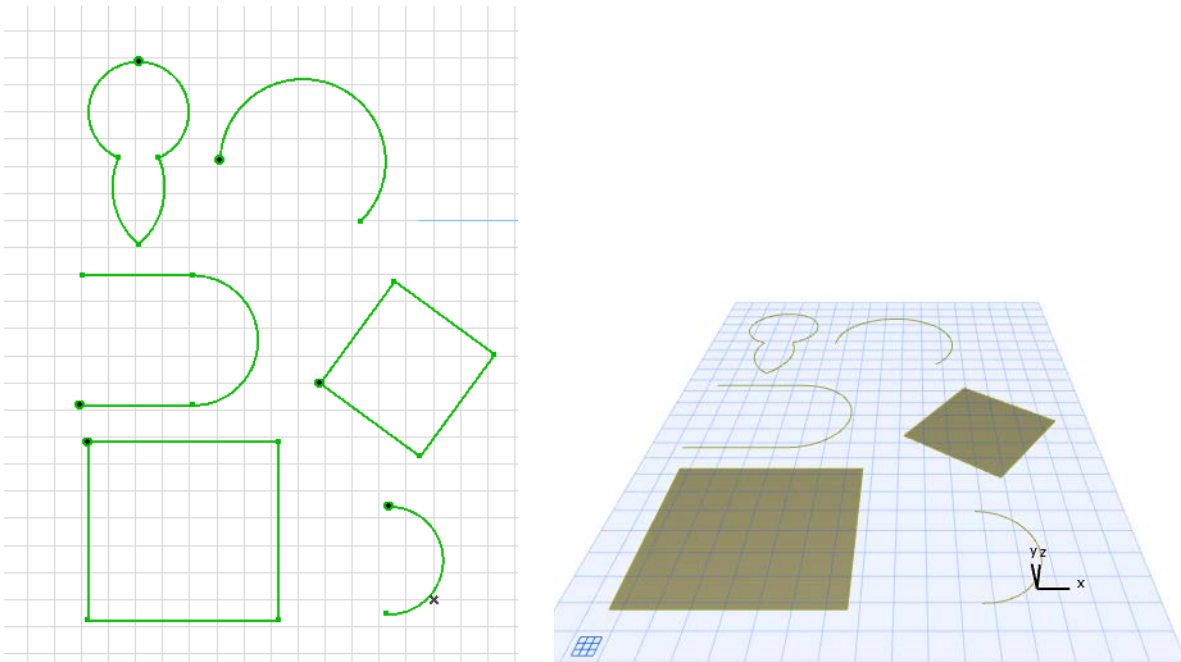
Choose the Morph Tool from the Toolbox, then choose a Geometry Method from the Info Box.



Use the following methods to create 2D lines, polylines, curves and closed polygons on the Floor Plan or in the 3D Window.

- Polygonal
- Rectangular and Free-Rotate Rectangular
- Curved - Centerpoint and Radius
- Curved - Circumference

The input process with these geometry methods works just like Archicad's 2D drawing tools.



The result, however, is a Morph face or a Morph line, rather than a 2D polygon or line: it is displayed in the 3D window, but has no geometric parameters (e.g. thickness). You can use the resulting 3D lines and surfaces for sketching or documenting purposes in the 3D window.

Free-Rotate Rectangle

The Free-Rotate Rectangle input method of the Morph tool allows you to draw the rectangle on any plane in space.

1. In the 3D window, choose the Free-Rotate Rectangle method.
2. Click twice to draw the first edge of the rectangle. You can draw this edge on any plane in space.
3. Click to place the opposite corner of the rectangle. You can snap to any point in space.

Create a Box Morph

Activate the Morph Tool and choose the Box geometry method.

In the 3D Window:

1. Choose the input plane. Either click to find the default plane, or choose another input plane based on existing elements or using the Editing Plane controls.

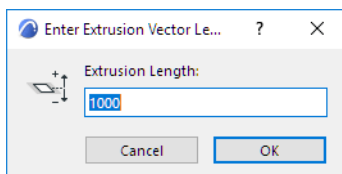
See [Reposition Editing Plane](#).

If you click in “empty space”, the default input plane is at the base height defined in Morph Default Settings.

2. Click twice to define the rectangular base of the Box Morph. (The base is always aligned with the grid. You can rotate the grid to place the box at a different angle, or you can place it and then rotate it afterwards.)
3. Drag the cursor to define the height of the Box Morph.
4. Click to complete.

On the Floor Plan or in Section:

1. Choose the Box geometry method from the Info Box.
2. Click twice to define the rectangular base of the Box Morph.
3. Enter an extrusion vector length - the distance that will be the height of the Box Morph.



Create a Revolved Morph

Use the Revolved method to create a Morph by defining its profile (either a default shape or a freehand profile), then revolving the profile around an axis.

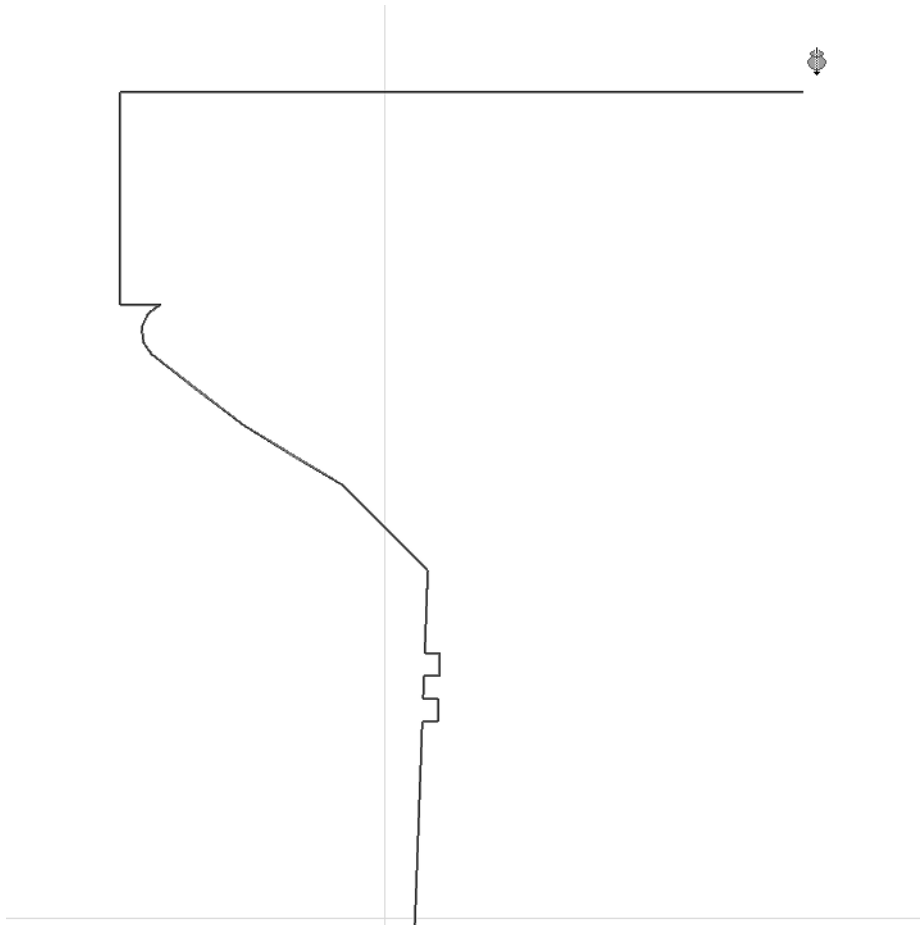
Activate the Morph Tool and choose the revolved geometry method.

On the Floor Plan or in Section:

1. Draw the Morph profile - a polyline or a closed polygon - with the usual polygon input methods. (The resulting Morph will have as many faces as the number of polygon segments.)

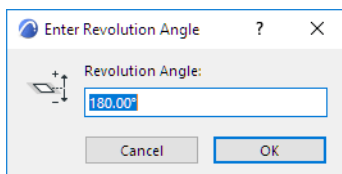
Note: You can use the Magic Wand to trace any closed polygon as your profile.

2. Double-click to complete the polyline or polygon.

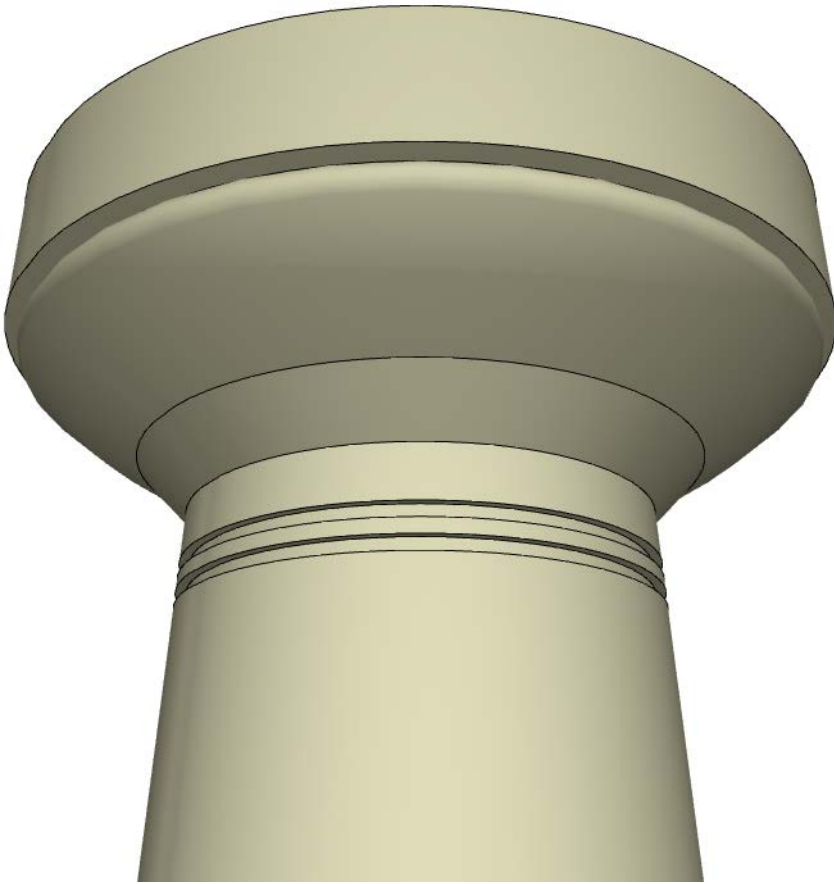


3. Click twice to draw the revolution axis.

4. In the appearing dialog box, enter the Morph's revolution angle. A positive number will revolve the Morph toward you. A negative number will revolve the Morph away from you.



5. Click OK to complete the Morph. View it in 3D.



In the 3D Window:

1. Choose the input plane. Either click to find the default plane, or choose another input plane based on existing elements or using the Editing Plane controls.

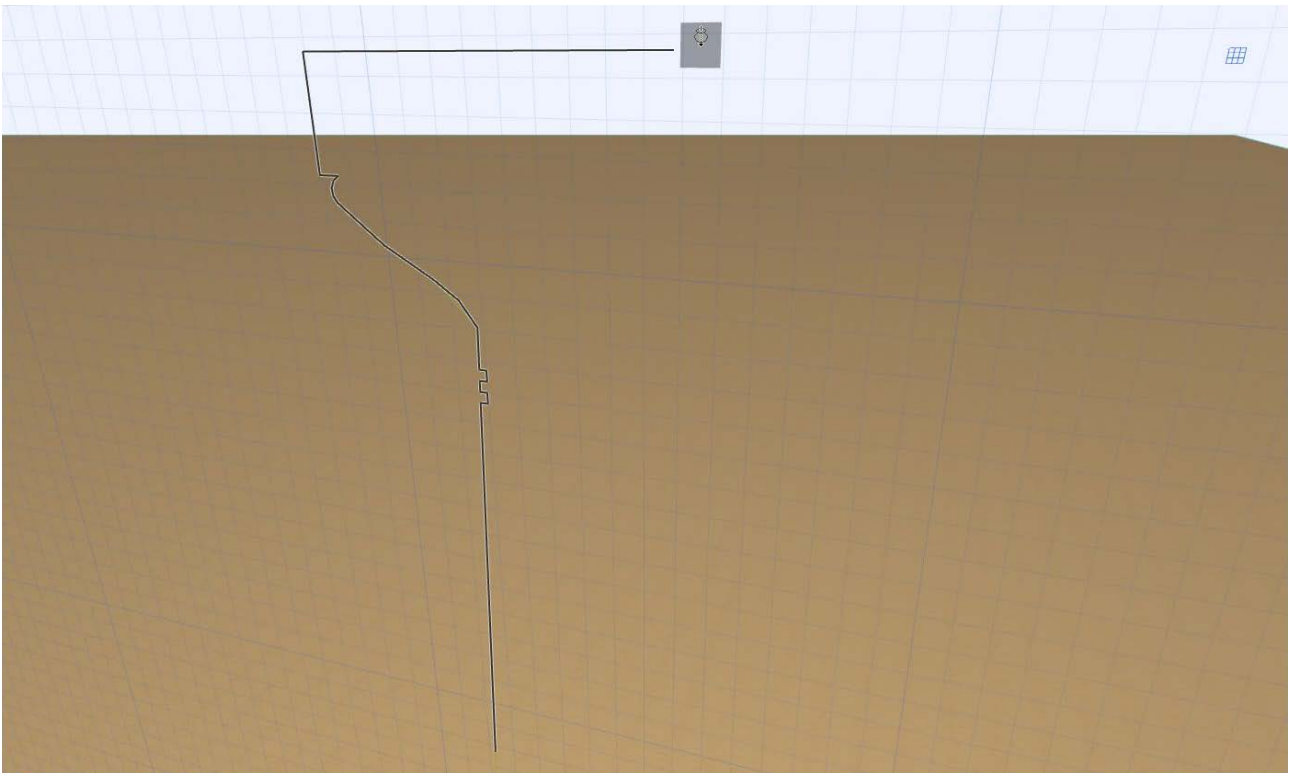
[See *Reposition Editing Plane*.](#)

This is the plane on which you will draw the profile and then its revolution axis.

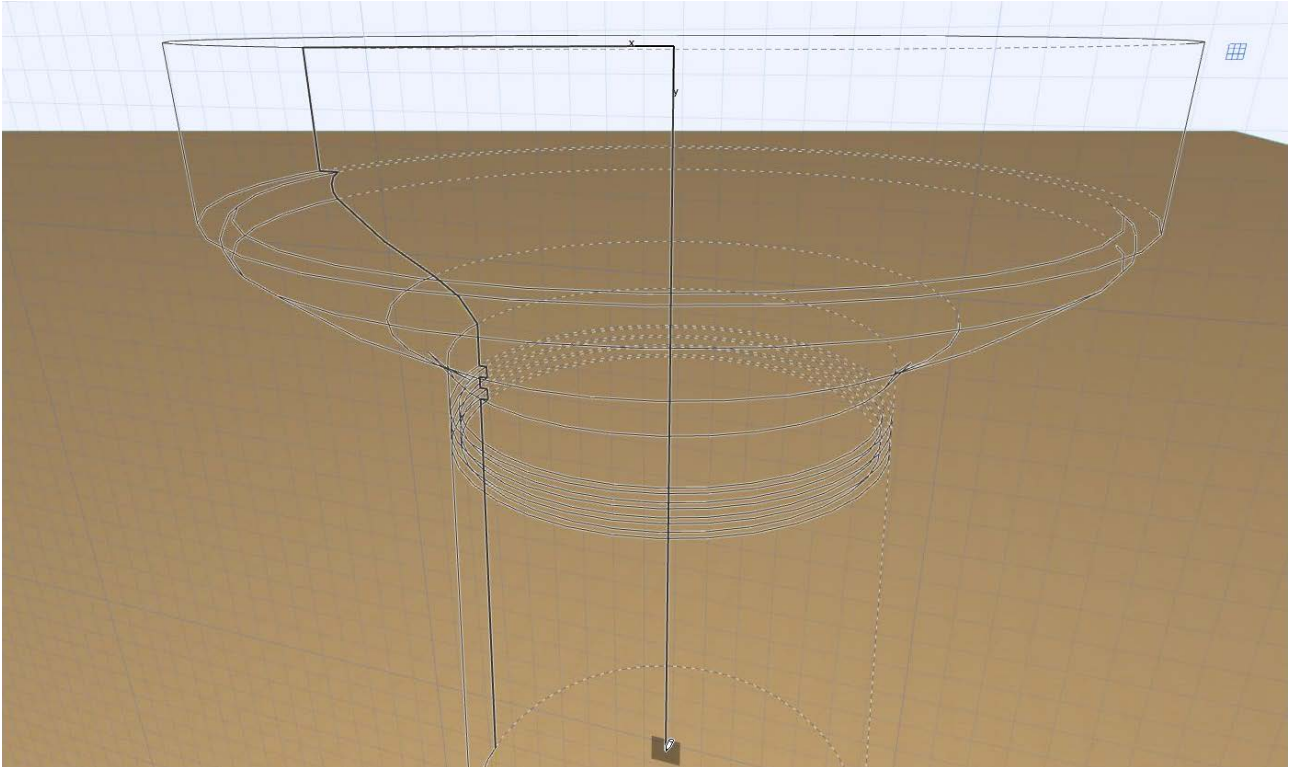
If you click in “empty space”, the default input plane is at the base height defined in Morph Default Settings.

2. Draw the Morph profile - a polyline or a closed polygon - with the usual polygon input methods.

Note: You can use the Magic Wand to trace any closed polygon as your profile.

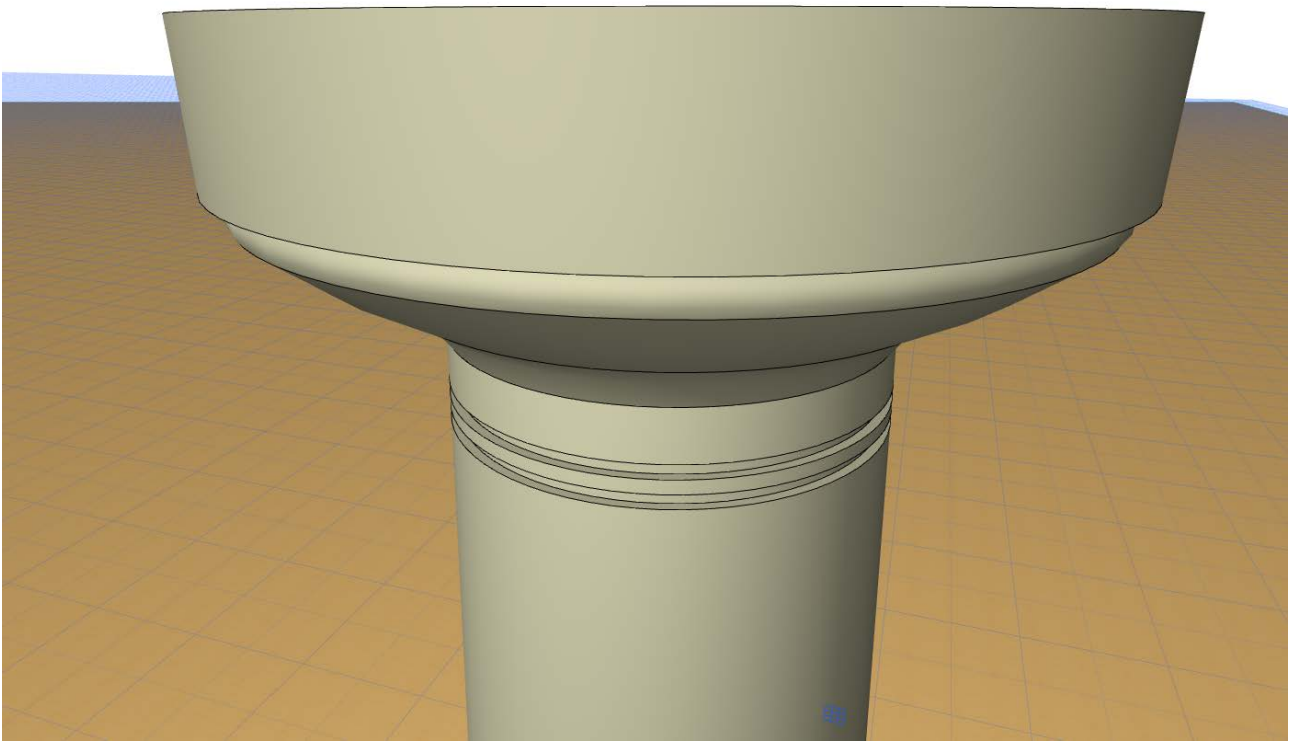


3. Complete the polygon.
4. Click twice to define the revolution axis.



5. Move the cursor to complete the revolution axis. (Navigate in the 3D view to find the best angle for seeing the feedback.)

6. The Morph is complete.



Create Morph from Existing Elements

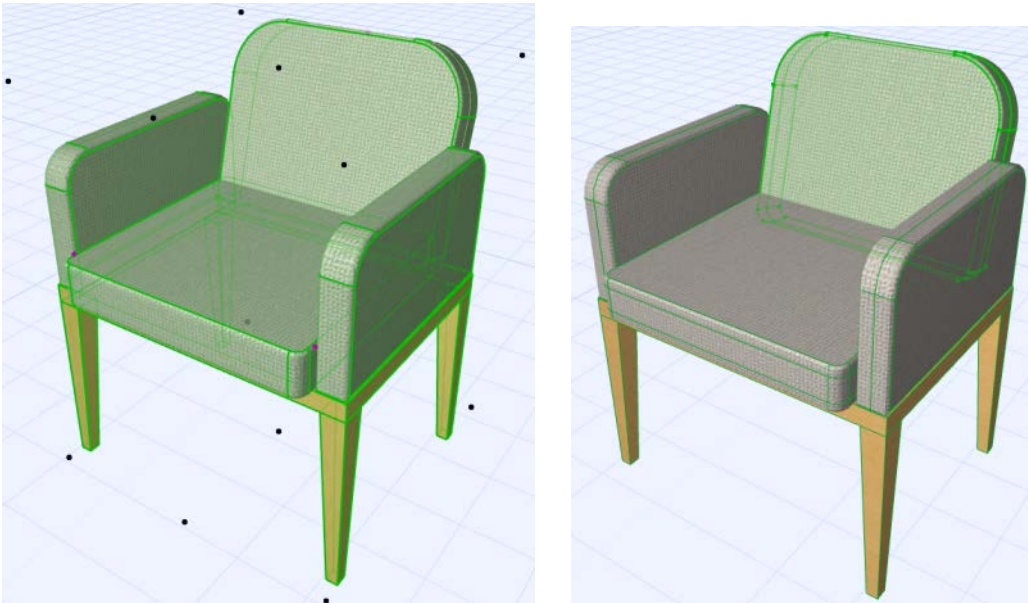
Convert Selection to Morph(s)

Use this command to create multiple Morphs out of selected elements: each selected element will become a separate Morph.

1. In any model window, select the elements to change into Morphs.
2. Use **Design > Convert Selection to Morph(s)**.
3. The selected elements, provided that they have a 3D representation, will be turned into Morphs.

[Watch the Video](#)

Example: Convert GDL Object to Morph



When you convert a GDL object to Morphs:

- GDL objects will be converted into independent Morphs.
- GDL sub-element components of other elements (e.g. door, window, wall end, skylight) will be converted into independent Morphs themselves.
- Similarly, the sub-elements of complex roofs and curtain walls are converted into independent Morphs.

Note: If the resulting Morph is very complex, you may run into editing and performance problems.

Notes on Attributes of Newly Created Morphs:

- If the transformed element was an object, its surface is added to the project's set of surface attributes, if the surface doesn't exist there already. Such a newly added surface attribute automatically receives a suffix of `_from Converted Object`, though this string is not considered when the program checks for duplicate names. (If the converted object's surface already exists in the project, the new Morph uses the existing surface.) Any texture attached to the surface is added to the embedded library.
- In Teamwork, creation of Surfaces and Fills requires the appropriate access right. If you do not have this right, you will see a warning message. The Morph will still be created, but the surface/texture will not be added to the project, and the Morph may thus look different than expected.

Basic Morph Editing

When you work with the Morph tool, you will typically place a standard shape, then graphically edit it into its final form.

Select the entire Morph, or select one of its sub-elements.

You can edit the Morph graphically using one of its sub-elements (face or edge). When you drag a face or edge, all of its connecting faces and edges move along with it.

The best way to learn this is to simply try out the editing options from the pet palette, using the 3D window. Don't forget that, in many situations of Morph editing, you can reset the editing plane to any position.

Basic Morph Editing: Push-Pull

Basic Morph Editing: Bulge

Basic Morph Editing: Add

Basic Morph Editing: Tube

Transforming Morphs: Custom Face Settings

Morph Edge Type: Hard, Hidden, Soft

Basic Morph Editing: Offset Morph Edge(s)

Basic Morph Editing: Curve Edge

Basic Morph Editing: Move Morph Sub-Element

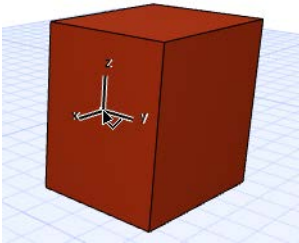
Basic Morph Editing: Add Node to Morph

Basic Morph Editing: Fillet/Chamfer a Morph

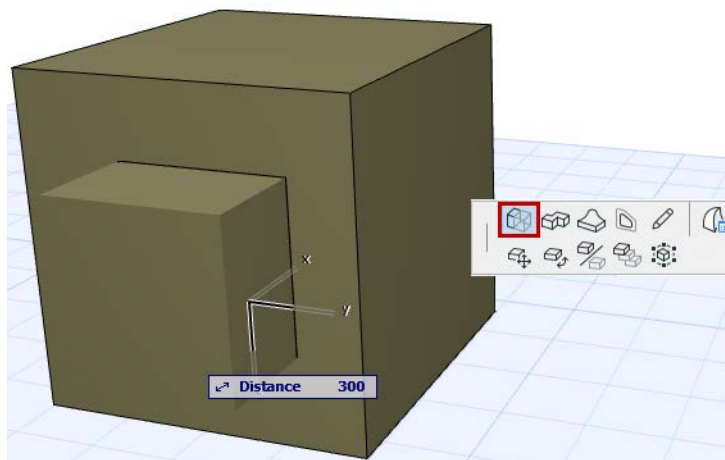
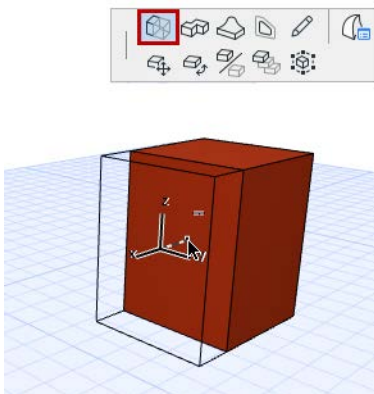
Custom Resolution for Fillet

Basic Morph Editing: Push-Pull

1. Click on the flat face of a Morph. (Multi-select sub-elements to edit them simultaneously.)
2. From the pet palette, choose the **Push-Pull** command.



3. Drag the face (the one you clicked) to extend the Morph shape perpendicularly to the clicked face: pull to extend the Morph; push to “compress” it.



Create a Morph-Shaped Hole in Another Morph

This method works if you push a Morph all the way through another Morph, provided that both ends of the “host” Morph are parallel faces.

For other kinds of geometries, use Boolean Operations on Morphs.

[See Boolean Operations with Morphs.](#)

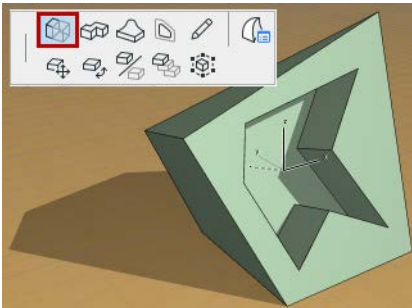
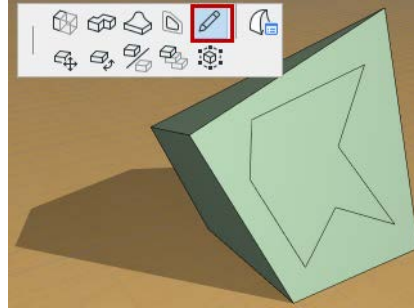
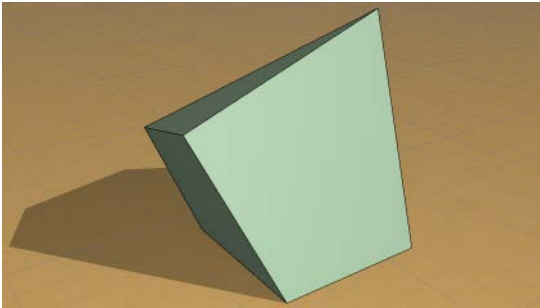
1. On an existing Morph, use the Add command to draw a new Morph in the shape of the hole you want.

[See Basic Morph Editing: Add.](#)

The new Morph becomes a sub-element of the existing Morph (since you used the Add command to draw it.)

2. Click on the sub-element.
3. Use Push-Pull to push the sub-element through the Morph.
4. Click the cursor on an opposite edge to complete the Push.

5. The hole is created.

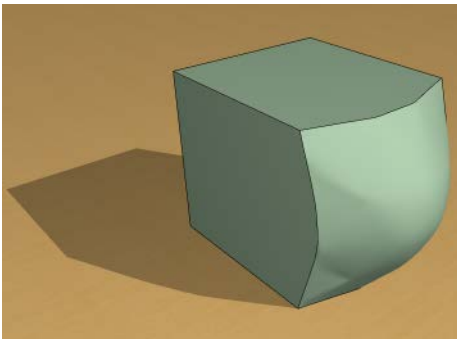
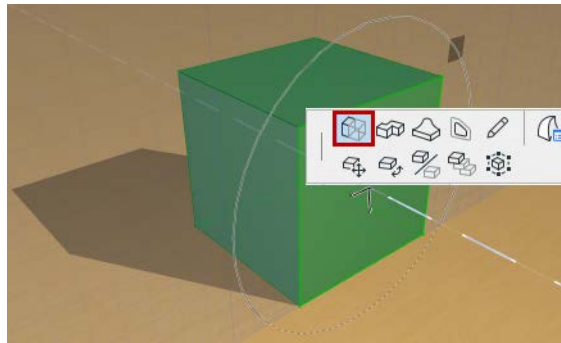
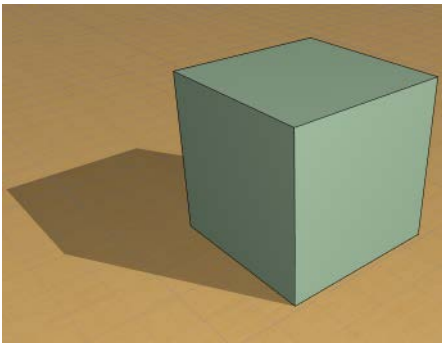


Basic Morph Editing: Bulge

1. Click on any face of a Morph.

Note: Choose the clicked point carefully, because it will be the centerpoint of the circle out of which you will draw the bulge.

2. From the pet palette, choose the **Bulge** command.
3. Drag the cursor to define the radius of the bulge.
 - A small radius will give you a smaller, sharper outcropping.
 - Use a larger radius for a more gentle rounded result.
4. Click to complete the radius.
5. Now drag the cursor to bulge the Morph in or out.

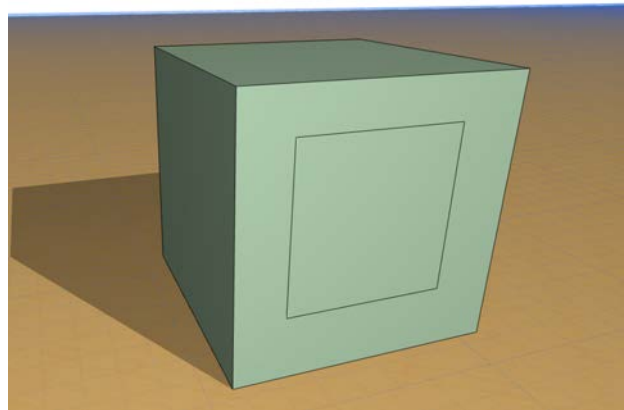
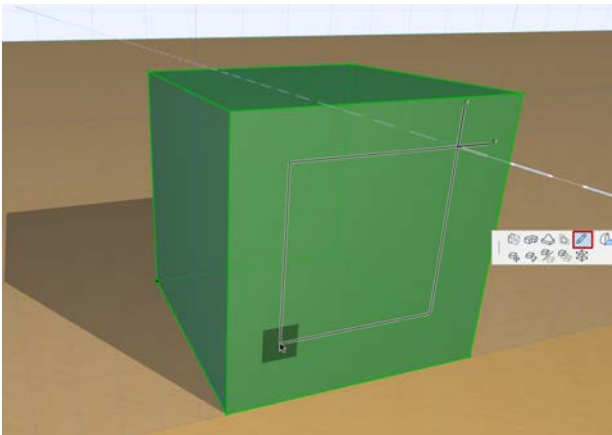
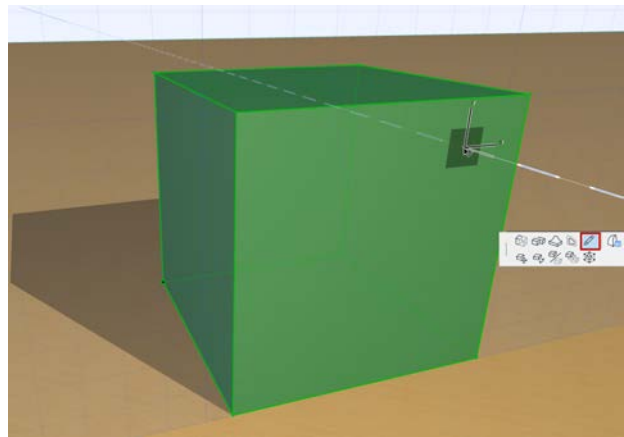
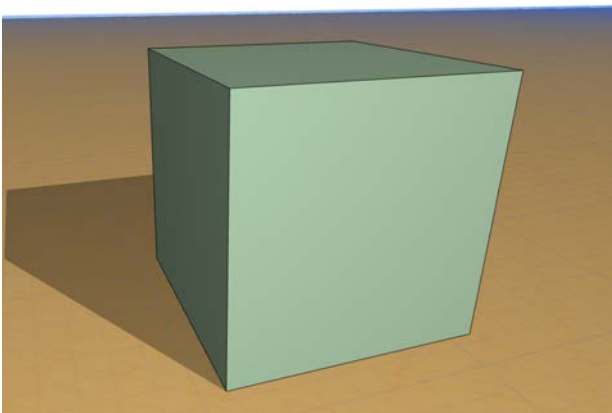


Basic Morph Editing: Add

Use the **Add** command to expand an existing Morph by drawing an additional Morph shape onto it. The result will be a single Morph that behaves as a single element.

Note: Alternatively, you can draw a separate Morph onto an existing one, without using the Add command; in this case, the new Morph will operate independently of the old one. Later, if needed, you can use **Create Morph from Selection** or the “Union” Solid Morph Operation to merge these separate Morphs into one.

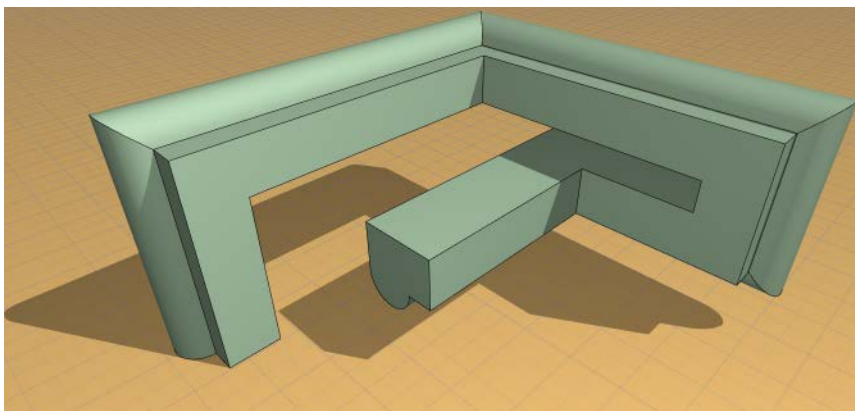
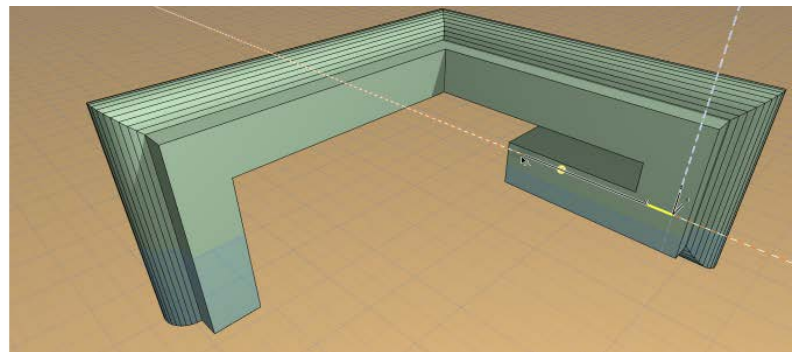
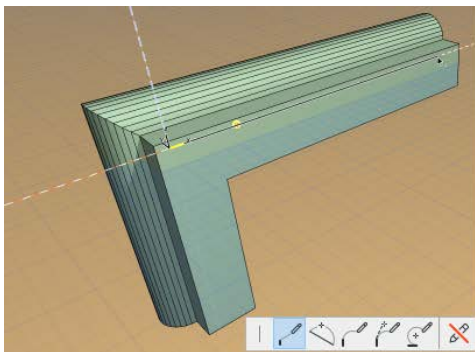
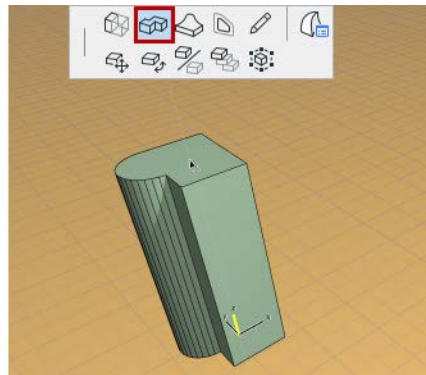
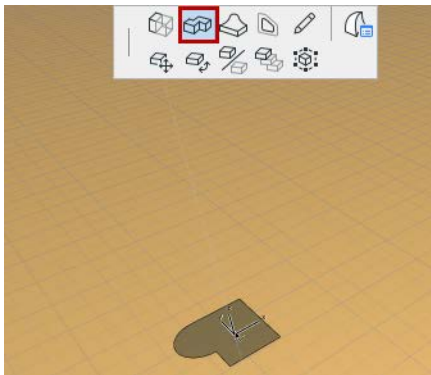
1. Click anywhere on a Morph or its sub-element. The new Morph input will begin at the point you clicked.
2. From the pet palette, choose the **Add** command.
3. Choose a geometry method from the Morph Info Box.
4. Draw the desired Morph.



Basic Morph Editing: Tube

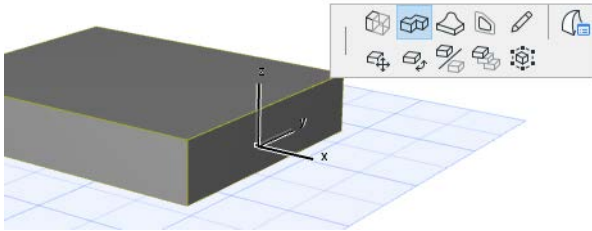
Use the **Tube** command to extend any Morph face into a tube, changing directions and planes as many times as you like.

1. Click anywhere on a Morph face.
2. From the pet palette, choose the **Tube** command.
3. Start drawing the Tube, clicking to place each segment in turn.
 - Note:** The first segment must be drawn perpendicular to the clicked face.
 - You can use the Magic Wand to trace a path for the Tube around a closed polygon (provided that the polygon plane and the tube profile plane are not in the same plane.)
 - Change the editing plane as needed.
 - To undo the last segment, use the Go back command from the pet palette.
4. Double-click to complete the Tube.

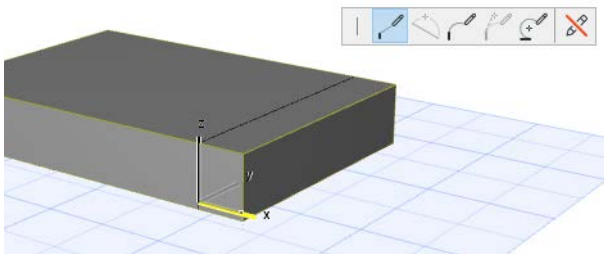


Tube Reference Point

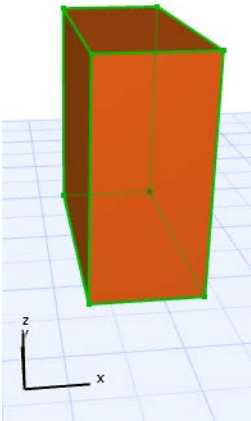
The point you clicked on the Morph face to bring up the pet palette is the reference point of the Tube (the point by which you drag the tube).



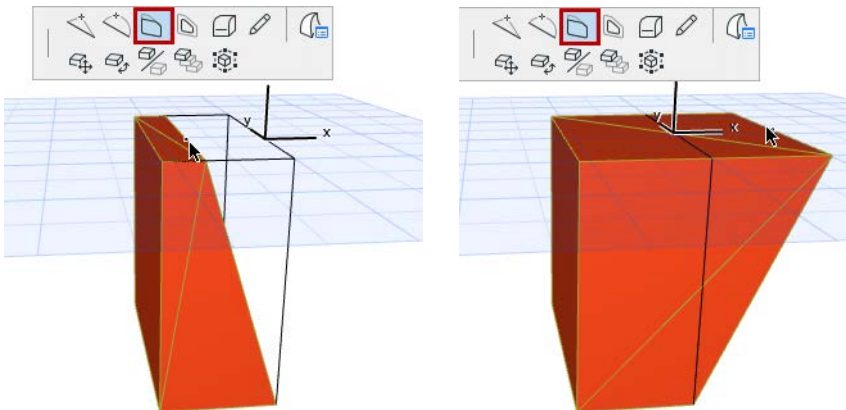
However, once you select the Tube command from the pet palette, you can click anywhere else on the face (or its contour) to use that newer clicked point as the reference point:



Basic Morph Editing: Offset Morph Edge(s)



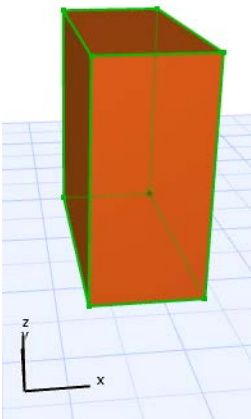
1. Select a Morph.
2. From any edge, bring up the Pet Palette and choose “Offset Edge”.
3. Drag the cursor, or choose a different editing plane. The Morph edge, plus its adjoining faces, will move along the chosen editing plane.



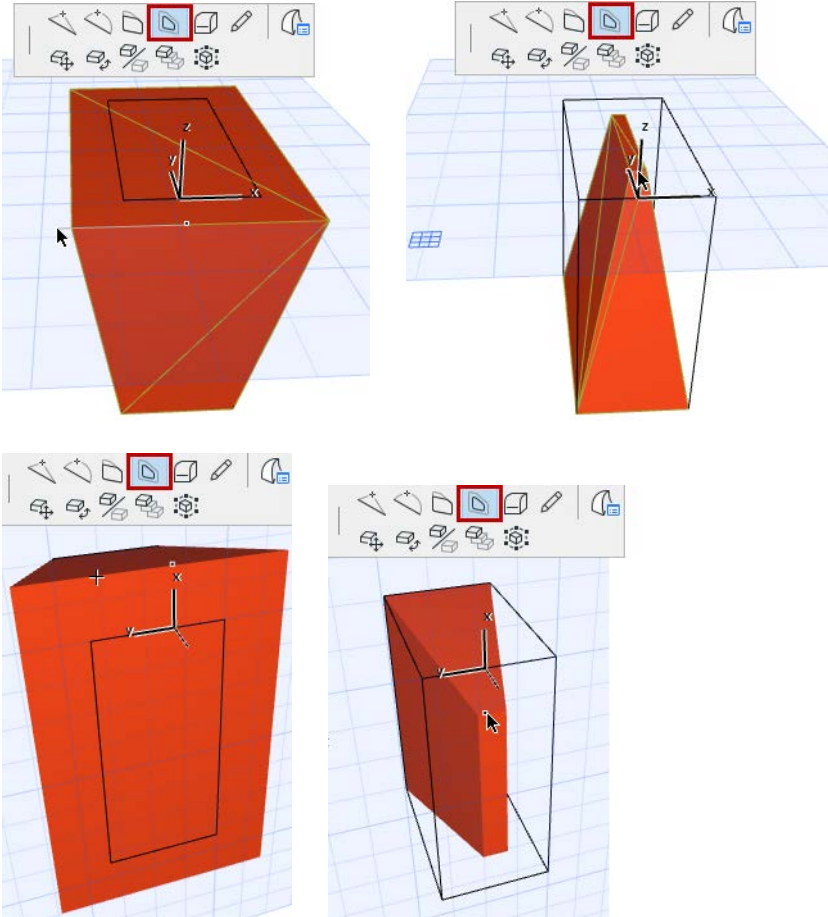
4. Click to complete.

Offset All Morph Edges

1. Select a Morph.

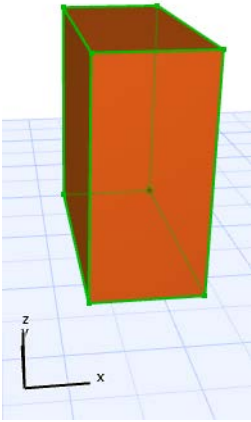


2. From any edge, bring up the Pet Palette and choose “Offset All Edges.”
3. Drag the cursor, or choose a different editing plane (only the adjoining planes are available). The face adjoining the clicked edge (the face that lies on the editing plane) will be offset on all sides along the chosen editing plane.
4. Click to complete.

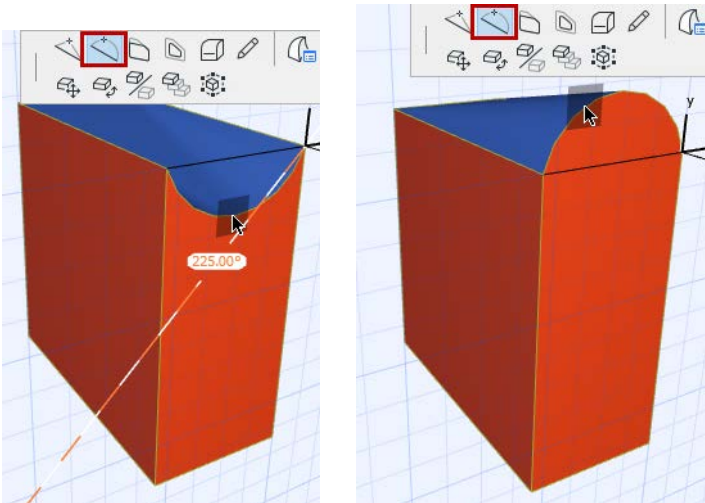


Basic Morph Editing: Curve Edge

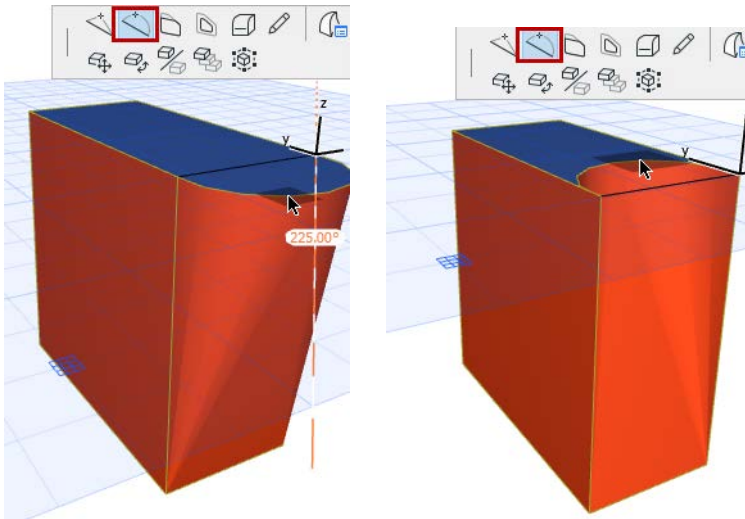
1. Select a Morph.



2. Click the edge you want to curve to bring up the Pet Palette, and choose “Curve Edge.”
3. Drag the cursor, or choose a different editing plane. The clicked edge, plus the adjoining face, will be curved along the chosen editing plane.



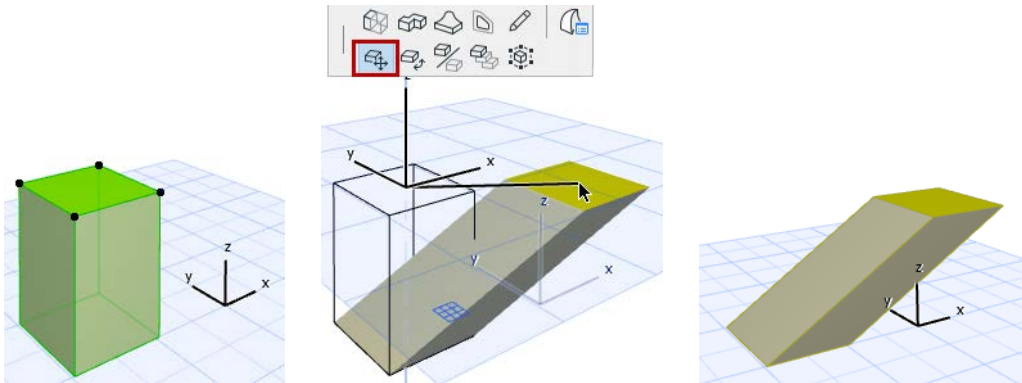
Here, the same operation done on a different Editing Plane:



Basic Morph Editing: Move Morph Sub-Element

Use this command to transform the Morph by dragging it by its sub-element(s).

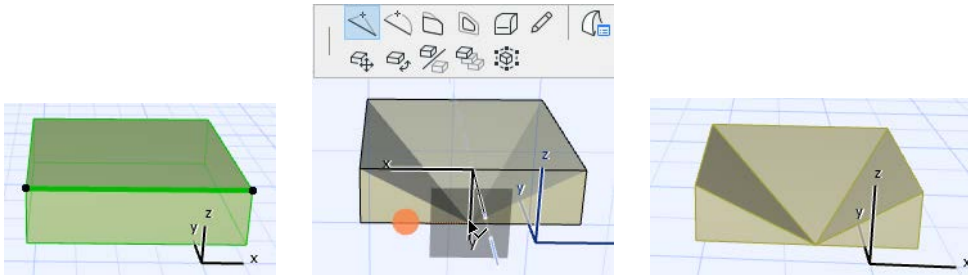
1. Select a sub-element.
2. Use the Drag command from the pet palette.
3. Move the sub-element (and its connected faces/edges) along the editing plane. Change editing plane if needed.



While moving, you can snap to the direction that is perpendicular to the editing plane.

Basic Morph Editing: Add Node to Morph

1. Bring up the pet palette on the edge of a selected Morph (or on a selected edge).
2. Use the **Add Node** command from the pet palette.
3. Move the node (and its connected faces/edges) along the editing plane. Change editing plane if needed.



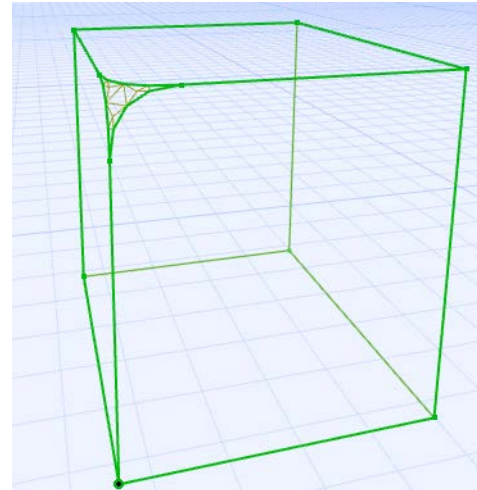
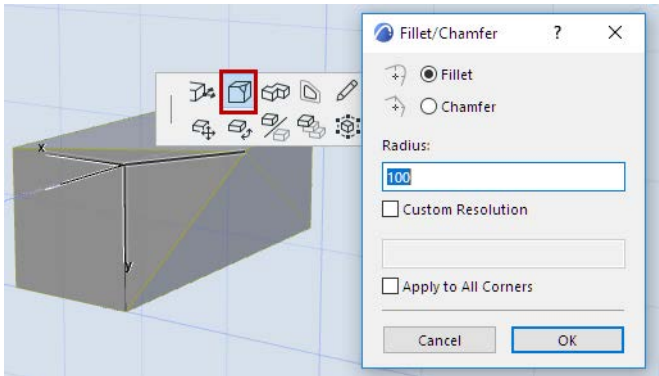
Basic Morph Editing: Fillet/Chamfer a Morph

You can fillet or chamfer the corners and edges of Morphs, in 2D or 3D windows.

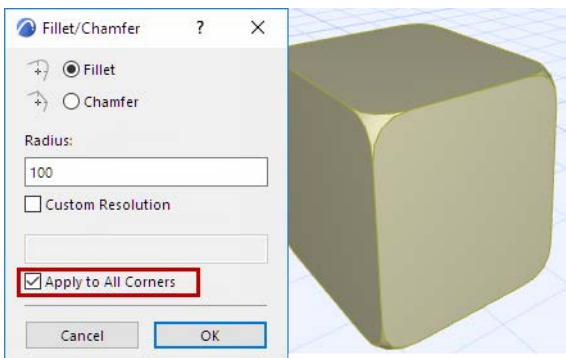
Note that a fillet or chamfer operation has the effect of created one or more new Morph faces. These new faces will always “inherit” the default surface of the original Morph.

Fillet/Chamfer a Morph Corner

1. Select a Morph and bring up the pet palette at one of its corners.
2. Choose the Fillet/Chamfer Corner command and apply the desired options.

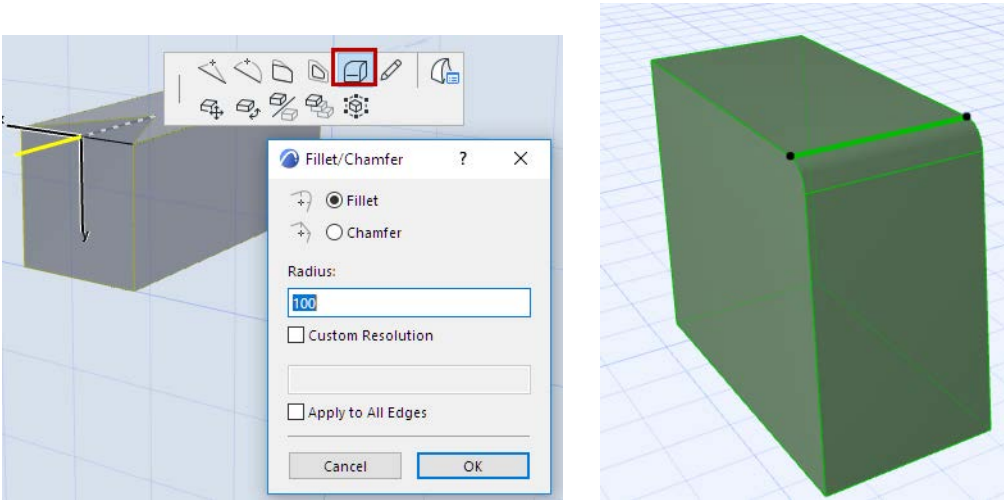


Check the box to apply to all corners of the Morph:

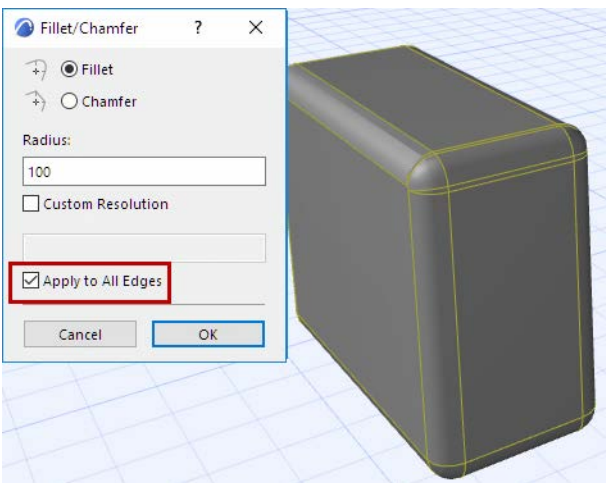


Fillet/Chamfer Morph Edges

1. Bring up the pet palette on the edge of a selected Morph (or on one or more selected edges).
2. Choose the Fillet/Chamfer Edge(s) command and apply the desired options.

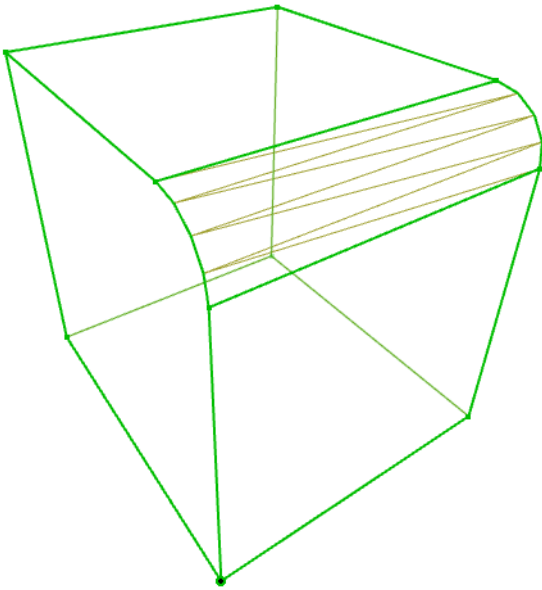


If you have selected an entire Morph, you can check the box to apply the fillet/chamfer to all edges:

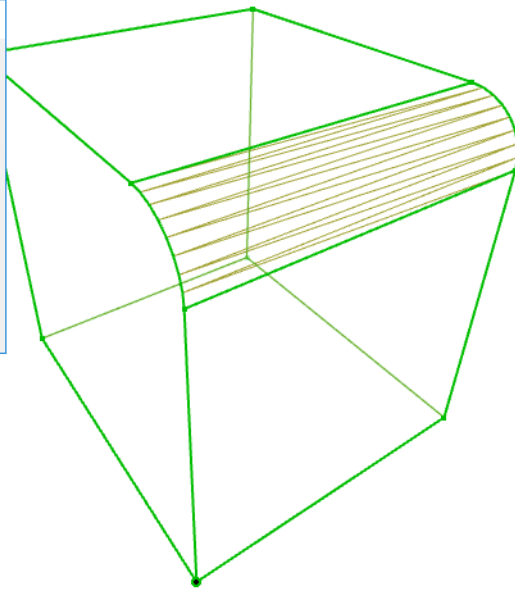
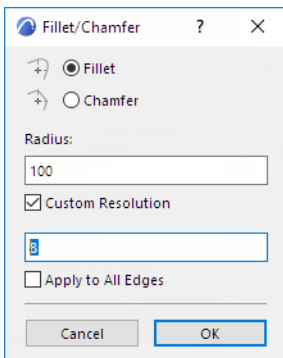


Custom Resolution for Fillet

By default, the Fillet resolution is set to 4 (this is best displayed in wireframe):



For different results, adjust the Fillet resolution up or down:



Useful Commands for Transforming Morphs

As you edit the Morph into the shape you need, try out the following commands:

Transforming Morphs: Smooth & Merge Faces

Intersect Morph Faces (Generate Edges within a Morph)

Transforming Morphs: Curve & Merge Edges

Combine Morph Faces for Smooth Editing Effect: Use “Soft” Edges

Transforming Morphs: Box Stretch a Morph

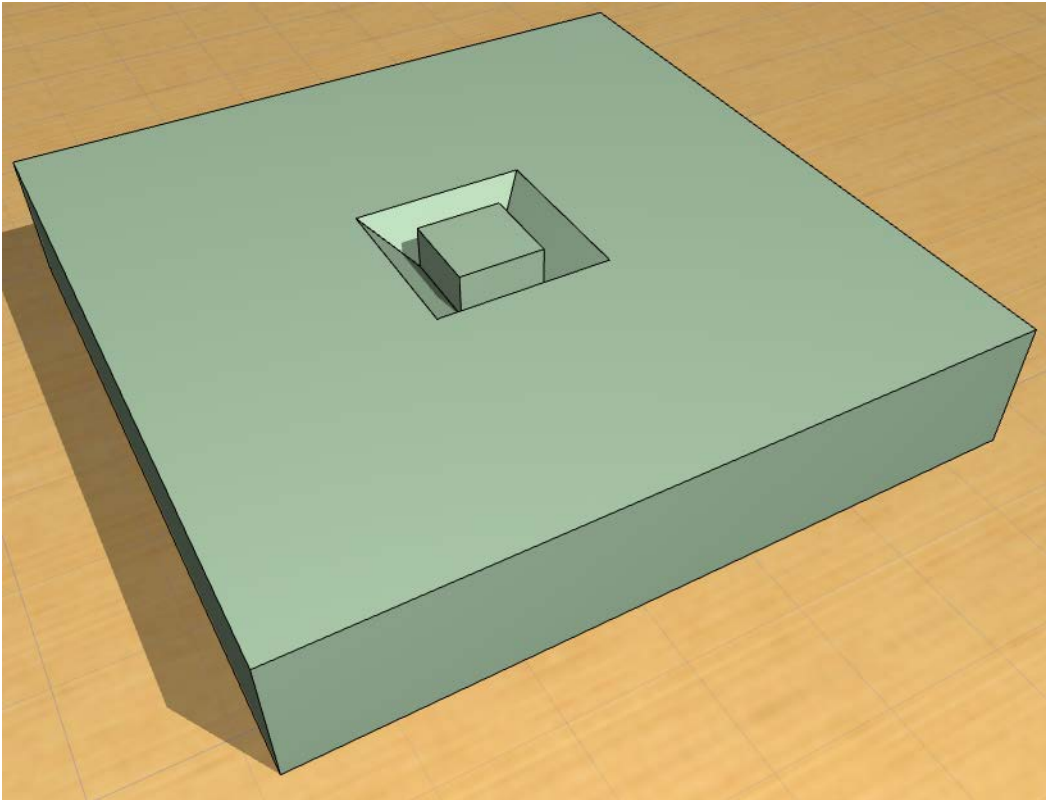
Transforming Morphs: Cover with Faces

Transforming Morphs: Smooth & Merge Faces

Use this command if you have done a basic angular-shaped model, and wish to achieve a more rounded effect with just one click.

Smooth A Non-Solid Morph

For example, we start with the following hollow Morph.

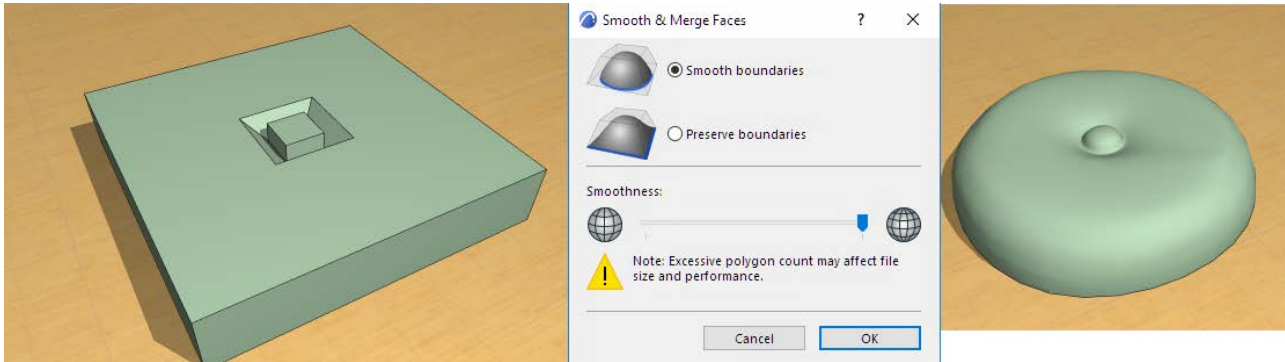


To achieve a smoother effect:

1. Select the Morph or a sub-element.
2. Use **Design > Modify Morph > Smooth & Merge Faces**.
3. The **Smooth & Merge Faces** dialog box appears.

Since the current (non-solid) Morph has outer boundaries (edges that are not adjoined by any faces), you can choose how to treat these boundaries:

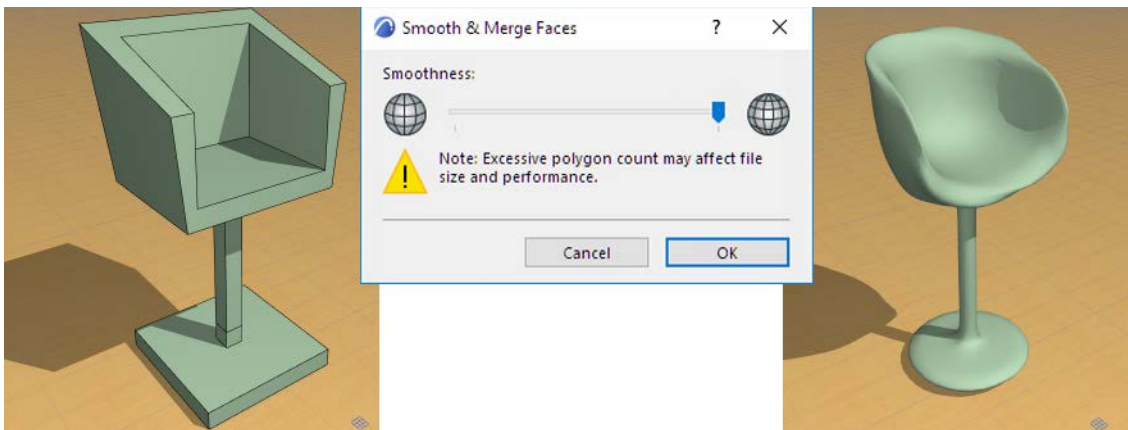
- **Smooth boundaries** means that the bounding edges of the selected element(s) or sub-elements will also take part in the smoothing:



- **Preserve boundaries** means that the bounding edges of the selected element(s) or sub-elements will stay intact, and only the inner parts are smoothed.
- **Subdivision level:** Use the slider to adjust how much triangulation (subdivision) you wish to achieve. Be aware that a higher subdivision level, used on complex or multiple Morphs, can lead to a high polygon count that can affect model size and Archicad performance.

Smooth a Solid Morph

With a solid Morph, you have no free-hanging edge boundaries, so all selected parts take part in the subdivision:

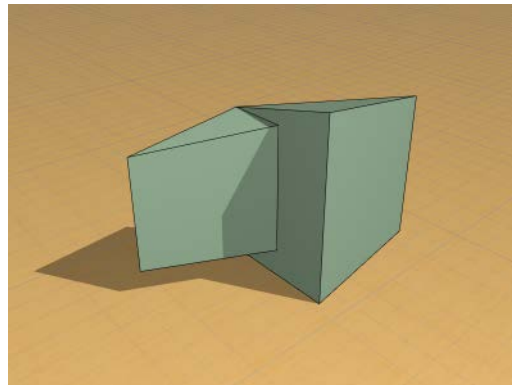
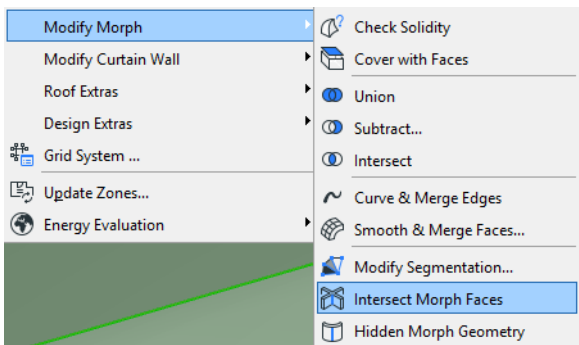
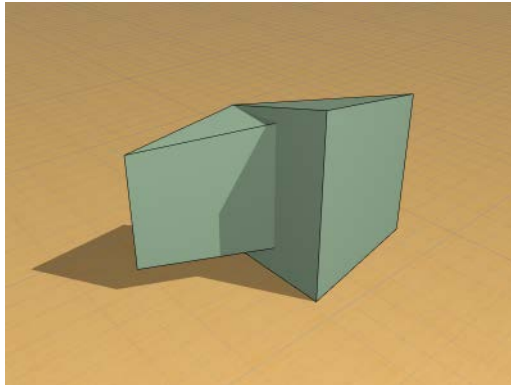
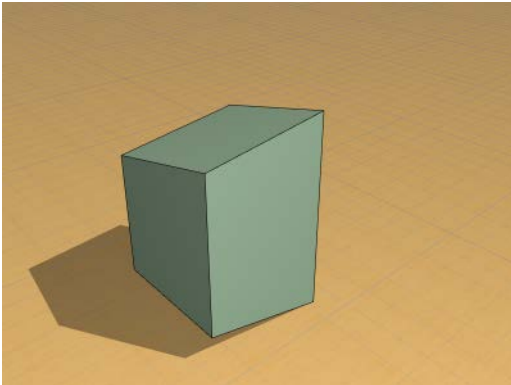


Intersect Morph Faces (Generate Edges within a Morph)

While editing a Morph, its shape may be transformed such that a single face is intersected by other parts of the Morph. At this point, the new (self-)intersection is not yet delimited by any edges.

If you henceforth want to treat such intersecting faces separately, use the **Design > Modify Morph > Intersect Morph Faces** command.

This command works on a single selected Morph only.

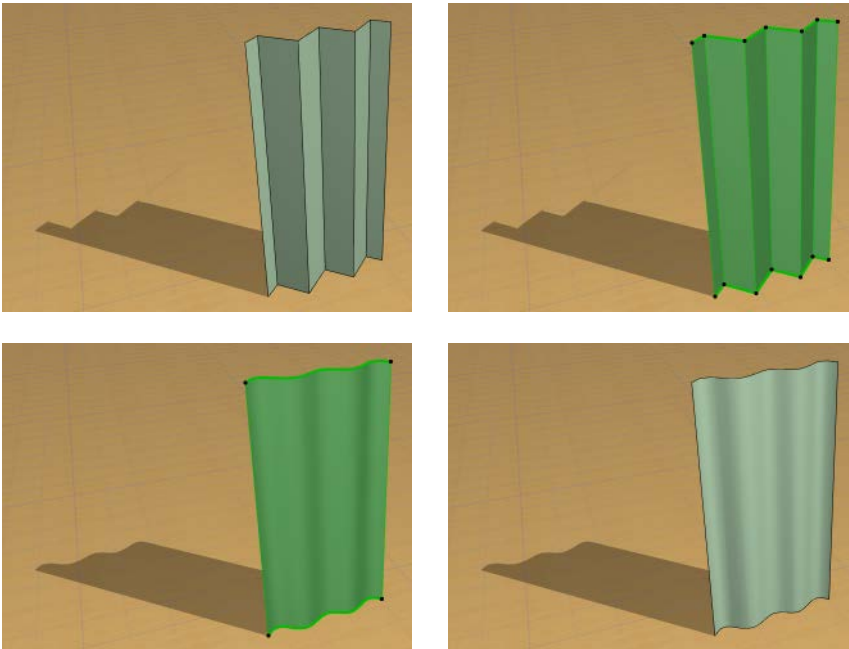


Transforming Morphs: Curve & Merge Edges

Use this command to segment one or more straight Morph edges to achieve a smoother, curved effect.

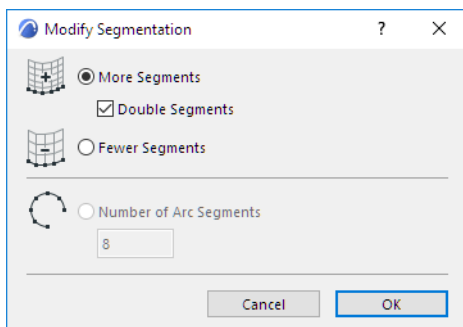
This feature is also useful to merge multiple curved Morph edges.

1. Select one or more Morph edges; or select one or more faces.
2. Use **Design > Modify Morph > Curve & Merge Edges**.
3. If needed, use **Design > Modify Morph > Modify Segmentation** to correct the effect.



Vary Smoothness Level of Morph Edges

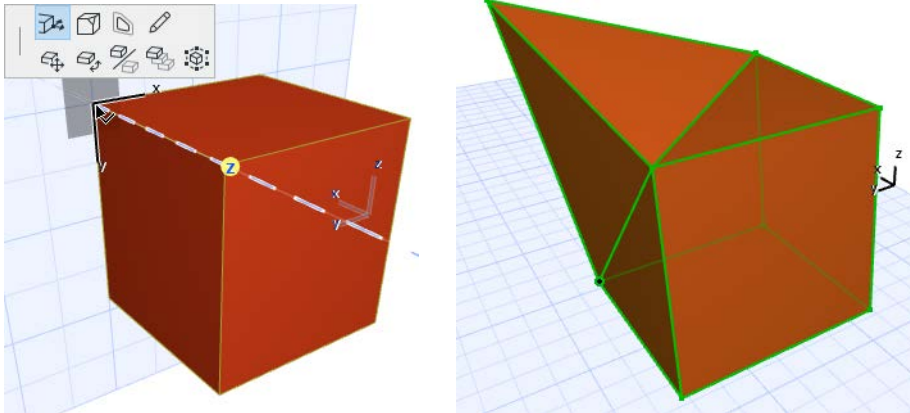
If you want to vary edge smoothing within a single Morph, you can adjust the Smoothing Level of individual Morph edges: select an edge, then use the **Design > Modify Morph > Modify Segmentation** command. This will reset the Smoothness behavior of the selected Morph edge only.



Combine Morph Faces for Smooth Editing Effect: Use “Soft” Edges

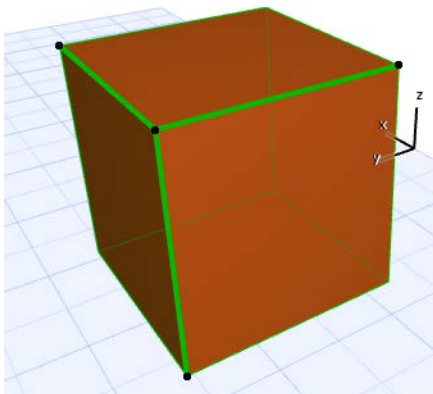
Morph editing often involves selecting and editing a sub-element (edge, node or face), which in turn affects the connecting faces of the Morph.

As the Morph changes shape, the connected faces, joined by a “hard” edge by default, behave as individual units, retaining their own geometries.

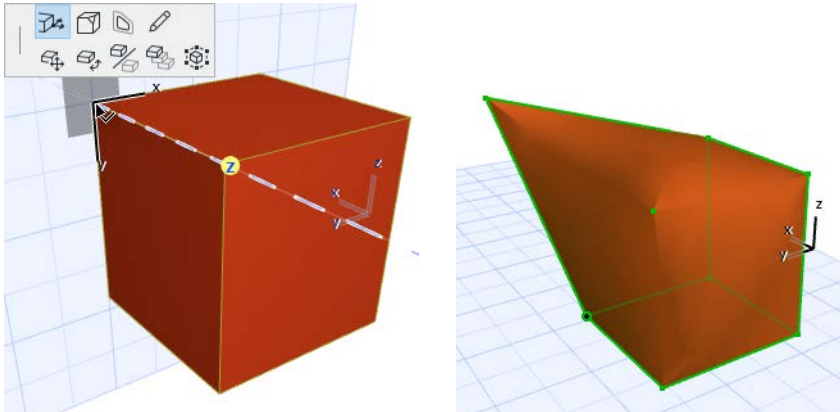


However, if you want the connected faces to “morph” smoothly, you can combine the distinct faces so that they henceforth behave as a unit while you edit them.

To do this, change the Edge settings (for any selected edges) from Hard (the default) to Soft. The easiest way to do this is to select any Edges, then click the Soft icon from the Info Box.



Faces connected by a “soft” edge behave as a single surface in Morph editing operations, and the smoothing algorithm is applied to them as a whole, resulting in a different editing effect:

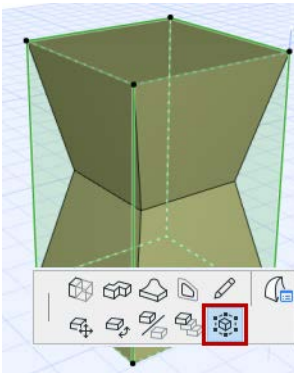


Transforming Morphs: Box Stretch a Morph

Use this command from the pet palette of a selected Morph, to stretch the entire Morph by its bounding box.

1. Select a Morph.
2. From the pet palette, choose the **Box Stretch** command.

A bounding box appears around the Morph.

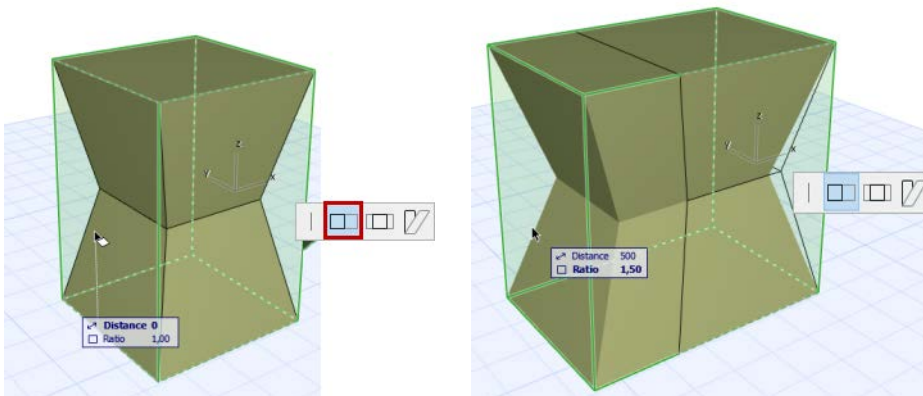


3. Click on an edge, face or node of the bounding box.
A “stretch” pet palette appears with three options.
4. Choose one of these three stretch varieties: **Stretch**, **Stretch from Center**, or **Skew**.
5. Then drag to transform your Morph.

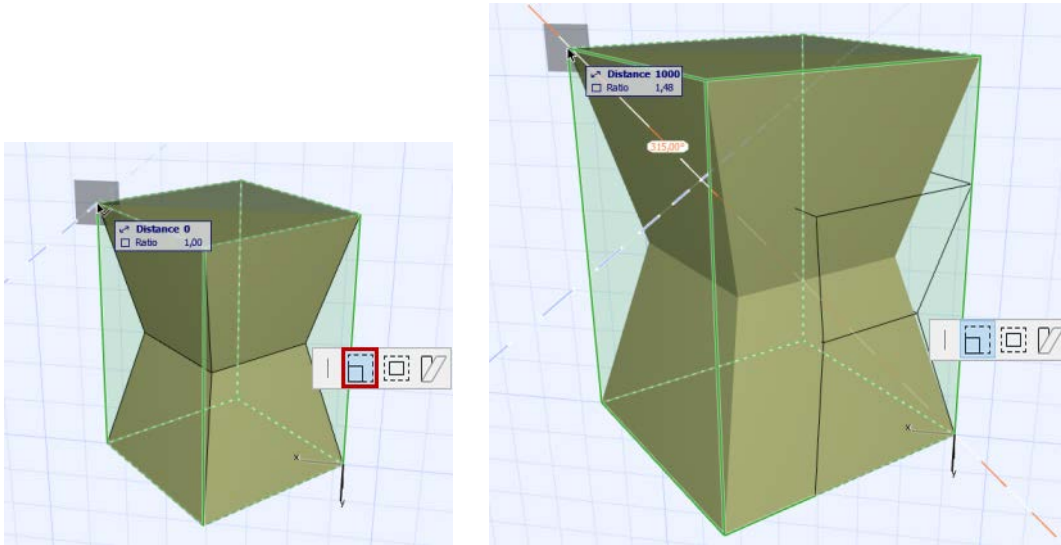
[Watch the Video](#)

Depending on which part of the box you clicked, the stretching result will vary:

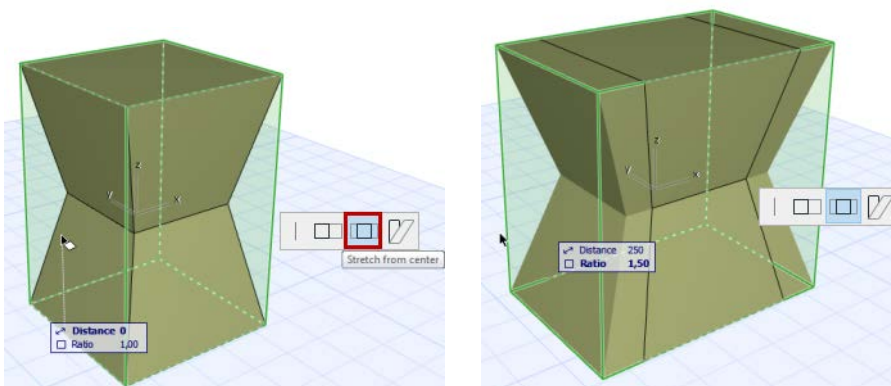
The plain **Stretch** (clicked on box face):



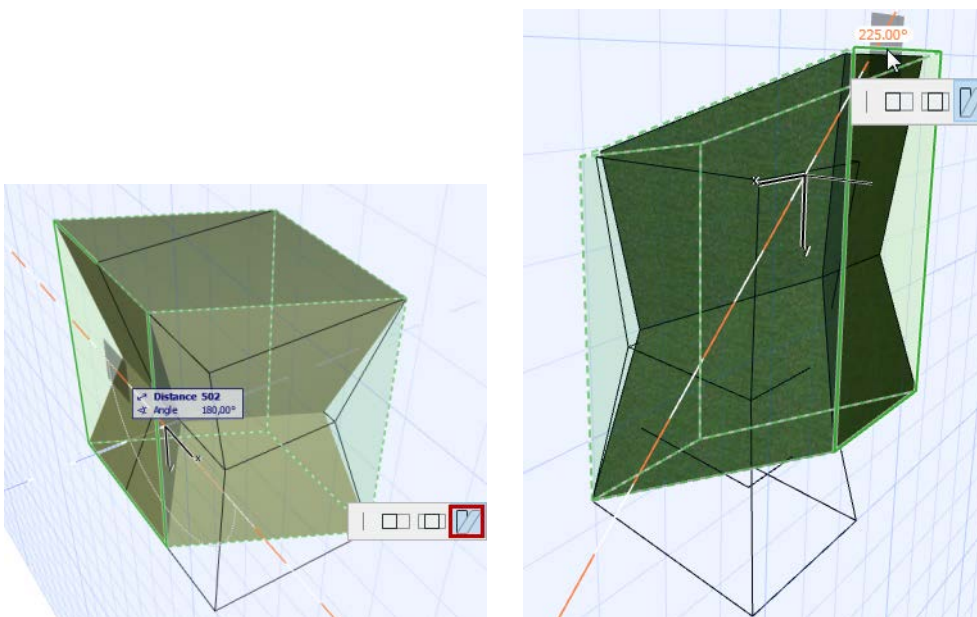
The plain **Stretch** (clicked on a box node):



The **Stretch from Center** (clicked on box face):



The **Skew**:

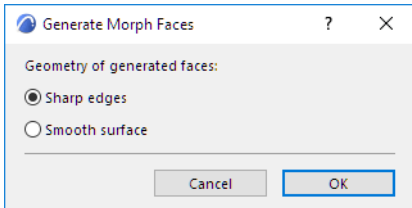


Transforming Morphs: Cover with Faces

Use this command (**Design > Modify Morph > Cover with Faces**) to add faces onto an existing Morph.

Select a Morph, or a series of Morph edges that form a closed polygon (this polygon can lie on any planes in space).

Use **Design > Modify Morph > Cover with Faces**.



From the appearing dialog box, choose whether the new faces should be based on

- Sharp edges
- Smooth surface

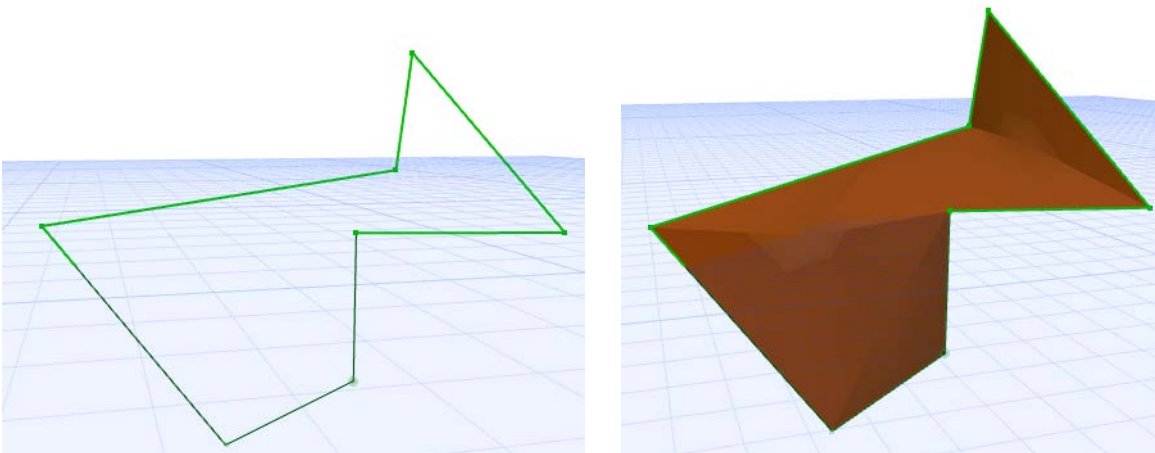
On an entire Morph:

If you have closed holes in the Morph, faces are generated to cover these holes.

On a series of selected edges forming a polygon:

- A Morph face is generated.

While you can also use the Add command from the pet palette, the Generate Faces command is different because it works on any plane in space.



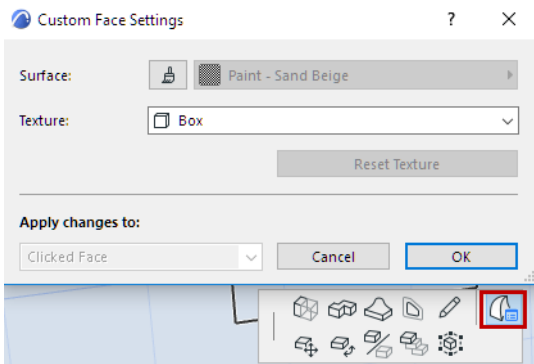
[Watch the Video](#)

Transforming Morphs: Custom Face Settings

You can apply custom **surface** and **texture mapping** settings to one or more Morph faces.

Note: By default, the surface applied to all Morph faces derives from the Building Material assigned to the Morph. You can override *all* Morph faces, using the Override Surface control in the Model Panel of Morph Settings. The Custom Face Settings offers a further way to customize the surface of *any one or more* Morph faces.

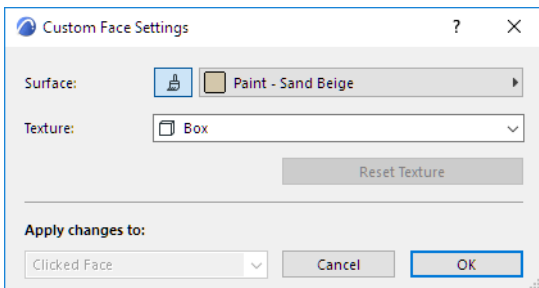
1. Select the face(s), or the entire Morph.
2. From the pet palette, choose the **Custom Face Settings** command.



3. Choose a custom surface and/or a different texture mapping:

Surface: Click the Surface override button, then choose a custom surface for the selected Morph face(s).

Texture: Choose a different texture mapping for the selected Morph face(s).



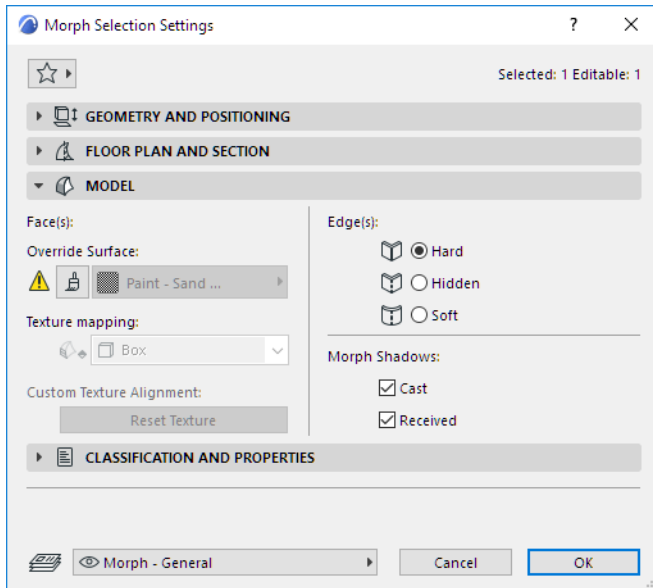
4. **Apply Changes to:** Once you have set the options as needed, choose the faces to which to apply these changes:

Note: These options are only available if you selected the entire Morph before bringing up the pet palette. If you selected only one or more faces, then your changes are applied only to the selected faces, and you have no other options here.

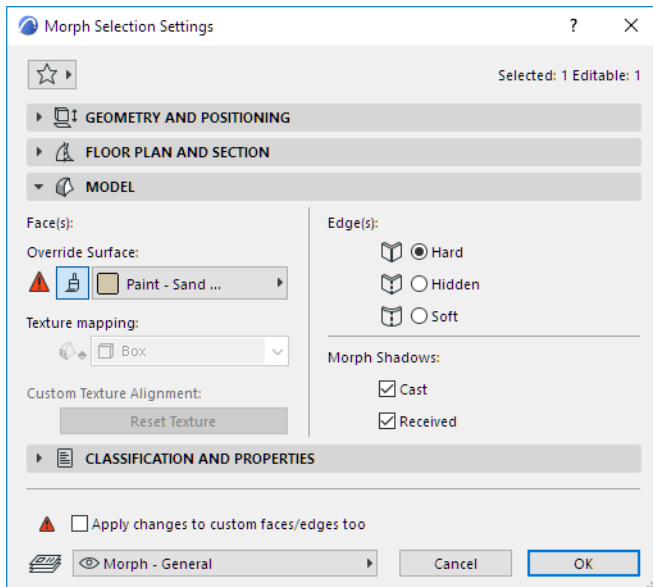
- **Clicked Face** will apply the chosen options (surface, texture mapping) only to the face you clicked when bringing up the pet palette.
 - **All Similar Faces** will apply the chosen options to all faces in the selected Morph which had the same initial surfaces as the one you changed. For example, if you change the of a face from yellow to red, all faces in this Morph whose surface was yellow will also change to red.
 - **All Faces** will apply the chosen options to all faces of the selected Morph.
5. Click **OK** to apply changes.

Custom Morph Surface: Feedback in Morph Settings

If you have customized any of the Morphs’s **Face surfaces** (in the Custom Face Settings dialog box), a yellow “custom” icon appears alongside the surface setting control in Morph Settings, indicating that one or more face surfaces have been changed from the general setting.



If you later change the Surface setting in Morph Settings while the dialog box is still open, the yellow patch (indicating “custom face”) gets a red shadow. This means that the Morph’s surface setting is going to change: do you want to *apply this override* to the custom faces too? If so, check the box at the bottom of Morph Settings (Apply changes to custom faces/edges too). If you do not check the box, your custom faces remain the way you set them.



Morph Edge Type: Hard, Hidden, Soft

- Morph edges, if **hard**, can be selected and edited.
- **Hidden** edges are simply hard edges that you have manually selected and set to be hidden.
- **Soft** edges are automatically generated during editing and smoothing operations, and within any curved Morph shapes; they are hidden by default and cannot be selected.
- Soft edges can also be defined manually by the user, as when you combine two faces into one by softening the dividing edge.
- A soft edge is an internal sub-element that divides segmented faces. Such faces are displayed (in Hardware acceleration and rendering) as smooth and curved.

See [Combine Morph Faces for Smooth Editing Effect: Use “Soft” Edges](#).

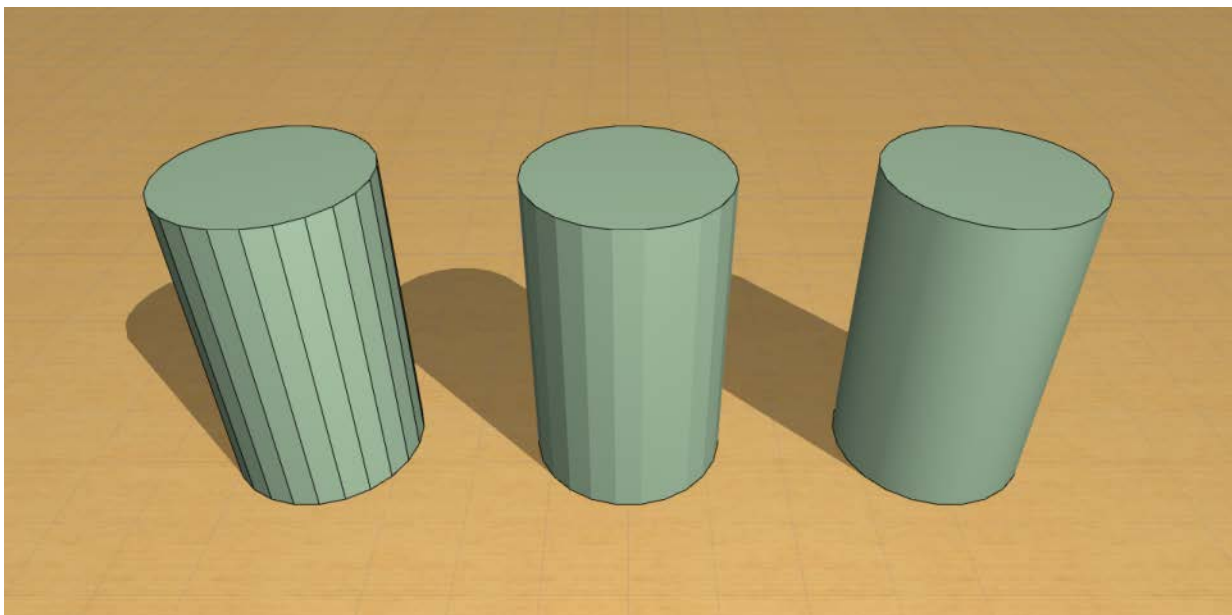
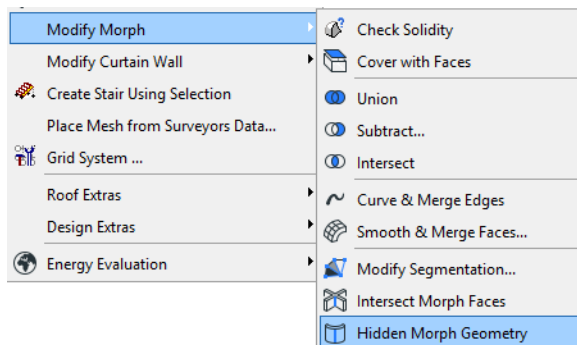
[Watch the Video](#)

Hard vs. Soft Edges: Show Hidden Morph Geometry

As you edit a Morph, you may wish to redefine an edge to be hard instead of soft, so that you can select it to modify the Morph shape.

To view (and be able to select) all edges, including soft ones:

Use **Design > Modify Morph > Show Hidden Morph Geometry**.

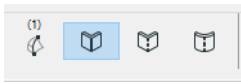


Change Edge Type

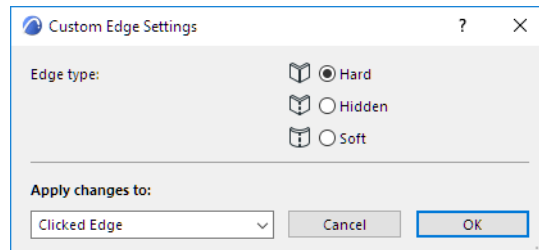
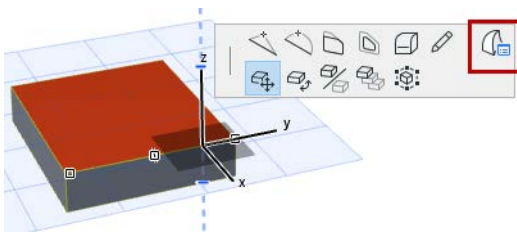
You can change the Edge type of any selected edge(s) of a Morph.

For more information on using this function, see [Combine Morph Faces for Smooth Editing Effect: Use “Soft” Edges](#).

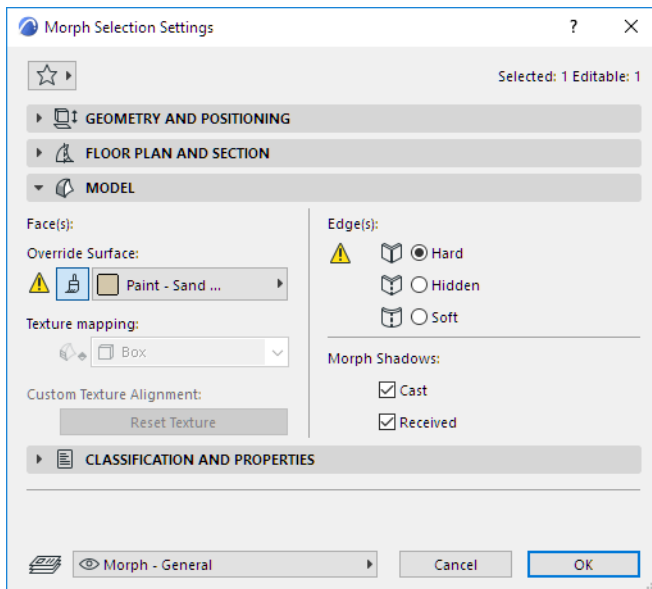
1. Select the edge(s), or the entire Morph.
2. Choose a different Edge type (Hard, Hidden, or Soft) using one of the following controls:
 - From the Info Box, or



- From the pet palette, by choosing the **Custom Edge Settings** command.



When you apply an Edge type that is different from the default Morph edge type, the Model Panel of Morph Settings will indicate that a custom edge type has been applied.



3. **Apply Changes to:** Once you have set the options as needed, choose the edges to which to apply these changes:

Note: These options are only available from the Custom Edge Settings dialog box, if you selected the entire Morph before bringing up the pet palette. If you selected only one or more edges, then your changes are applied only to the selected edges, and you have no other options here.

- **Clicked Edges** will apply the chosen edge type only to the edge you clicked when bringing up the pet palette.

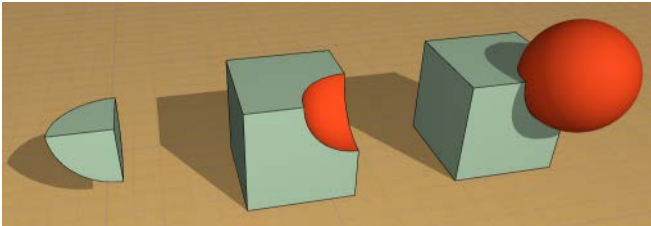
- **All Similar Edges** will apply the chosen options to all edges in the selected Morph which had the same initial setting as the one you changed. For example, if you change the Edge type from “Hard” to “Hidden,” all of this Morph’s hard edges change to hidden.
 - **All Edges** will apply the chosen type to all edges of the selected Morph.
4. Click **OK** to apply changes.

Boolean Operations with Morphs

Three Boolean operations are available from the **Design > Modify Morph** menu (or the context menu of selected Morphs):

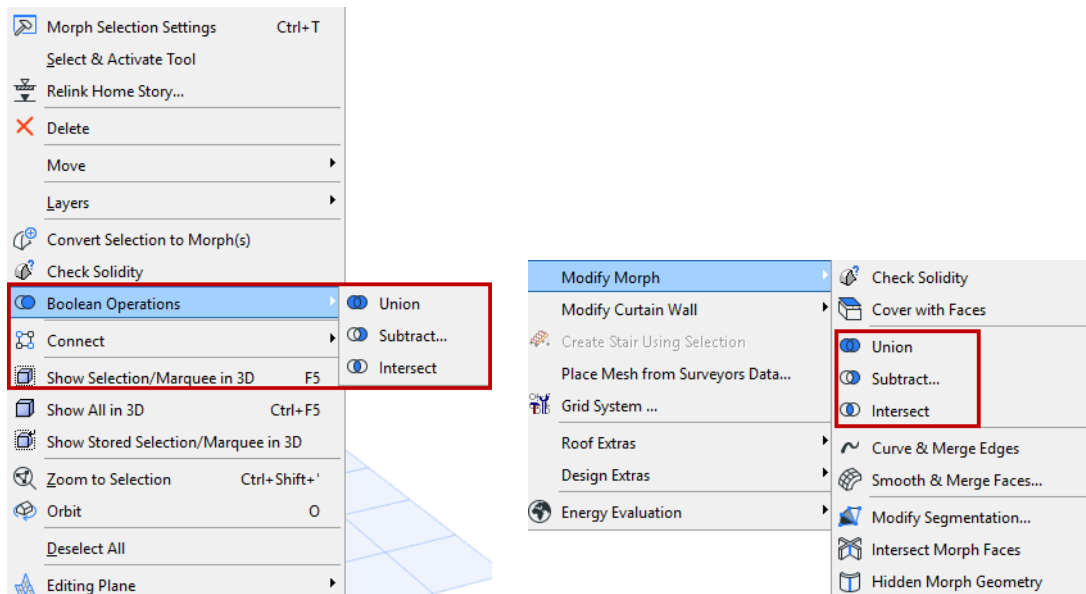
- Union
- Subtract
- Intersect

Note: You can also enable priority-based connections between Morphs and other elements. For information on how intersections (junctions) work among construction elements, including Morphs.



Morph Boolean Operations: Intersect, Subtract, Union

[See Element Intersections and Merge Elements: Roofs, Shells, Morphs.](#)



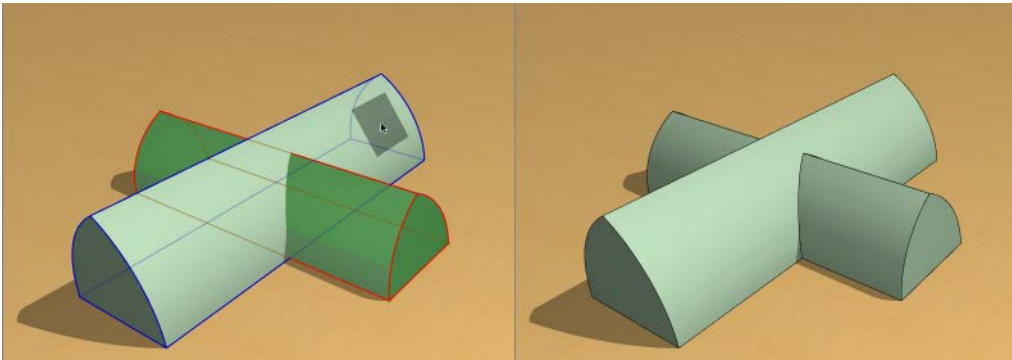
These commands are available if the last selected is a Morph; these operations can only involve Morphs. (If you want to operate on non-Morph elements, first convert those elements to Morphs.)

Unlike Solid Operations on other ArchiCAD elements, these operations are not associative - the final result of the operation is permanent.

Union

1. Select the Morphs you wish to merge.
2. Issue the **Boolean Operations > Union** command.

Note: If you have only selected one Morph, it is considered the operator. You are prompted to click another (Target) Morph with which to merge it.



3. As a result of this command, the Morphs in the selection will be merged into a single Morph element. The resulting Morph will use the attributes of the last selected Morph.

Subtract

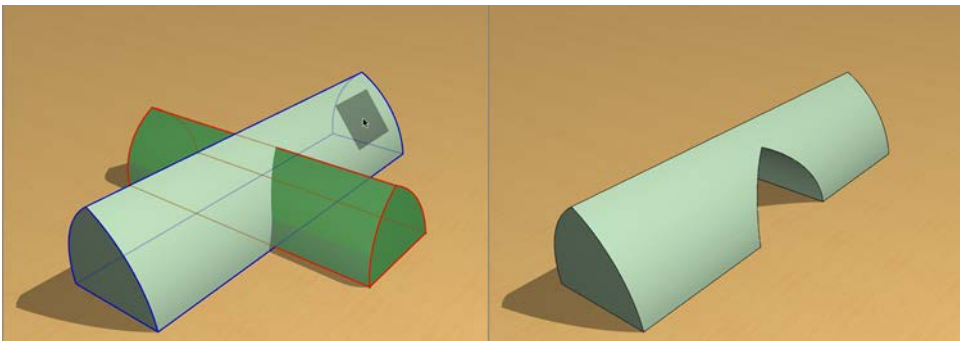
Use this Solid Morph Operation to subtract one Morph from another.

1. Select the Morph(s) you wish to subtract.
2. Issue the **Boolean Operations > Subtract** command (from the context menu) or **Design > Modify Morph > Subtract**.

Note: If your original selection contains any non-Solid Morphs, you are warned of this. You can either proceed anyway - this will leave the non-solid Morphs out of the operation - or ask to Show non-Solid Elements.

3. You are prompted to “Click a Morph from which to subtract the selection.” (The clicked target Morph can be a surface; it need not be solid.)

Note: If you try to click a target element that is not a Morph, the cursor changes to the “Do not Enter” sign and you get a warning to that effect.



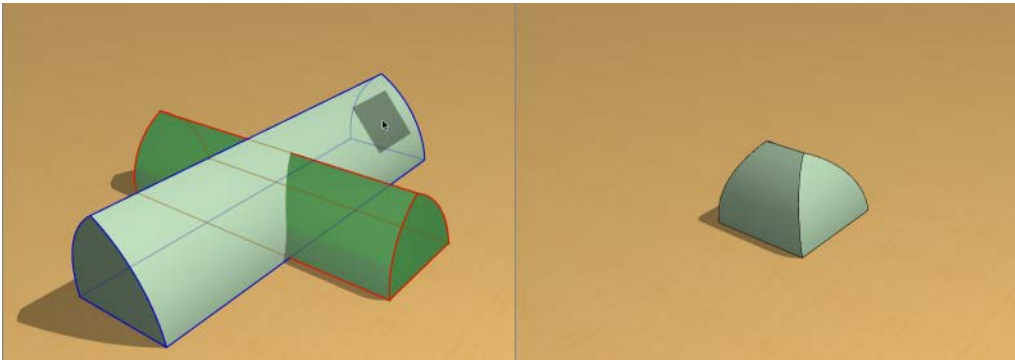
4. After the Subtract operation is completed, the Operator elements are deleted. The result is always a Morph.

Intersect

Use this command to intersect multiple solid Morphs and end up with a single Morph.

1. Select the Morphs you wish to intersect.
2. Issue the **Boolean Operations > Intersect** command (from the context menu) or **Design > Modify Morph > Intersect**.

Note: If you have only selected one Morph, that element is considered the operator. You are prompted to click another (Target) Morph with which to intersect it.



3. As a result of this command, the intersecting portion of the originally selected Morphs are retained, resulting in a single solid Morph.

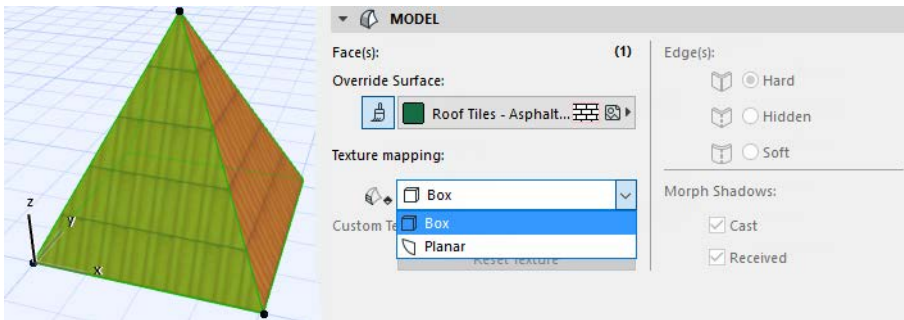
The resulting Morph will use the attributes of the last selected Morph.

Morph Texture Mapping and Alignment

Texture Mapping

If your Morph uses a surface that includes a texture, you can control how that texture is displayed on the Morph surfaces.

The Texture Mapping definition (either Box or Planar) is shown in the Model Panel of Morph Selection Settings.



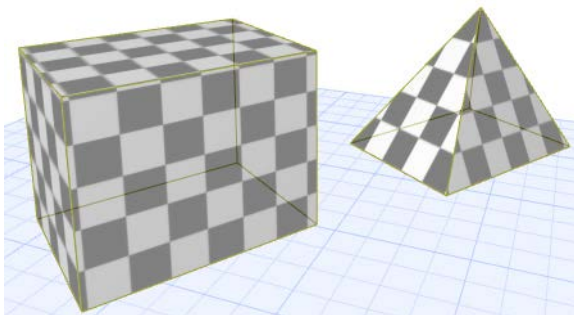
Box Mapping: this is the default, and the only option available if you have selected an entire Morph (see Box Mapping below).

Planar: This option can only be chosen if your selection includes one or more faces. The “Planar” definition means that you can vary the plane from which the texture is projected onto the selected face, as well as set an individual texture alignment and origin for the face.

Box Mapping

For a Morph selected as a whole, and for default Morphs, the texture mapping is always “Box.”

“Box” means the pattern is projected onto the Morph from six planes (as from a box). Any given plane gets its texture from the direction of the closest of the six planes.

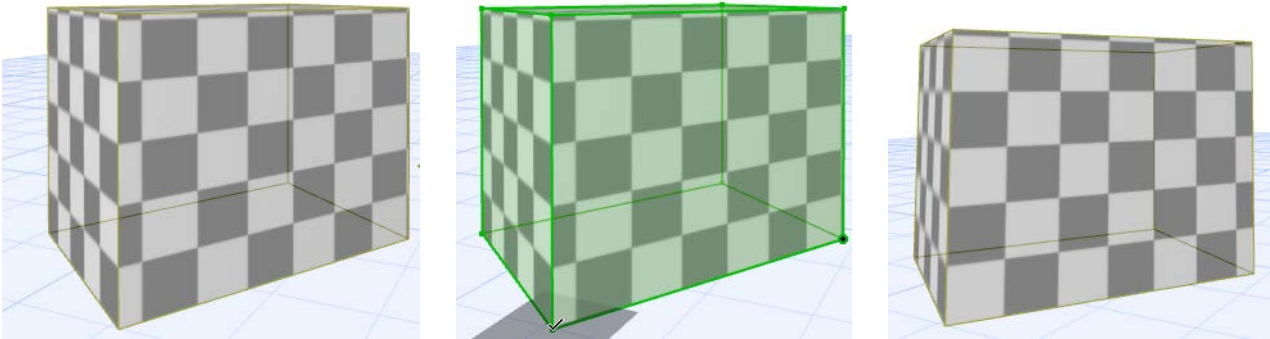


The origin of the texture is at Project Zero.

Set Origin of Box Mapping

1. Select one or more Morphs.
2. Use **Document > Creative Imaging > Align 3D Texture > Set Origin**.
3. Click at the point (e.g. a corner) where you want the texture pattern to originate.

- The origin is set. (If you selected multiple Morphs, the texture origin is in effect for all selected Morphs.)



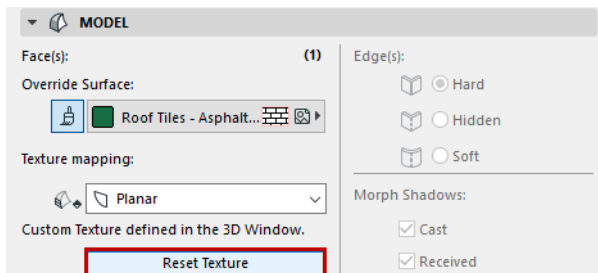
Set Direction of Box Mapping Texture

- Select one or more Morphs.
- Use **Document > Creative Imaging > Align 3D Texture > Set Direction**.
- In the appearing dialog box, enter a rotation angle or draw a vector to reset its direction.

Reset Box Mapping

For Box Mapping, “reset” means that the texture origin returns to Project Zero.

- Select one or more Morphs.
- Do one of the following:
 - Use **Document > Creative Imaging > Align 3D Texture > Reset**.
 - Click **Reset Texture** in Morph Selection Settings (Model Panel), or click the Reset button alongside **Custom Texture** in the Info Box.



- The texture origin is reset.

Planar Mapping

This option can be available only if your selection includes one or more Morph faces. The “Planar” definition means that you can vary:

- the plane from which the texture is projected onto the selected face
- the texture origin separately for any face
- the texture alignment separately for any face

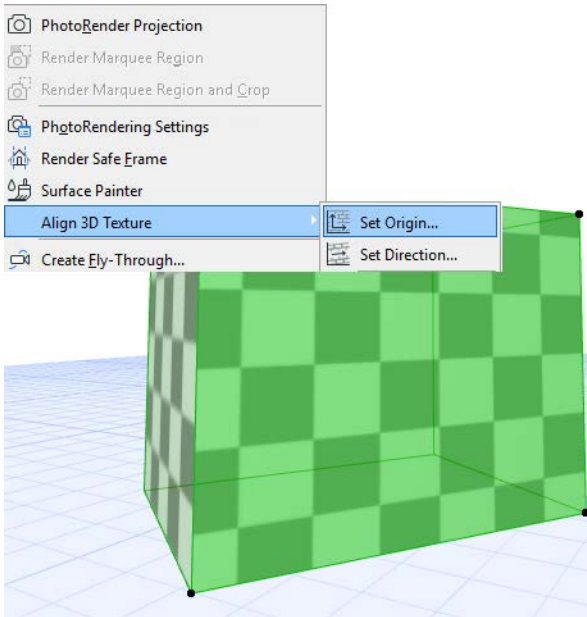
Set Texture Projection by Plane

- Select one or more Morph faces.
- Choose **Document > Creative Imaging > Align 3D Texture**. (The Texture Mapping control in Morph Selection Settings automatically changes to “Planar.”)

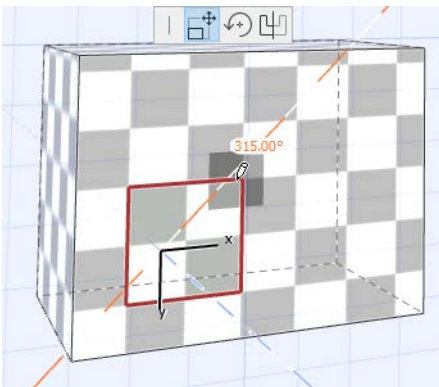
3. Choose one of the following commands: **Set Origin/Set Direction.**
4. An editing square appears on the Morph face, representing a unit of the texture pattern.
5. Vary the editing plane to vary the texture projection.

Set Origin or Alignment by Plane

1. Select one or more Morph faces.
2. Choose **Document > Creative Imaging > Align 3D Texture.** (The Texture Mapping control in Morph Selection Settings automatically changes to “Planar.”)
3. Choose one of the following commands: **Set Origin/Set Direction.**



4. An editing square appears on the Morph face, representing a unit of the texture pattern.
5. Click anywhere on the square. A pet palette appears with the “Move” and “Rotate” options.



6. Do one of the following:
 - Drag the square and click to place. The clicked point represents the origin of the texture pattern on that face.
 - Choose the Move icon to drag the texture and realign its origin (same effect as above).
 - Choose the Rotate icon to rotate the texture direction on that face.

7. Click anywhere outside the square with the hammer cursor to complete the operation.

[Watch the Video](#)

Fix Modeling Errors: Check and Solidify Morphs

What is a Non-Solid Morph?

A Morph is non-solid if it includes any edge segment that is not adjoined by exactly two faces. A Morph with a free-standing edge, or faces that do not connect precisely, will be non-solid.

Often, you cannot see the non-solid error if you are looking at a complex model.

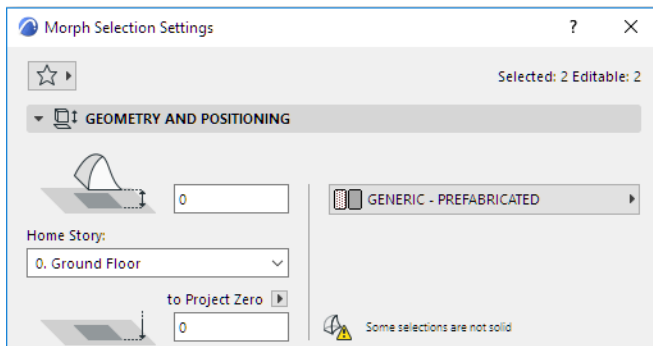
This may cause problems:

- if you calculate Morph volume in lists, a non-solid Morph will have a volume of zero;
- only solid Morphs can have a cut fill; a non-solid Morph in Section is shown without a fill.

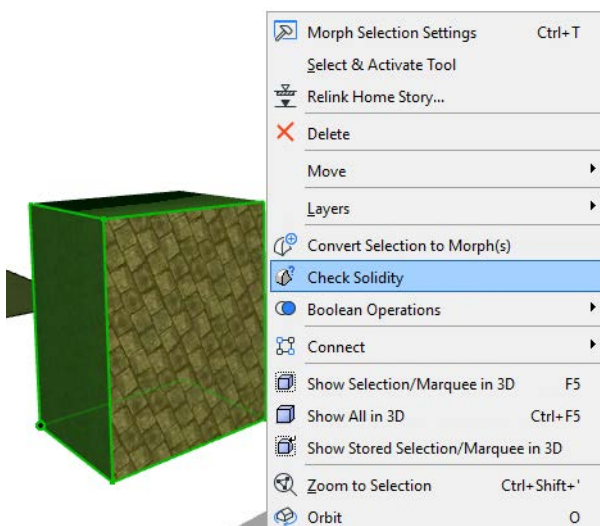
How To Fix a Non-Solid Morph: Use Solidify Command

To find and fix such modeling problems, do the following:

- Check the Info Box or Settings of any selected Morphs to see if they are solid.

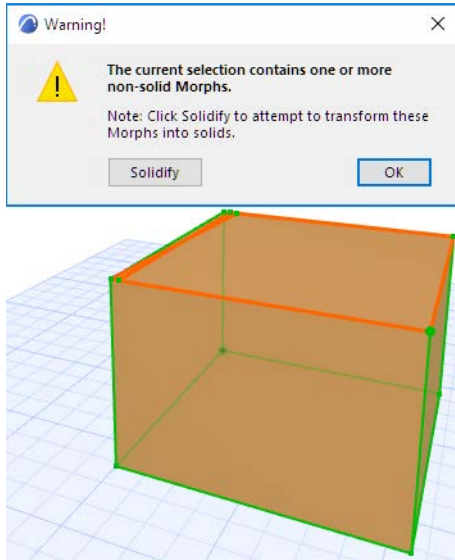


- Select any or all Morphs, then use the **Check Solidity** command from the context menu (or the same command from Design > Modify Morph).



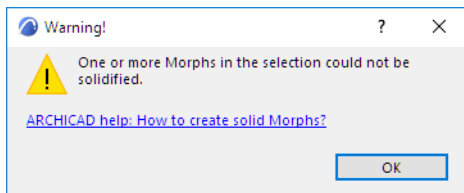
- If any of the selected Morphs is not solid, you get a warning, and the problematic Morphs and edges are highlighted in the model, as shown in the image below.

- To try to fix the error, click **Solidify**.



If the Solidify Command Cannot Fix The Error

If the Solidify command could not fix the non-solid Morph automatically, you get a warning.



The problematic Morphs and edges are highlighted in the model.

Try to fix the problem with graphical editing, adding Morph edges to fill in the gaps:

- Use the **Add to Morph** command (pencil icon) from the pet palette to add one or more new Edges where the gap is. Each time you add a new Edge, Archicad will automatically try to create adjoining solid faces within the Morph.
- In some cases, it is useful to draw another Edge on top of an existing Edge where the gap is, to generate automatic faces.
- When you are done adding edges, select the Morph and use **Check Solidify** again to see if the edits worked.

[Watch the Video](#)

Openings

Use the Opening Tool to place an Opening element in a Wall, Slab, Mesh, or Beam. Even if the host element is part of a Hotlink Module or reserved by another user - you can place the Opening.

An Opening is a hole that provides space for engineering items such as elevators and MEP elements. A limited-depth Opening can be defined, e.g. to create a Wall niche or Slab recess.

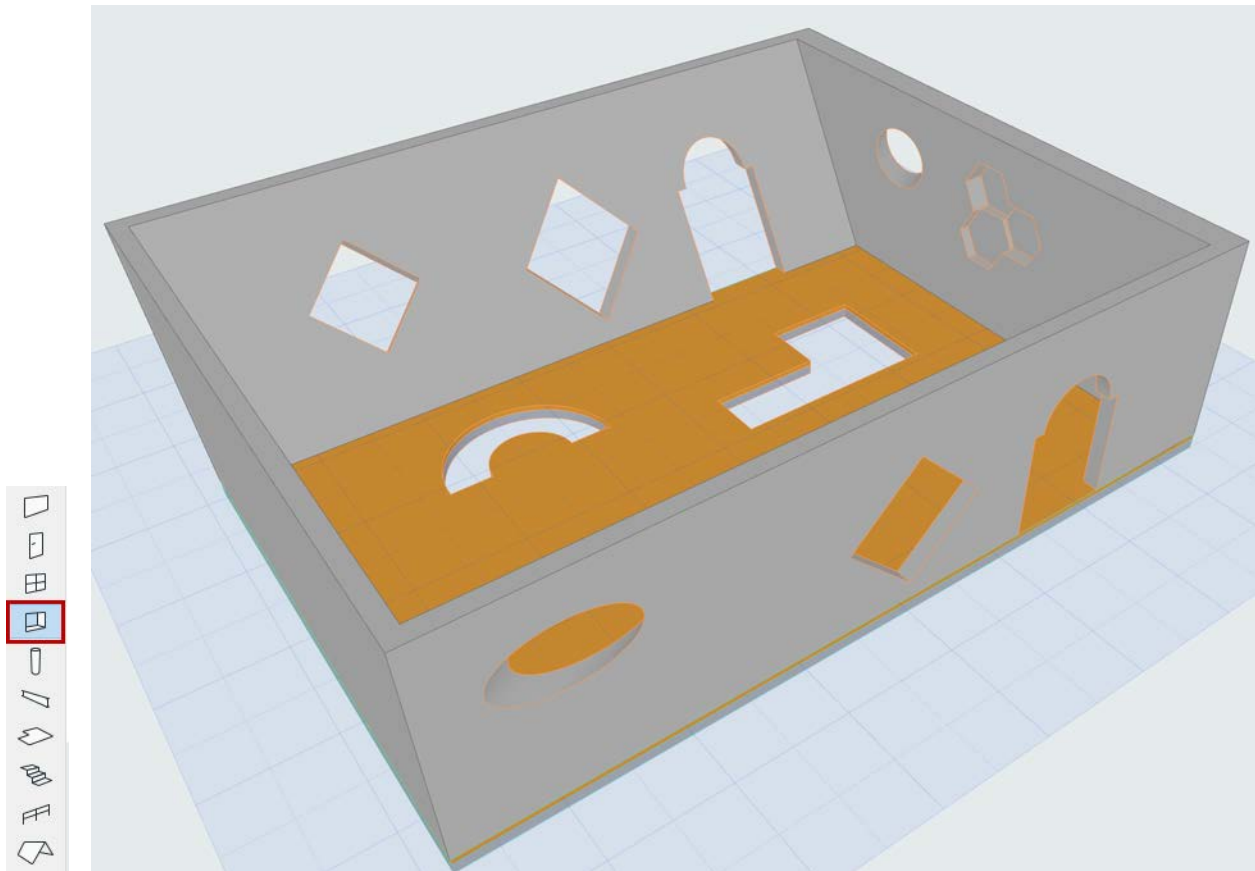
The polygonal geometry method enables Openings in custom shapes for any architectural purpose. You can also use the Magic Wand for polygonal Openings.

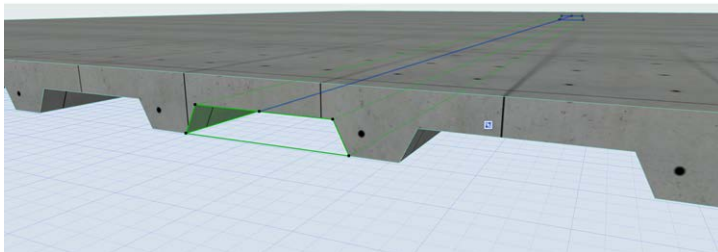
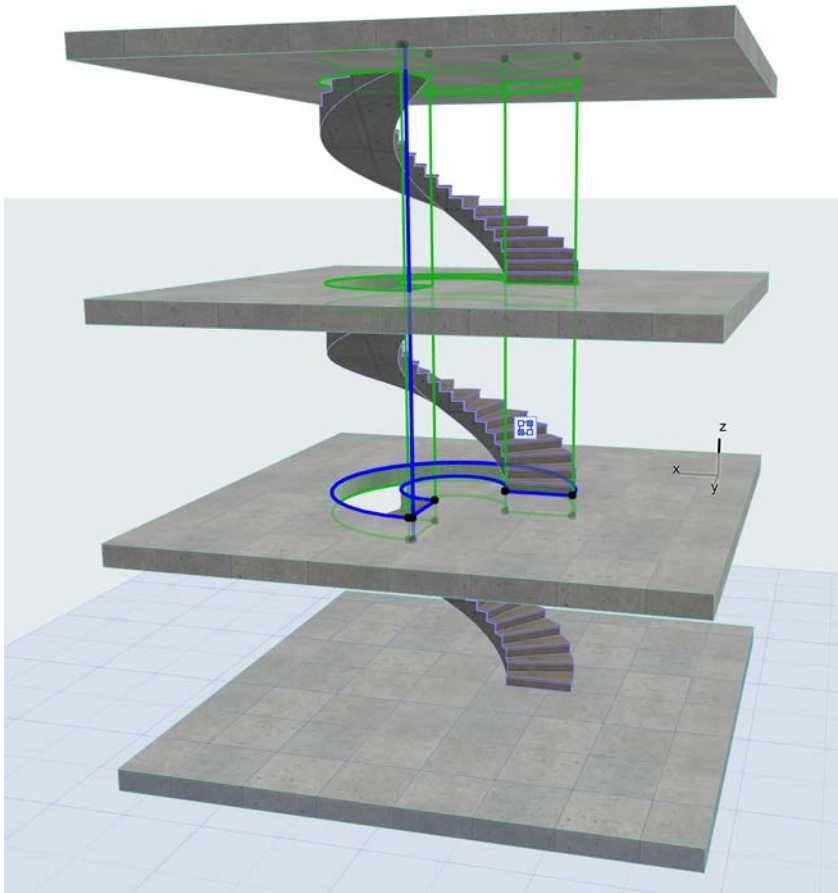
Every Archicad Opening is associated to an element: Wall, Slab, Mesh or Beam. The Opening's position is relative to that associated element, and moves along with it.

A single Opening can cut multiple elements at once. For example, a suspended ceiling modeled from three separate elements can be cut by a single Opening.

Like any other model element, each Opening has ID, Classification, and Property data. Schedule and Label these values as needed.

Set the show/hide status of Openings using Model View Options (similar to Door/Window).





Related Topics

[Create Opening](#)

[Opening Display on Floor Plan](#)

[Opening Display in 3D](#)

[Use Limited Opening to Create Niche or Recess](#)

[On-Screen View Options for Openings](#)

[Edit Opening](#)

[Manage Opening Connections](#)

[Edit Openings in Teamwork](#)

[Import/Save Openings with IFC](#)

[Create Openings from Selection](#)

[Watch the Video](#)

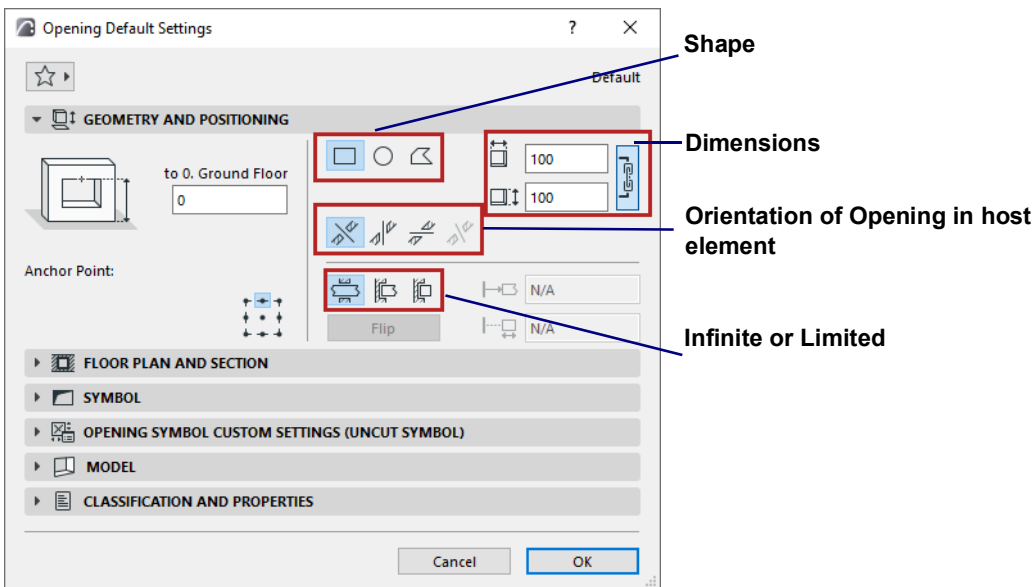
Create Opening

[Watch the Video](#)

1. Activate the Opening tool.



2. In Opening Settings or the Info Box, define the Opening's basic geometry, shape and anchor point, as needed. (Settings are described below.)



3. Hover over the associated element (e.g. Wall) into which to place the opening. The pre-placement highlight indicates the geometry of the Opening and its associated element.

If the cursor is above several potential associated elements, use **Tab** to cycle the highlight among them.

Notes:

- Even if the host element is part of a Hotlink Module or reserved by another user - you can place the Opening.
- Opening geometry is measured relative to its associated element, and moves along with it.

4. Click to place the Opening, depending on its shape and Orientation (see below).

Shape

Choose an Opening shape: Rectangular, Circular, Polygonal.

- If you edit the shape of a rectangular or circular Opening, it changes automatically to Polygonal.

Exception: An ellipsis Opening does not become a polygon.

- You can also use the Magic Wand to place a Polygonal opening.

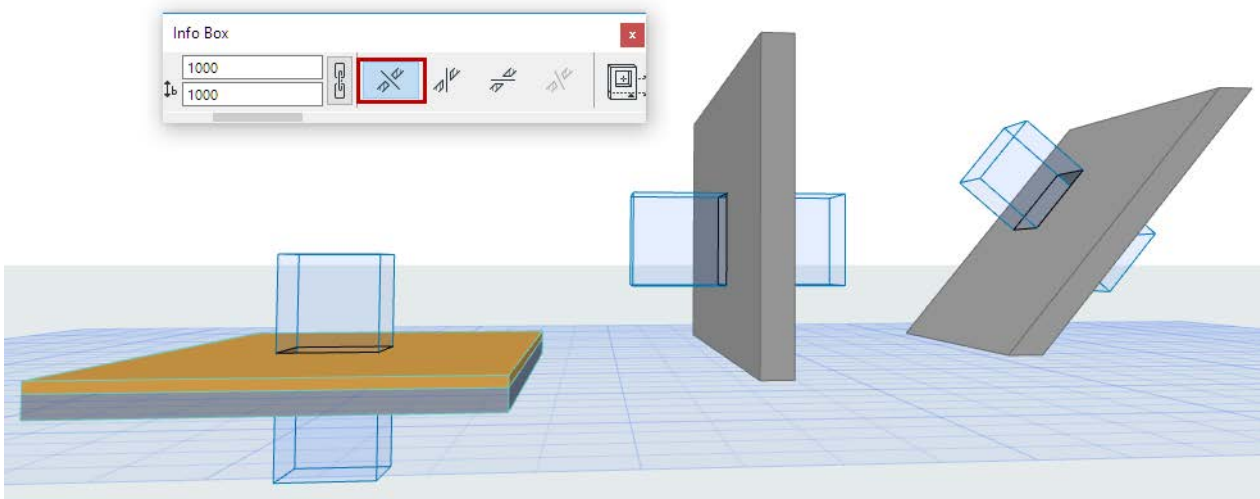
Orientation

Orientation defines the Opening extrusion relative to the associated element: Aligned, Vertical, Horizontal or Custom.

Aligned (default)

Aligned Opening is perpendicular to the host element's reference line or plane, in the most logical direction depending on the host:

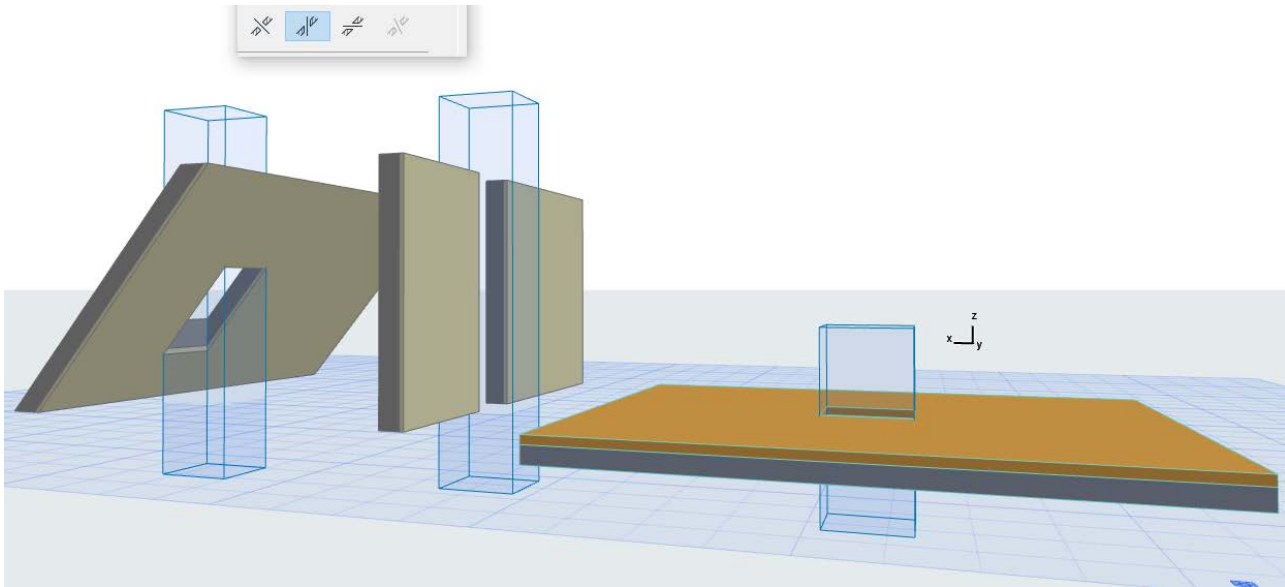
- Opening in a Slab or Mesh is vertical
- Opening in a straight Wall or Beam is horizontal
- Opening in a slanted Wall is perpendicular



An Opening placed with the Aligned method remains connected to the host element's reference line or plane.

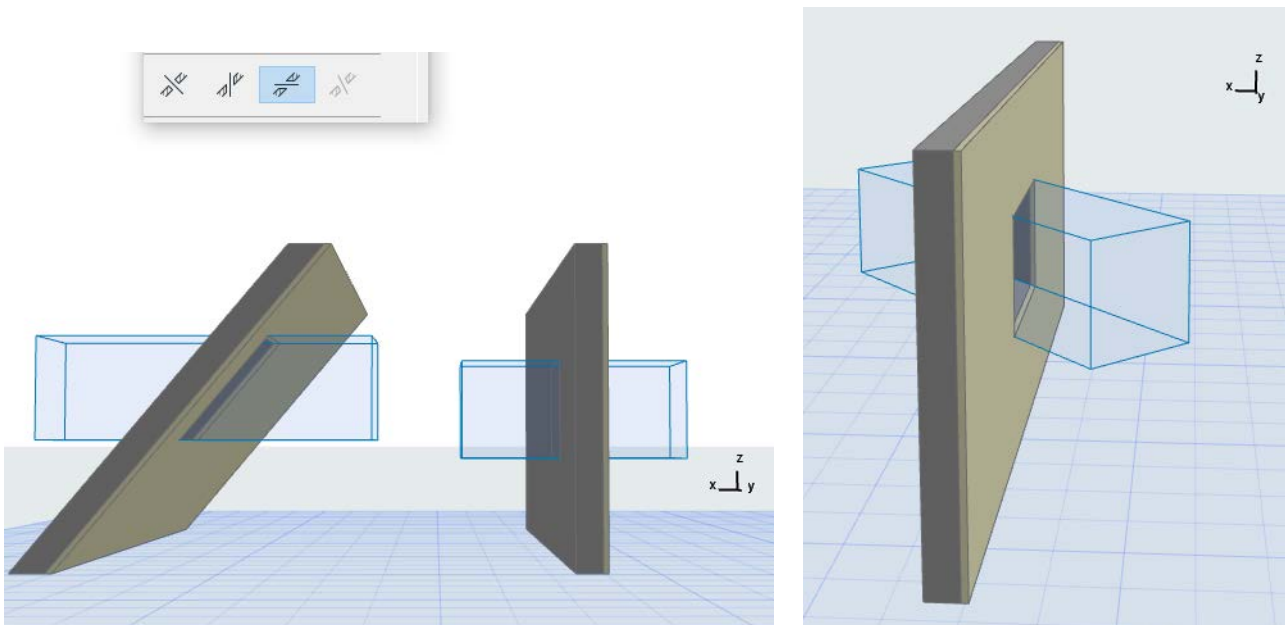
Vertical

Examples of vertical Openings:



Horizontal

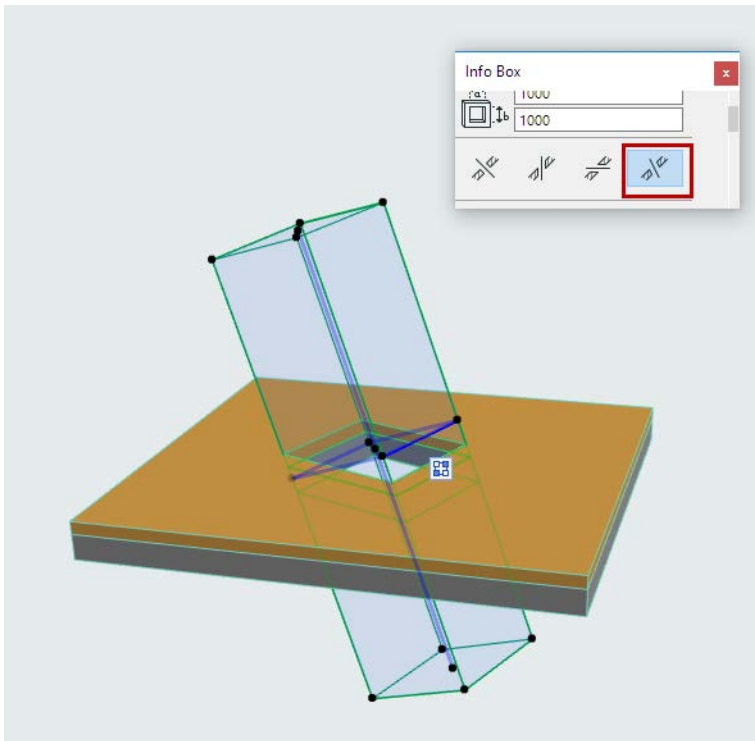
Click once to place the horizontal Opening, then click again to define extrusion direction:



Custom

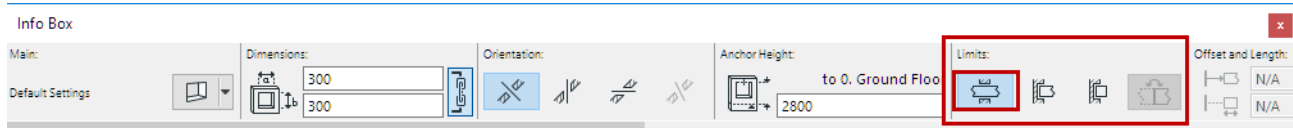
Drag, mirror or rotate an existing Opening into any position.

- Available for a selected, existing Opening. (Not available as default input method.)
- You can change a selected Custom-oriented Opening, then change its orientation to Aligned, Vertical or Horizontal. However, the previous Custom orientation is now lost.



Infinite or Limited Opening

By default, an Opening extends infinitely - as set at the **Limits** option in Info Box or Opening Settings.



An “infinite” Opening means that its length does not depend on the cut element’s geometry: if Wall thickness is increased, the Opening still cuts through it.

An Opening can also be defined as limited on one or both sides.

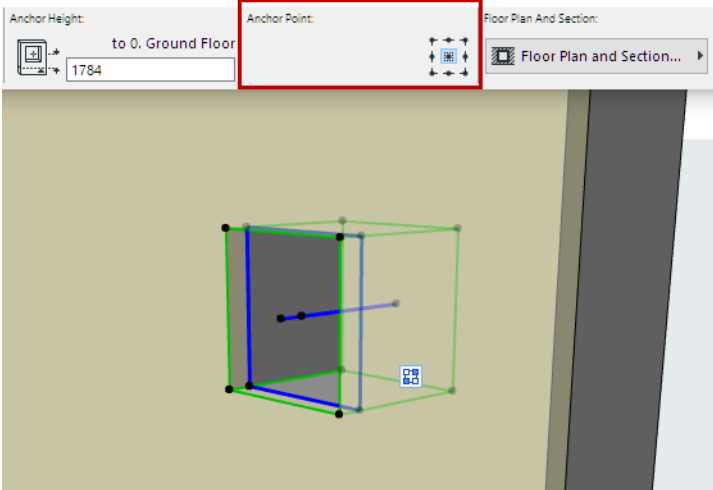
[See Use Limited Opening to Create Niche or Recess.](#)

Anchor Point

Position the Opening’s Anchor Point at any of the 9 points on the bounding rectangle.

In 3D, the blue extrusion vector indicates the anchor point’s position.

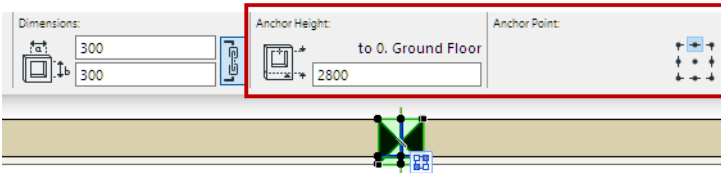
- If you resize the Opening numerically, the anchor remains fixed.
- If you resize the Opening graphically, the Opening centerpoint remains fixed - regardless of the anchor.



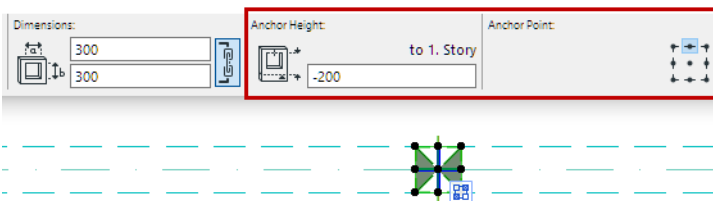
Opening Elevation

Opening elevation is measured from the associated element's Home Story to the Opening's Anchor point. This is the anchor height value.

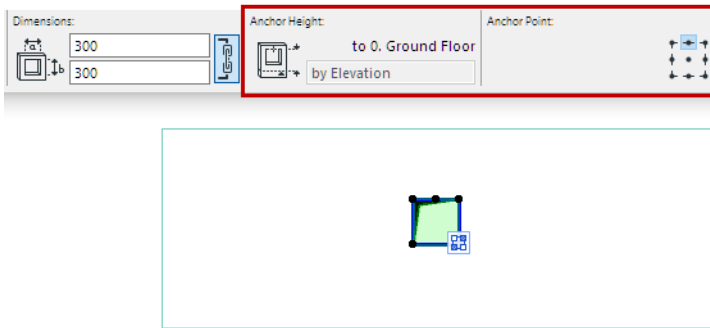
- In 3D: Click to place the opening graphically at any elevation.
- On Floor Plan:
 - Opening's anchor height in a vertical element (e.g. Wall) is measured to the associated element's home story.



- When placing an Opening placed in a Beam, anchor height is indicated from the story above.



- For an Opening placed in a Slab or Mesh: anchor height follows the elevation of the host element.



Opening Display on Floor Plan

Use Opening Settings to fine-tune the display of Openings on the Floor Plan, including its symbol, attributes, and Reference Axis.

Note: Openings are visible only if at least one of its Cut Elements is visible too.

[Watch the Video](#)

Topics in this Section:

[Floor Plan Symbol](#)

[Opening Display in Cut Elements](#)

[Divided Openings: Separated or Unified](#)

[Cut Surfaces](#)

[Opening Outlines on Floor Plan](#)

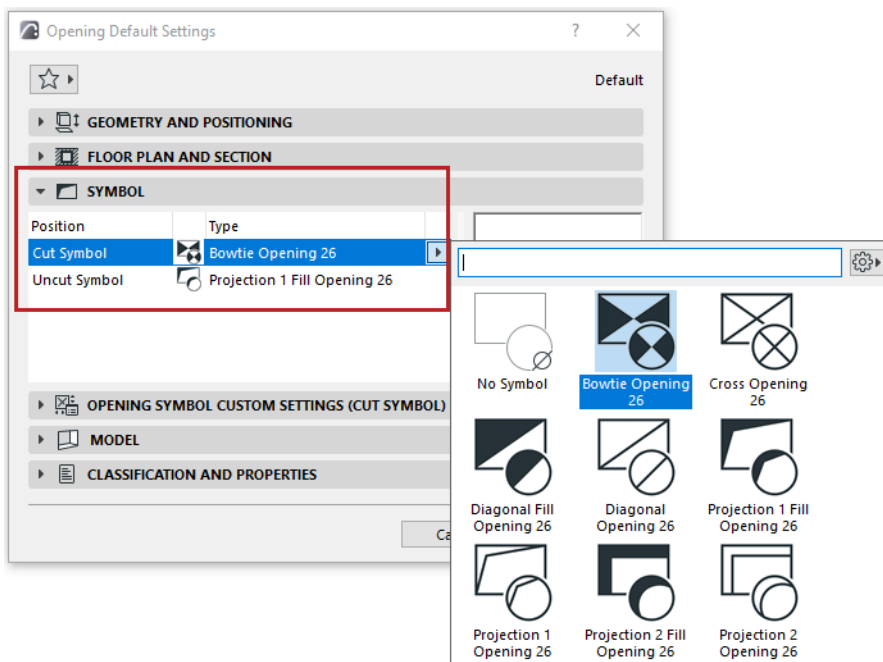
[Opening Cover Fills](#)

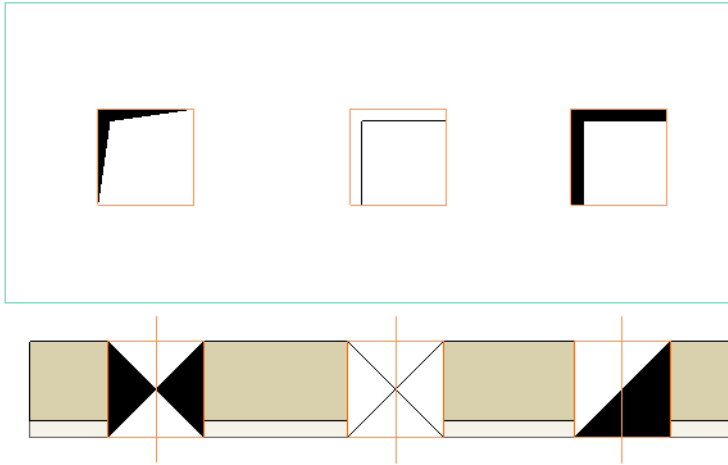
[Opening Reference Axis](#)

Floor Plan Symbol

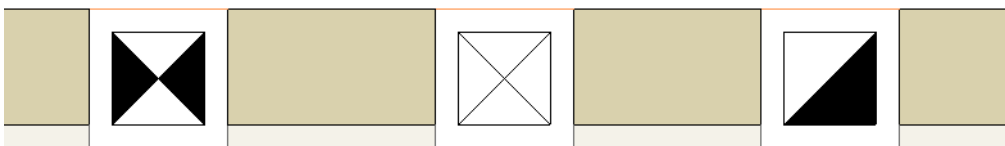
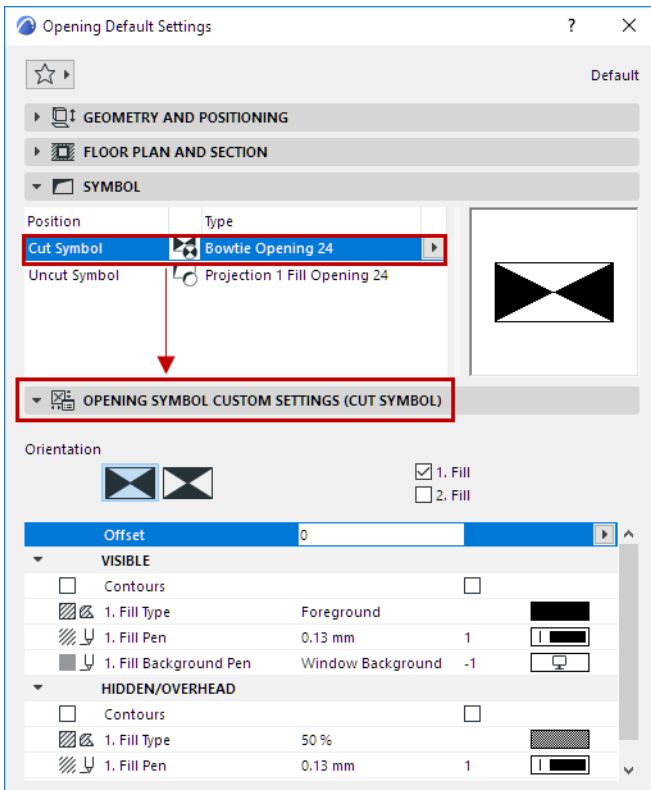
[Watch the Video](#)

An Opening can have two Floor Plan Symbols: one each for Cut and Uncut openings. Choose a default for each, using the **Symbol panel** of Opening Settings.

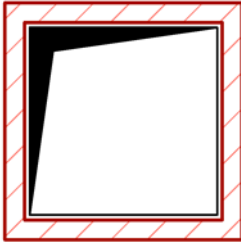




Fine tune your preferences and geometry preferences for any Symbol, using the Symbol Custom Settings panel:



Cut symbol, with offset and contour

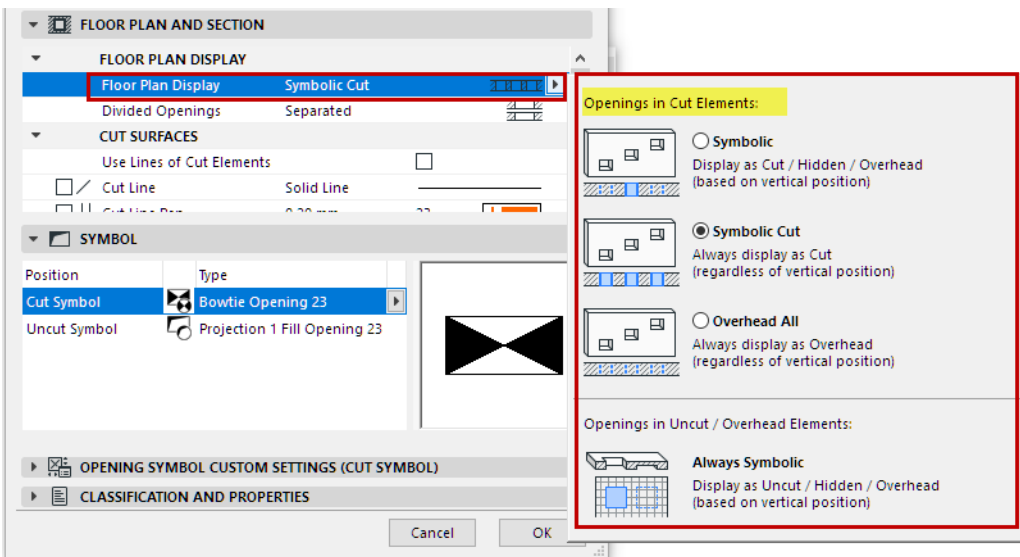


Uncut symbol, with offset and contour

Opening Display in Cut Elements

In Opening Settings: Use the Floor Plan Display controls to display openings in cut elements.

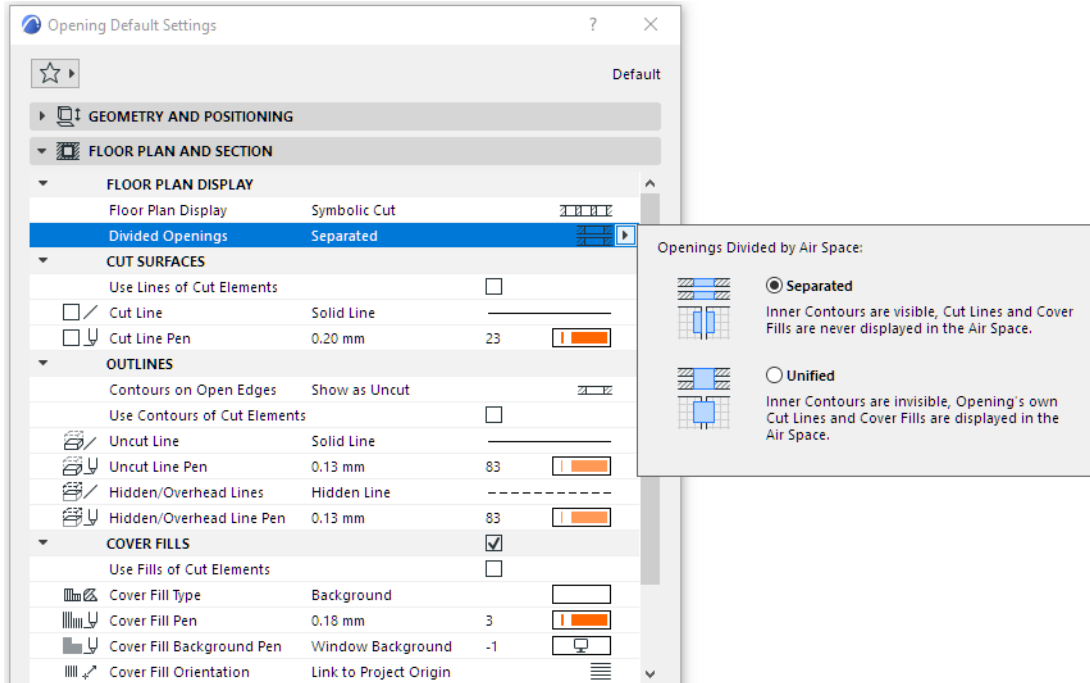
Note: Openings in Uncut/Overhead elements are always shown as Symbolic.



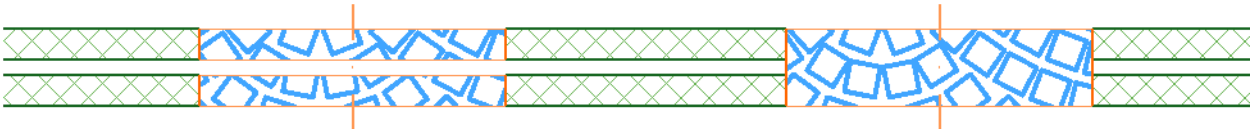
- **Symbolic Cut:** Always use Cut attributes (regardless of the Opening's vertical position)
- **Symbolic:** Use either Cut, Hidden or Overhead lines, based on the Opening's vertical position
- **Overhead All:** Always use Overhead attributes (regardless of the Opening's vertical position).

Divided Openings: Separated or Unified

For an Opening that cuts through multiple elements, or a composite element, separated by air space:



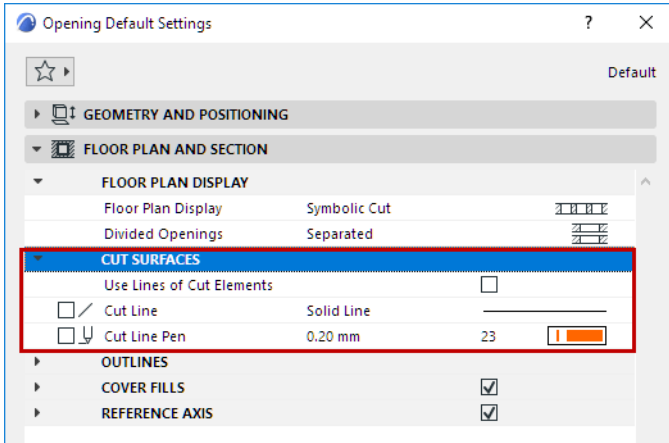
- **Separated:**
 - Inner contours (inside the air space) are visible.
 - Cut lines and cover fill are not shown.
- **Unified:**
 - Inner contours (inside the air space) are not visible.
 - Cut lines and cover fill are displayed.



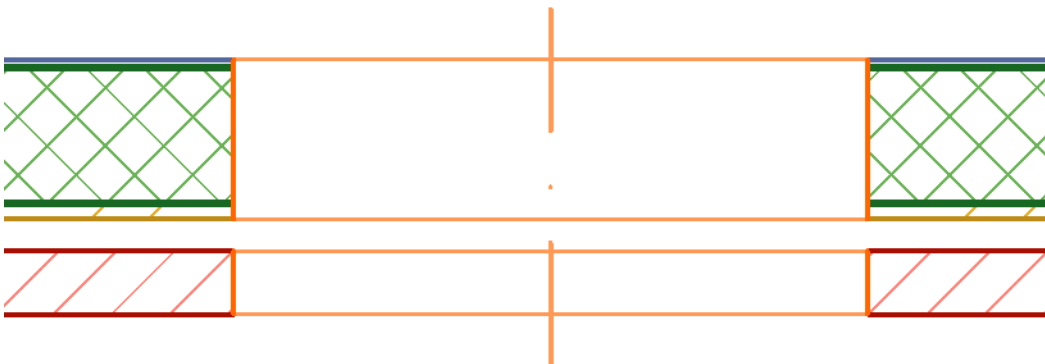
Openings Divided by Air Space: Separated vs. Unified

Cut Surfaces

For the element surfaces cut by the opening: check the box to **Use Lines of Cut Elements**. Otherwise, the Opening's own cut lines (defined in this dialog box) will be used.



Use Lines of Cut Elements

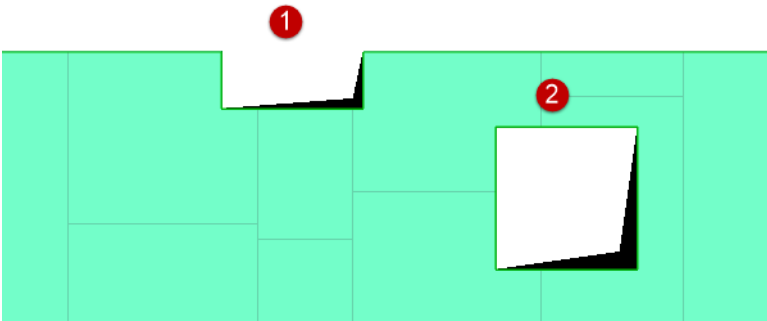


Use Opening's Own Cut Lines

Opening Outlines on Floor Plan

Contours on Open Edges

“Open Edges” are Opening contours that do not intersect any other element, and do not coincide with another element’s edge.



Openings with Open (1) and Bound (2) edges

Display these Open Edges as **Uncut**, **Overhead** or **Hide**:

Opening Default Settings			
Default			
GEOMETRY AND POSITIONING			
FLOOR PLAN AND SECTION			
FLOOR PLAN DISPLAY			
Floor Plan Display	Symbolic Cut		
Divided Openings	Separated		
CUT SURFACES			
OUTLINES			
Contours on Open Edges	Show as Uncut		
Use Contours of Cut Eleme...		<input type="checkbox"/>	
Uncut Line	Solid		
Uncut Line Pen	0.13 mm	83	
Hidden/Overhead Lines	Hidden		
Hidden/Overhead Line Pen	0.13 mm	83	
COVER FILLS			
REFERENCE AXIS			

Contours on Open Edges:

- Show as Uncut**
Show contours on open edges as uncut.
- Show as Overhead**
Show contours on open edges as overhead.
- Hide**
Hide contours on open edges.

Contours on Bound Edges (eg. where the Opening's contour coincides with the Slab contour) are always displayed as the associated element's contour. This option has no effect.



Open Edges as Uncut - Show as Overhead - Hide



Open Edges as Uncut - Show as Overhead - Hide

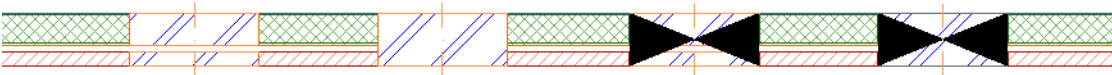
Note: This setting has no effect on Openings that coincide with e.g. a Slab edge. Such edges are “Bound edges,” and their display (Overhead or Uncut) depend on the Slab’s contour display.

Opening Contours

Define attributes for Uncut and Overhead contours. You can **Use Contours of Cut Elements**. Otherwise, the Opening’s own Uncut/Overhead attributes (defined in this dialog box) will be used.

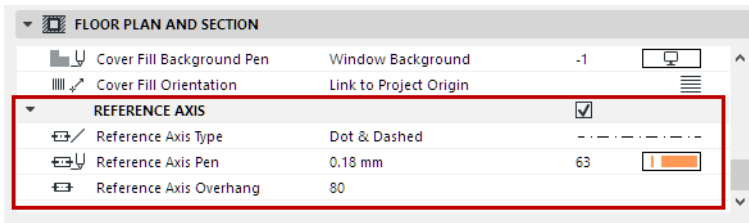
Opening Cover Fills

See [Assign a Cover Fill](#).



- For “separated” display of divided openings, the cover fill is not shown in the air space.
- The Opening’s symbol is placed on top of the opening (including its cover fill, if any.)

Opening Reference Axis

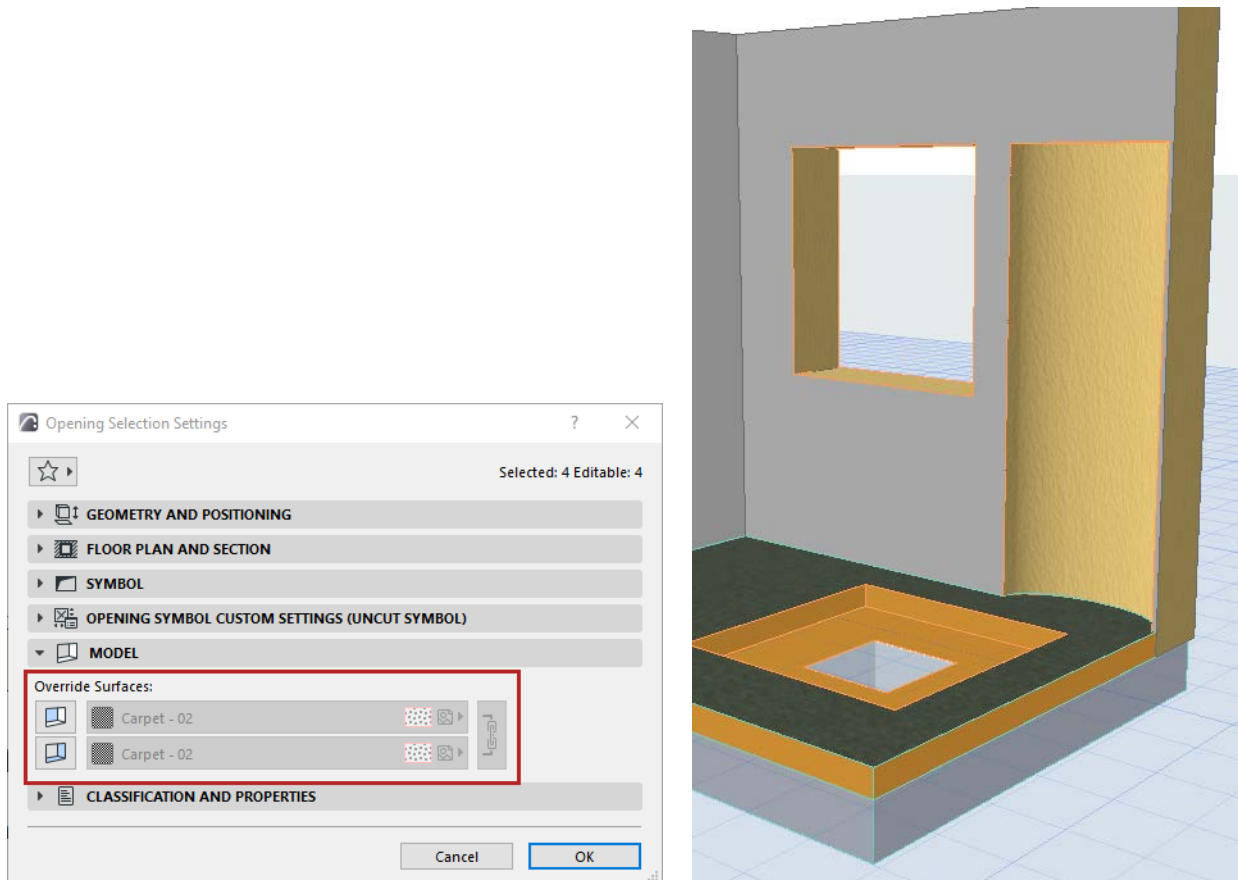


Use these controls to define the Reference Axis of the Opening as shown on Floor Plan and Section.

Note: To show/hide Opening Reference Axes by view, use Model View Options. See [Model View Options for Construction Elements](#).

Opening Display in 3D

By default: the Opening displays the host element's surface, with no override.



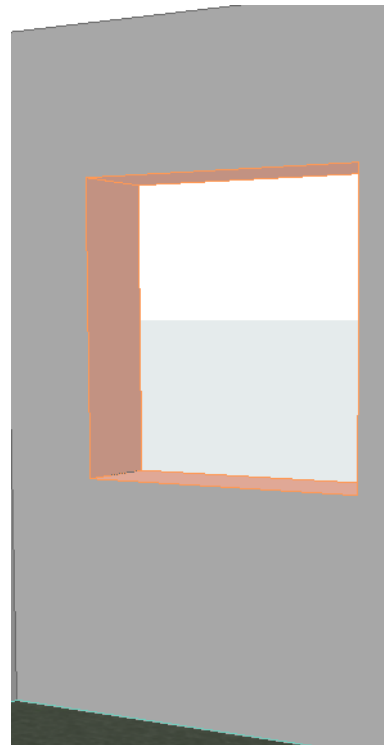
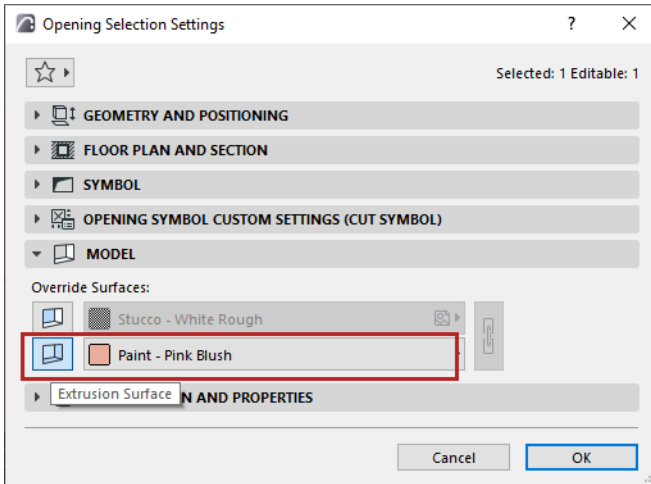
Override Opening Surface

If needed, you can apply a different surface to the Opening extrusion and/or ends.

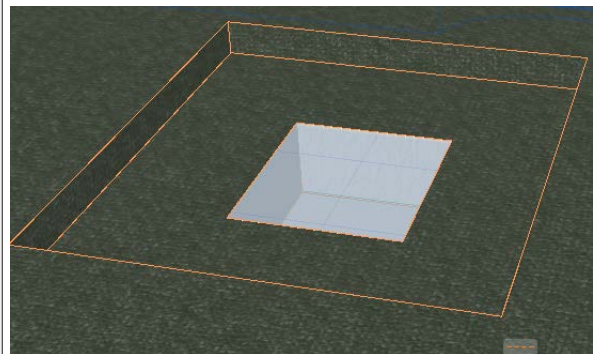
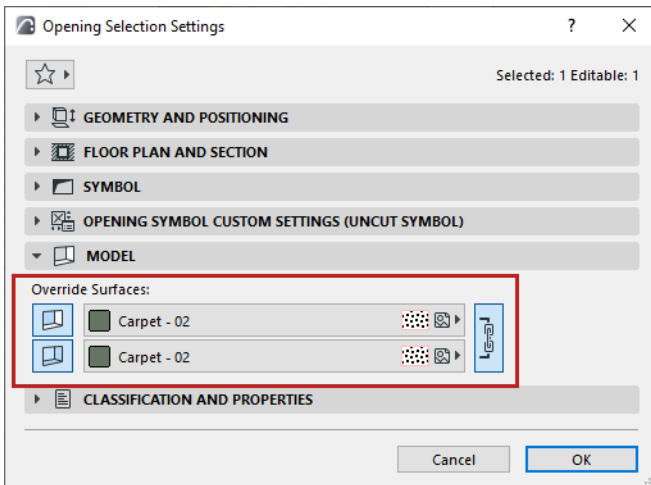
- **Extrusion:** the sides of the Opening body, where it cuts through the host elements
- **Ends:** one or two ends of a limited Opening body, where it meets but does not cut through the host element (e.g. niche or recess).

1. Select the Opening
2. Open the Override Surfaces control
 - in the Info Box, or
 - in the Model Panel of Opening Settings
3. Activate the toggle of the ends surface and/or extrusion surface
4. From the pop-up, choose the desired surface.

Note: Click the Chain icon ("Link Surfaces") to apply the same surface to ends and extrusion.



Override Opening surface - extrusion



Override Opening surfaces - both extrusion and end

Related Topic

[Opening surface in Surface Schedule](#)

Use Limited Opening to Create Niche or Recess

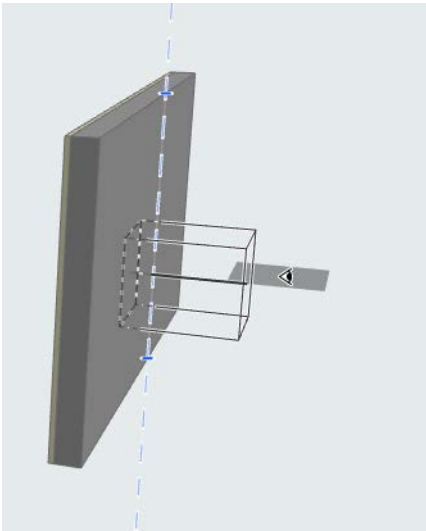
A limited Opening can be used, for example, as a Wall Niche or a Slab Recess.

Limit One Side

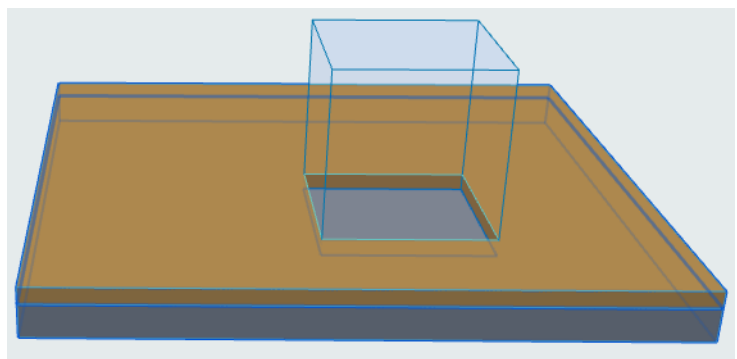
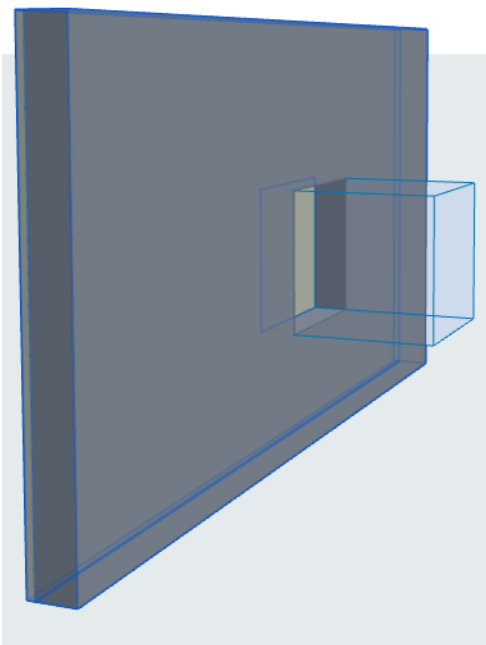
1. In Opening Settings or the Info Box, choose **Limit One Side**.



2. Click on the associated element into which to place the opening.
3. Move the eyeball cursor to see the placement options (e.g. on either side of the host Wall's reference line).



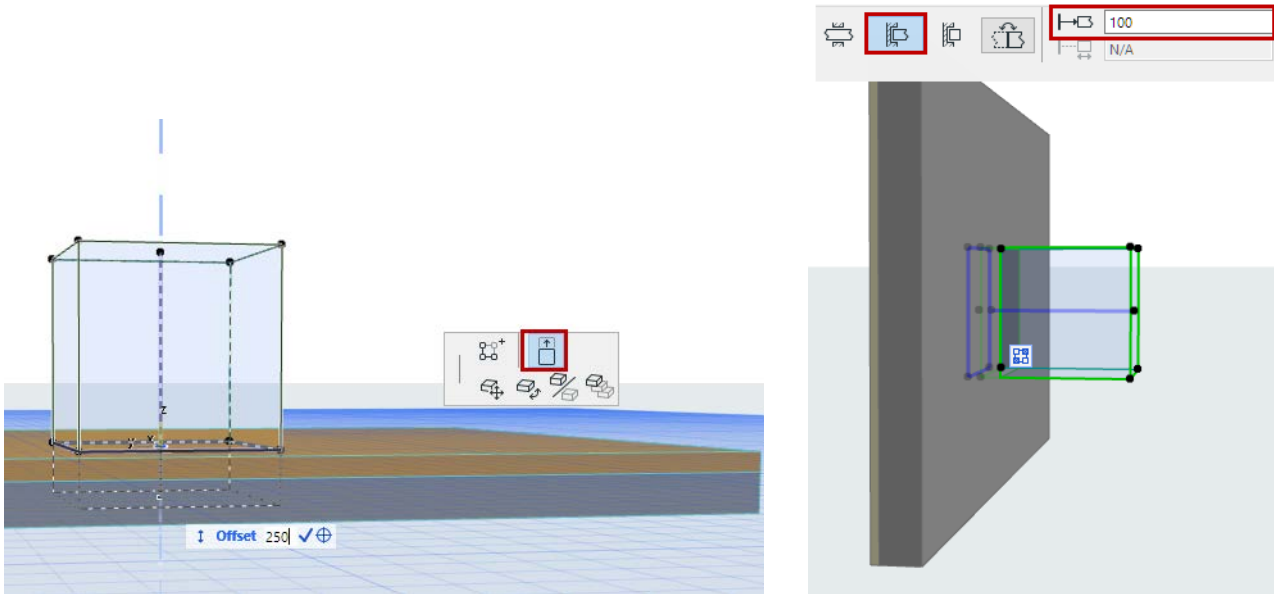
4. Click to place. The Opening's limited side coincides with the host's reference line/plane.



Offset Host Element's Reference Line/Plane

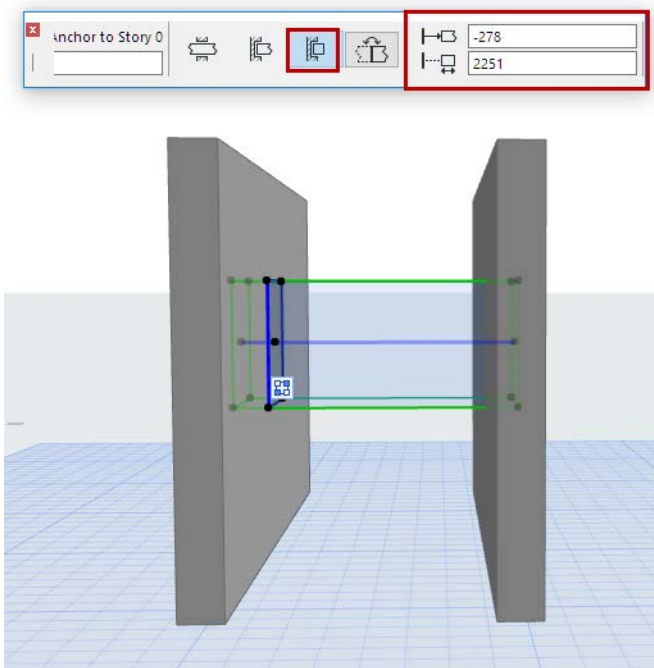
Do one of the following:

- Use **Offset from Reference Plane** command from pet palette
- Enter offset in Settings or Info Box



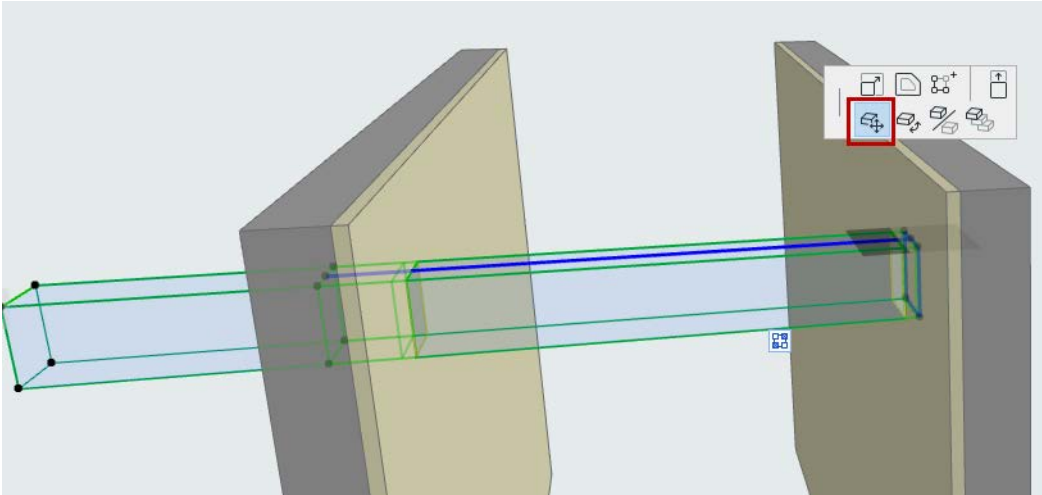
Limit Both Sides

1. In Opening Settings or the Info Box, choose **Limit Both Sides**.
2. Optionally, offset the Opening from the host's reference line/plane (enter positive or negative offset in Settings or Info Box).
3. Enter the desired Opening length, or edit it graphically.



Move Opening's Reference Line/Plane

For Openings placed with the Horizontal or Vertical method (but *not* the Aligned method): You can use the Drag command from the pet palette to reposition the Opening's reference line/plane.



Flip Opening

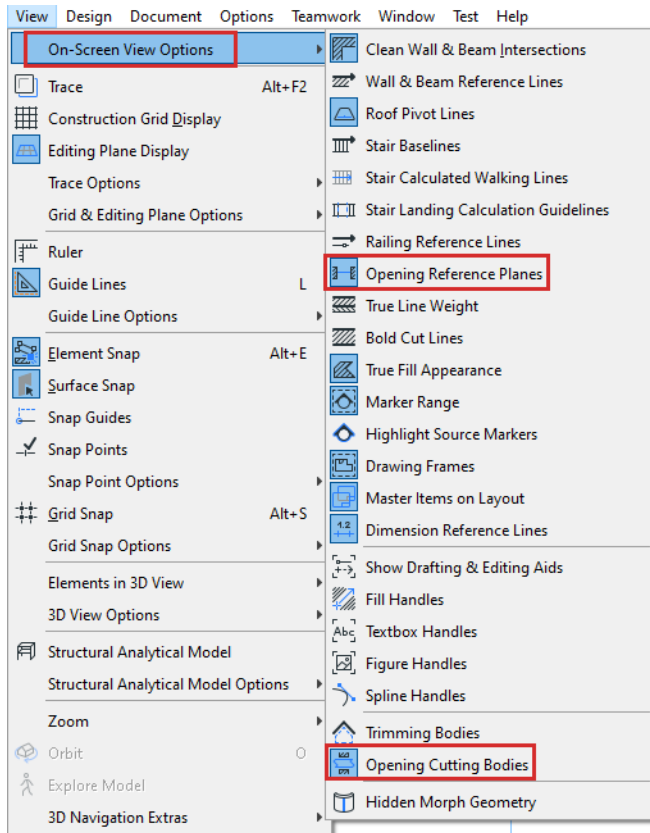
To reverse the direction of a limited Opening, click **Flip** in Opening Settings or the Info Box.

On-Screen View Options for Openings

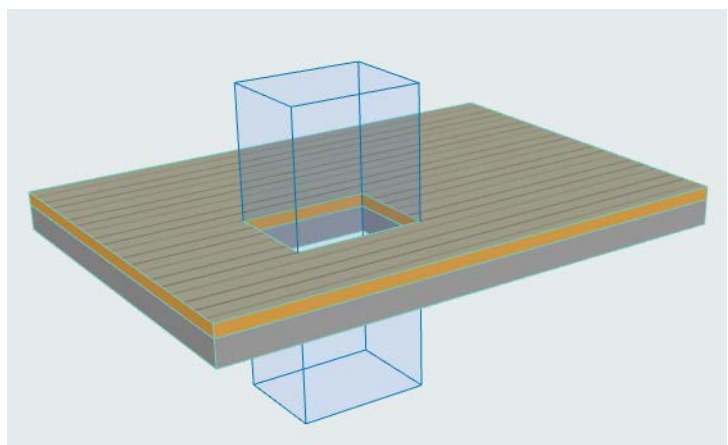
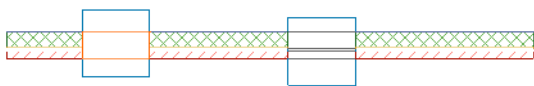
[Watch the Video](#)

Turn these options with one of the following:

- **View > On-Screen View Options**
- the On-Screen View Options toolbar.



Opening Cutting Bodies



Opening Reference Planes

This toggle ensures that an Opening will have an on-screen display on the Floor Plan, even if it is located entirely outside of its associated host element.

Edit Opening

[Watch the Video](#)

Drag-Rotate-Mirror

As with other construction elements, use **Drag**, **Rotate** and **Mirror** to freely modify the Opening position in any view. As a result, the Opening orientation setting (seen in the Info Box) may change to “Custom”.

Copy or Duplicate Opening

You can only copy/paste an Opening if you also copy/paste its associated element.

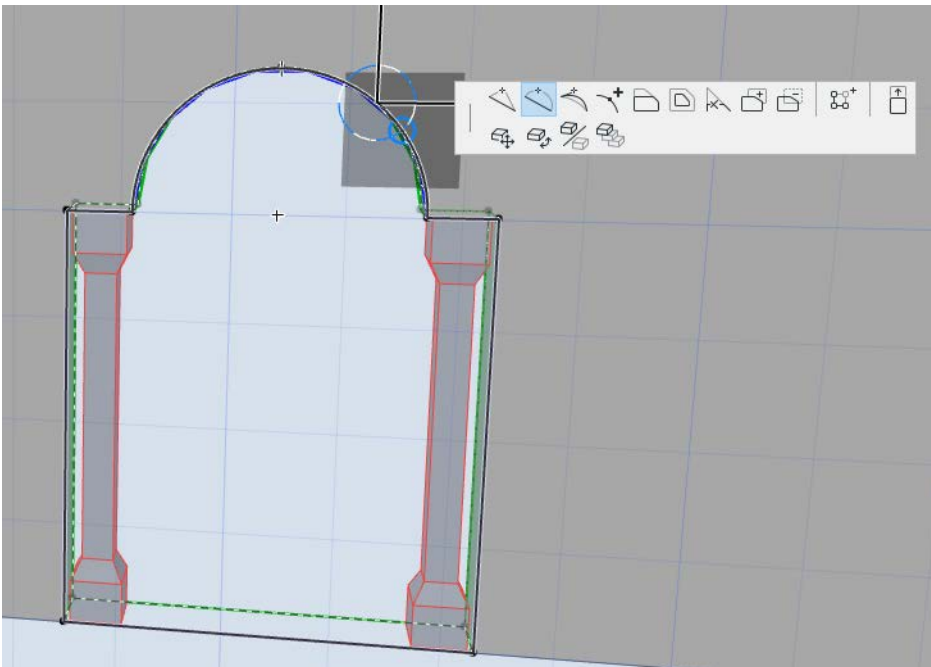
To duplicate an Opening (without moving or copying the associated element), use **Drag a Copy**

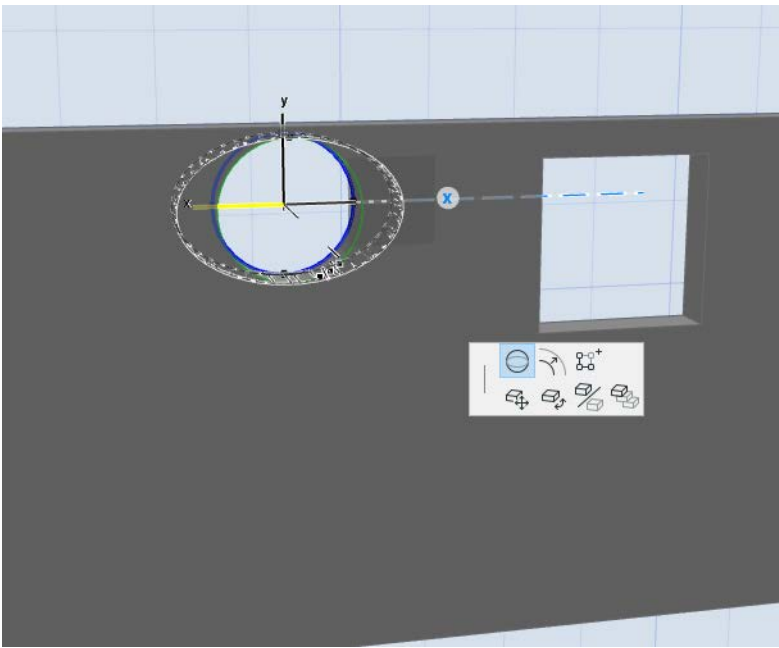
Graphical Editing

Use pet palette commands.

If you edit the shape of a rectangular or circular Opening, it changes automatically to Polygonal.

Exception: An ellipsis Opening does not become a polygon.





Manage Opening Connections

Manage an Opening and its cut elements using the commands of the Connections grip. (Each Opening is associated to a single element, but a single Opening can cut multiple elements at once).

Note: Opening geometry is measured relative to its associated element, and moves along with it.

Topics in this Section:

[Cut Additional Element \(Command\)](#)

[Cut Additional Element \(Use Connection Grip\)](#)

[See List of Elements Cut by the Opening](#)

[Redefine Opening's Associated Element](#)

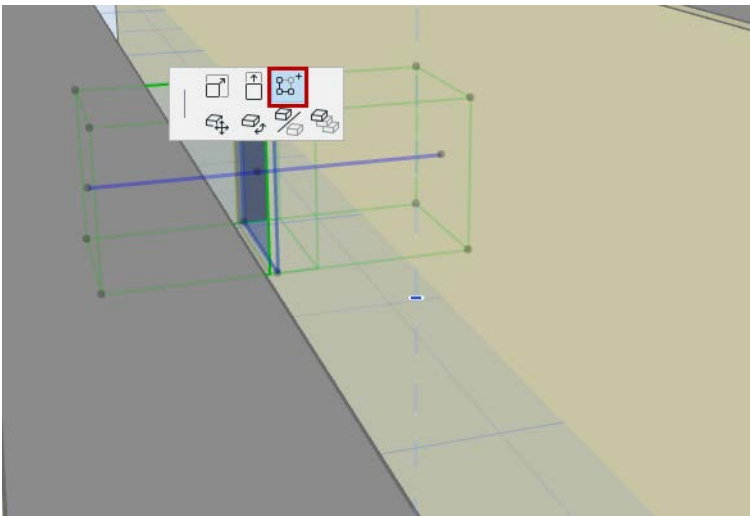
[Delete Cut Element Relationship](#)

Cut Additional Element (Command)

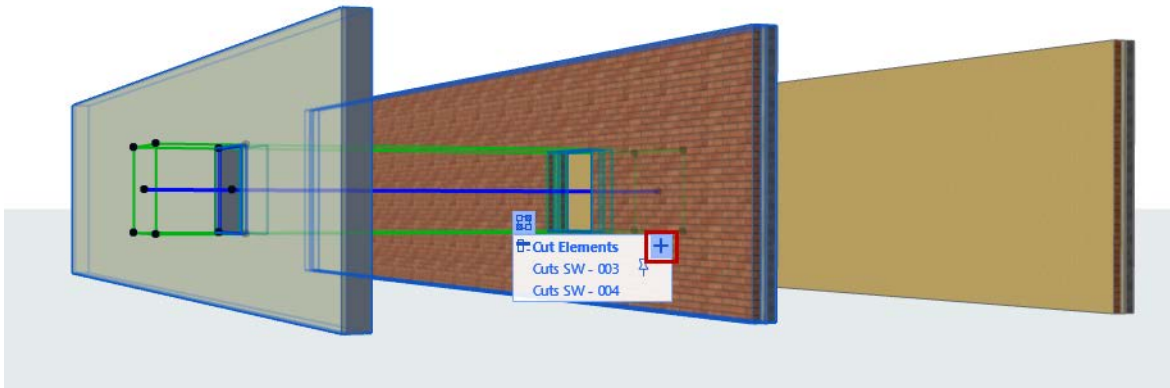
1. Select the Opening.
2. From the pet palette or context menu (Connect submenu), choose **Cut Additional Element with Opening**.
3. Move your cursor to the next element you wish to cut. Eligible elements are shown with a pre-selection highlight.
4. Click to cut the element.

Cut Additional Element (Use Connection Grip)

1. Select the Opening.
2. Click the Connections grip.



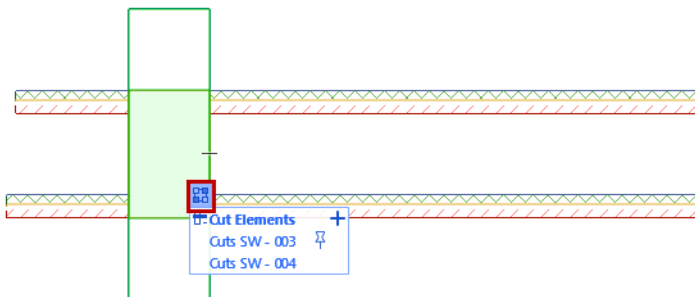
3. Hover over the list header (Cut Elements) and click the Plus sign.



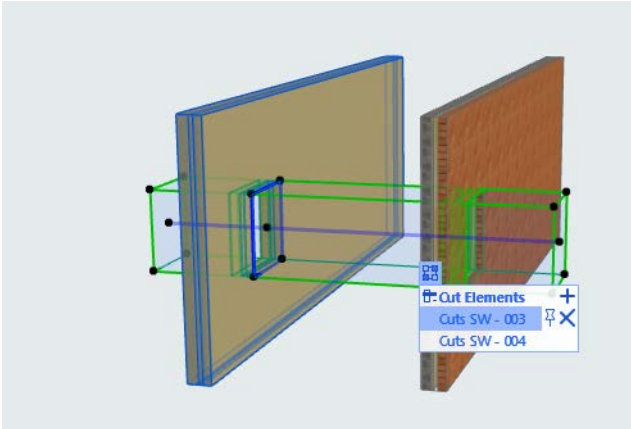
4. Move your cursor to the next element you wish to cut. Eligible elements are shown with a pre-selection highlight.
5. Click to cut the element. It is added to the Opening's Cut Elements list.

See List of Elements Cut by the Opening

1. Select a single Opening. The **Connections grip** appears.
Note: The Connections grip does not appear if multiple elements are selected.
2. Click the Connections grip to display a list of the elements cut by the Opening.
3. **Pinned Element:** this is the Opening's associated element. Opening geometry is measured relative to this associated element, and moves along with this element. Reassign the pin to any of the listed elements. (See [Redefine Opening's Associated Element.](#))



4. Move your cursor through the list: As you hover over any listed item, the corresponding cut element is highlighted in the project



To highlight *all* of the project elements cut by the selected Opening, hover over the header (**Cut Elements**) in the connection list.

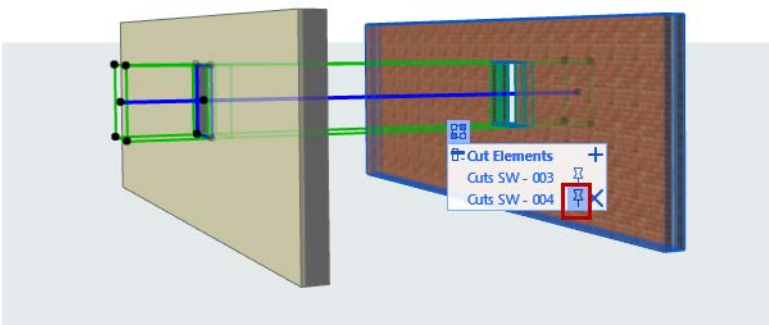
If the listed cut element is not visible in the current view (e.g. it is on a hidden layer, or on a different story), that item is listed in italic type.

Redefine Opening's Associated Element

To redefine an Opening's associated element:

1. Select the Opening.
2. Click the Connections grip.
3. In the list, hover over the Cut element that you want to redefine as the Opening's associated element.

Note: Opening geometry is measured relative to its associated element, and moves along with it.
4. Click the Pin icon.

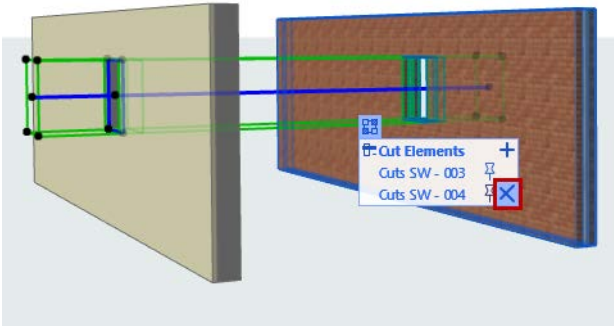


Redefine Opening's Associated Element

Delete Cut Element Relationship

1. Select the Opening.
2. Click the Connections icon.

3. In the list, hover over the element whose Cut you want to delete, and click X



Delete Cut

Note: If you delete the Opening's cut to its associated element (the one with the pin), then the Opening is deleted.

Edit Openings in Teamwork

To edit the geometry and settings of an Opening element, you must first reserve the Opening.

For example, the following interactions require that you reserve the Opening:

- Cut additional element
- Delete Cut element relationship
- Redefine the Opening's associated element (reassign Pin)

However, you do not have to reserve its associated or cut elements. For example, you can create new Openings in elements that are part of a Hotlinked Module without reserving them.

Import/Save Openings with IFC

Import IFC Openings as Archicad Openings

Imported IFCOpeningElements are converted to Openings in Archicad provided that:

- The Archicad host element is a Wall, Slab, Mesh, or Beam
- The incoming Opening is empty (does not contain a Door/Window)
- The incoming Opening shape is rectangle, circle or ellipse

IFC Openings that do *not* meet these conditions are converted to a Morph, which is in a Solid Element Operation with the host element.

Convert IFC Provision for Void to Archicad Opening

An IFC Provision for Void is a simple Building Element Proxy with a special IFC Attribute (Provision for Void) assigned to it. An IFC Provision for Void cannot contain a Door or Window.

Architects receive Provisions for Voids from other design disciplines as an IFC model. This model might contain other model elements, or just the Provision for Voids.

The Building Element Proxies that represent IFC Provisions for Voids are imported to Archicad as Morphs, and classified as Provisions for Voids.

[See also *Create Openings from Selection*.](#)

Export Archicad Openings to IFC

As IFC Opening Element

Each Archicad Opening is exported as a single IFC Opening Element (IfcOpeningElement), with a single host element.

If the Archicad Opening cuts multiple elements, then:

- The Opening part cut from the host (associated) element is exported as an IfcOpeningElement. It inherits all metadata (classification, properties, ID).
- The Opening parts cut from additional elements are represented in IFC by holes (created by solid element subtraction).

As IFC Provision for Void

Provisions for Voids that are modeled using Morphs in Archicad can be classified and exported to IFC as Provisions for Voids.

Exporting Host Elements to IFC

When exporting the host (associated) elements of Openings (e.g. Wall/Slab) to IFC, you can choose to export their gross geometry - that is, without the holes. To do this, use the “Export gross geometry” checkbox in the IFC Translator for Geometry Export. The opening elements are exported separately (IfcOpeningElement), and the exact subtraction can be carried out in IFC by the receiving application (e.g. for more accurate cost calculation).

[See *Export gross geometry of elements \(Export holes separately\)*.](#)

Create Openings from Selection

[Watch the Video](#)

Use this command to transform selected Archicad construction element(s) into Openings.

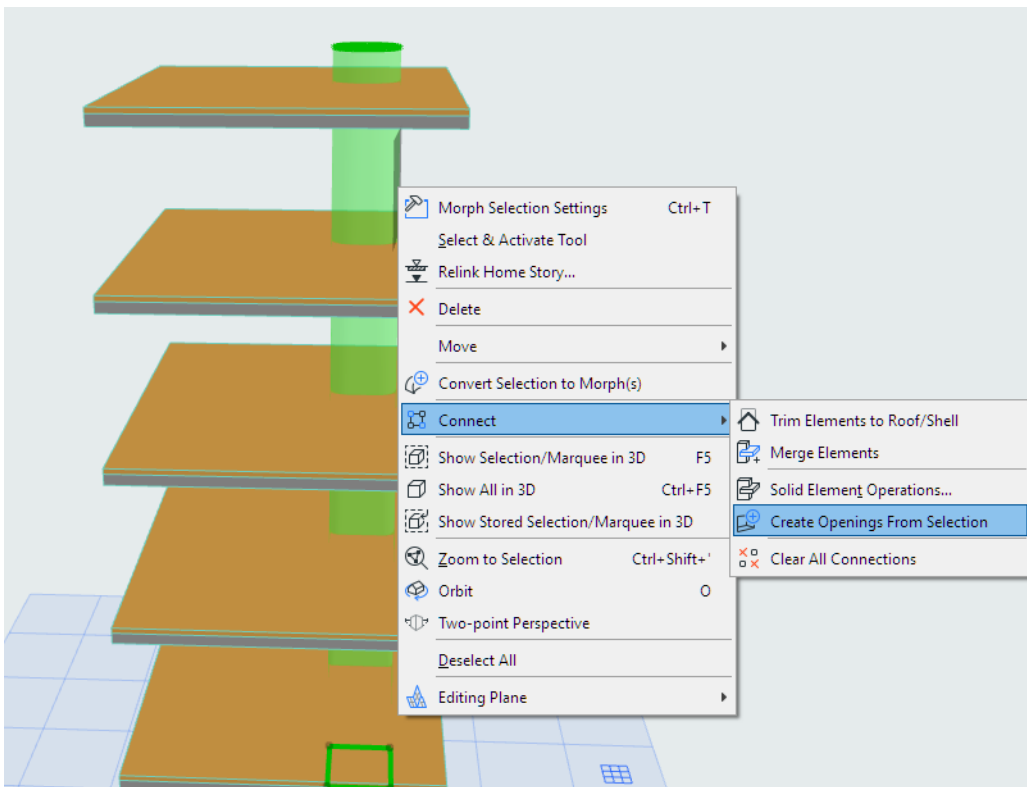
A selected Archicad element will turn into an Opening if it collides with an eligible element (Wall, Slab, Mesh, Beam).

For example: Morphs, MEP elements, and Stair elements (with headroom On) are typically transformed into Openings.

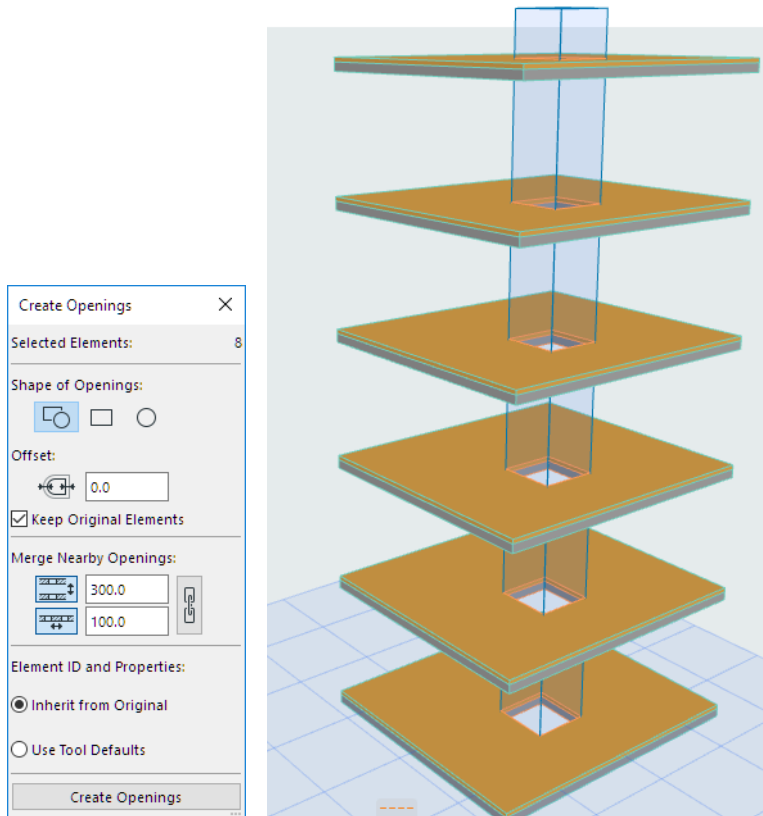
Note: If the two intersecting elements do not collide (i.e. because of automatic intersection or Solid Element Operation), no Opening is created.

For example, an imported file may contain a multi-story Morph representing mechanical voids.

1. Select the element (here, the Morph).
2. Use the **Create Openings from Selection** command
 - from the context menu
 - from **Design > Connect** menu



- Set options in the appearing palette, then click **Create Openings** at the bottom of the palette.



Shape

- Automatic: Openings will match the original element geometry as closely as possible
- Rectangular or Circular

Offset

Optionally, add an offset to the Opening.

Keep Original Elements

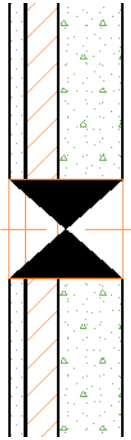
Uncheck to delete the original elements.

Merge Nearby Openings

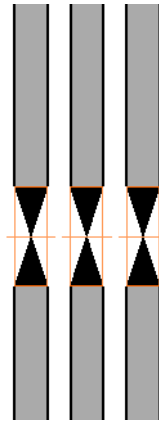
Check this to merge Openings that are closer than the tolerance distance defined here.

- Merge lengthwise: This can be useful in parallel Walls that are close together, or Slabs on top of one another.
- Merge sideways: For example, with MEP pipework, in which parallel pipes are very close to each other.

If you do not check this box, then a separate Opening is created in each host element.



Merge nearby Openings



Do not merge nearby Openings

Element ID

For the newly created Opening element(s):

- Retain the ID of the original element (e.g. MORPH-002), or
- Assign a new ID based on Opening Tool defaults

Model Element Connections

Archicad enables you to create complex structures by combining model elements with each other.

There are several types ways to connect elements:

- **Automatic Priority-Based Junctions:** Colliding construction elements will generally create correct intersections in all views, based on the elements' (or skins') respective Intersection priorities, as defined in their Building Materials.
- **Trim** (use a Roof or Shell to trim any model element geometrically)
- **Merge Elements** (to connect Roofs, Shells and Morphs with other elements, enabling priority-based junctions to take effect)
- **Solid Element Operations (SEO)**
- **Join** (between one or more Walls and a Curtain Wall)

What all of these functions have in common is that they create an associative connection between two elements. "Associative" means that even though the connected elements remain separate elements, if you change one element's geometry, then the connected element will automatically adjust itself as needed.

Each function is described in detail in the following sections:

[Element Intersections](#)

[Trim Elements to Roof or Shell](#)

[Join Wall to Curtain Wall](#)

[Solid Element Operations](#)

[Merge Elements: Roofs, Shells, Morphs](#)

[Managing Element Connections](#)

Element Intersections

When they intersect each other, Archicad model elements will “clean up” in predictable, largely automatic ways. The following sections explain how model elements intersect in Archicad:

[Basic Intersection Principles](#)

[Specific Element Intersections and Junctions](#)

[How to Make Sure Elements Collide for Best Cleanup Results](#)

[Intersecting Elements That Don't Clean Up](#)

[Best Practices for Intersections: Conceptual Design Phase](#)

[Fine-Tune Intersections](#)

[Legacy Intersection Mode for Older-Version Projects](#)

Basic Intersection Principles

In Archicad, the basic principle of intersections (or junctions) is that the construction elements must collide in space. All construction elements will clean up correctly in all views, provided that they **collide**.

- In the case of **Shells, Morphs** and **Roofs**, they must collide, and they must also be **merged** or **trimmed** via the commands of the Connect menu.

[See Merge Elements: Roofs, Shells, Morphs and Trim Elements to Roof or Shell.](#)

The resulting intersection depends on the elements' **Building Material priorities**, set in the Building Materials dialog box. The element (or skin) with the higher-priority Building Material will cut the one of a lower priority.

[See Intersection Priority \(Building Material\).](#)

This works for the following elements: Wall, Beam, Column, Slab, Roof, Shell, Morph – whether basic, complex profile, or composite. (All these elements use at least one Building Material.)

Automatic Intersections in Archicad

	Wall	Beam	Column	Slab	Roof	Shell	Morph
Wall	Collision and Ref. Line	Collision	Collision or Wrapping	Collision	Connect	Connect	Connect
Beam		Collision or Ref. Line	Collision	Collision	Connect	Connect	Connect
Column			SEO	Collision	Connect	Connect	Connect
Slab				SEO	Connect	Connect	Connect
Roof					Connect	Connect	Connect
Shell						Connect	Connect
Morph							Connect

Priority Support

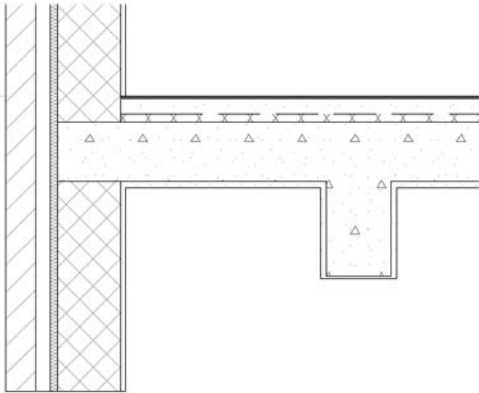
Intersection display in 3D Model

	Building Material Priority (Automatic)	Skin-level
	Building Material Priority (Requires Merge/Trim)	Skin-level
	No Priority Support; Element Cut Only	Element-level only

- Intersections are **model-based**: the connections physically modify the geometry down to the skin level. This means that they will be correct in all model views, including the 3D window and in lists or schedules, as well as on Floor Plan and Section
Note: Intersections that include elements in symbolic display in Floor Plan may not match the 3D model representation; elements whose display is based on their 3D projection will be correct.
- **Layer Intersection Groups** provide an additional means of control: elements whose layers have differing intersection numbers never intersect.

Note: Layer Intersection numbers have nothing to do with element priorities; they serve merely as tags or labels (e.g. 1 or 2) for grouping elements.

See [Use layers to prevent intersections.](#)



Junction cleanup based on skin priority

Specific Element Intersections and Junctions

See the following sections for specifics on element intersections:

[Wall-Wall Intersections](#)

[Beam-Beam Intersections](#)

[Wall-Beam; Wall-Column; Column-Slab; Column-Beam, Wall-Slab](#)

[Wall-Slab Intersection: Slab Finishes and Opening Thresholds](#)

[Intersections involving Shell, Roof, or Morph](#)

[Intersecting Elements with Identical Priorities](#)

[Line Elimination Between Model Elements](#)

[Curtain Wall Frame Intersections](#)

Wall-Wall Intersections

Requires Reference Line intersection

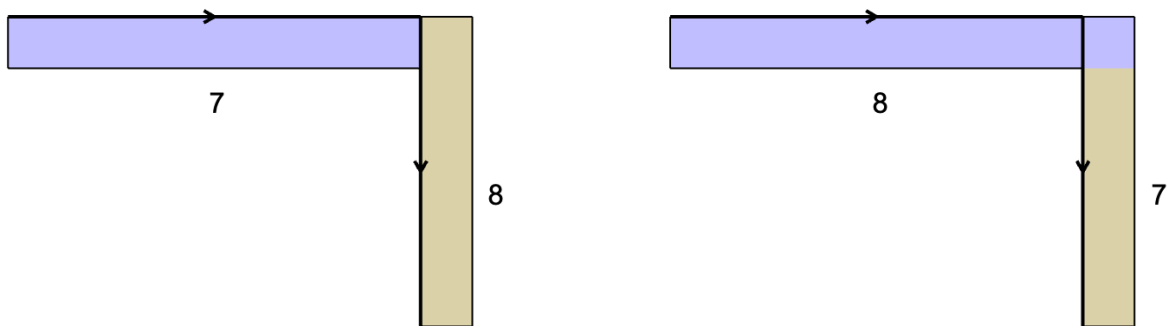
For Wall-Wall junctions, it is not enough that they collide: their reference lines must also intersect. The element (or skin) with the higher-priority Building Material will cut the one of a lower priority.

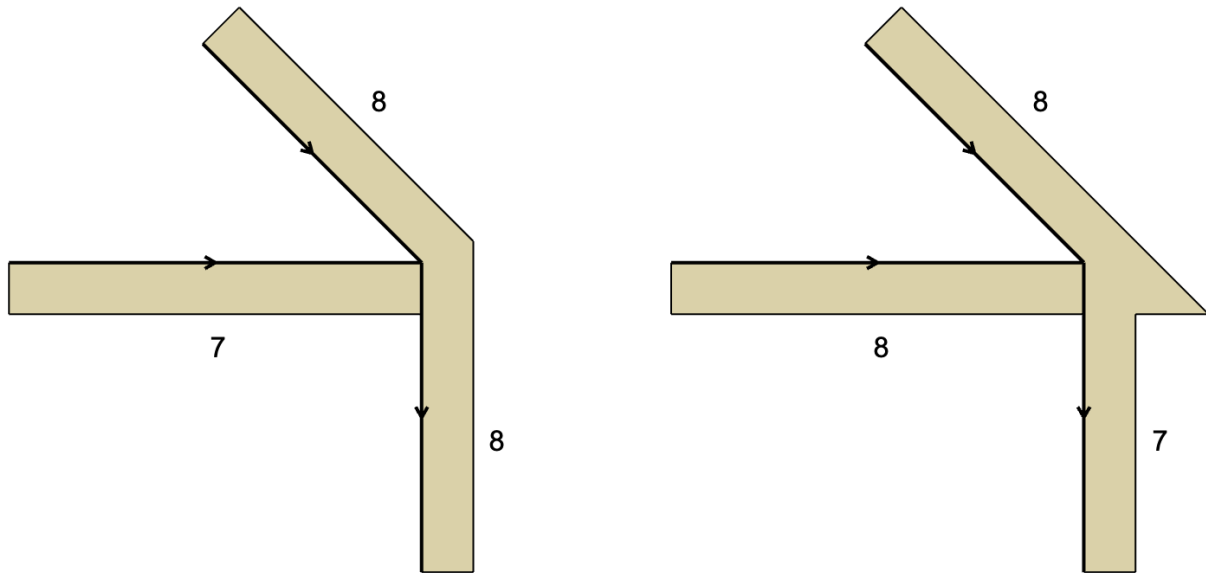
For Equal-Priority Walls: Set Junction Order

If two or more Walls meet (their reference lines intersect), use **Junction Order** to control the element level junction. Changing Junction Order determines which two Walls meet first, before others join the intersection.

Set **Junction Order** in the Model Panel or Info Box of Wall Settings.

[Watch the Video](#)





Beam-Beam Intersections

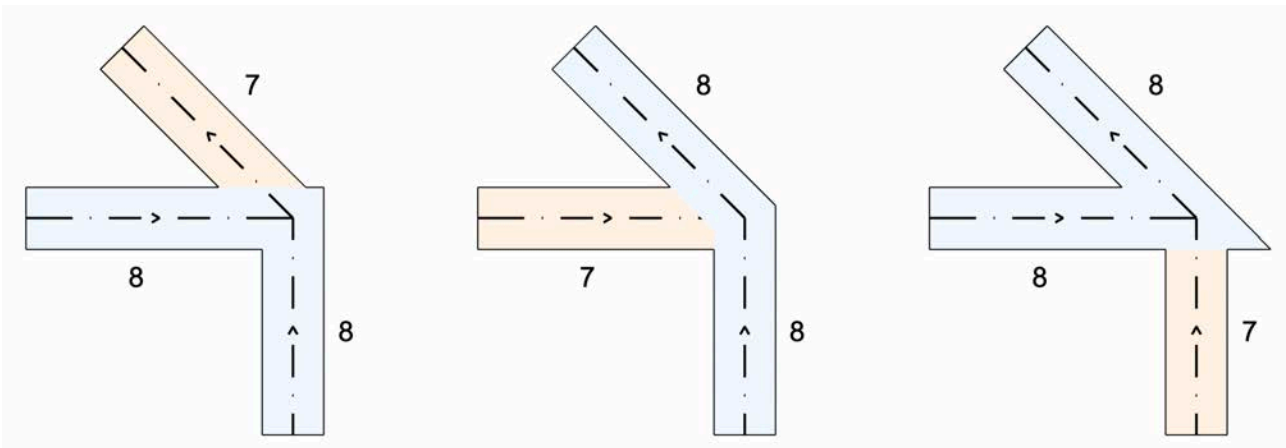
Requires Geometry Collision or Reference Line intersection.

For maximum flexibility in Beam-Beam intersections, make sure their reference lines intersect. In this case, the Beams do not actually have to collide. The Beams will auto-stretch into each other along their reference lines.

You can also intersect beams by simply colliding them, even if the reference lines do not intersect. In this case, however, the Beams will stop each other; the Beams will not auto-stretch further along their reference lines. The element (or skin) with the higher-priority Building Material will cut the one of a lower priority.

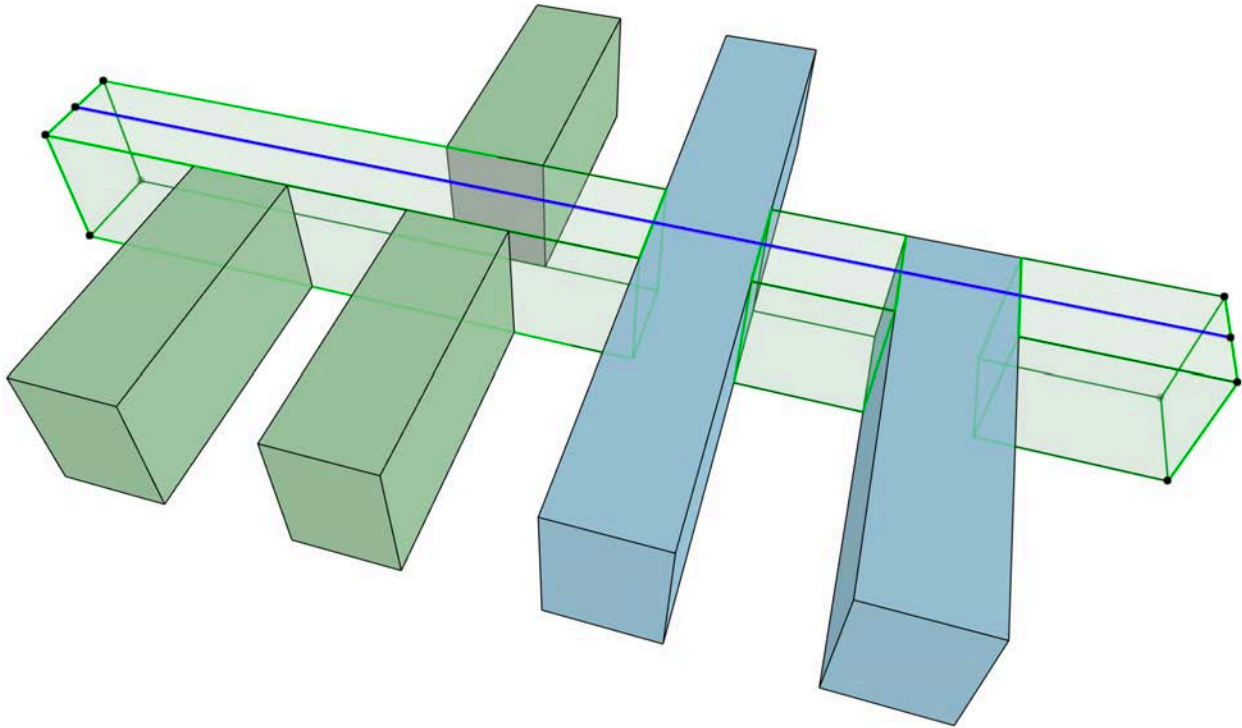
[Watch the Video](#)

If two or more Beams meet (their reference lines intersect), then use **Junction Order** to control the junction.



Here, the Beams use Junction Orders as follows:

- Transparent Beam = 8
- Weaker green Beam = 7
- Stronger blue Beam = 9



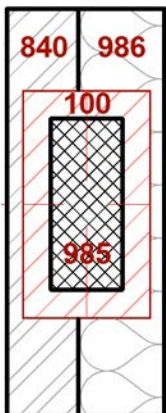
Wall-Beam; Wall-Column; Column-Slab; Column-Beam, Wall-Slab

Requires Geometry Collision.

The element (or skin) with the higher-priority Building Material will cut the one of a lower priority.

Exception for Wrapped Column intersections with Composite Walls:

When placing a “Wrapped” Column that intersects a composite Wall, the Column will break through the core skins of the composite Wall regardless of skin priorities.



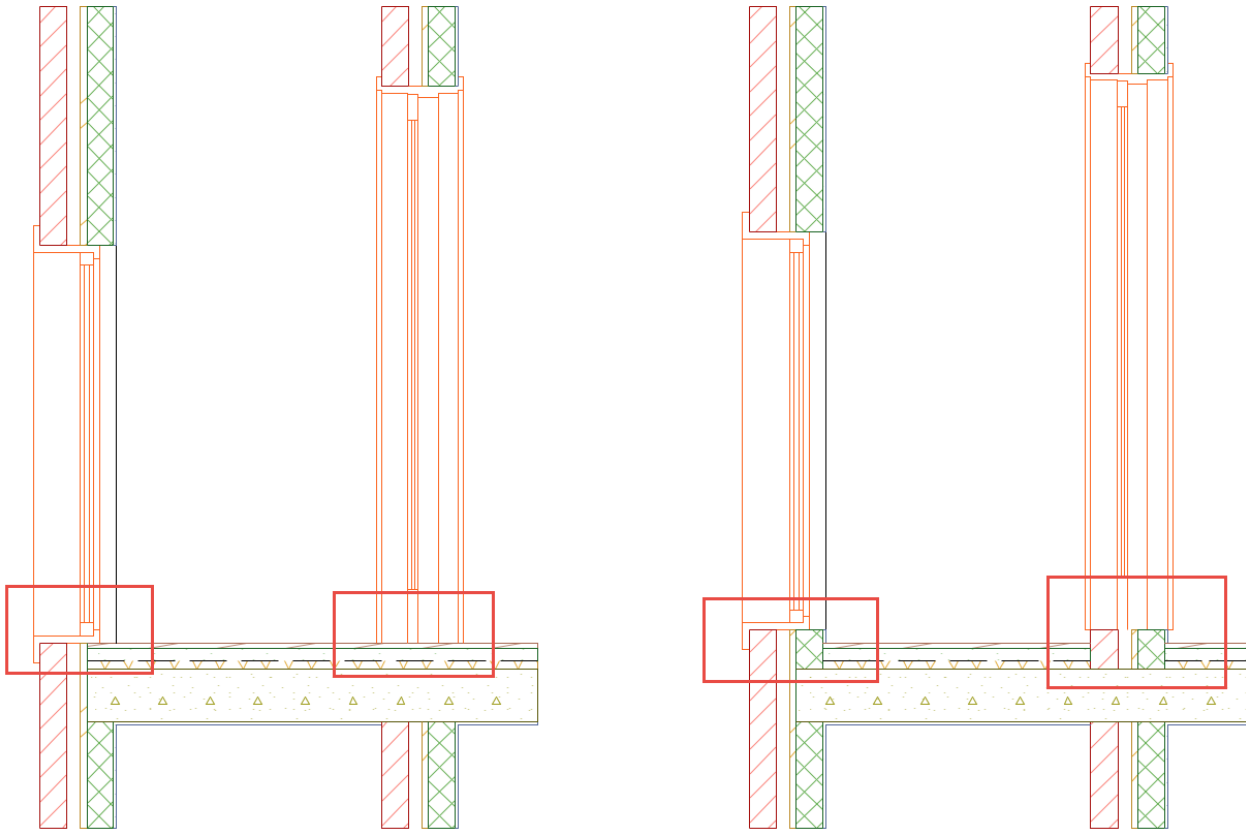
Wrapping Wall skins will be added to the column veneer in 3D. However, they will not be calculated in listings.

[Watch the Video](#)

Wall-Slab Intersection: Slab Finishes and Opening Thresholds

A special case pertains, to allow continuous finishes between rooms within an interior openings threshold.

When an Opening (Door or Window) in a Wall is flush with the top of a Slab, then the Slab's finish skins will continue through the Wall regardless of skin intersection priorities (down to the Slab's core skin). If the Opening is not flush with the Slab, then the regular priority-based connections are in effect.



In the image:

Left: Window and Door are flush with the Slab's top. Therefore, the Slab's finish skins do not form priority-based junctions with the Wall's skins. Instead, the Slab's finish skins remain continuous.

Right: Window and Door are not flush (they are elevated above the Slab), so the usual rules for priority-based connections are in effect between the Slab and Wall skins.

Intersections involving Shell, Roof, or Morph

Requires Geometry Collision, and Trim or Merge command.

If your intersection includes a Shell, Roof or Morph, you must use **Design > Connect > Merge Elements** to ensure correct intersections.

If a Shell or Roof acts as a trimming element (**Design > Connect > Trim Elements to Roof/Shell**) it will be merged with the trimmed element; you do not need to use the Merge Elements command again to achieve correct intersections.

Once merged, the element (or skin) with the higher-priority Building Material will cut the one of a lower priority.

Related topics:

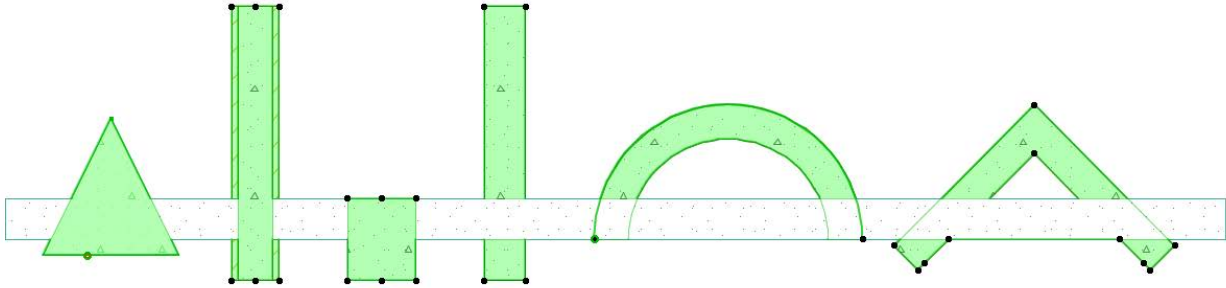
[Trim Elements to Roof or Shell](#)

[Merge Elements: Roofs, Shells, Morphs](#)

Intersecting Elements with Identical Priorities

When various, colliding construction elements have the same intersection priority (for example, elements that have the same building material), the following intersection order will prevail:




Morph > Column > Beam > Slab > Wall > Shell > Roof




Line Elimination Between Model Elements

If two elements (same or different construction element types) meet in a model view, such as Section, Elevation and 3D Document, the following line elimination rules apply.

- Same Building Material: Both lines are eliminated, hatches are visually merged
- Different Building Materials
 - Different Line Weight: The line with the thicker pen will remain, regardless of Line Type
 - Same Line Weight, different Lines: The solid line will be kept, and the other lines are eliminated.

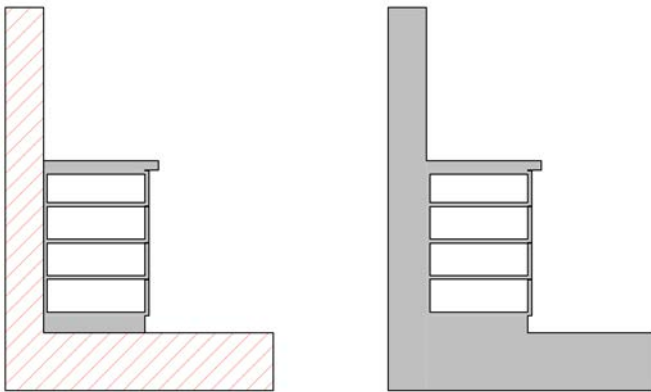
Walls in Section: Which Lines Are Eliminated?	
Same Building Material Both lines are eliminated	
Different Building Materials Same Cut Fill	
Different Building Materials Same Line weights Solid line is kept , other lines eliminated	

Walls in Section: Which Lines Are Eliminated?	
Different Building Materials Different Line weights Thicker line is kept (regardless of Line Type)	

Notes:

- These rules don't apply on the Floor Plan, where all lines are drawn. To determine which line is visible, you can use Display Order. [See Display Order.](#)
- For Composite and Complex Profile elements, line control is customizable by the user in the attribute settings dialogs.

If one or both elements do not use a Building Material (e.g. two objects), the above logic is applied to the Cut Fills. Thus, identical Cut Fills between two objects will eliminate both lines.



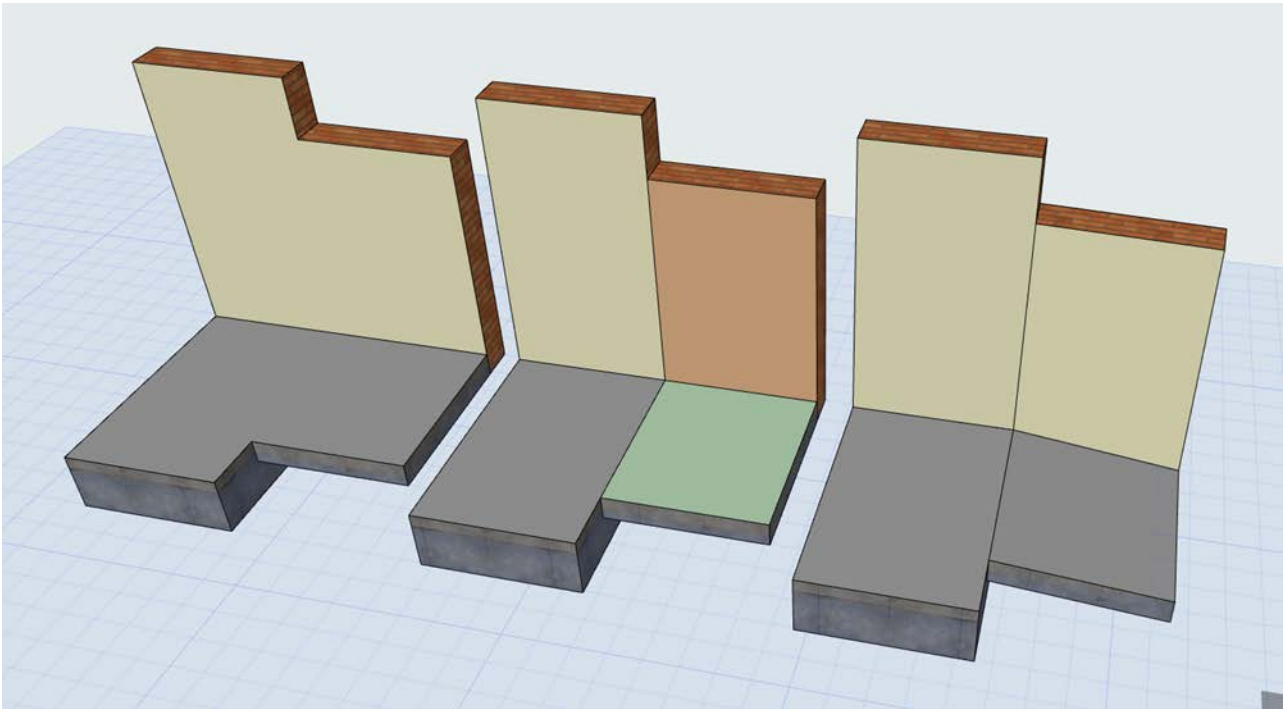
Wall/Slab and Kitchen Cabinet in Section - from left:

Cabinet has different Cut Fill; Cabinet has the same Cut Fill.

- If you want to retain the dividing line, change the Layer Intersection Group number of one of the elements. [See Use layers to prevent intersections.](#)

In Elevations and 3D views, dividing lines will always be eliminated between two elements when:

- they have the same surface, and
- the surfaces are co-planar.



Walls/Slab/Roof in 3D - from left: Same Surface, co-planar; Different Surfaces, co-planar; Same Surfaces, in different planes

How to Make Sure Elements Collide for Best Cleanup Results

Use the following techniques and features to ensure that your elements collide as needed.

Define Wall/Slab Core as the Reference Line/Plane

Place the Slab reference plane at the top of the core.

Place the Wall reference line along the Wall core (outside or inside).

You can assign a Slab's reference plane, or a Wall's reference line, to run along the element core, in addition to its faces.

This way, when you edit the Wall or Slab (e.g. adds skins or changes the thickness), the Wall/Slab core connections remain constant, and the elements' positions relative to their reference line remain unchanged.

Related topics:

[Wall Reference Line](#)

[Slab Reference Plane](#)

Link Wall/Column/Zone Tops to Story Above

You can link the tops of Walls, Columns and Zones to a story. Use this feature to ensure that these elements stay properly connected to the structure or ceiling above them even after you edit story heights.

[See *Define Toplink for Wall, Column, Zone or Stair*](#).

Adjust Elements to Slabs

Use this function (**Edit > Reshape > Adjust Elements to Slabs**) to easily adjust elements (Walls, Columns, or horizontal Beams) to specific levels of one or more Slabs that are located above and/or below them.

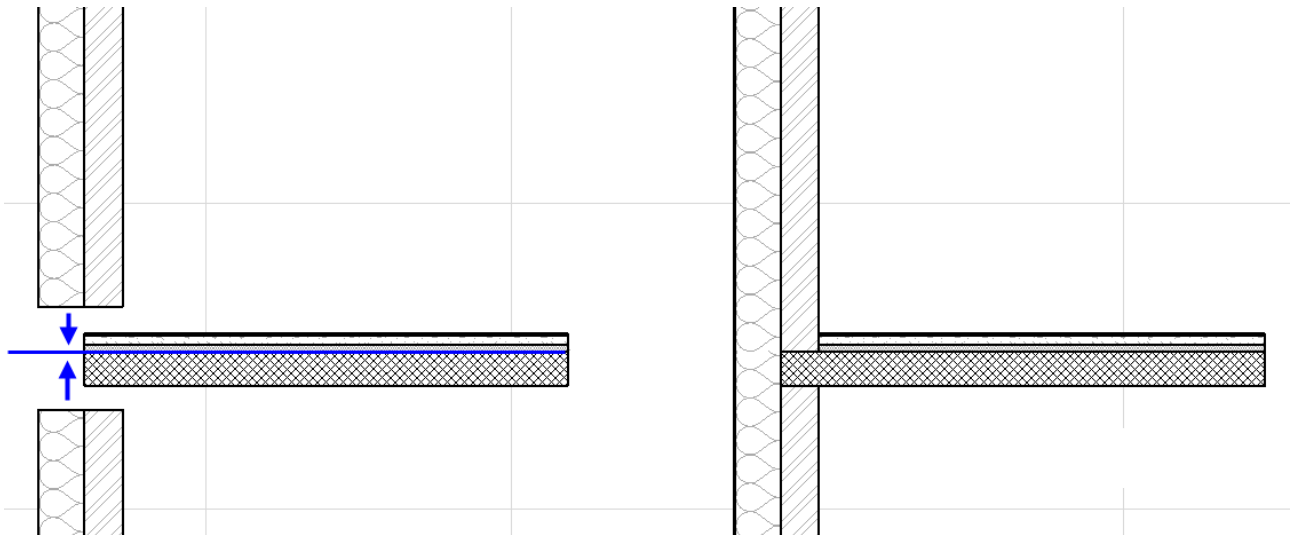
Suppose you are migrating an older-version Archicad file in which the Wall bases are placed on the tops of Slabs. As of Archicad 17, the Walls can extend past the Slab's finish level to its core, which is what you want.

To speed up this process, select multiple Walls and then use the **Adjust Elements to Slabs** command to extend them all to the Slab core with a single click.

[See *Adjust Elements to Slabs*](#).

The intersections will now clean up nicely, provided that your Building Materials are consistent. The element (or skin) with the higher-priority Building Material will cut the one of a lower priority.

Note: If a Door/Window is linked to the bottom of a Wall, the Adjust Elements to Slabs command will automatically recalculate the offset to retain the opening position.



Anchor Door/Window to a Story

When placing new Doors/Windows, it is best to anchor them to a Story level rather than to the base of the Wall, to ensure they remain in place.

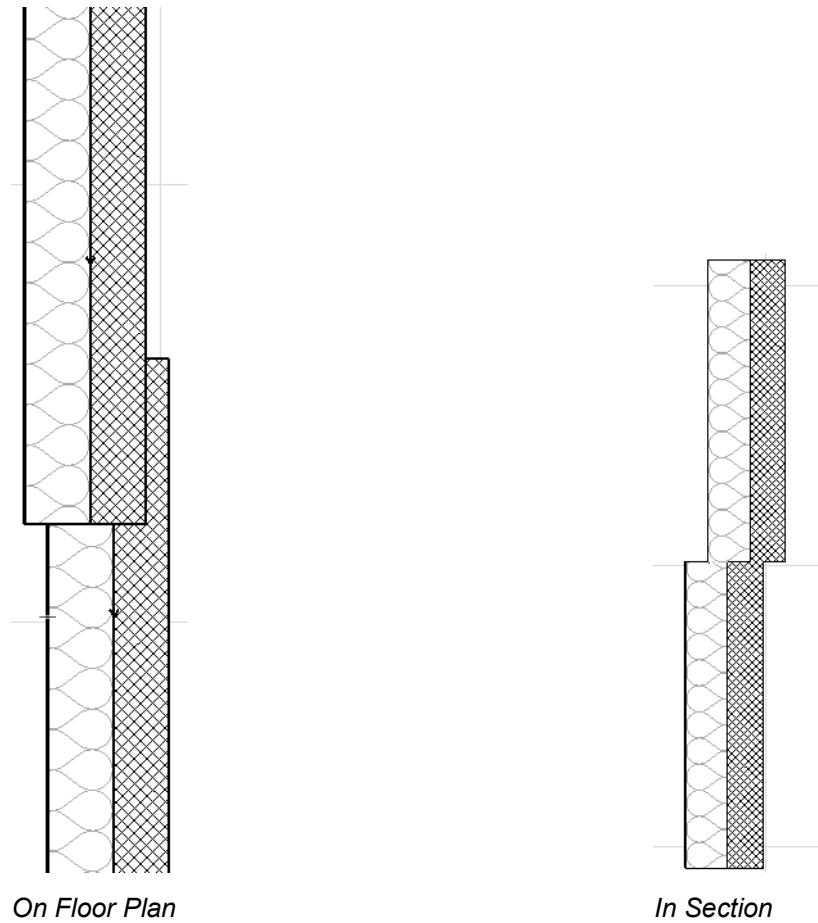
However, if your Door/Window is linked to the bottom of a Wall, the Adjust Elements to Slabs command will automatically recalculate the offset to retain the opening position.

Let the Cursor Help You

When you work with composite elements, the cursor can snap to each separate skin and component in all model views. This way you can make sure that you are aligning elements correctly to the core of other elements.

Intersecting Elements That Don't Clean Up

- Wall-Wall intersections clean up only if their reference lines intersect. (Collision is not supported.)



On Floor Plan

In Section

- Column-Column intersections do not clean up when they collide.
- Use a profiled column or the Merge Profiles function to achieve special Column intersections.
- When placing a Column that will intersect a Composite wall, your choice of Geometry method – wrapped or unwrapped – will affect the display of the wall/column in the Floor Plan.
- Slab-Slab intersections do not clean up when they collide.
- Mesh is not supported.
- Objects are not supported.

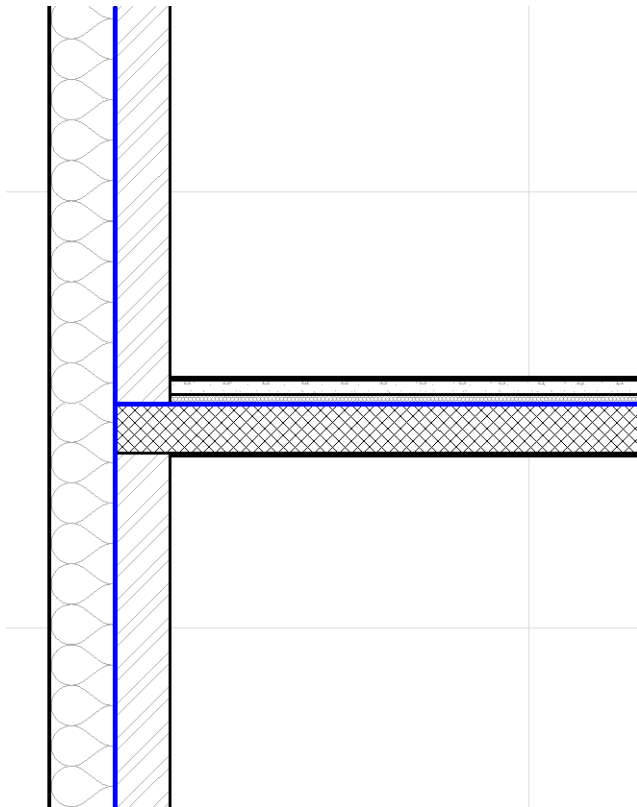
Best Practices for Intersections: Conceptual Design Phase

In this phase, you often can only estimate what the final structure will be. For maximum flexibility, and to avoid modeling errors as the design undergoes changes (e.g. structures are replaced or composite layers are edited), try the following:

Use Composite Slabs with At Least Two Skins

This way, you can add finishes to the structures on either side, without the details falling apart.

Note: In the early design phase, a composite that consists of a single skin will also work well, if the top of the monolithic structure is considered as the core top)



Use Wall Reference Lines Consistently

- **Flip Wall along Reference Line:** This command flips (or mirrors) a selected Wall along its reference line. The reference line stays in place, but the body of the Wall flips to the other side. The openings do not change their opening direction.

For example, you may use this command if you initially modeled the project using single-skin Walls to represent core structures, then start adding additional skins at a later design phase.

[See *Flip Wall Along Reference Line*.](#)

- **Modify Reference Line:** This command moves the reference line without changing the Walls' position on the Floor Plan.

For example, use this command if you are migrating an older project in which a Wall Reference Line was set to Outside Face: modify the reference line to Core Outside, without moving the geometry of the Wall itself.

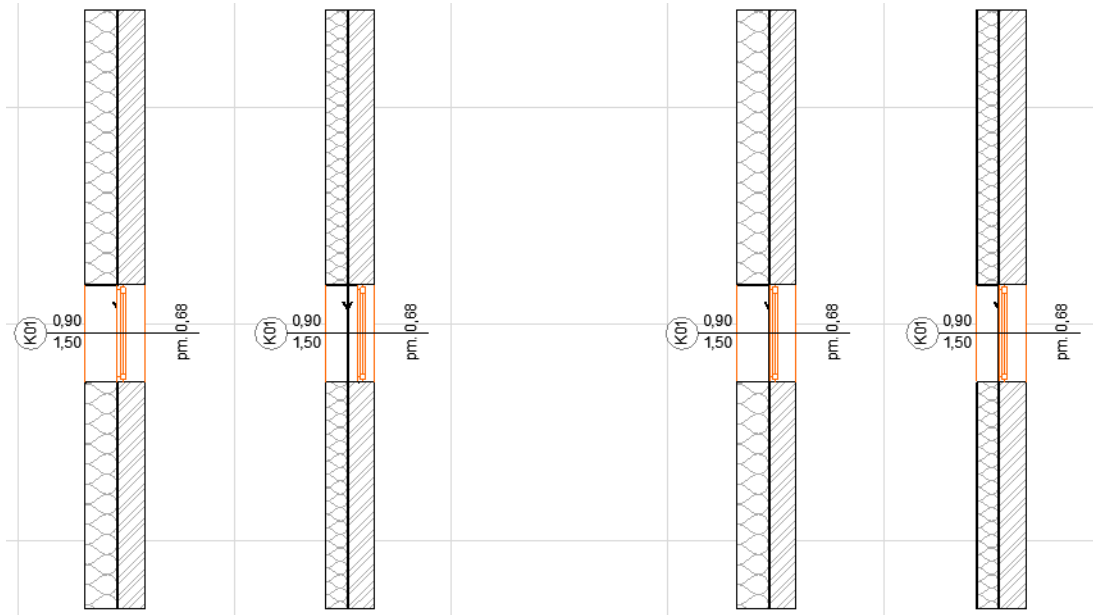
[See *Modify Wall Reference Line*.](#)

Anchor Door/Window Reveal to Wall Core

Anchor the Door/Window reveal to either the Wall face or the Wall core.

For example, you anchor a window to a composite Wall's core. If you later make changes in the composite structure (e.g. thicker insulation), the reveal depth will automatically adjust to maintain the window's distance relative to the Wall's core skin.

[See *Anchor Reveal*.](#)



On the right, the window position adjusts automatically to the change in Wall thickness, because it is anchored to the Wall core

Fine-Tune Intersections

To adjust the intersection of elements (or their skins/components), you can do one of the following:

Change Building Material Priority (Affects All Project Elements)

1. Select the element that is problematic.
2. Open the Building Materials dialog box.
3. Select a Building Material, then use the Intersection Priority slider to change the priority.

Note: Any changes you make here will affect all project elements which use that Building Material!

[See also *Intersection Priority \(Building Material\)*.](#)

Duplicate Building Material (Apply to Single Element)

If you want to adjust the priority in just one element of the project, you can create a duplicate Building Material with a different priority.

1. Select the element that is problematic.
2. Open the Building Materials dialog box.
3. Select a Building Material.
4. Click **New** and make sure you are creating a Duplicate. Give the duplicate Building Material a new name.
5. The new duplicate is selected in the Building Material dialog. Adjust its priority as needed.
6. Assign this new Building Material to the problematic element.

Legacy Intersection Mode for Older-Version Projects

If you open an older-version project (Archicad 16 or earlier), the Legacy Mode for element intersections will be turned on by default to maintain the appearance of your model.

In Legacy Mode, the automatic and Building Material priority-based intersections implemented as of Archicad 17 will not take effect in 3D. Connection methods used in Archicad 16 and earlier versions will be used. You may prefer this when working on an older project, to ensure that your existing intersections remain intact.

Legacy Mode is controlled in **Options > Project Preferences > Legacy**.

Parametric Objects

When you place an Object (as opposed to other elements in Archicad), you are placing an instance of an external file located in an object library. Archicad is shipped with a standard object library containing hundreds of preconfigured, editable objects (also known as GDL objects or Library parts). For the most part, you will use Archicad tools to place objects from this standard library.

Objects are parametric. This means that you can freely configure the placed instance, using the **Object Settings dialog box**, or the dialog boxes associated with the object-type tools, without actually modifying the source file for the object.

See [Settings of Library Part Elements](#) and [Object Tool Settings](#).

Topics in this section:

[About Parametric Objects](#)

[Where to Find Objects for Use in Archicad](#)

[BIM Components](#)

[Search for Library Part](#)

[Placing an Object](#)

[2D and 3D Representation of Objects](#)

[Selecting a Placed Object](#)

[Stretching Objects](#)

[Parameter Transfer Between Objects](#)

[Graphical Editing Using Editable Hotspots](#)

[About GDL Object Subtypes](#)

[Create Patch Object](#)

[Create Custom Library Parts and Components](#)

[Open and Edit Library Part](#)

[Scripting Custom Objects](#)

[Further Resources on GDL Object Creation](#)

Related Topics:

[Ramp Object](#)

[Import RFA as GDL Object](#)

[Import Rhino 3DM File as Object](#)

[Library Part Maker](#)

About Parametric Objects

Objects can be placed in the Project

- using one of Archicad's dedicated tools (Object, Lamp, Door, Window, Skylight, Corner Window, Wall End, Curtain Wall Accessory or Junction)
- automatically by specific commands or Add-ons (Markers, Labels, RoofMaker and TrussMaker elements)

See [Settings of Library Part Elements](#).

When you open an Archicad Project, Archicad automatically runs a check to ensure that all needed libraries and library parts are up to date and available. If there are any issues, the Action Center or Library Manager provides information.

See [Troubleshooting Library Issues](#).

You will find most of the objects you need for your project in the Archicad Library - the factory-shipped library of standard objects. The objects are organized into folders and sub-folders to make it easy to locate what you need. The Archicad Library is comprehensive, although localized versions of Archicad libraries will contain special objects which are localized for the standards of particular countries.

When you activate a tool that is dedicated to a particular Object subtype (i.e. Door), the Settings Dialog box accesses only the relevant part of the Archicad Library. For example, activating the Window Tool gives you access to all the Windows in the Archicad Library, but not Doors or Mechanical Objects.

See also [About GDL Object Subtypes](#).

All of these files can be opened in Archicad with the **File > Libraries and Objects > Open Object** command and they can be created in Archicad with the **File > Libraries and Objects > New Object** command. You can also create objects graphically, using other Archicad elements, and then saving them as an Object file.

See [Create Custom Library Parts and Components](#).

Where to Find Objects for Use in Archicad

- The standard Archicad library, shipped as part of your program, contains hundreds of predefined standard objects.
- Using Archicad's Library Manager, you can load additional libraries or add individual library parts as embedded objects at any time. You can also access libraries from BIMclouds.

For more information, see [Library Manager](#).

- Through the Settings dialog box of Library Part elements, as well as Surfaces, you can access BIM Components (BIMcomponents.com) while browsing for library parts. BIM Components is a searchable and interactive depository of objects and surfaces contributed by Archicad users.

For more information, see [Search for Library Part and BIM Components](#).

For information about importing surfaces, see [Get more surfaces](#).

- On the Internet, you will find a growing number of websites dedicated to GDL Objects. See especially the **GDL Center**, Graphisoft's GDL knowledge base, at gdl.graphisoft.com.
- A number of third party programs or Add-ons let you create or edit GDL Objects. These objects contain GDL scripts as well as other, application dependent data. Consult your dealer for availability information.
- DXF libraries are widespread in the CAD world and cover almost every area of engineering and design. Many manufacturers offer symbols or models of their products in DXF format. Archicad can easily import these libraries.
- You can use existing elements within Archicad to create and save custom objects.

For more information, see [Create Custom Library Parts and Components](#).

- In some cases, you can sculpt in other 3D applications and import the 3D model.

For more information, see [add-on specific documentation in the Graphisoft Documentation folder](#).

- If you want to try your hand at GDL scripting, start with [Scripting Custom Objects](#). See also the **GDL Center** at gdl.graphisoft.com.

BIM Components

BIM Components, at BIMcomponents.com, is a public website containing GDL objects and surfaces. Anyone can access the portal to browse, search, view and even download these files for free.

To upload, comment on, and rate the objects or surfaces, users must first register using a valid email address. It is not necessary to be an Archicad user to access or upload to BIM Components.

What Can You Do On BIM Components?

- **Search.** Find specific objects or surfaces, using an interface just like that of Archicad Object Settings. Objects/surfaces are shown with preview picture, description and an interactive 3D preview, with which you can rotate the object and modify its parameters.
- **Browse.** If you like, use the “tag cloud” (word cloud) to browse the most popular items.
- **Register.** The process is simple - provide your email address or an existing Graphisoft ID. After successful registration, you will receive a Registration Number, which you will need in order to share an object or surface from Archicad.
- **Share.** You can share both individual GDL Objects and LCF (Library Containment) files, as well as surface attributes. (Use LCF if you want to share a whole family of objects at once.) As part of the Share process, you can specify an attribute set by selecting your language template, ensuring that the object will work for others the way it works for you.
- **Participate.** Comment on and rate the items you find to help other users find the most suitable one, and to help developers create even better content.

For more information on all these functions, go directly to the BIM Components Portal at BIMcomponents.com.

Obtain BIMcomponents.com Registration Number

Before sharing an object to BIMcomponents.com, you must obtain a registration number. Once you receive the registration number, you need only enter it once; after that, Archicad will remember it. (It is also listed in the License Information dialog box, available from the Archicad Help menu.)

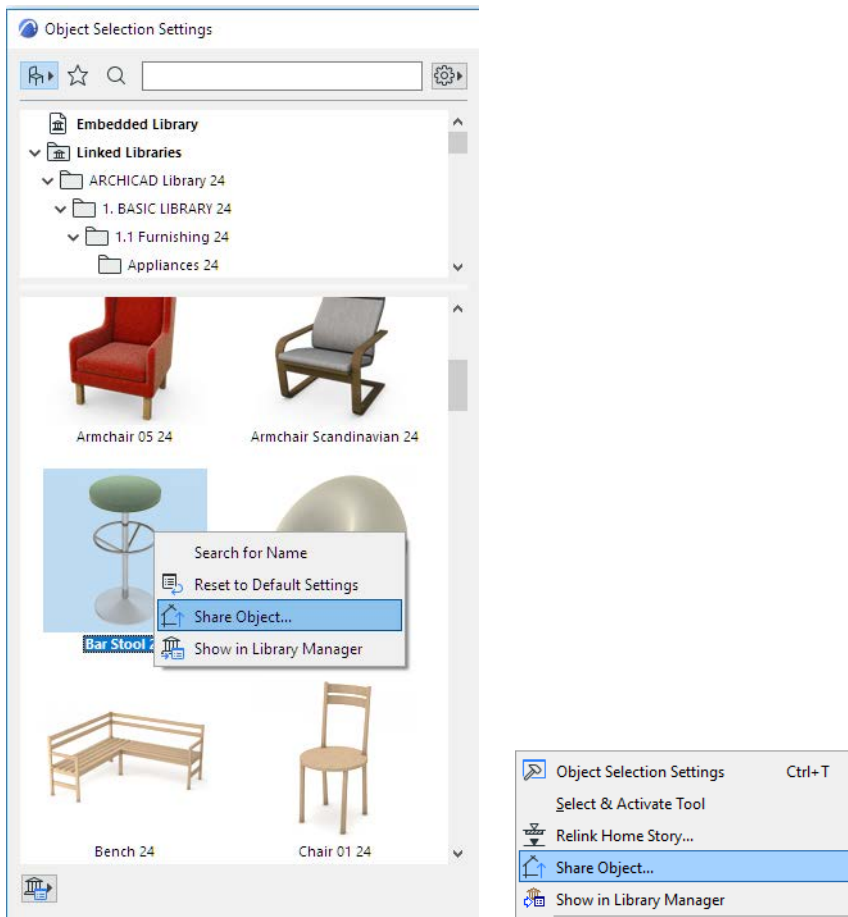
Share Object on BIM Components

Note: Before sharing an object to BIMcomponents.com, you must obtain a registration number. (See above.)

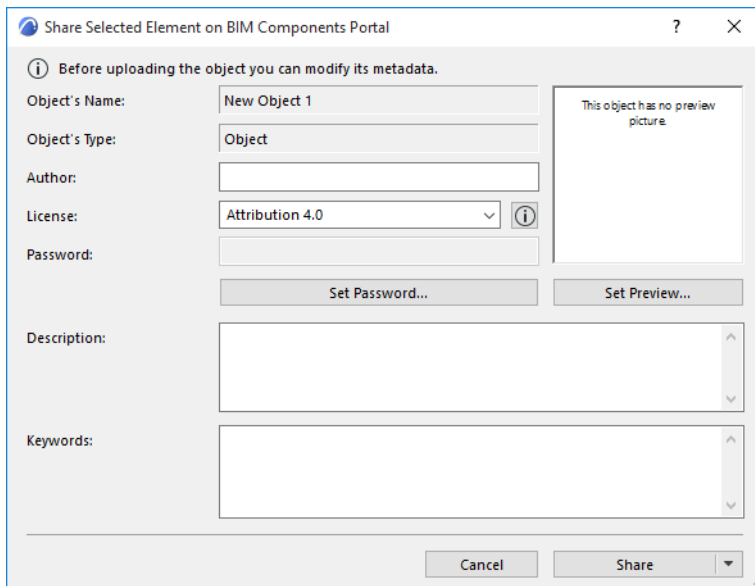
To share an object on BIM Components, do one of the following:

Select an object from the Settings dialog box (Library view), then use the **Share Object** command from one of these places:

- the context menu
- the icon at the bottom of the dialog box
- the **File > Libraries and Objects > Share Object** command
- The **Share Object** command from the context menu of a placed object.



The **Share Selected Element** dialog box appears.



Note: Before the Share process can proceed, a background check is run to ascertain that the object's parameters (e.g. copyright definition) and the user's license information are correct.

Edit the following information as needed:

1. Enter an **Author**. (If you leave this blank, the name you used to register on BIMcomponents.com will be used as the author.)
2. Assign a **License**. The following license options are available (for a complete description of any license option, click the Info button from the dialog box)
 - **Public Domain Dedication 1.0**. An element with this license belongs to the public domain; anyone can copy, modify or distribute it without permission.
 - **Attribution-NoDerivs 3.0**. An element with this license can be copied and distributed, even for commercial purposes, but only with attribution, and the object cannot be modified.
 - **Attribution-NoDerivatives 4.0**: This is a newer version of the Attribution-NoDerivs 3.0 license.
 - **Attribution 4.0**: An element with this license can be further shared (copy and redistribute the material in any medium or format) and adapted (remix, transform, and build upon the material for any purpose, even commercially)
 - **Attribution-ShareAlike 4.0**: An element with this license can be further shared (copy and redistribute the material in any medium or format) and adapted (remix, transform, and build upon the material for any purpose, even commercially), but the contributions must be distributed under the same license as the original.

Note: Another type of copyright, “Commercial,” can be assigned from the GDL Editor window. Objects with a commercial copyright, however, cannot be shared to BIMcomponents.com.

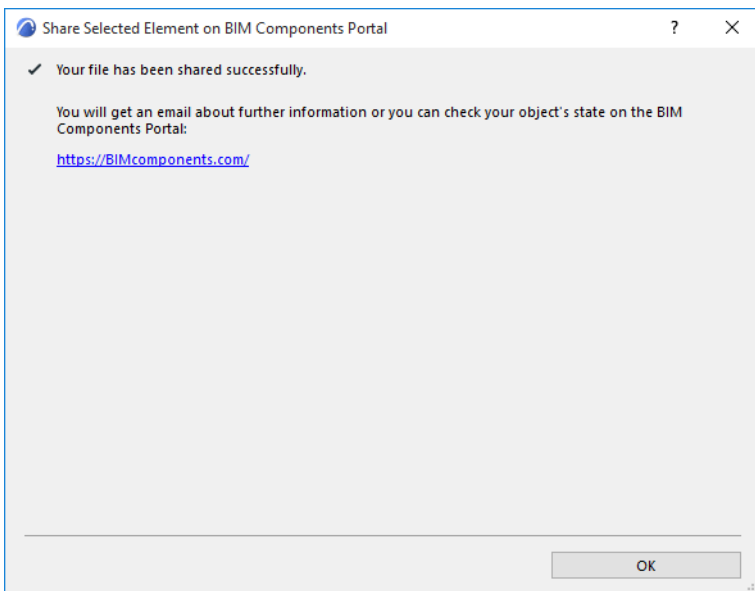
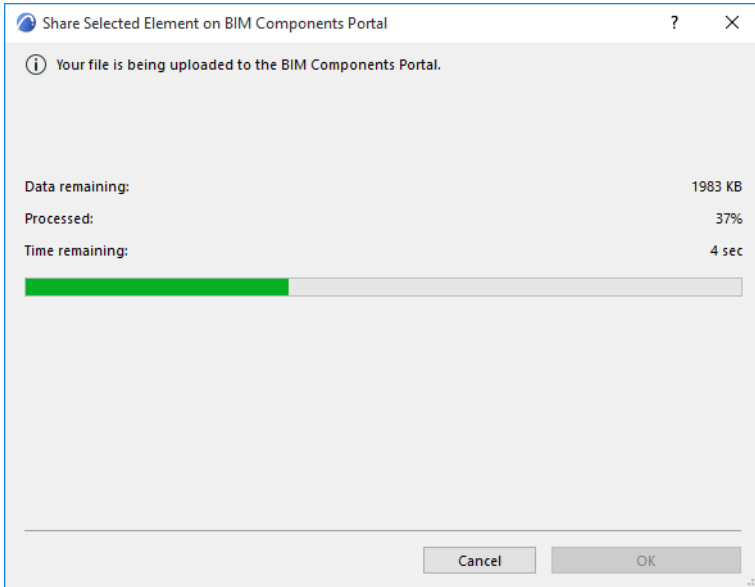
[See GDL Object Editor.](#)

3. **Password** (not available with Public Domain license). Click **Set Password** to define an optional password. Such objects can be freely viewed and downloaded, but the user cannot open the object to edit or copy its script without a password. If the object already has a password, the Change Password button allows you to modify it.
4. **Description and Keywords**. Enter a description and Keywords to describe the object. Archicad’s object search function will search (among other fields) these fields for the entered search terms. While optional, a detailed description is recommended, including the functions and purpose of the object, so that other users will be able to locate it.
5. When you are ready, click one of the following:
 - **Share**; or
 - **Share and Save**. (This saves any changes to the object data in your Embedded Library as well as to BIMcomponents.com.)

Note: the Share and Save command is not available in the following cases:

- if you are working in Teamwork but have not reserved the object, or
- if the object has been opened from a Library Container File (.lcf)

During the Share process, you cannot keep working in Archicad. A dialog box shows the progress.



Search for Library Part

To search for library parts, use the search field at the top left of the browser section of its Settings dialog box. This search function is available in the Object Settings dialog box, as well as the Settings of any tool that places Library Parts: Window, Door, Skylight, Wall End, Corner Window, Lamp, MEP tools.

For information on searching for Favorites from other dialog boxes:

[See *Search Favorites*.](#)

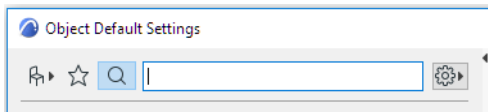
For information on searching and loading surfaces from catalogs, using the Surfaces dialog box:

[See *Get more surfaces*.](#)

Enter Search Mode

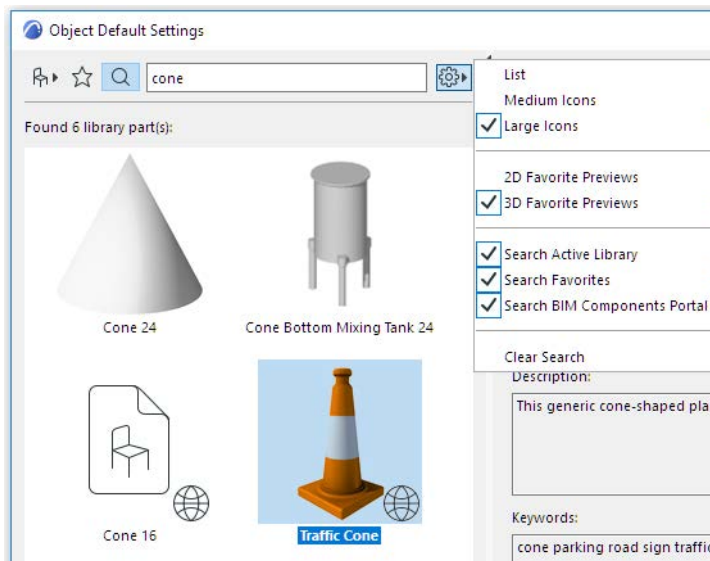
Do one of the following:

- Click into the Search field
- Click the Search icon (magnifying glass)
- Click Ctrl+e (Mac: Cmd+e)



Define Scope of Search

Use the pop-up control at right to define where to search (at least one of the three options must be checked):



- **Search Active Library:** Search in linked libraries, the Embedded Library, and BIMcloud Libraries you have downloaded
- **Search Favorites:** Search only among the saved Favorites for this tool
- **Search BIM Components Portal:** Search only on BIM Components Portal, a searchable and interactive depository of objects contributed by Archicad users or uploaded by Graphisoft.

For more information, see [BIM Components](#).

Start Search

Enter any string of characters to search for.

Once you press Enter or end your input (no input for 1 second), the search process will begin.

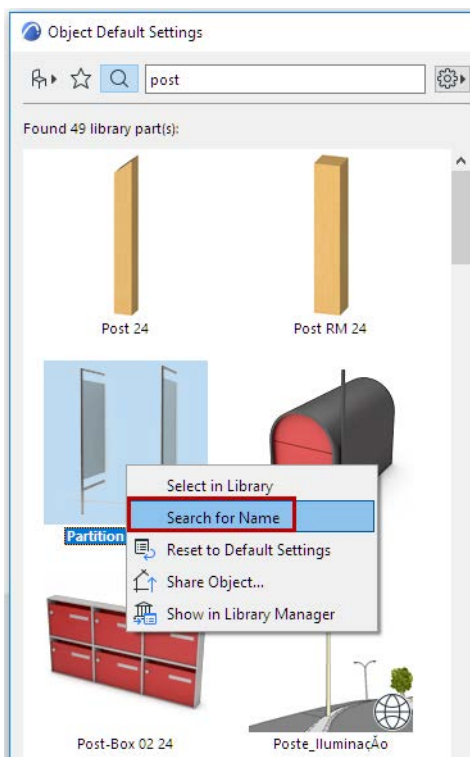
The search engine will search for the string in the following locations:

- Library Part name
- Library name
- Description
- Subtype
- Name of author of Library Part
- Keywords

See also [Search Favorites](#).

Hints for Using Search Terms

- Use quotation marks around a phrase to search for exactly that phrase, rather than the individual words
- From the context menu of a Library Part in the Settings dialog box, use **Search for Name** from the context menu. This will copy the name and add it to the Search field. For example, use this to search for Favorites based on a particular Library Part.



Display of Found Library Parts

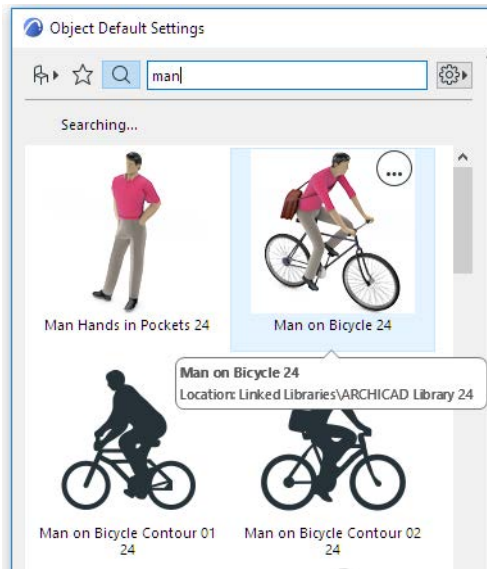
The **Found Library Parts** panel in the browser area displays your search results.

Only those library parts are found which are used by the active tool. In other words, if you are searching in the Object Settings dialog box, Objects will be found - no Doors or Windows.

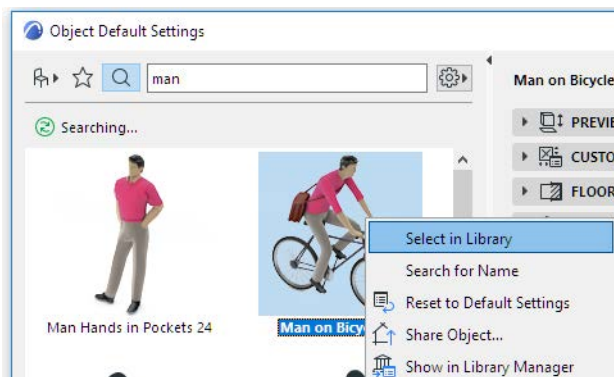
Depending on your where the library parts were found ([see Define Scope of Search](#)), the options differ:

Results from Active Library:

- Click a library part to apply its settings and see them in the panels on the right
- Hover the cursor over a library part to see its name and the library where it is located

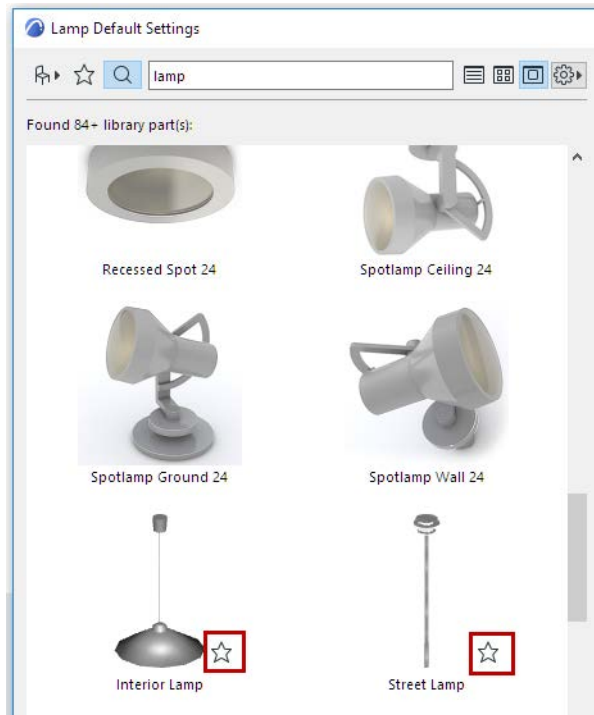


- Use **Select in Library** to switch to the library view and see this in the context of its library hierarchy



Results from Favorites

- Favorite search results are displayed with a star icon



Results from BIM Components Portal

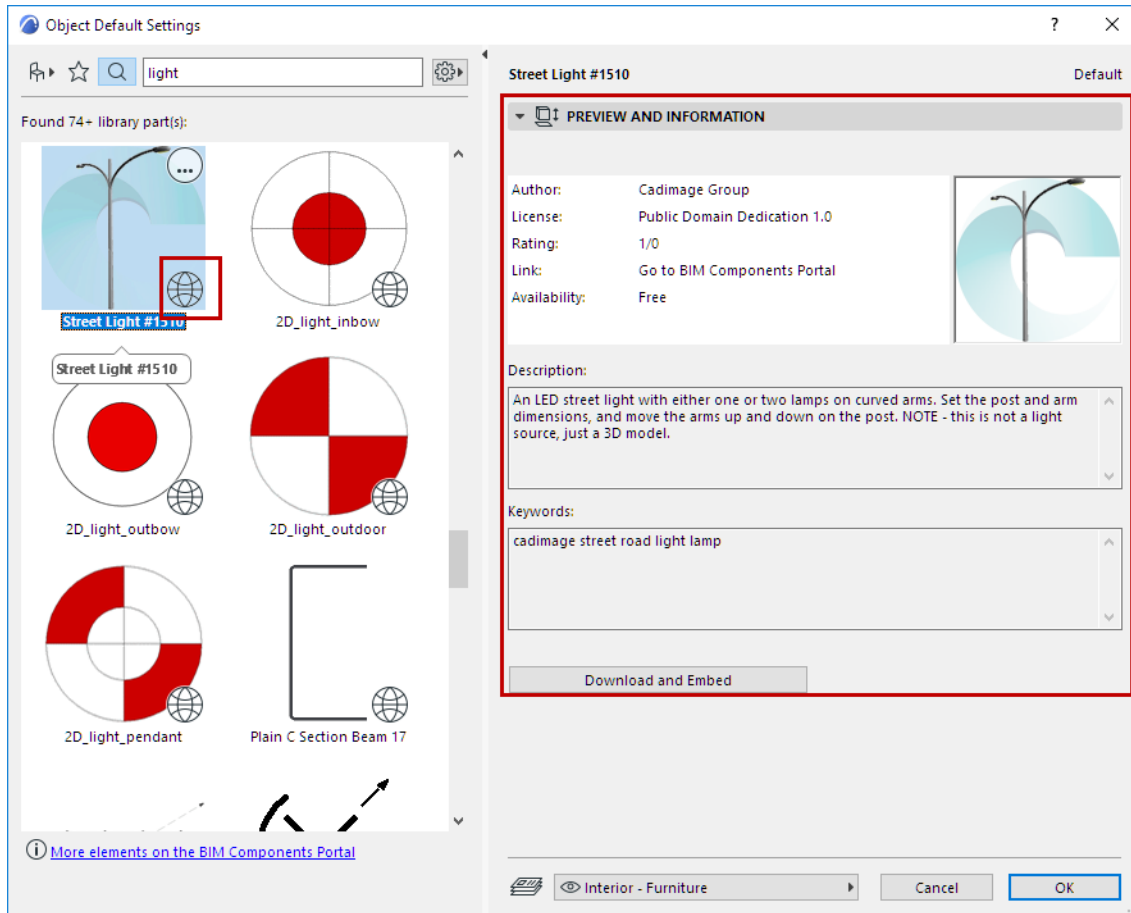
- Found BIM Components are displayed with a globe icon.

Exception: Some of the BIM Components-based hits will be marked with a key. Such objects are available only to users with some type of premium service agreement. (This may vary by market; contact your distributor.)

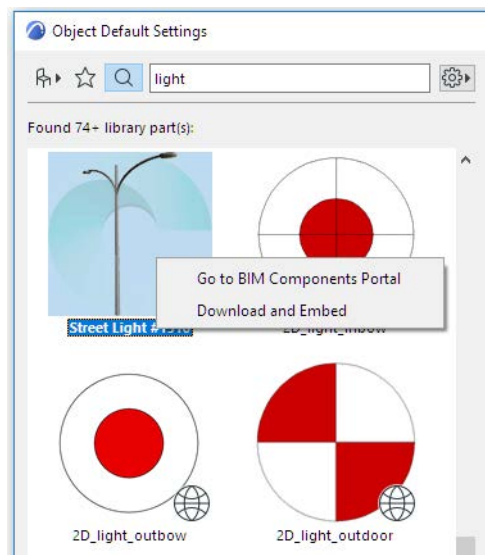
Note: The “Found Library Parts” section will display a maximum of 50 of all the library parts found on BIM Components; if more are found, you can see the remaining hits by clicking on the **More elements...** link at the bottom of the dialog box.

- Hover the cursor over the component to see its name.

- Click to see its Preview and Information the panel on the right.



- Context menu commands:



- **Go to BIM Components Portal:** Click to enter BIMcomponents.com.

See [BIM Components](#) for more information about this web page.

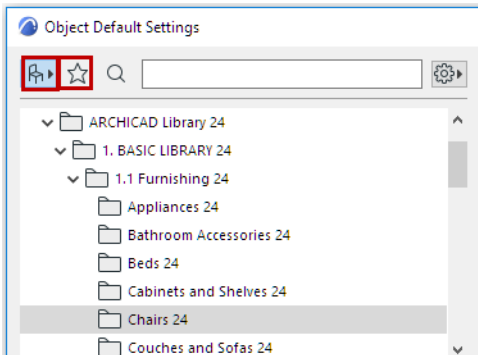
- **Download and Embed:** Click to download the library part to your Embedded Library. You can access this same command from the Preview and Information panel at right.

While the download is underway, you cannot work in Archicad. Click **Cancel** to abort the download process.

Exit Search Mode

To exit search mode, do one of the following:

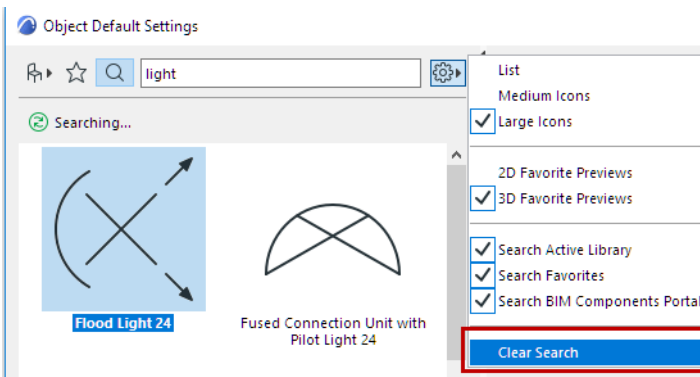
- Click on either of the other two icons at the top (Library view or Favorites view).



- Close the Settings dialog box

Even after you exit Search mode, Archicad will remember the last-used Search and its results, which will be shown the next time you enter Search mode.

To clear the Search, use **Clear Search** from the pop-up menu:



Placing an Object

Typically, you will follow these steps when placing an object:

1. Select the corresponding tool in the Toolbox and open its settings dialog box by double-clicking the tool's icon.
2. Browse for the appropriate element in the available library folders.
3. Adjust the default settings default to suit your particular situation or purpose. The external file is not affected in any way by the changes you make in the Object (Lamp, Door, etc.) Settings dialog box.
4. Click at the desired location and place the customized instance of the object.

Note: Objects can be placed in a Section/Elevation window, but they are considered as graphic symbols only. (No corresponding 3D model element is generated.)

See also [2D and 3D Representation of Objects](#).

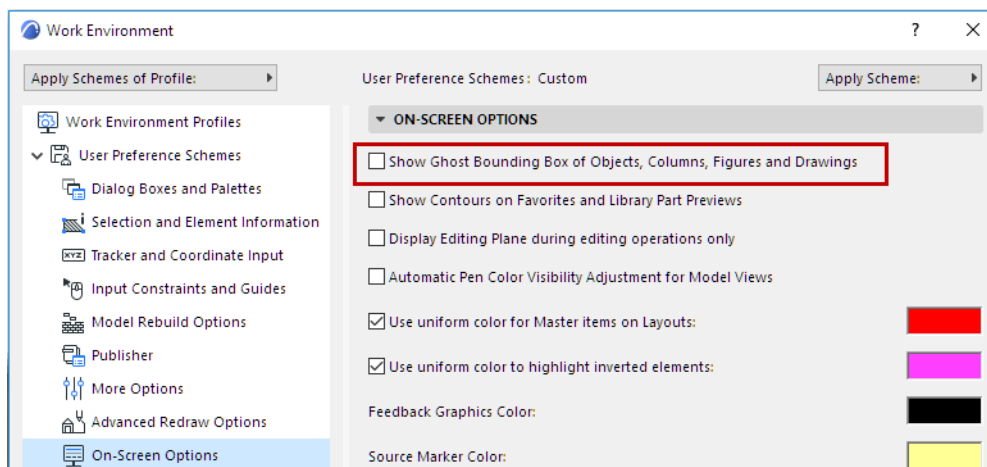
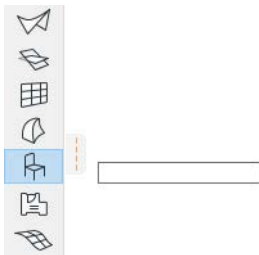
Placement Anchor

The primary hotspot (object's placement anchor) can be defined on the Preview and Positioning Panel of Object Settings. Before placing the Object or Lamp, make sure that you have selected the appropriate hotspot to serve as the anchor.

See [Object Preview and Positioning Panel](#).

Placement Aids

When the Object or Lamp tool is active and you place your cursor inside a construction window, the ghost bounding box of the element is shown and it follows the movements of the cursor. This also allows you to check the active hotspot used for positioning the Object or Lamp. You can switch this ghost box feature off in **Options > Work Environment > On-Screen Options**.



When positioning the Object or Lamp, you can use numerical input, gravitation, mouse constraints or the grids. This allows you to fit fixtures or furniture to corners, specific positions, or to each other with great accuracy.

Geometry Methods

Four geometry methods are available in the **Info Box** for placing Object and Lamp type Library Parts.



- The **Orthogonal** method automatically places Library Parts in alignment with the normal grid lines unless you specify a rotation angle in the Settings dialog box before placement.
- To place a **Rotated** Library Part, first define a reference point by clicking any point. Use the resulting rubber band line to place the rotation vector. This rubber band line can be constrained using any of the drafting modifiers or enabled Mouse Constraint angles.
- The **Diagonal** input method works like the rectangle geometry method used for polygonal elements.
- The **Rotated diagonal** input method works like the rotated rectangle method used for walls, slabs, etc.

The last two methods let you define the A and B parameters of the Object or Lamp graphically. (For most objects, these parameters represent the Length and Width of the object on the Floor Plan.) Note that they are only available if the GDL Object can be freely stretched; some Objects may have fixed or proportionally set dimension parameters.

The result of this technique also depends on the primary hotspot (anchor). As a rule of thumb, if a hotspot at the corner of the object is selected, you can define both the A and B parameters graphically. If the hotspot is along a side of the object, you can only define one of the parameters graphically and the other parameter will be taken from the Settings dialog box. When choosing a hotspot within the bounding box of the symbol, none of the parameters can be set graphically and the element will be placed as if using the simple orthogonal or rotated methods.

Selecting a Placed Object

Use the regular selection techniques, including the Find & Select function, to select objects or groups of objects.

Unlike when selecting walls or other elements constructed on the fly, the cursor can only snap to predefined hotspots of the object symbol. Objects can be placed, selected, and fitted to other elements by these hotspots.

2D and 3D Representation of Objects

Model View Options for Library Parts

Depending on which libraries your project has loaded, use Model View Options (**Document > Model View > Model View Options**) to assign global preferences for the display of certain library parts and their parameters.

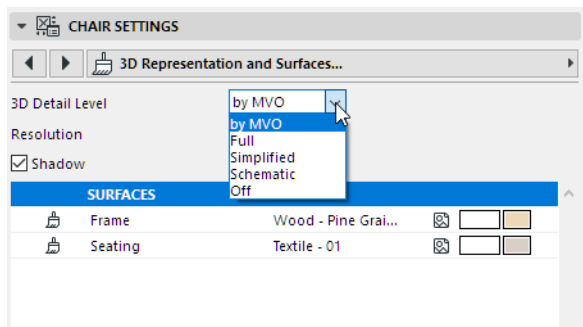
See [Miscellaneous Settings for Library Parts](#).

By default, Objects and Lamps use the detail level settings defined in Model View Options (**By MVO**). For a specific object/lamp, if the control is available, you can override its detail level in Element Settings (use the 2D or 3D Representation controls).

3D Detail Level

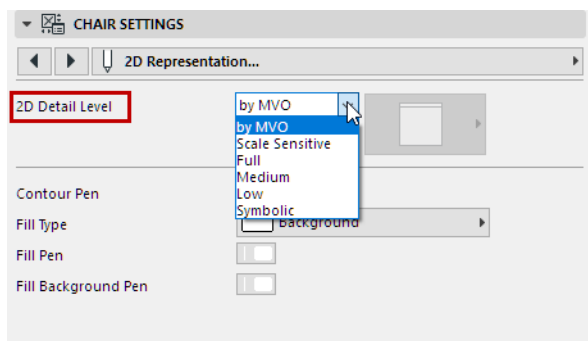
Where available, this control lets you overwrite the MVO-based settings for an object: Choose Schematic, Simplified, or Full.

If you choose Off, the object is not shown in 3D.

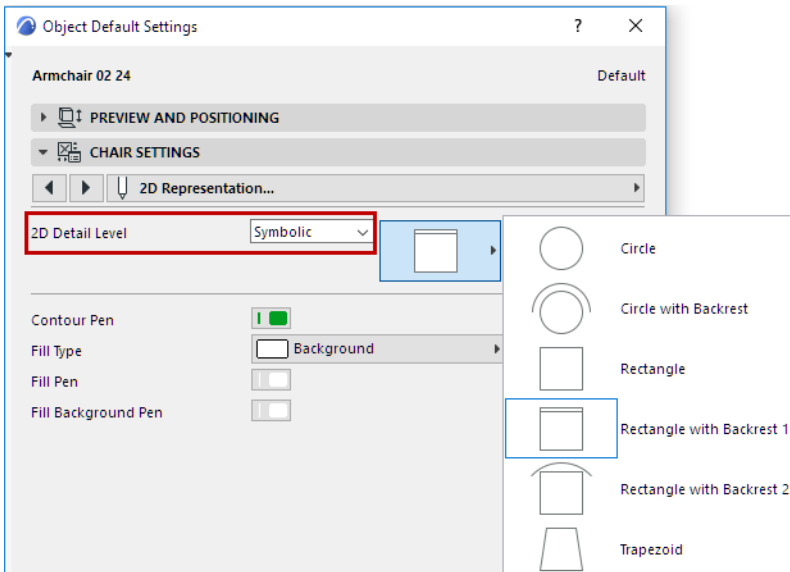


2D Detail Level

- If you choose Scale-sensitive, the object's display varies with the scale setting in **Document > Floor Plan Scale**.



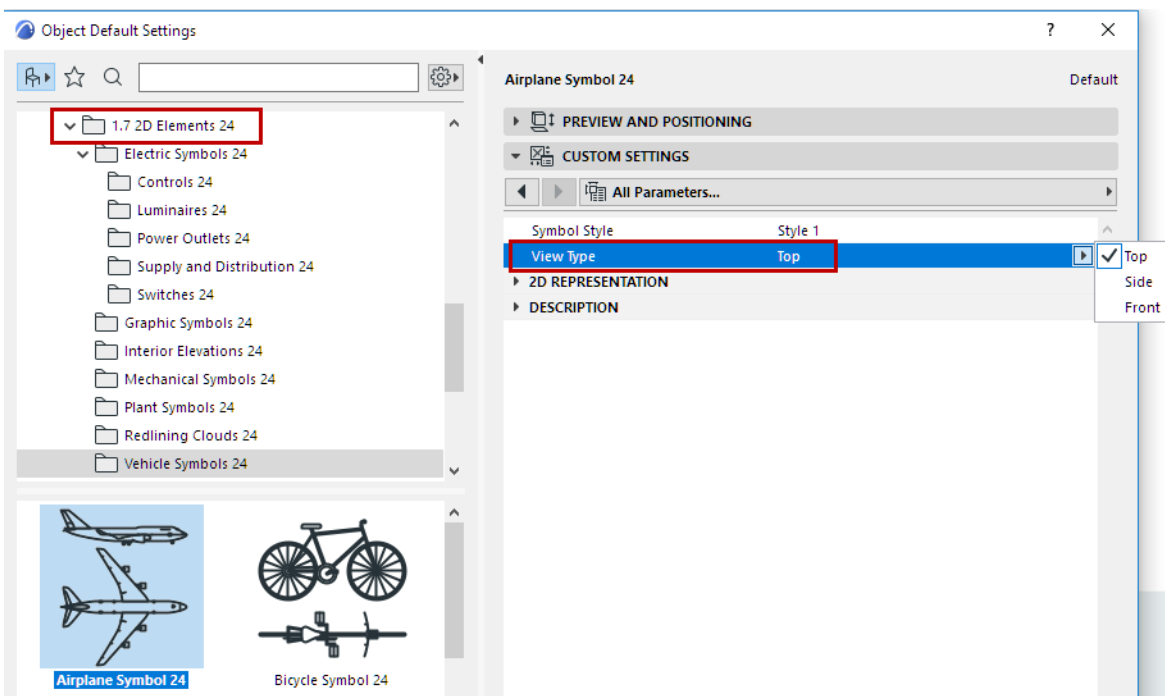
- If you choose **Symbolic**, the **Symbol Type** parameter becomes available, where you can choose a fixed symbol to display the object in 2D views regardless of the scale.



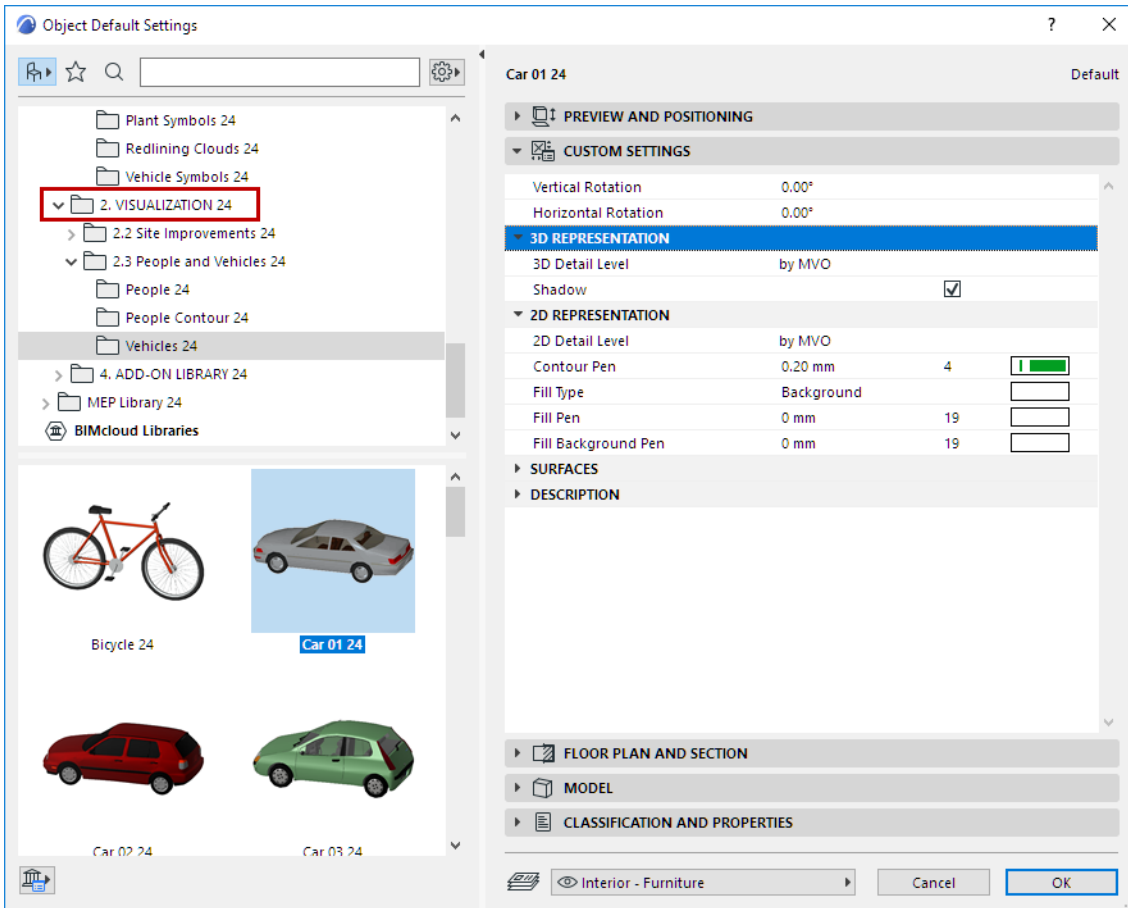
2D vs. 3D Objects

Plants, People and Vehicles can be found both in the 2D Elements and the Visualization folders. The People and Plants in the 2D Elements folder are shown in elevation view and are meant to be placed in Section/Elevation/IE windows. Using these simple representations saves time.

Vehicles are also 2D only elements, but they have two or more View Types: a top view for the Floor Plan and side, front and/or rear views for Sections/Elevations.



The counterparts of these elements in the Visualization folder are full 3D representations meant to appear in the 3D Window and especially in PhotoRendered pictures. The Visualization folder also contains a couple of Picture elements used for texture mapping. They are displayed as simple fills in shaded views and only appear in photo-renderings. If you place and activate a Camera, the picture will always face this camera to avoid distortion.

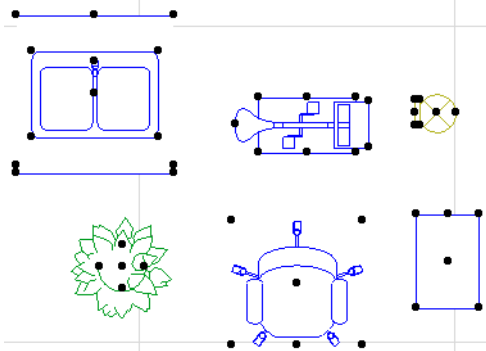


Some complex elements may take a long time to display in fully detailed 3D - for example, realistic trees and other plants that consist of a very large number of polygons.

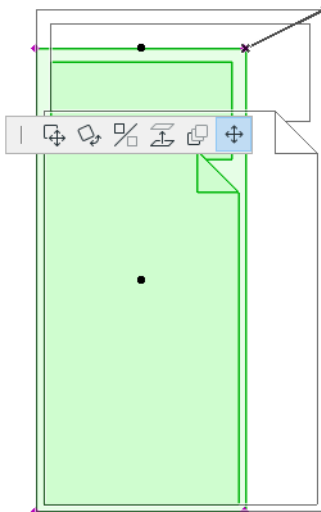
Stretching Objects

Objects, including Doors and Windows, can be stretched by their hotspots.

Select the Object to display its hotspots.



You can either use the **Stretch** command (**Edit > Reshape > Stretch**) or the pet palette's Move Node icon to stretch/shrink the Object by a node.



Note that you stretch the **A** and **B** parameters of the symbol, not its bounding box. Using parametric 2D scripts, you can easily create objects whose symbol size will not match the **A** and **B** parameters. If you make such a symbol, make sure that you define hotspots at the corner of an **A*B** sized rectangle. These will enable you to stretch that symbol.

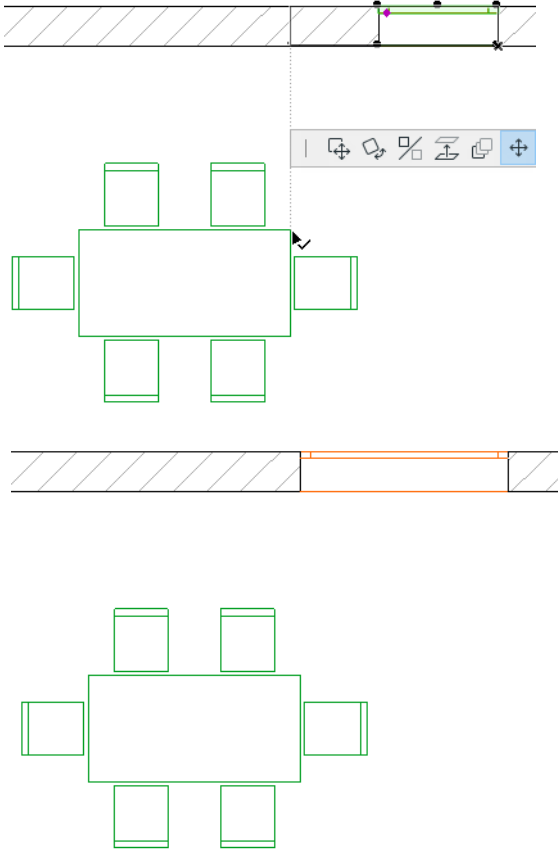
[See also *Graphical Editing Using Editable Hotspots*.](#)

Stretch Door/Window to Reference Point

When stretching **Doors & Windows**, you can use reference points outside the host wall. The endpoint of the stretch vector will be projected perpendicularly or orthogonally (depending on what you have set for in the Control Box) to define the new opening edge.

[See *Projection of Cursor with Mouse Constraints*.](#)

This allows you to position a Door or Window with respect to an element within a room, or in another room, as shown below.



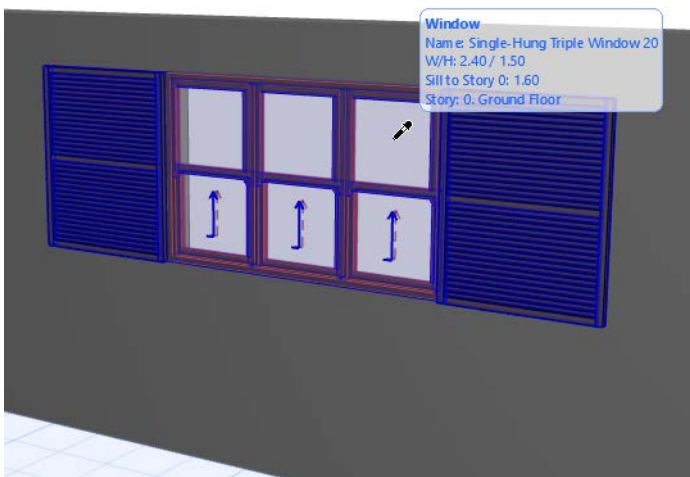
Parameter Transfer Between Objects

After you have adjusted an object's parameters, Parameter Transfer offers a way to transfer your edits to a different object of *the same subtype* (e.g. transfer parameters between two window objects).

[See also *Parameter Transfer for other ways to use this feature.*](#)

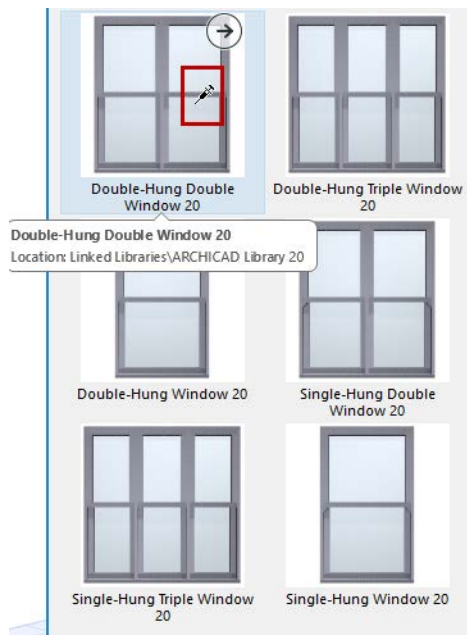
To transfer parameters between objects of the same type, do the following:

1. In the object's Settings dialog box, make parameter modifications as needed, then place the object.
In our example, we modify the Triple Sash window by adding shutters, then place the window.
2. Use the Parameter Transfer shortcut (Alt+click) to pick up the parameters of the placed object (here, the Triple Sash window).

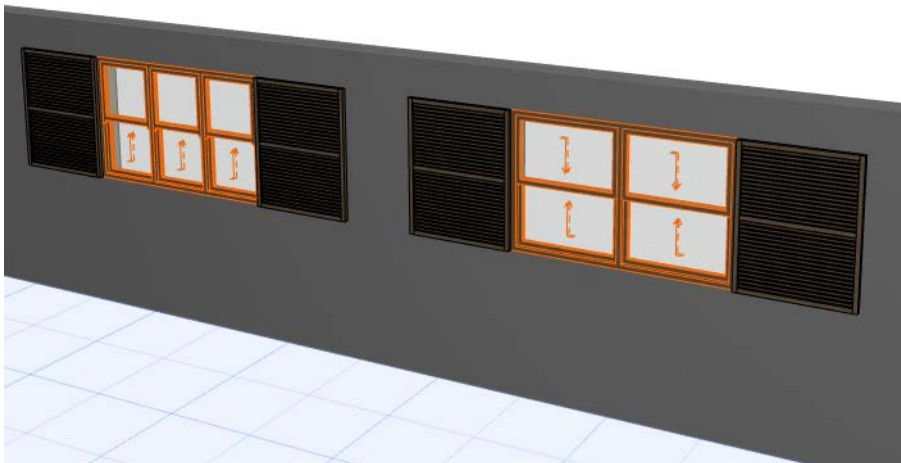


3. Open the Settings dialog box.

4. Move your cursor to the next object you wish to place (here, the Double Sash window) - and press Ctrl+Alt (Mac: Cmd+Opt). The cursor will change to a syringe shape.



5. Now click on the object to transfer the parameters. Parameters of your originally modified object (Triple Sash window) will be transferred to the newly activated object (the Double Sash window).
6. Place the object using the changed parameters.
In our example, the Double-Sash window has inherited the parameters of the Triple-Sash window, including the shutters.

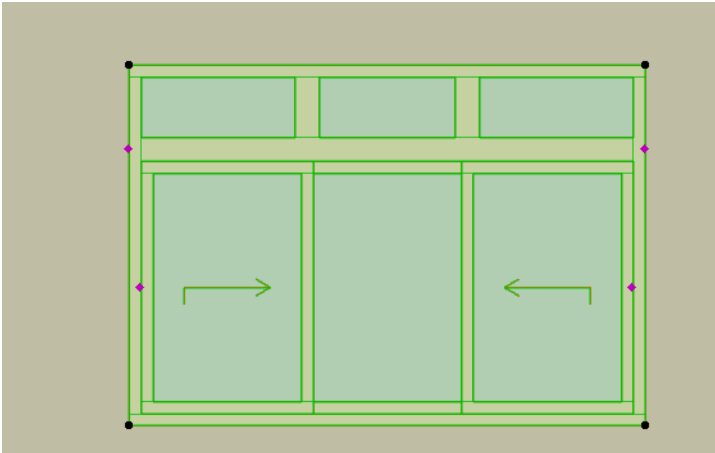


These modifications to the object are in effect only until you close the dialog box and place the object; the object's default settings will not change.

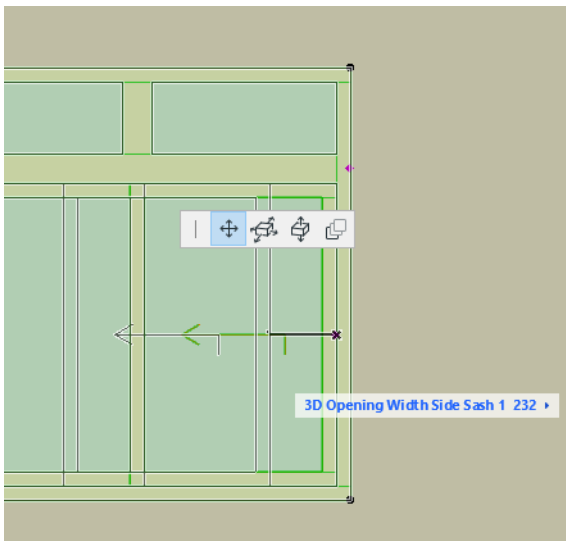
Graphical Editing Using Editable Hotspots

Depending on their programming, **GDL Objects** can have smart editable hotspots (selection dots). You can recognize an editable GDL hotspot by its diamond shape. If you select such an object, click an editable hotspot and choose the stretch or Move Node icon in the pet palette (whichever is appropriate). Clicking an editable GDL hotspot produces the Tracker displaying the editable parameter. Click Tab to enter a value for the GDL parameter.

The color of editable GDL hotspots can be set in **Options > Work Environment > Selection and Element information**.



The diamond-shaped hotspot serves to alert you that in addition to graphical editing, direct numerical editing through the Tracker (for example, setting a window’s sash-opening width by typing it in) is also available through that hotspot (provided that the applicable pet palette option has been selected). In the image below, the Tracker pops up showing the “Sash Opening Width” parameter if the Stretch icon is chosen from the pet palette.



The diamond-shaped hotspot and associated editing palette is also available in the 3D window. In the image below, you must choose Move Node from the pet palette to activate the Sash Opening Width parameter.



Note: The diamond-shaped hotspot appears *only* for editing options that produce a specific parameter editing possibility in the Tracker. No other type of function is necessarily associated with this type of hotspot.

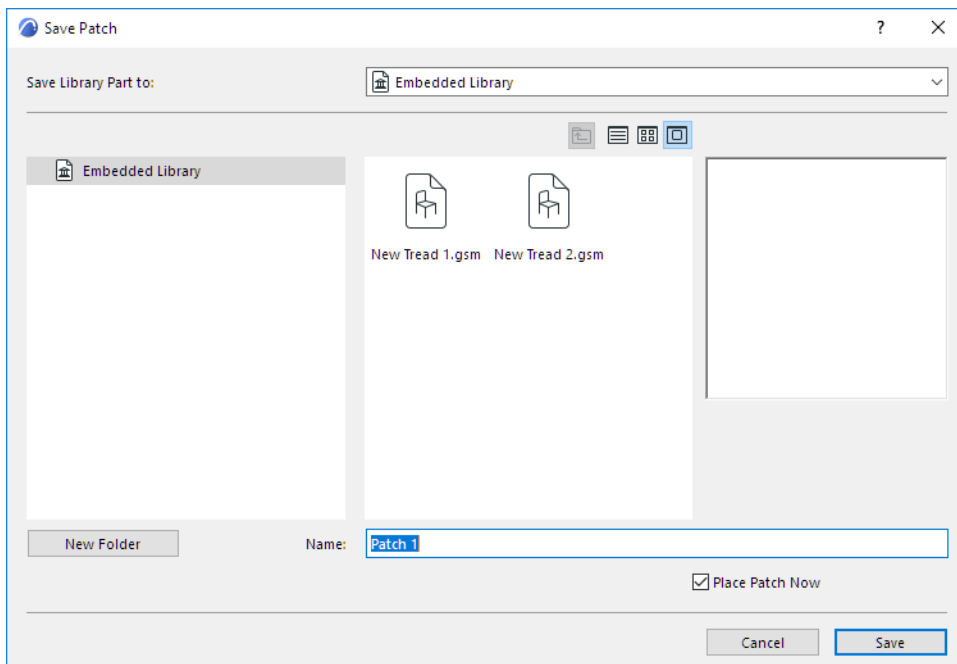
Create Patch Object

With the **Document > Annotation > Create Patch** command, you can save the area enclosed by the **2D Marquee** as a parametric 2D Library element. You can later use this to “sticker” your plans in order to provide the necessary detail in Floor Plan and Section views while saving time on modeling.

To create a Patch:

1. Define a marquee area in the Floor Plan window.
2. Use **Document > Annotation > Create Patch**.
3. In the appearing dialog box, specify the location and name of the created element.

Note: The **Place Patch Now** option is checked by default; the new patch object is placed automatically.



Note: Objects created this way will be enhanced with parametric behavior, specified by a special template in the Archicad Library. If this template is missing from the loaded libraries you will still be able to save patches, but they will be plain stickers with no special intelligence. If Archicad fails to locate the Patch Template, it will notify you, and you will have the option to either proceed or cancel the operation.

About GDL Object Subtypes

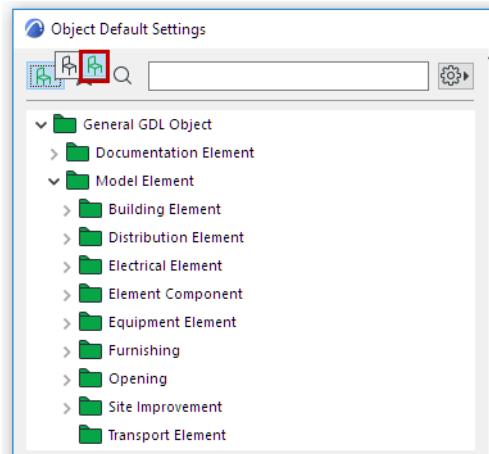
All GDL Objects have a subtype, organized in a tree hierarchy that generally corresponds to the IFC (Industry Foundation Classes) entity structure.

Note: IFC stands for Industry Foundation Classes. To read more about IFC, see [Working with IFC](#)

Subtypes allow you to organize your library parts into a logical system. Add-ons also use subtypes to identify their corresponding library parts.

Locate Element by Subtype

In the Object Settings dialog box, select **Subtype View** to browse by object subtype.



Changing an Object's Subtype

To change the subtype of a placed object, select it, open it (**File > Libraries and Objects > Open Object**) and click the **Select Subtype** button at the top of the dialog box.

Changing the Object's subtype modifies its behavior and adds the parameters and functionality needed for the new subtype.

More about Subtypes

Archicad subtypes are controlled by subtype object templates. When you create a new GDL Object, Archicad will allow you to choose one of the available subtypes for your element.

Each subtype has a set of compulsory parameters that define the behavior or functionality of the elements that belong to it. When you use the Parameter Transfer between objects, this will only work between two objects of the same subtype.

[See *Parameter Transfer Between Objects*.](#)

Some of the subtypes include additional features called handlers. These define whether the elements that belong to the subtype can be accessed as general objects by clicking the Object tool's icon and choosing them in the Object Settings dialog box, or as specific objects that either have their own tool icon in the Toolbox (for example Door, Window, Corner Window, Skylight, Label) or are placed by special commands or functions (for example Zone Stamps and RoofMaker objects).

Related topics:

[Subtype Hierarchy Dialog Box \(GDL Master Window\)](#)

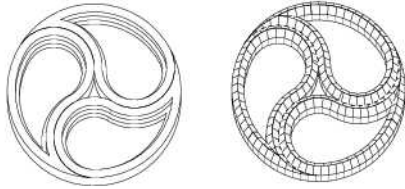
[Settings of Library Part Elements](#)

Search for Library Part

Create Custom Library Parts and Components

If you cannot find the library part you need, you can create your own GDL object. You need not do any programming; a few techniques using familiar Archicad tools lets you draw and save a customized object easily, using regular construction elements as custom building blocks (regardless of their originally intended purpose).

Custom components are sub-parts of library parts, such as doorknobs. These, too, you can create graphically and save for use with existing library parts.



Tracery of a Gothic window modeled with slabs and roofs

Related Topics:

[Save 2D Symbols as Objects](#)

[Save 2D Elements as Label](#)

[Save 3D Elements as Objects](#)

[Save Rectangular Doors and Windows from 3D Model Elements](#)

[Save Custom-Shape Doors or Windows](#)

[Create Custom Components for Doors, Windows, Curtain Wall, Stair, Railing](#)

[Create Custom Components: Example Workflows](#)

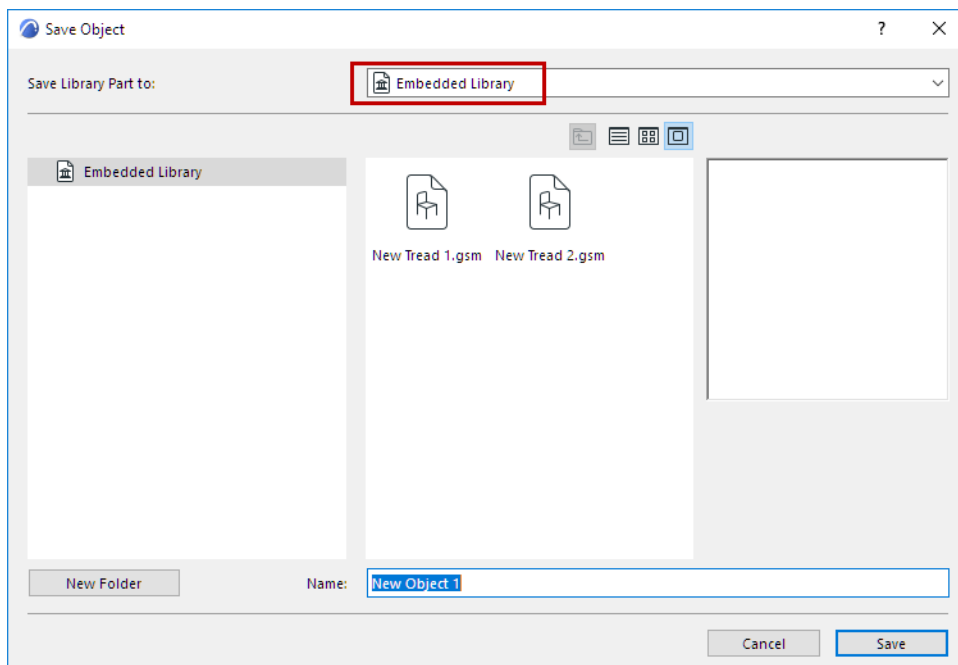
Save 2D Symbols as Objects

If you find that you need to place a 2D pattern many times in a project, you may wish to create a 2D symbol, then save it as an object.

1. Draw the desired 2D symbol in the project using 2D tools (line, text, fill etc.).
2. Put hotspots on each point where you want the cursor to find the object.
3. Select the desired drawing elements.
4. Use **File > Libraries and Objects > Save Selection as**, and choose **Object**.
5. In the **Save Object** dialog which appears, enter a name and choose the desired folder location for the object:
 - to the Embedded Library (default choice. It is recommended to save customized, project-specific objects to your Embedded Library.)
 - browse for a linked library
 - to a BIMcloud library that has been added to the project

Note: In Teamwork, you must reserve the BIMcloud Library folder before saving to it.

Optionally, click New Folder to create a new folder within the selected library folder.



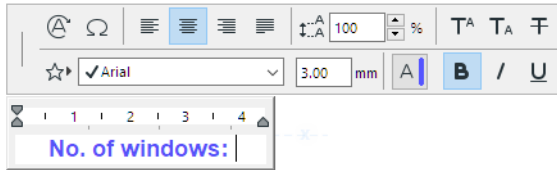
6. Place the object with the Object tool. Note that the created object is stretchable, but will be distorted after stretching since it was not scripted using parametrics.

For information on editing the resulting GDL library part, see [Editing Custom Library Parts and Components](#).

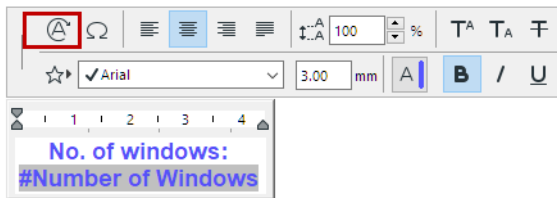
Save 2D Elements as Label

Use this function to create your own symbol Label, consisting of any combination of symbols or custom text, including Autotext.

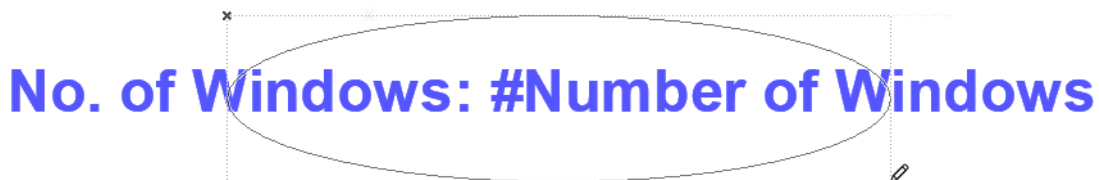
1. Use the Text tool to place a Text box.
2. Insert the desired text.



Use Autotext as needed to display element-specific information (e.g. #NumberOfWindows).



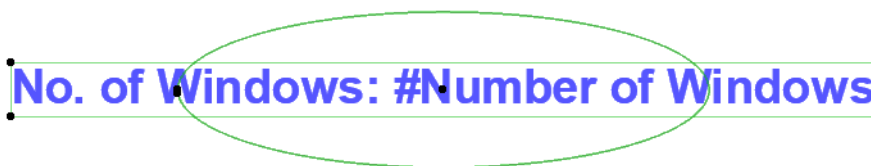
3. If the Label should have a frame, use a 2D tool (e.g. Line tool) to create a frame of any shape around the text.



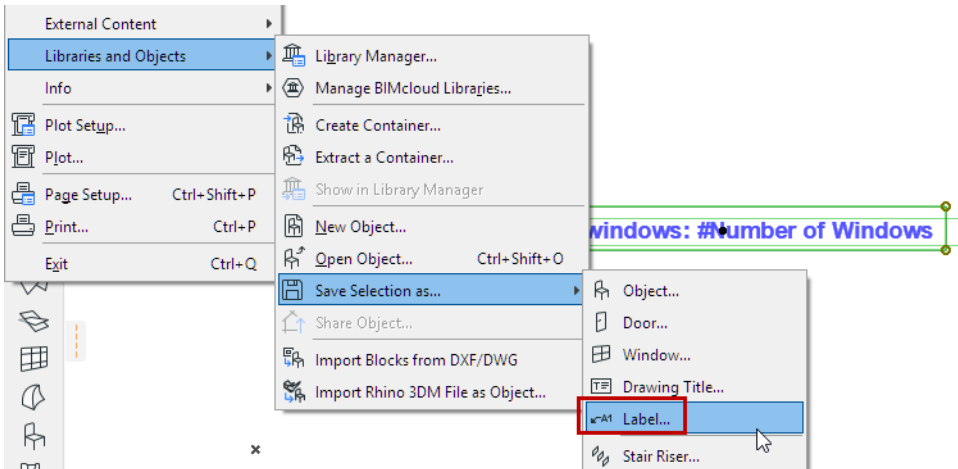
Hints:

- If the label contains Autotext, consider the estimated size of the resulting Autotext value (e.g. number of digits or length of expected text), and draw the frame accordingly.
- It may be a good idea to make the centerpoint of the text block and the frame element coincide, and to use a center anchor point for the text block.

4. Select all the 2D elements.



5. Use File > Libraries and Objects > Save Selection as, and choose Label.



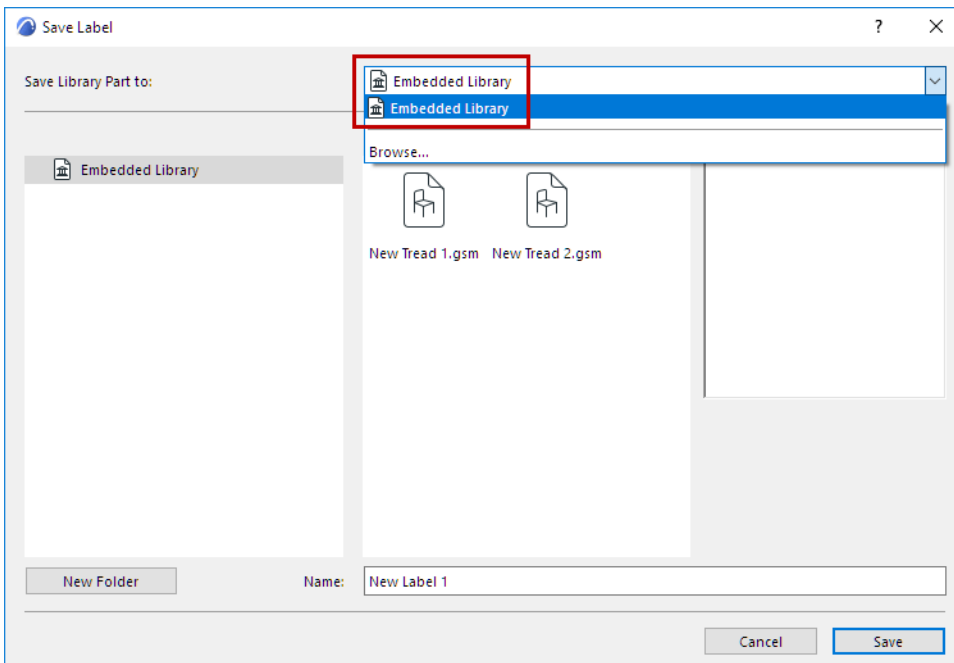
6. In the Save Label dialog which appears, enter a name and choose the desired folder location for the object:

- Embedded Library (default choice. It is recommended to save customized, project-specific objects to your Embedded Library.)
- A BIMcloud library that has been added to the project

Note: In Teamwork, you must reserve the BIMcloud Library folder before saving to it.

- A linked library

Optionally, click New Folder to create a new folder within the selected library folder.

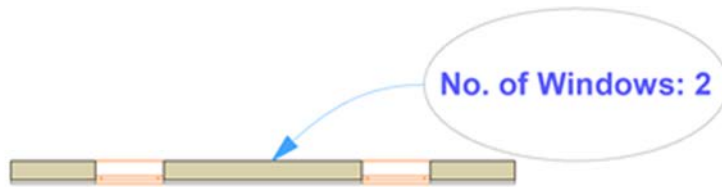


7. The Change Label's Basic Settings dialog box appears. Here, you can define the new Label's Fills and Pens. Even after placing the Label, you can further fine-tune these attributes in Label Settings.

To add information to the new Label (for example, keywords to make it easy to search for), click **Change Details** at bottom left.

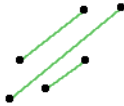
8. Click OK to close the dialog box.

9. Place the new Label. If associated to an element, its Autotext information is shown as applicable.



Example: Fixed Glass Symbol Label

In this example, the custom label consists of a simple line drawing to indicate fixed glass. This label has no frame.



It is applied to all the fixed-glass panels of a Curtain Wall (using **Documentation > Annotation > Label Components of Selected Elements**).

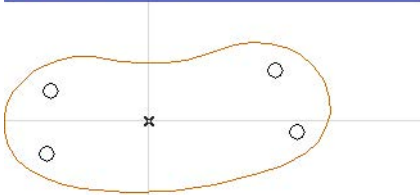
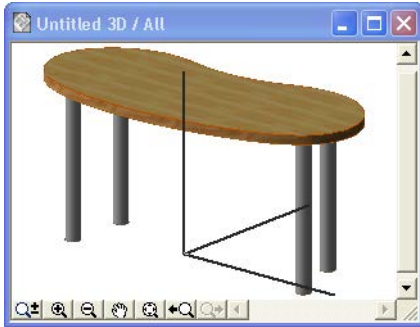


See also [Labeling Curtain Walls](#).

Save 3D Elements as Objects

1. Create the desired 3D elements within the project using Archicad construction tools (slab, wall, mesh, roof etc.).

For example, you can create a table whose tabletop is made from a slab and legs are made with the column tool.



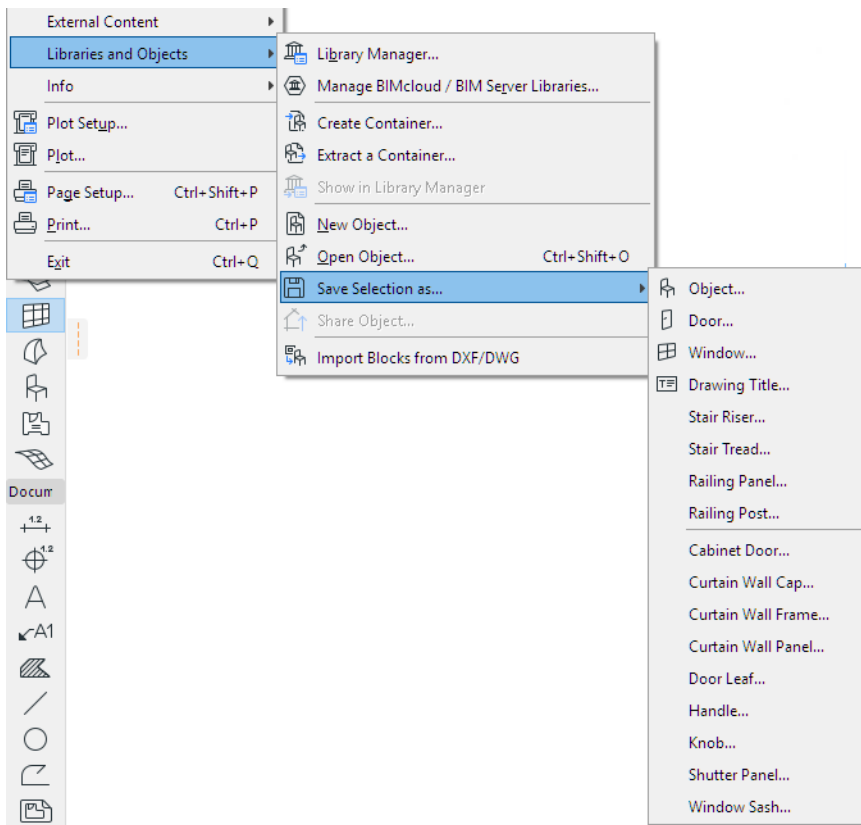
2. Select the desired elements either on the Floor Plan or in the 3D window.

Note: In the 3D window, unless you set up a special 3D projection (e.g. axonometry side view), the object will always be saved using its top view.

To use a special projection, see [Setting an Object's 3D Projection](#).

Note: If you do not want to show all the lines of the model in the 2D symbol, use the hidden line or the shading representation from **View > 3D View Options**.

3. Select, then save the object with the **File > Libraries and Objects > Save Selection as** command, and choose **Object**.

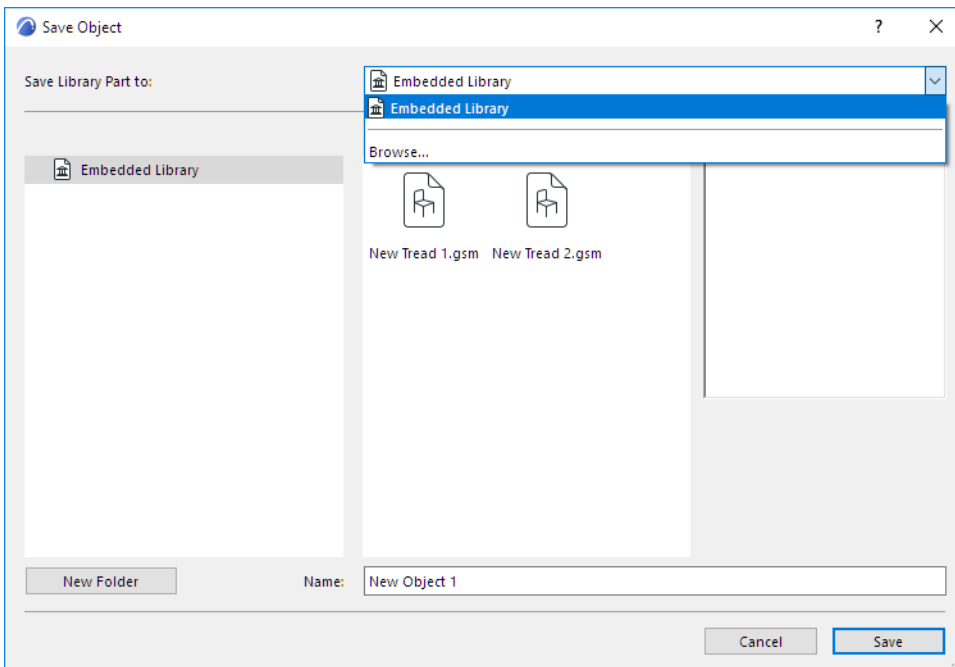


When saving an object from the 3D window, you are reminded that its top view will be used as the floor plan view when you place it, unless you set up a special 3D projection (e.g. axonometry side view).

To use a special projection, see [Setting an Object's 3D Projection](#).

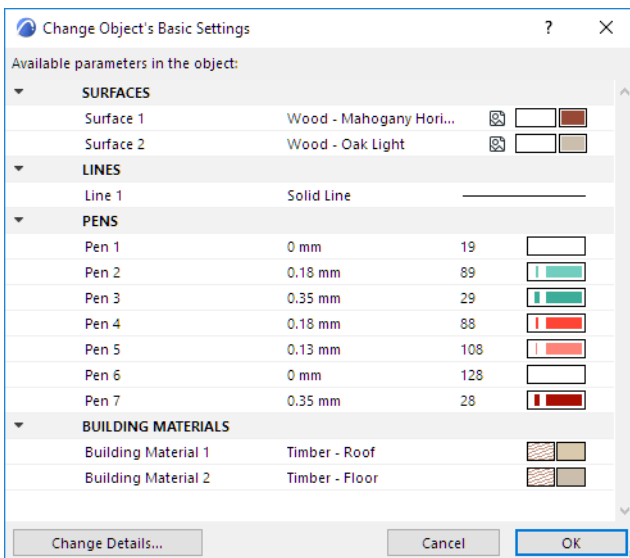
4. In the **Save Object** dialog which appears, enter a name and choose the desired folder location for the object:
- to the Embedded Library (default choice. It is recommended to save customized, project-specific objects to your Embedded Library.)
 - browse for a linked library
 - to a BIMcloud library that has been added to the project

Note: In Teamwork, you must reserve the BIMcloud Library folder before saving to it. Optionally, click New Folder to create a new folder within the selected library folder.



5. Click **Save**.
6. The **Change Object's Basic Settings** dialog box appears. Here, you can define the new library part's default Surfaces, Fills and Pens. Even after placing the library part, you can further fine-tune these attributes in Object Settings.

Also, to add information to your new library part (for example, keywords to make it easy to search for in Object Settings or BIM Components), click **Change Details** at bottom left.



7. Click **OK**.
8. Place the object with the Object tool. Note that the created object is stretchable, but will be distorted after stretching since it was not scripted parametrically.

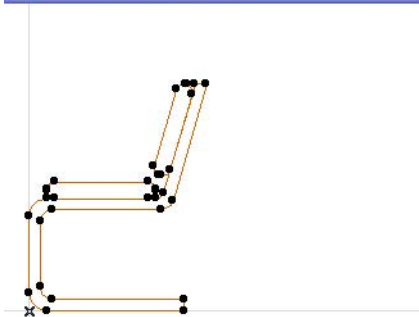
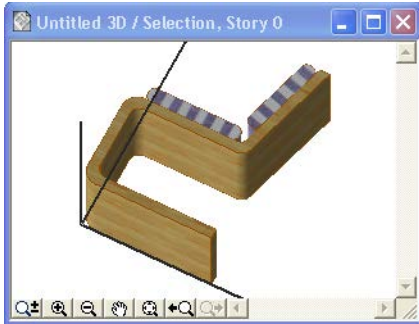
For information on editing the resulting GDL library part, see [Editing Custom Library Parts and Components](#).

[Watch the Video](#)

Setting an Object's 3D Projection

In the **View > 3D View Options > 3D Projection Settings** dialog box (Parallel projection), set the desired projection. Archicad will automatically rotate the library part according to this setting.

For example, if your model is lying on its side ([see the chair object below](#)) set the following: side view, azimuth=90°.

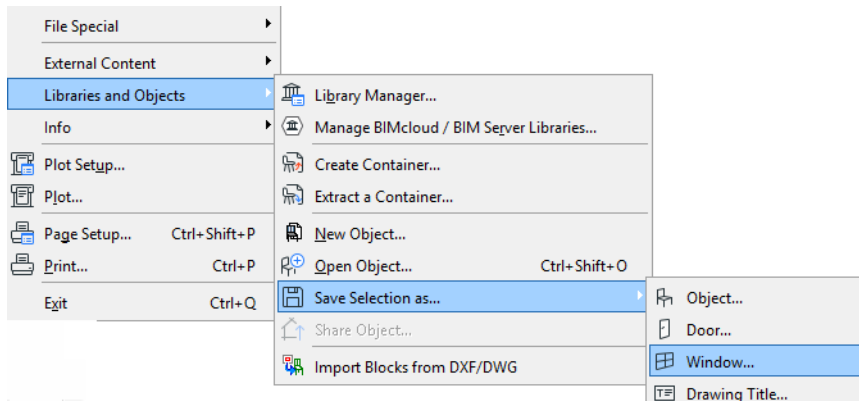


Save Rectangular Doors and Windows from 3D Model Elements

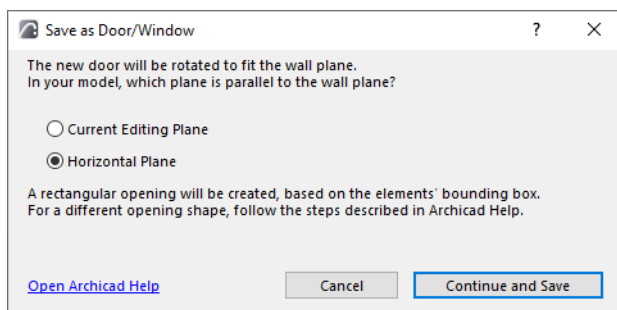
1. Create the desired 3D model of the door or window in the project using Archicad construction tools (slab, wall, mesh, roof etc.). It is recommended to draw the element on the XY plane at zero level.
2. Select the elements in the 3D window. You can use any 3D projection settings.

To use a special projection, see [Setting an Object's 3D Projection](#).

3. In the 3D window, save the selected elements with the **File > Libraries and Objects > Save Selection as** command, and choose Window or Door.



4. You are prompted to specify which plane of the model represents the host wall plane (elevation plane), into which the new Door/Window will be rotated: either the current editing plane or the horizontal plane. (This is significant if, for example, you are saving an edited Morph element which was originally a door, and which is therefore not lying in the horizontal plane of your model.)



5. In the **Save Door** (or Save Window) dialog which appears, enter a name and choose the desired folder location for the new opening:

- to the Embedded Library (default choice. It is recommended to save customized, project-specific objects to your Embedded Library.)
- to a BIMcloud library that has been added to the project

Note: You must reserve the BIMcloud Library folder before saving to it

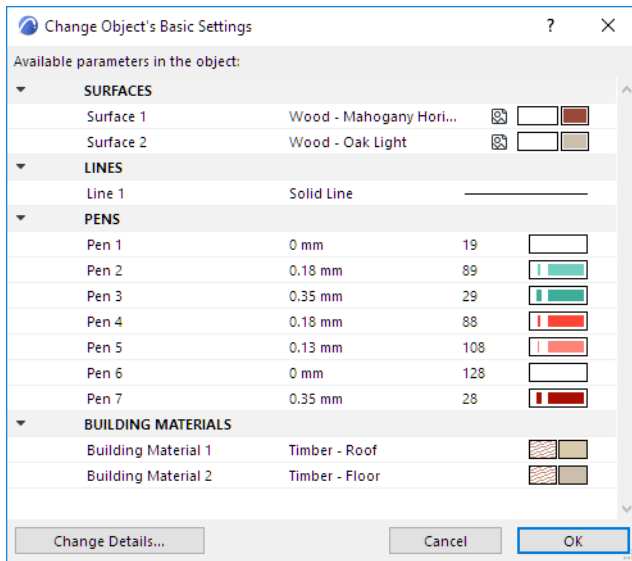
- browse for a linked library

Click New Folder to create a new folder within the selected library folder.

6. Click **Save**.

7. The **Change Object's Basic Settings** dialog box appears. Here, you can define the new library part's default Surfaces, Fills and Pens. Even after placing the library part, you can further fine-tune these attributes in Object Settings.

Also, to add information to your new library part (for example, keywords to make it easy to search for in Object Settings or BIM Components), click **Change Details** at bottom left.



8. Click **OK**.
9. Place it with the Window (or Door) tool. Note that the created object is stretchable, but will be distorted after stretching since it was not scripted parametrically.

For information on editing the resulting GDL library part, see [Editing Custom Library Parts and Components](#).

Save Custom-Shape Doors or Windows

Use this method to create a door or window having a custom shape.

For rectangular shapes, see [Save Rectangular Doors and Windows from 3D Model Elements](#).

In this process, you must create

1. a custom shape that will serve as the door/window itself, and
2. another shape that will serve as the hole or niche into which the custom door/window will be placed.

The shape of the hole is defined by a freely drawn slab or a roof, which is then assigned an ID of **Wallhole** or **Wallniche**. Slabs and roofs bearing these IDs will be displayed as a hole or a niche in the wall. (**Wallhole** and **Wallniche** are GDL commands used to create holes or niches in walls.)

To save a custom-shape door or window, do the following:

1. Create the 3D model of your custom shape window or door on the floor plan as described at the previous section.

See [Save Rectangular Doors and Windows from 3D Model Elements](#).

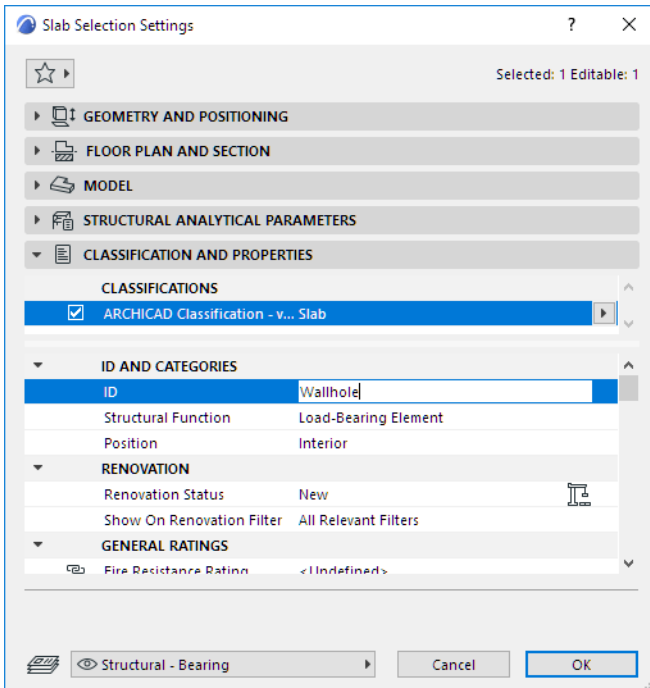


2. Using the Slab or Roof tool, create another element whose contour matches the outer contour of the desired wall hole or wall niche.

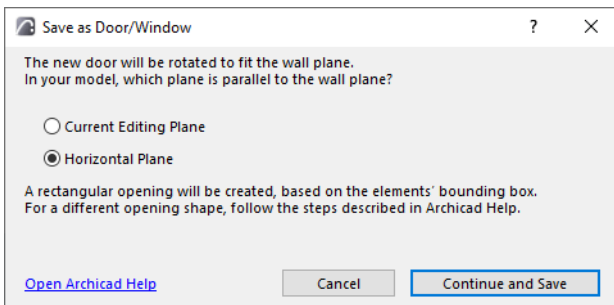


3. Select this Slab or Roof and open its settings dialog.
4. Go to the **Classification and Properties** panel and change the **ID** field to **Wallhole** if you want to make a hole and **Wallniche** if you want to make a niche in the wall.

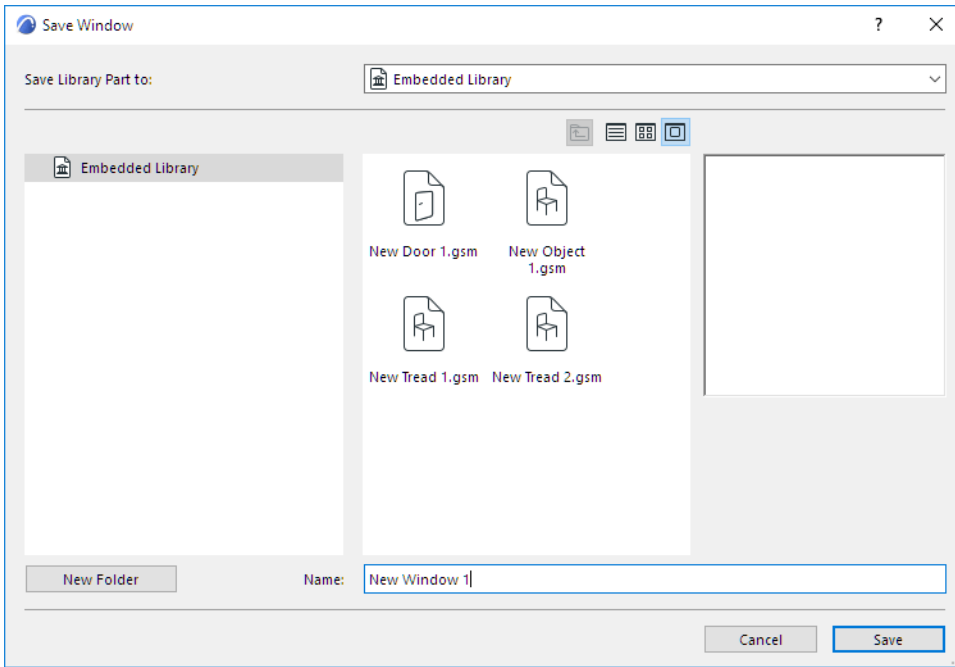
Note: The Wallhole-type element - regardless of its thickness - will always cut through the wall into which it is placed. The Wallniche-type element, however, will cut into the Wall to an extent that depends on its thickness, as set in Slab/Roof Settings. If you are making a Wallniche, be sure to set the element's thickness to the desired value.



5. Move the Wallhole (Wallniche) element onto your custom window elements.
6. Select all the elements that will compose your custom-shaped door/window: that is, the door/window element(s) plus the hole/niche element.
7. Save the selection using the **File > Libraries and Objects > Save Selection as** command, and choose Window or Door.
8. You are prompted to specify which plane of the model represents the host wall plane (elevation plane), into which the new Door/Window will be rotated: either the current editing plane or the horizontal plane. (This is significant if, for example, you are saving an edited Morph element which was originally a door, and which is therefore not lying in the horizontal plane of your model.)



9. In the **Save Door** (or Save Window) dialog which appears, enter a name and choose the desired folder location for the new opening:
 - to the Embedded Library (default choice. It is recommended to save customized, project-specific objects to your Embedded Library.)
 - to a BIMcloud library that has been added to the project
 - browse for a linked library
 Click New Folder to create a new folder within the selected library folder.

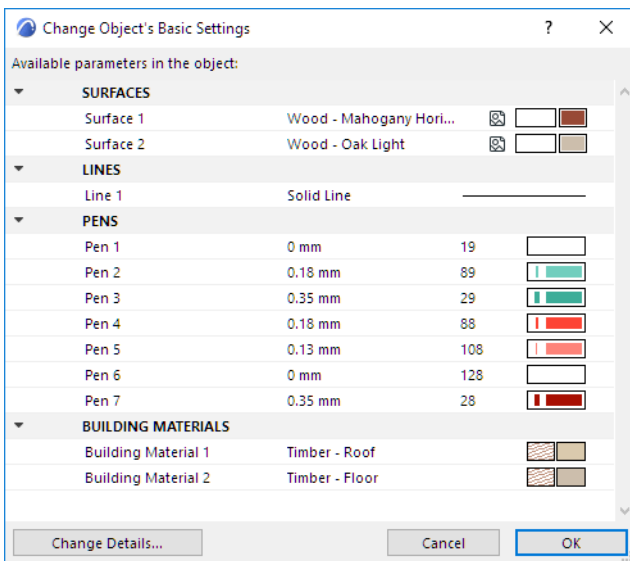


10. Click **Save**.

11. The **Change Object's Basic Settings** dialog box appears. Here, you can define the new library part's default Surfaces, Fills and Pens.

Note: You can further fine-tune these attributes in Object Settings after you place the library part.

Also, to add information to your new library part (for example, keywords to make it easy to search for in Object Settings or BIM Components), click **Change Details** at bottom left.



12. Click **OK**.

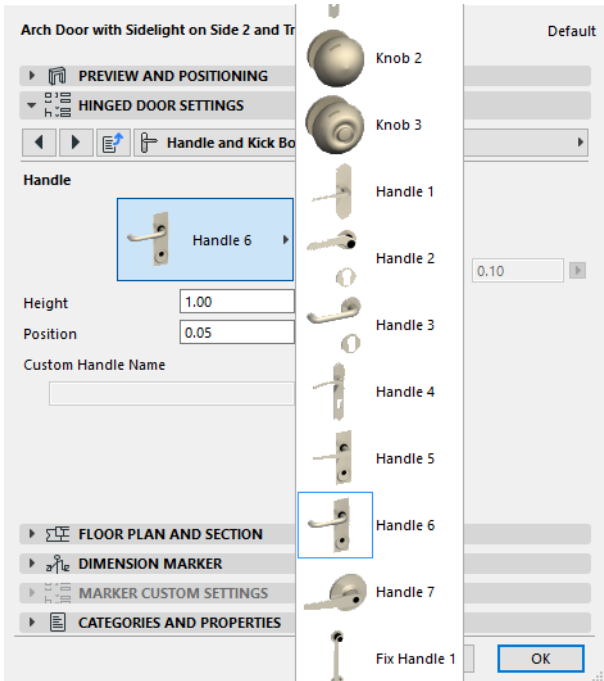
13. Place the newly created Window/Door with the Window or the Door tool. Note that the created object is stretchable, but will be distorted after stretching since it was not scripted parametrically.

You can combine multiple slabs or roofs to create complex holes or niches. Since you can define inclination angle for the roofs, you can even cut non-perpendicular holes in walls.

For information on editing the resulting GDL library part, see [Editing Custom Library Parts and Components](#).

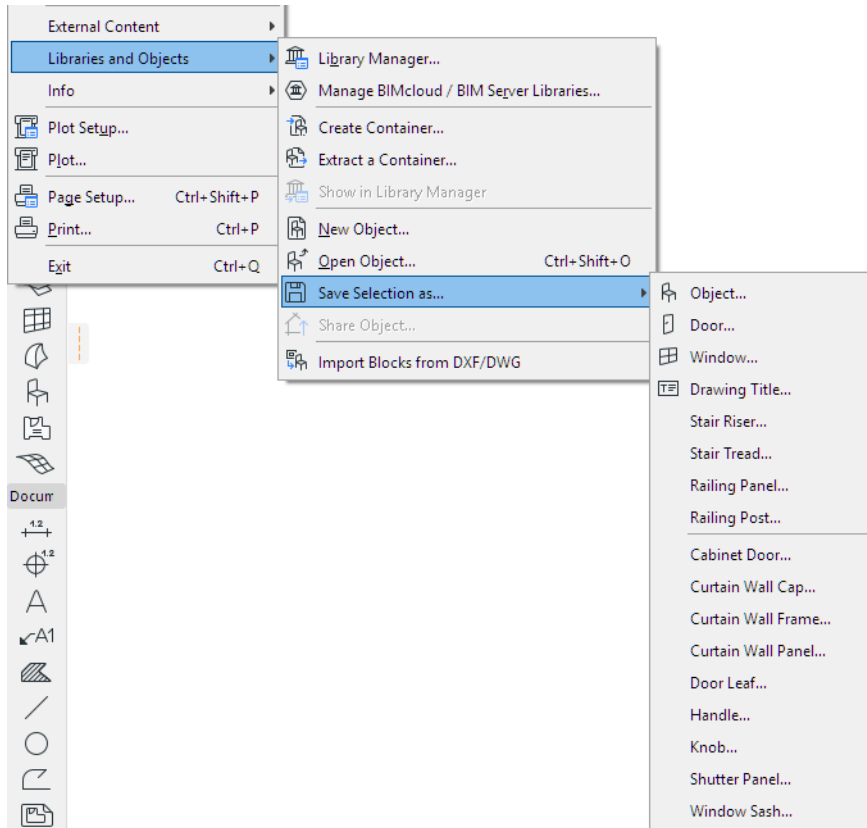
Create Custom Components for Doors, Windows, Curtain Wall, Stair, Railing

Doors, Windows, and the hierarchical design elements (Curtain Wall, Stair, Railing) are GDL objects with many parameter choices. For example, doors in the Archicad library offer you a variety of choices for door panels and doorknobs:



However, your design may need a different kind of door component - one which is not available in your library. In this case, you can draw a model element and save it as a library part, defining it as a custom component (e.g. door handle) that can then be used with your placed doors.

The possible custom components are listed under **File > Libraries and Objects > Save Selection as:**



For example, you can save a slab as a custom door leaf. The resulting GDL object is saved as a Door Leaf in the project's Embedded Library. All Doors that use Door Leafs will be able to use this custom Door Leaf.

- When applying the custom component, Archicad will automatically resize it, if necessary. For example, you may create a Door Leaf-type Custom Component that is 1000 by 2000 mm in size. If you have a Door that is 800 by 2000 mm in size, with its Trim Width 50 mm on all sides, the Door Leaf will be resized to 700 by 1950 mm. The program will stretch/shrink all parts of the Door Leaf Custom Component proportionately to fit this size.
- The automatic resize is NOT applicable to Custom Components saved as Stair and Railing sub-components. For Stair and Railing components, you must create the component in the precise size that you wish to use it in the project.

[See *Create Custom Components: Example Workflows*](#)

[Watch the Video](#)

Create Custom Components: Example Workflows

The following sections walk you through some common custom component creation workflows.

[Example: Create Custom Door Leaf](#)

[Example: Create Custom Frame or Cap Profile for Curtain Walls](#)

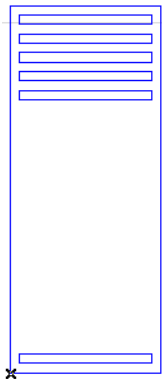
[Custom Sub-Elements for Stairs and Railings](#)

Example: Create Custom Door Leaf

Suppose you wish to create a custom leaf for a door. Follow these steps:

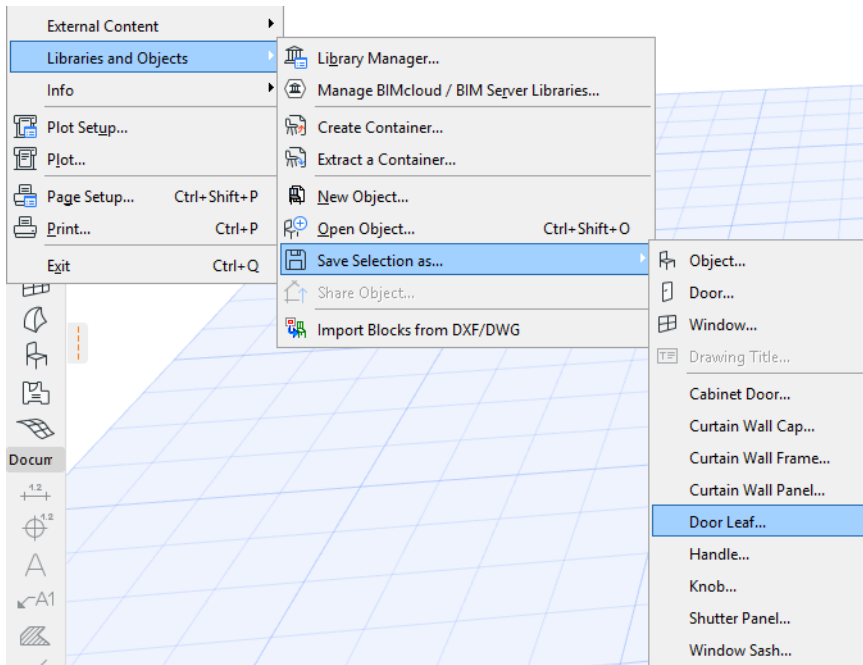


1. Using Archicad Construction Elements (such as Slabs), draw the desired door leaf on the Floor Plan or in the 3D window.
2. In case of Door/Window custom panels, what you draw in the X-Y plane of the Floor Plan Window will be 'stood upright' in the Window/Door library part (rotated 90 degrees around the X axis – as described in the GDL Reference Guide). For other custom component types, this condition does not apply.
The centerpoint of the resulting object is the center of the drawn elements' bounding box.



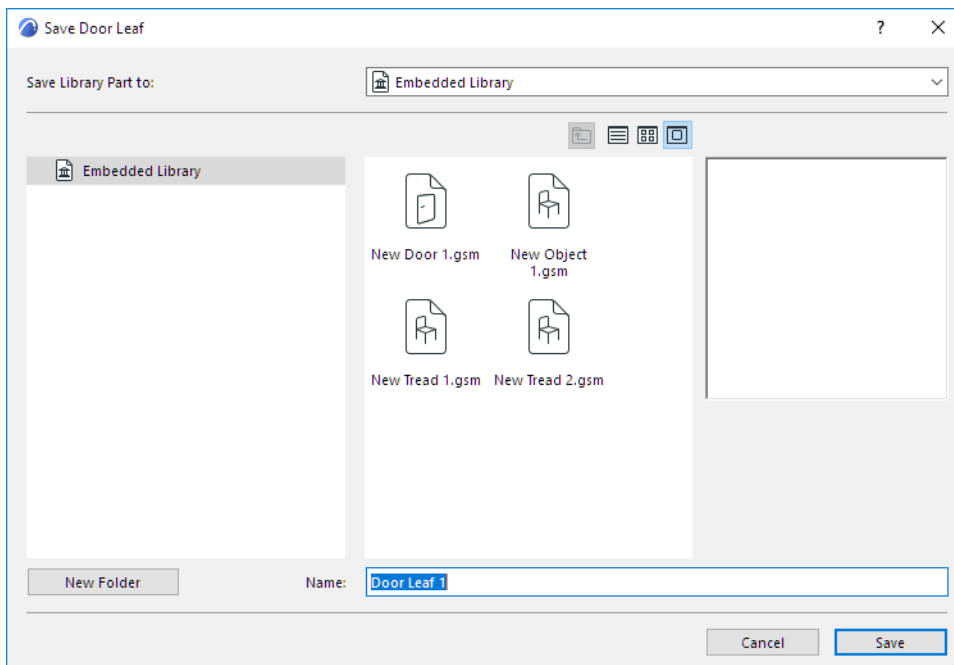
Note: You should set the attributes (Pen, Surface etc.) values of the drawn elements to the desired values *prior* to saving it. These values will be applied to the custom component when it is actually used later in the Project.

3. Select the drawn element(s) and choose the **File > Libraries and Objects > Save Selection as > Door Leaf**.



(If you have no elements selected, the command is gray.)

4. The Save Door Leaf Dialog comes up.



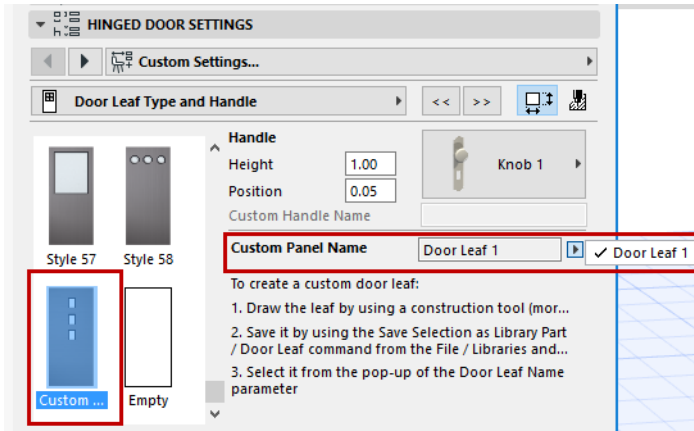
5. Specify a name for the new library part, and choose the desired folder location in which to save it:
- to the Embedded Library (default choice. It is recommended to save customized, project-specific objects to your Embedded Library.)
 - to a BIMcloud library that has been added to the project

Note: You must reserve the BIMcloud Library folder before saving to it

- browse for a linked library

Click **New Folder** to create a new folder within the selected library folder.

6. In the Door Settings Dialog, set the Door Leaf style to Custom Leaf.
7. From the Custom Panel Name pop-up, choose the custom leaf you wish to use (if there is only one custom component defined, the parameter will be automatically set to that value). Then press **OK** to accept changes.



The custom Door leaf is applied to your Door.

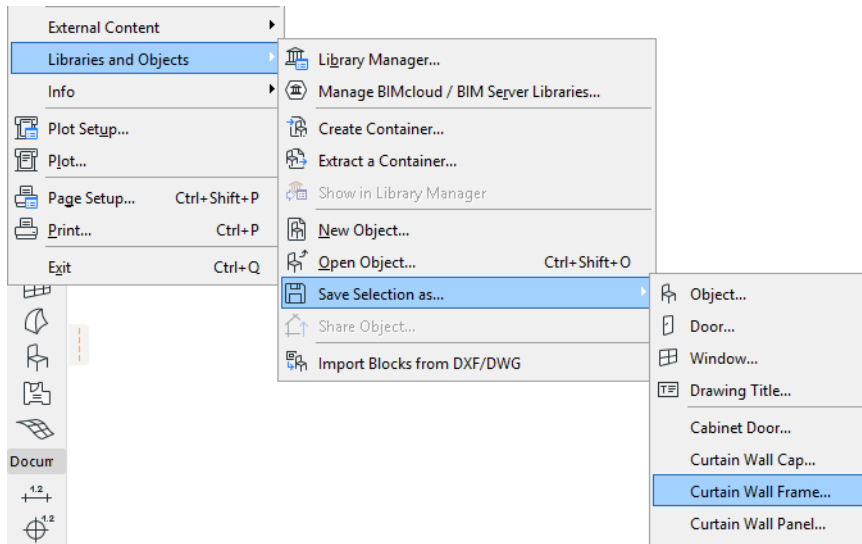


For information on editing the resulting GDL library part, see [Editing Custom Library Parts and Components](#).

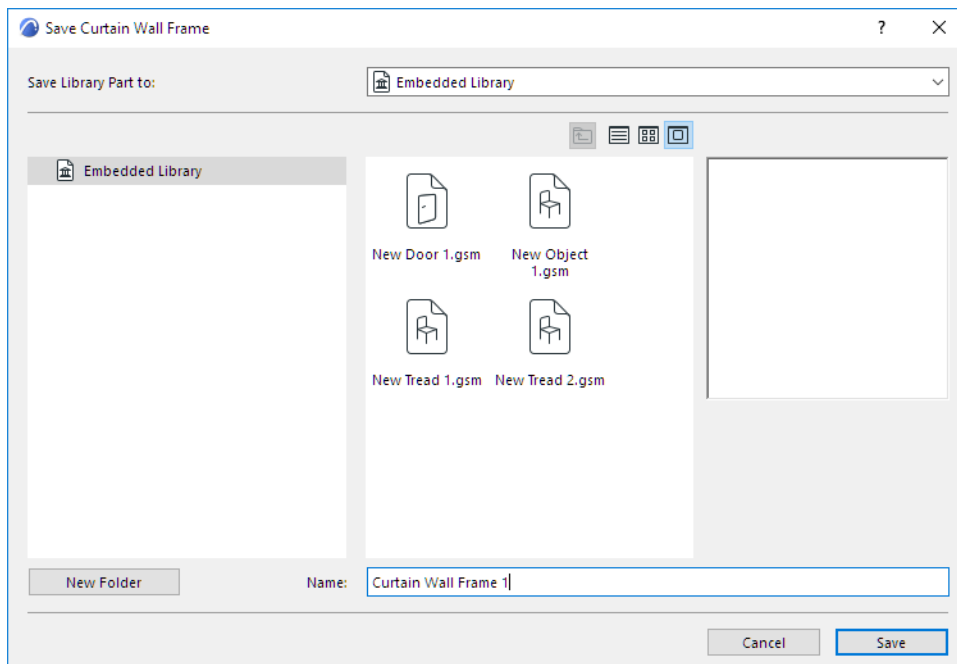
Example: Create Custom Frame or Cap Profile for Curtain Walls

Suppose you wish to customize the profile of your Curtain Wall Frame, or to add a custom Cap. Follow these steps:

1. Using Archicad construction elements, draw the custom Curtain Wall frame on the Floor Plan or in the 3D window.
2. Select the drawn element(s) and choose the **File > Libraries and Objects > Save Selection as > Curtain Wall Frame**.



3. The Save Curtain Wall Frame dialog box comes up.

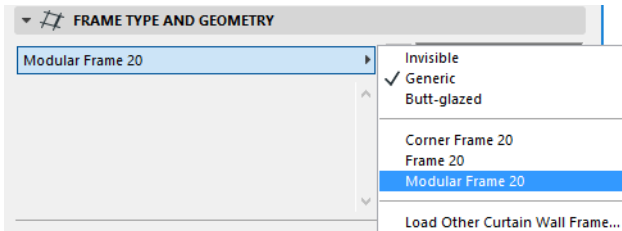


4. Specify a name for the new library part, and choose the desired folder location in which to save it:
 - to the Embedded Library (default choice. It is recommended to save customized, project-specific objects to your Embedded Library.)

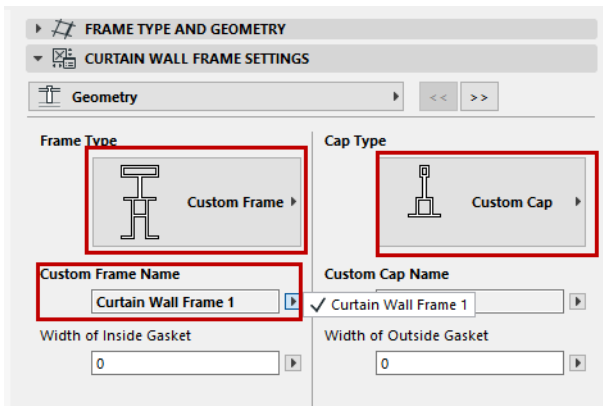
- to a BIMcloud library that has been added to the project
- Note:** You must reserve the BIMcloud Library folder before saving to it
- browse for a linked library

Click New Folder to create a new folder within the selected library folder.

5. Select the Curtain Wall whose Frame you wish to customize.
6. In Curtain Wall Selection Settings (either “Mullion” or “Transom”), choose Modular Frame.



7. In the Curtain Wall Frame Settings panel, select “Custom Frame” and/or “Custom Cap” from the Type pop-ups.



Your saved custom library parts are listed in the pop-ups below.
 Click **OK** to replace the selected Frames with the custom component(s).

Editing Custom Library Parts and Components

There are two ways to edit a GDL object saved from Archicad elements: either edit the script, or edit the original elements:

Edit the Script

See [Open and Edit Library Part](#).

Edit the Original Elements

Edit the original Archicad elements from which you have saved the GDL library part, then re-save the library part (overwriting its previous version.)

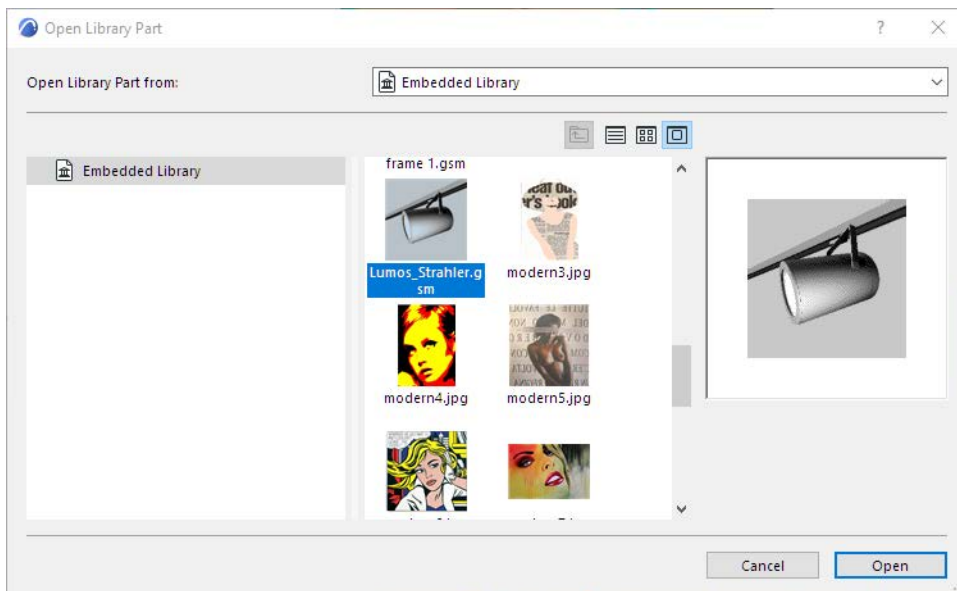
To keep the original Archicad elements, you can do one of the following.

- place them on a separate layer, then hide that layer
- save them in a separate PLN file
- save and export them as a module (.mod file)

See [Save Selection as Module](#).

Open and Edit Library Part

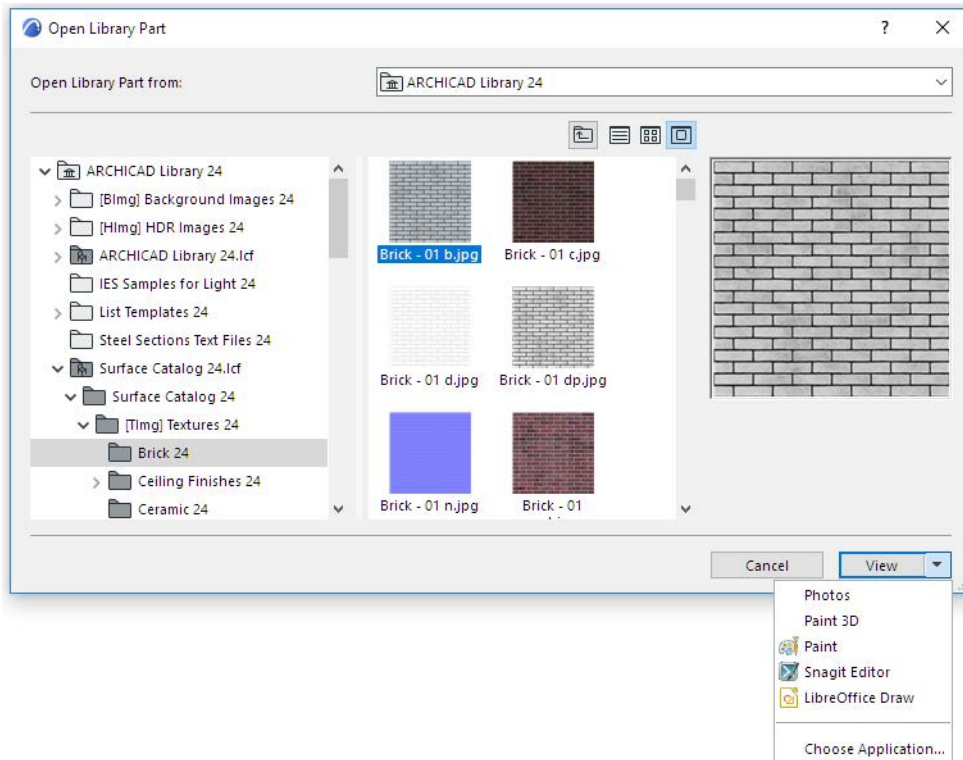
1. Go to **File > Libraries and Objects > Open Object**.
2. In the Open Library Part dialog box, select the library part. (In Teamwork, you must reserve the library part.)
3. **If it is an editable GDL Object:** Click to **Open** the GDL Object Editor. (Reserve the editing window, if it is not reserved already), and make the necessary edits.



[See GDL Object Editor.](#)

4. **If it is a non-editable GDL Object** (e.g. located in a linked library): Click **View** to see its settings in GDL Object Editor. You cannot modify the object settings here. Use Library Manager to extract the library part to a local folder, then edit it and reload it to the project.
5. If it is another type of editable library part: Click the dropdown to choose the application in which to open and edit the selected library part.

[Watch the Video](#)



6. Close the window and save the modifications.

Teamwork users will update their projects with the modified BIMcloud library data as soon as they do a **Send & Receive**.

Solo users will update their projects with the modified BIMcloud library data as soon as they do a **Reload & Apply** from Library Manager.

Access Library Part in BIMcloud Library

You can access a library part in a BIMcloud Library only if:

- You have reserved the object
- Your access rights include “Library Part – Delete/Modify”
- The library part is located in a regular folder, and is *not* a PLA or LCF file

Note: In Library Manager, the contents of PLA and LCF files are shown expanded into a tree view. To access the contents (a library part), select the item, choose **Duplicate** or **Export**, and provide a folder location. The folder is then opened, with the content available for editing.

Otherwise, the Object Editor window of such an object is Read-Only.

Edit a Placed GDL Object

To edit the parameter script of a GDL object already placed in your plan, select it, then use **File > Libraries and Objects > Open Object**

Use the appearing GDL Object Master Window to modify existing objects in a single environment. In this window, you can add or edit the **Parameters**, **Components**, **Descriptors**, **Scripts** and the **2D Symbol** of the GDL Object.

For details, see [GDL Object Editor](#).

Scripting Custom Objects

The easiest way to create custom objects is graphically, using Archicad tools:

[See the topics at *Create Custom Library Parts and Components*.](#)

However, another option is to create custom objects using GDL programming language and Archicad's scripting environment. This may be your option if you want to go beyond all of the above possibilities, or if you need Library Parts for which the shape can be controlled with user-definable parameters.

[See the *GDL Center*, Graphisoft's GDL knowledge base, at \[gdl.graphisoft.com\]\(http://gdl.graphisoft.com\).](#)

When editing object parameters, you may wish to activate the Edit GDL Library Parts toolbar (**Window > Toolbars > Edit GDL Library Parts**) for quick access to frequently used commands. You can add it to your Archicad menu structure using the Work Environment dialog box.



The creation of new Objects and the modification of existing ones is handled in a single environment: the GDL Master Window. In this window, you can add or edit the **Parameters, Components, Descriptors, Scripts** and the **2D Symbol** of the GDL Object.

[For more details, see *GDL Object Editor*.](#)

Each GDL Object can include a scale-sensitive symbol representing it in floor plan view, a set of parameters defining the possible variations of the product family and a 3D script describing the complex geometry of the element. The 3D view of the GDL Object is generated from the 3D script that takes into account the parameter choices made by the user.

Scripting in GDL is not limited to geometry; you can create elements with many useful attributes, including cost, availability, and texture.

[For information on how Archicad projects handle custom attributes of imported GDL objects, see *Custom attributes of GDL objects*.](#)

Some GDL Objects do not contain any geometric data. For example, macros can be text-only files that other Objects can refer to, without needing to include the same script in all of these Object files, while Property Objects contain descriptive data only that is used to calculate element quantities.

[For information on Property Objects, see the *Archicad Calculation Guide*.](#)

To modify an object already placed in your plan, select it, then choose **File > Libraries and Objects > Open Object**. The GDL Object Master Window appears.

For Further Information

[See *Further Resources on GDL Object Creation*.](#)

Further Resources on GDL Object Creation

Books and documentation are available on GDL and object library development.

- See the **GDL Center**, Graphisoft's GDL knowledge base, at gdl.graphisoft.com.
- The **GDL Reference Guide** will help you learn to apply Archicad's powerful Geometric Description Language for dressing up your Project with objects, ornaments and other details of your own design. It includes a section on Basic Technical Standards.

See its PDF version in the Archicad > Documentation folder or in the Archicad Help > Documentation menu.

- David Nicholson Cole's **GDL Cookbook**, aimed at both entry-level and advanced GDL programmers, is available free at <http://www.nottingham.ac.uk/~lazwww/cookbook/>.

Dedicated Object Tools: Doors, Windows, Skylights, Wall Ends

The following object-type tools in Archicad are analogous to the general Object tool, but have a number of specific parameters and behaviors.

[Doors/Windows](#)

[Corner Windows](#)

[Skylights](#)

[Wall Ends](#)

Related Topics:

[Openings](#)

[Ramp Object](#)

[Save Custom-Shape Doors or Windows](#)

[Create Custom Components for Doors, Windows, Curtain Wall, Stair, Railing](#)

[Library Part Maker](#)

Doors/Windows

The handling and behavior of Windows and Doors are quite similar, therefore they will be described together. In Archicad, Doors and Windows simulate the look and behavior of real-life Windows and Doors. They are always placed into walls.



Doors and Windows cut real, see-through openings into the wall, so that 3D visualizations are more accurate and lifelike. The glass surface lets light in and you can look through the Windows and glass Doors in PhotoRendered views. In elevations, glass panes are represented as solid shapes.

The geometry of the Window or the Door is defined by the information included in the Library Part. While some generic elements allow a large amount of freedom in modifying the size and shape of the Window or Door before or after placing it in the project, more specific designs corresponding to real-life manufacturer catalog items can only be placed as defined, with restricted variation possibilities.

To choose a door or window, open Door or Window Settings, where you can choose Doors and Windows from the loaded libraries. Use the Settings dialog to adjust the parameters as needed.

For information on setting parameters of Doors and Windows, see [Door/Window Tool Settings](#).

Archicad has separate tools for Corner Windows and Skylights. See these sections:

[Corner Windows](#)

[Skylights](#)

Related Topics:

[Placing Doors or Windows](#)

[Floor Plan Display of Doors/Windows](#)

[Setting the Window/Door Plane in Slanted or Complex Walls](#)

[Sill or Header Heights](#)

[Sill Height Using Subfloor Thickness](#)

[Anchor Reveal](#)

[Creating an Empty Opening](#)

[Moving Wall Openings](#)

Doors/Windows in Curved Wall

Door/Window Tool Settings

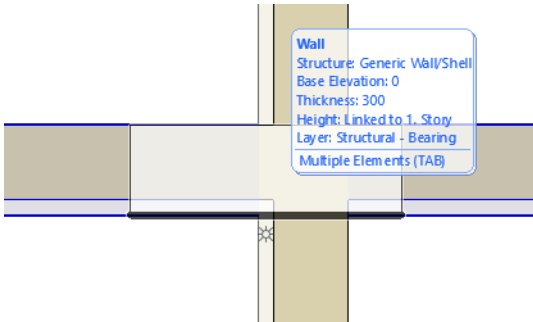
Placing Doors or Windows

Windows and Doors can only be inserted into Walls; they cannot be placed independently in the project.

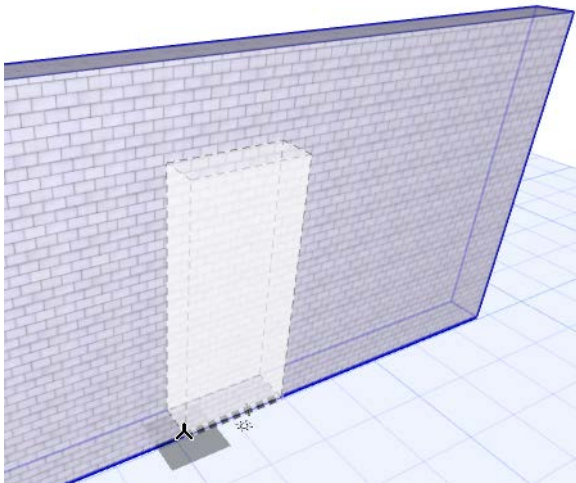
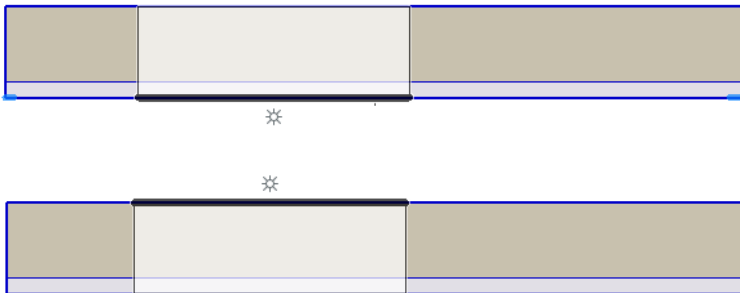
Follow these steps:

1. In the Floor Plan or 3D Window, activate the Window or Door Tool, and choose the desired Window/Door object from the library.
2. As you move the cursor over the target Wall, you will see a preview of the opening at its current location.

Note: If you hover the cursor in a place where several Walls intersect, click Tab to toggle among them until the one you want is highlighted.



3. The sun icon and thick line represent the “outside” (**reveal side**) of the Wall
To switch the reveal side, just move the cursor to the opposite side of the Wall.



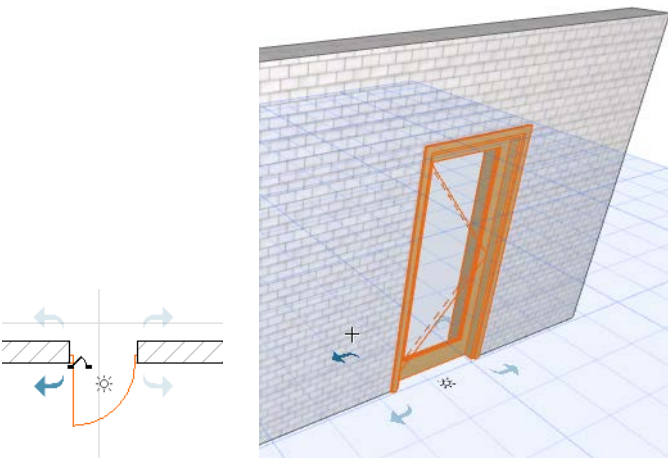
4. Click to place the opening at the desired place.

The **Anchor Point** determines whether you will place the Window or Door by its centerpoint or either side. As you place the opening into a Wall, you can switch Geometry Methods on the fly (use the Anchor Point control in the Info Box, or its shortcut.)



5. Next, you will choose the **opening direction**: toward the inside or outside of the wall, either on the right or the left: each possibility is shown by an arrow on screen.

- Move the cursor around to any of these four positions; the feedback shows you what you will get when you click.

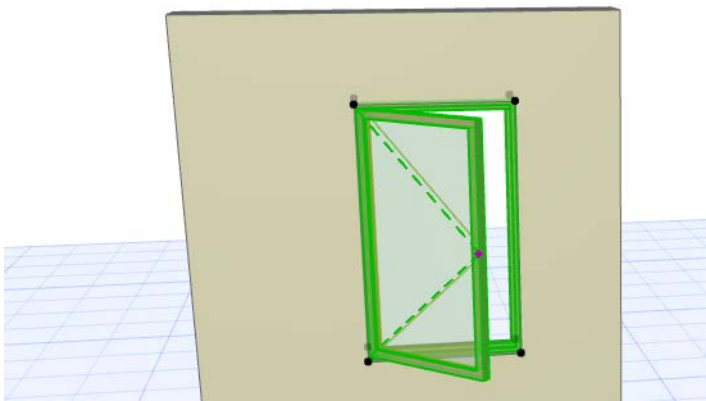
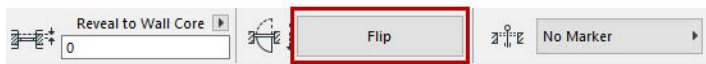
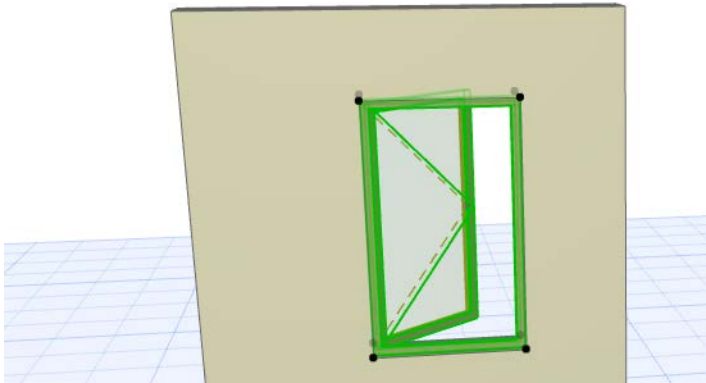


- The sun icon represents the **reveal side** that you chose with the first click. You can still change the reveal side now: press **Tab** to toggle the two reveal sides for this opening.

6. Click to place the opening.

Flip Opening Direction

The **Flip** button (in the Info Box or Door/Window Settings) switches the opening direction of a selected door/window from one side to the other, while keeping the frame in place.



In Section/Elevation/IE Windows:

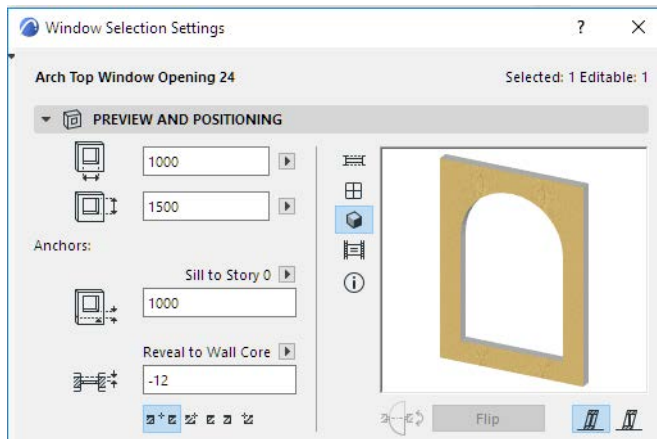
You can create new construction elements only by duplicating existing ones, using the Multiply or Drag a Copy functions.

Previews in Door/Window Settings

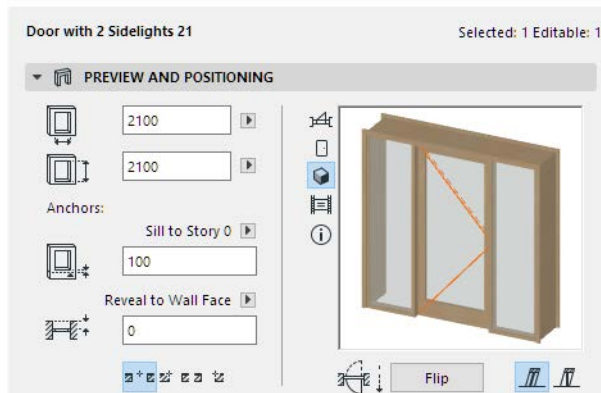
The Preview picture in Door/Window Settings uses some Wall settings to provide useful context.

For example:

- For an Empty Opening, the 3D Preview picture also indicates the thickness and surfaces of the surrounding Wall, to provide some context. (In default Door/Window Settings, the default Wall parameters are used for the Preview.)



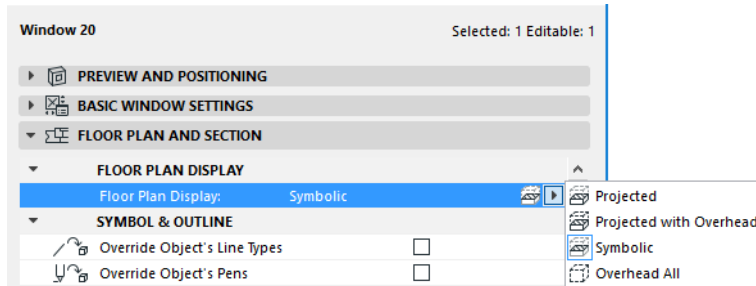
- When showing Casing, Frame and/or Sill in the 2D/3D Previews, the thickness of the Wall setting is used.



Floor Plan Display of Doors/Windows

Floor Plan Display: Symbolic vs. Projected

Use the Floor Plan Display control in Door/Window Settings to define how to display the opening:



- **Symbolic** (default setting): This is the object's standard symbol.
- **Overhead All**: the entire element's outline is shown with its overhead attributes.
Note: Define Overhead attributes (line type, pen) in the Floor Plan and Section panel of Door/Window Settings.

In case of a slanted wall, however, you might prefer a more realistic display of the cut window: choose either Projected or Projected with Overhead to see all parts of the window in a slanted wall.

- **Projected**: shows cut part of element's 3D model, plus its uncut (downward) part.
- **Projected with Overhead**: shows cut part of element's 3D model, plus the element's overhead part (i.e. the part of the element that is above the Floor Plan Cut Plane, plus its uncut (downward) part).

Attributes of Doors/Windows

There are two ways to define the attributes (line type, pen, fill) of Doors/Windows on the Floor Plan:

1. Use the **GDL attributes** (line type, pen, fill) defined in Door/Window Settings (Parameters panel or the Custom Settings panel of the Graphical Interface). This method gives you a great many line/fill parameters, giving you greater control over the precise and detailed display of the opening and each of its components separately.
2. Use the **2D attributes** set for the opening on the **Floor Plan and Section** panel of Door/Window Settings. This method involves a reduced number of parameters for simplicity.

For details, see [Override GDL Object Attributes of Doors/Windows](#).

Model View Options for Doors and Windows

The Door Options and Window Options settings in the Construction Elements panel of Model View Options (**Document > Model View > Model View Options**) allow you to show Doors and Windows on the Floor Plan in full, with contours only, or as empty openings, or to show or hide their dimensions or markers.

See [Model View Options for Construction Elements](#).

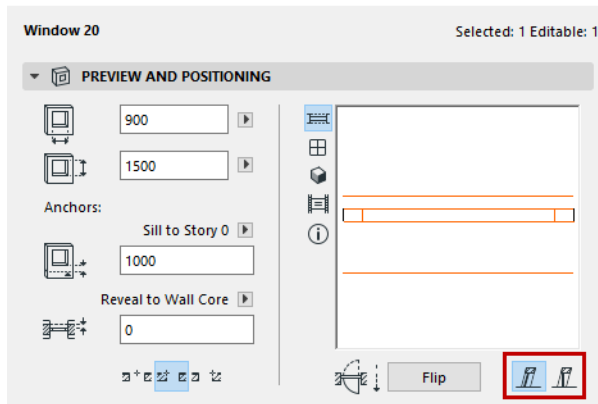
To define the display of Door/Window handles and opening lines, use the Options for GDL Objects panel of Model View Options.

See [Miscellaneous Settings for Library Parts](#).

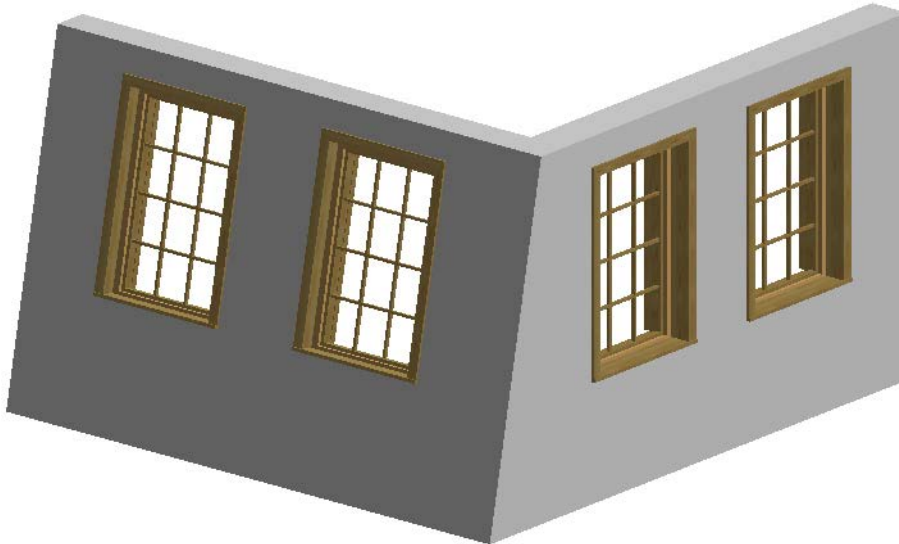
Setting the Window/Door Plane in Slanted or Complex Walls

Windows and Doors can be placed in any type of wall, including polygonal, slanted and complex shapes.

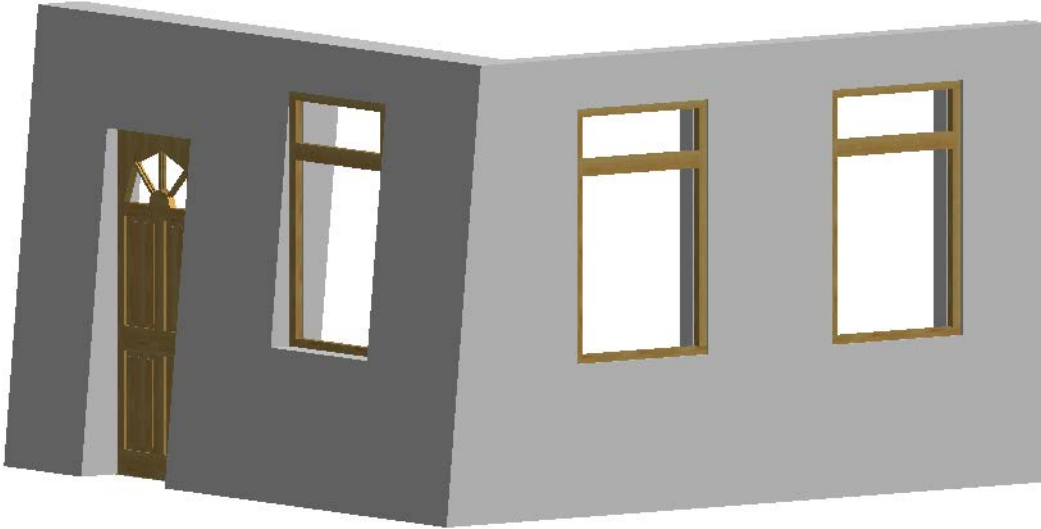
If your project contains slanted walls, use the Door/Window **Opening Plane** icons ([Door/Window Preview and Positioning Panel](#)) to make the plane of your openings either associated to the wall plane or remain vertical.



- **Associated to Wall:** the slant of the Window/Wall will be associated to the Wall into which they are placed. In slanted Walls, the Window/Door slants at the same angle as the Wall. In Double Slanted Walls, the Window/Door slants at the same angle as the side of the Wall into which the Window/Door was placed. Thus, if you move the Window/Door to the other side of the Wall, its slope will change accordingly.



- **Vertical:** the Window/Door will be vertical regardless of the type of Wall it is placed into.

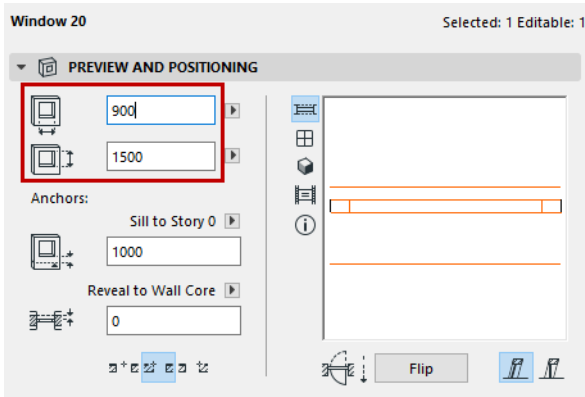


In **Complex** Profile Walls, the slant of the Window/Door is determined in the Profile Manager: use the **Opening Reference** Layer in the Design Layers panel to define the slant of Windows/Doors set into the Complex Wall.

For more information, see [Complex Profiles](#).

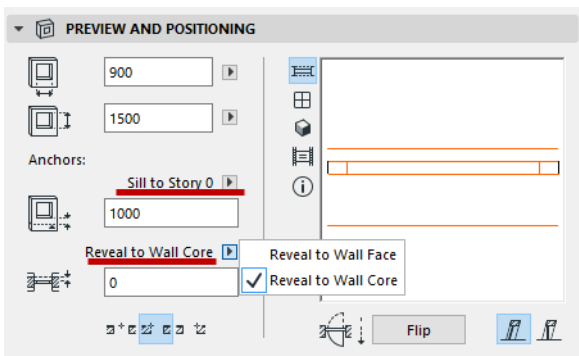
Sill or Header Heights

Use the Preview and Positioning Panel of Door/Window Settings, or the Info Box, to set the height and width of a door/window.



Every time you change a window/door height, the change will affect either the sill or the header. Depending on how you choose to anchor the opening, sill or header height can be measured from the wall base, the wall top, or a given story height.

In the **Preview and Positioning** panel, the pop-up lets you choose any one of these as a point of reference for the sill/header value.

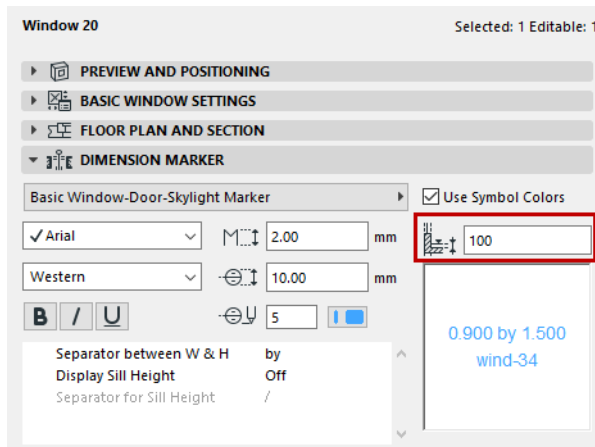


- Sill to Wall Base
- Sill to Wall Top
- **Sill to (current) Story:** the height of the Sill will be measured relative to the level of the Current Story (the last story open on the Floor Plan). To choose a different Story, click the **Select Story** option and choose a story from the dialog box.
- Header to Wall Base
- Header to Wall Top
- **Header to Story:** same method as the **Sill to Current Story** options, but you must provide the Header height. To choose a different Story, click the **Relink Anchor Story** option and choose a story from the dialog box
- **Relink Anchor Story:** If you choose this option (select the Story name from the list in the appearing dialog box), the chosen story's height will serve as the Anchor from which the Window/Door should be measured. This option may be used, for example, when a Glass Curtain Wall is created and all individual panes must have their heights set to a common height.

Sill Height Using Subfloor Thickness

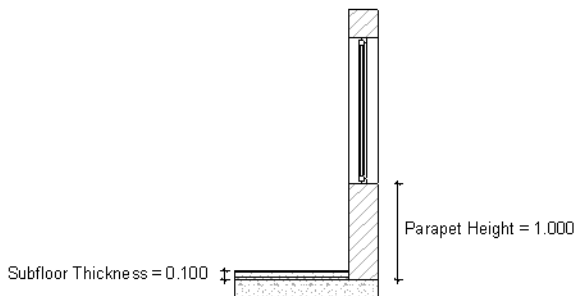
Use the Subfloor Thickness field, in the Dimension Marker Panel of Door/Window Settings (or the Info Box), to fine-tune the marker display of your sill height value if your floor level takes into account carpeting or parquet flooring, and if your wall base is not at this floor level.

Archicad measures the height of a door/window sill from the wall base (or a given Story level, as set in the Preview and Positioning Panel.) But if you want the marker display of this value to reflect the real floor level, enter this additional thickness in the Subfloor Thickness field.



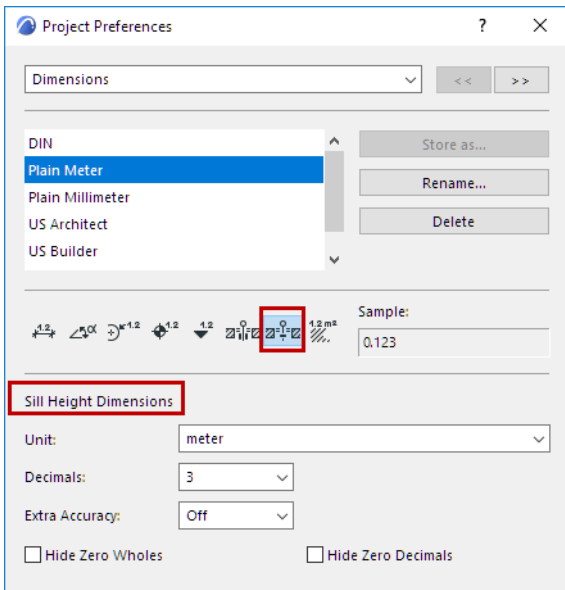
Archicad will still measure the sill value from the wall base, but the door/window *marker* will show a sill height value that subtracts the subfloor thickness from the entered sill value.

For example, say the Wall Base is at -100 mm below the floor level (Subfloor Thickness is 100 mm). The window sill (parapet) height is 1000 mm, but your effective Sill Height, for the purposes of construction documentation, is really 900 mm from the floor. So the window's sill height, as displayed in the Floor Plan marker, will show 900 mm.

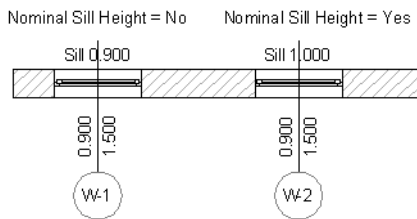


Note: Entering a value into the Subfloor Thickness field does not affect the actual positioning of the door/window within the wall; it is merely an alternate way to display sill/header height in the door/window marker.

Note: Sill height markers use dimension units set specifically for this type of value: go to **Options > Project Preferences > Dimensions**, and choose Sill Height Dimension.

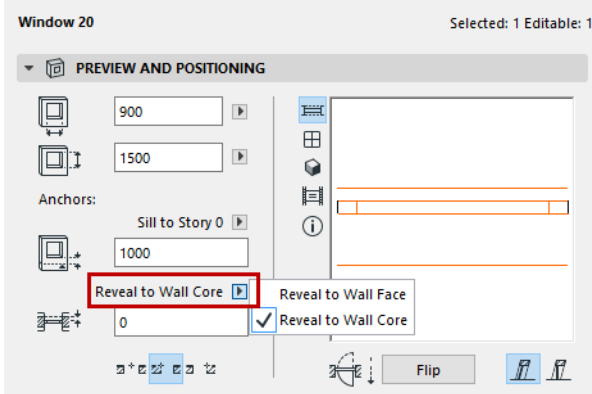


Some markers have a checkbox to display the **Nominal Sill Height** (e.g., the displayed Sill value does not take subfloor thickness into account). The illustration below shows the effects of unchecking and checking this box.



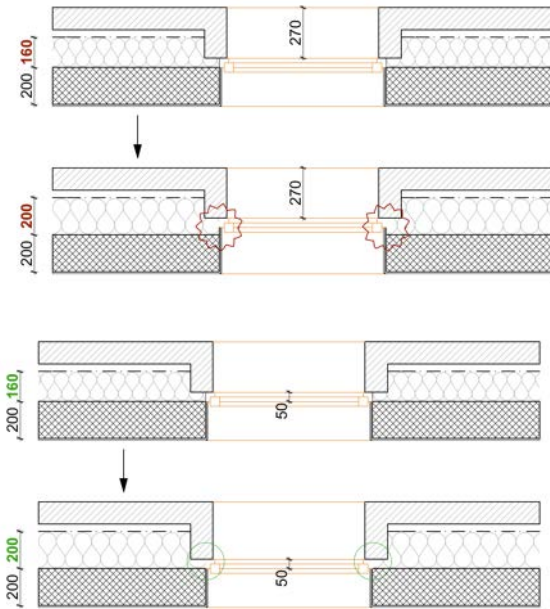
Anchor Reveal

Anchor the Door/Window reveal to either the Wall face or the Wall core (the closest core skin, if there are several core skins). This is relevant if you are using a composite Wall which has a core.

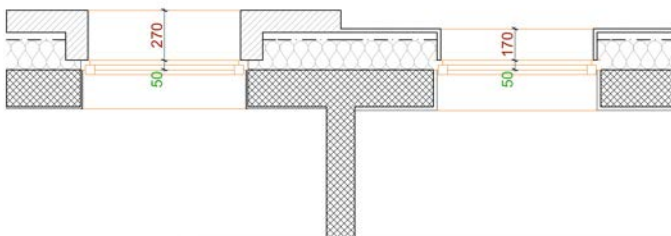


You may wish to anchor openings to the Wall core if

- you later make changes in the Wall's composite structure (e.g. thicker insulation). The reveal depth will automatically adjust to maintain the window's distance relative to the Wall's core skin.



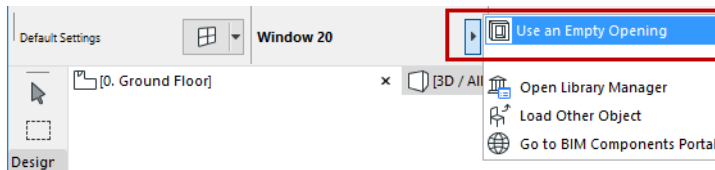
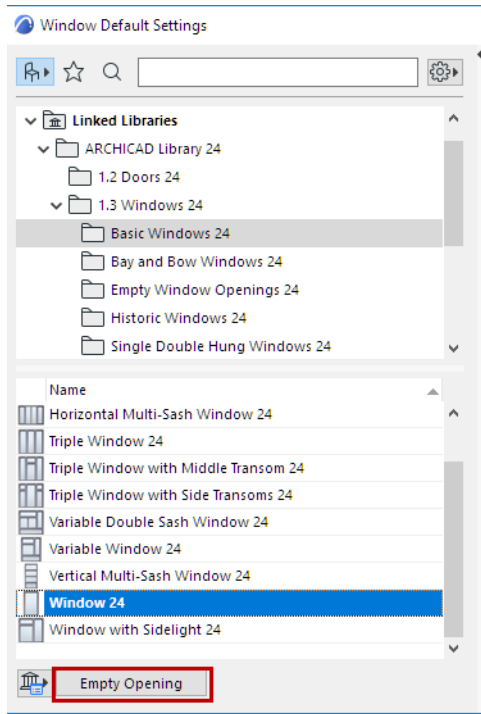
- you place openings into several similar composite Walls (that only differ in the finishes, e.g. a brick finish and a plastered wall). The openings will be aligned; you don't have to adjust Reveal depths in order to keep them aligned.



Note: If you place an opening whose reveal is anchored to the wall core, then change the composite Wall so that it no longer has a core, then the reveal anchor will change to “Wall Face,” and the reveal depth will be automatically recalculated to retain the opening in its original location.

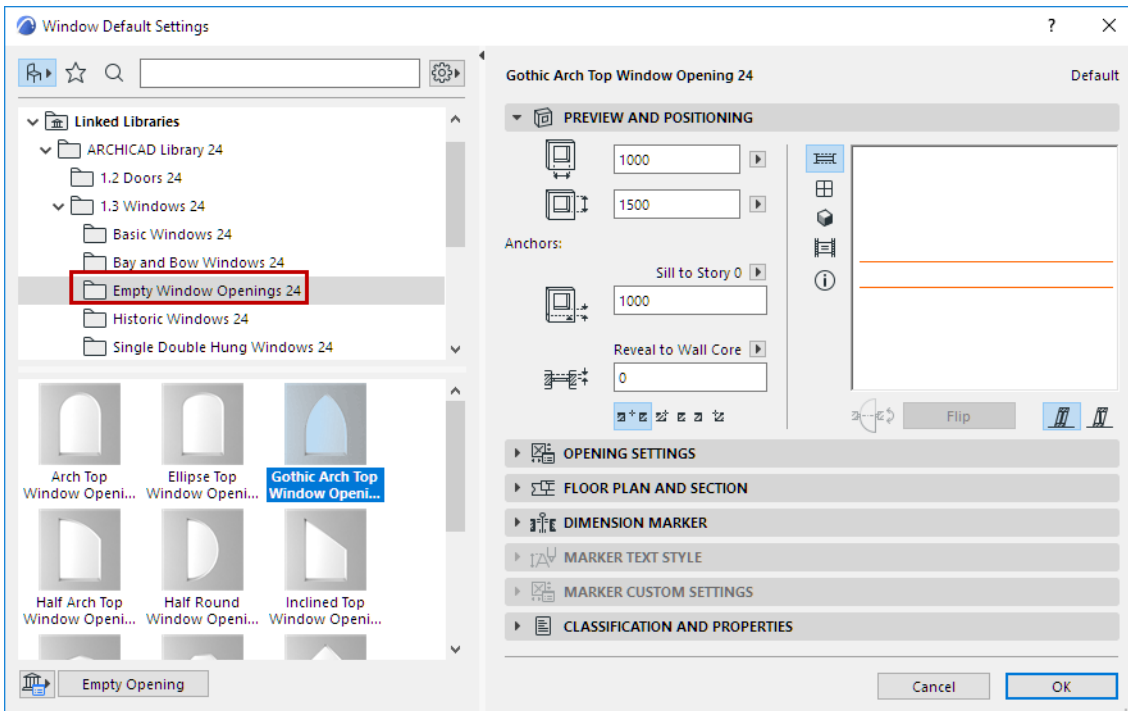
Creating an Empty Opening

To create a simple opening without placing a library part type Door or Window, click the Empty Opening button at the bottom of the Library Part Settings dialog box (or from the Info Box.)



Place the opening in the wall, as you would a Window or Door. Adjust the size and other parameters of the selected opening in the Preview and Positioning Panel of its Settings dialog box.

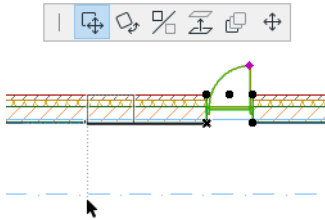
For empty openings of varying shapes, with extensive parameters, choose a window from the Archicad library's Empty Openings folder.



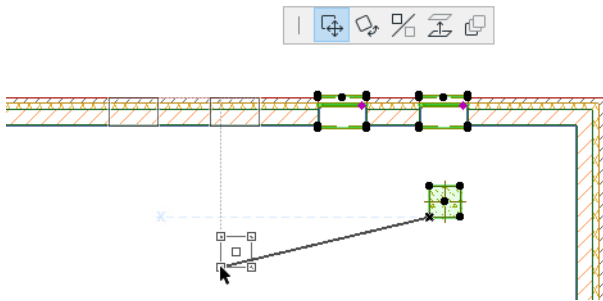
Moving Wall Openings

Use the **Drag**, **Rotate**, **Mirror** and **Elevate** commands to move wall openings (i.e. Doors and Windows). All these operations are available in the Floor Plan, Section/Elevation/IE windows and the 3D Window, for slanted and straight Walls only.

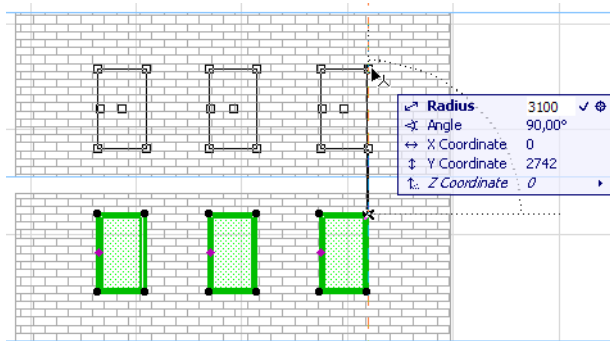
Door/Windows do not need to be dragged by their hotspots. After selecting the Window/Door, choose the drag, move or multiply command, then click anywhere in the view to define the starting and ending points of the operation's vector.



Windows/Doors may be dragged/moved/copied along with other Element types in one step. In the following image you can see that the Column is dragged. The Windows, selected together with the column, will also be dragged within their Wall by a distance that equals the component of the Drag vector.

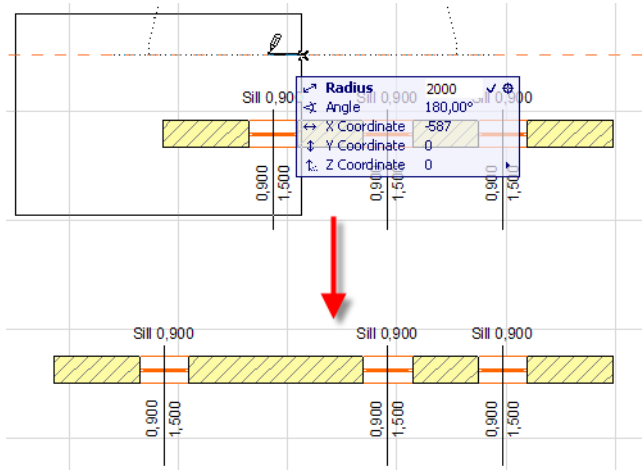


- Windows/Doors may be moved across multiple Wall elements. You can drag or copy a Window from one Wall and end up in another Wall element.



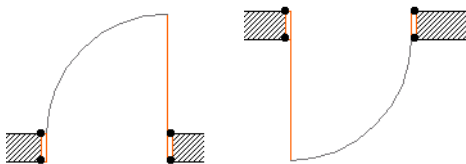
Moving an opening across multiple elements works only if the Wall elements involved have their reference-side wall planes all in the same plane. If there are two vertical Walls involved, their Reference Lines must be aligned in a single vertical plane. If they are slanted Walls, their Reference Wall sides must align in a single slanted plane. The thicknesses and other attributes of these Walls, however, do not need to be identical; the Walls involved do not even need to be connected to each other.

Also, when you stretch a Wall with the Marquee, the Windows/Doors included in the Marquee will also be dragged/copied, so they retain their position relative to the end of the Wall.



Rotating a Window or Door

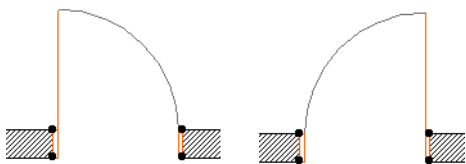
To rotate Doors and Windows, select the opening and click the **Rotate** command (from Edit menu or pet palette). The selected opening will rotate by 180 degrees around its center (there is no need to draw any vector). Any number of openings can be rotated at the same time.



Note: Rotated text blocks, labels, dimensions and fill polygon area labels are always readable from the bottom and the right side of the drawing.

Mirroring a Window or Door

To mirror a Door or Window, select the opening and click the **Mirror** command (from **Edit** menu or pet palette, or use **Ctrl+M**). The pencil cursor appears. Click either on an edge of the host wall or outside the wall. The Door/Window will be mirrored (with its opening direction flipped laterally) along the axis where you click. If you click outside the host Wall, the mirroring axis is the line perpendicular to the Wall at the clicked point. (With trapezoidal walls, the mirroring axis is perpendicular to the reference axis.)



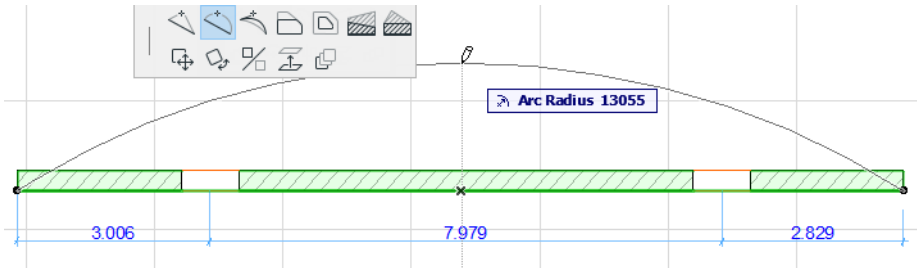
- Mirroring can create an opening that falls entirely beyond the end of the wall. You can then select and move it in the same plane as the original host wall.

Stretching Doors and Windows

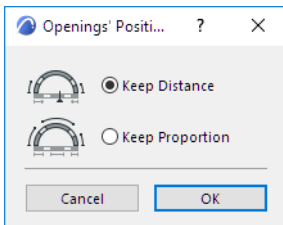
See [Stretching Objects](#).

Doors/Windows in Curved Wall

Suppose you have placed windows into a straight Wall (at the given distances from either end of the Wall), then later curve the Wall (as in the picture below).

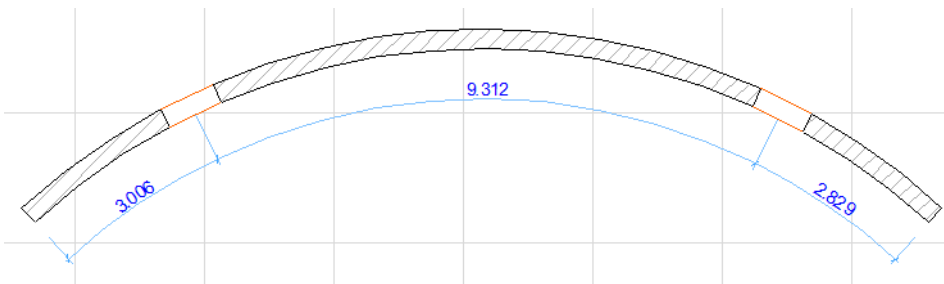


The appearing **Openings' Position** dialog box allows you to choose how to handle the openings' new position in the curved Wall:

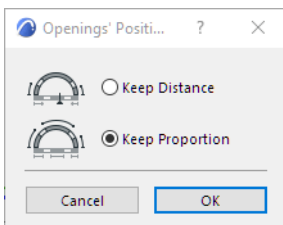


Keep distance: This option will ensure that the openings' distance from the nearest Wall end remains unchanged. (Openings are measured from their center point.)

Note: The Wall segment that you click to curve will be the segment that is stretched.



Keep proportion: This option will keep constant the position of the openings relative to the Wall length.



Corner Windows



Archicad has a separate **Corner Window** tool.

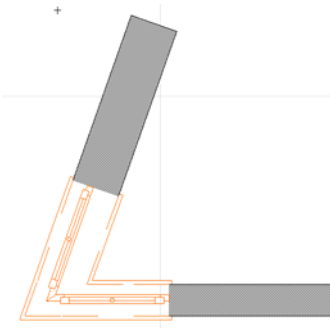
(For INT users: By default, the Corner Window tool is located in the “More” section of the Toolbox.)

The Corner Window Settings dialog box is similar to the Door and Window Settings dialog boxes.

For more information, see [Door/Window Tool Settings](#).

There is no separate object type associated with the tool; most of the windows in the standard Archicad library can be placed using the Corner Window tool.

Corner Windows are placed at straight wall corners.

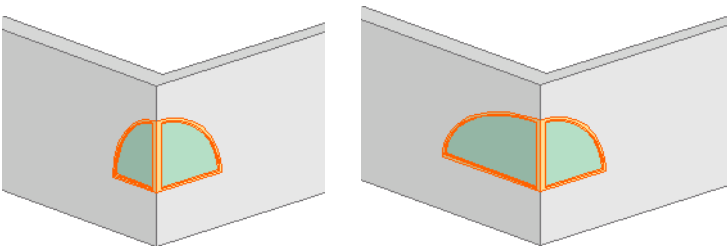


Please note:

- Corner Windows cannot be placed in a Profiled Wall.
- Corner Windows cannot be placed in trapezoid or curved Walls.

When you wish to place a Corner Window at a wall corner, you can click anywhere on the wall: Archicad will automatically snap the window to the nearest corner of the wall and create a mirrored copy of it on the adjoining wall. The second window’s parameters and properties are identical to those of the first. Their angle and position are automatically adjusted.

- You can only place one pair of corner windows at any given corner.
- The angle between the two walls cannot be equal to either 0 or 180 degrees.
- The placed windows can be stretched manually and individually without breaking the link between the two windows.



- Corner Windows turn into simple Windows if:
 - You drag away, mirror or rotate any of the two placed Windows.
 - The angle between the two Walls is changed to 0 or 180 degrees.

- One of the Walls is elevated (since Windows are part of the host Walls, the two Windows' elevations become different).
- One of the connecting Walls is deleted or dragged away.

For more information, see [Door/Window Tool Settings](#).

Skylights

Skylights are analogous to Doors and Windows. They can be intelligently placed into Roofs and Shells, using the dedicated Skylight tool.



The Skylight placed on the surface of a Roof or Shell will be automatically inserted into the parent element. When placed into a Roof, it is adjusted to the correct Roof pitch angle.

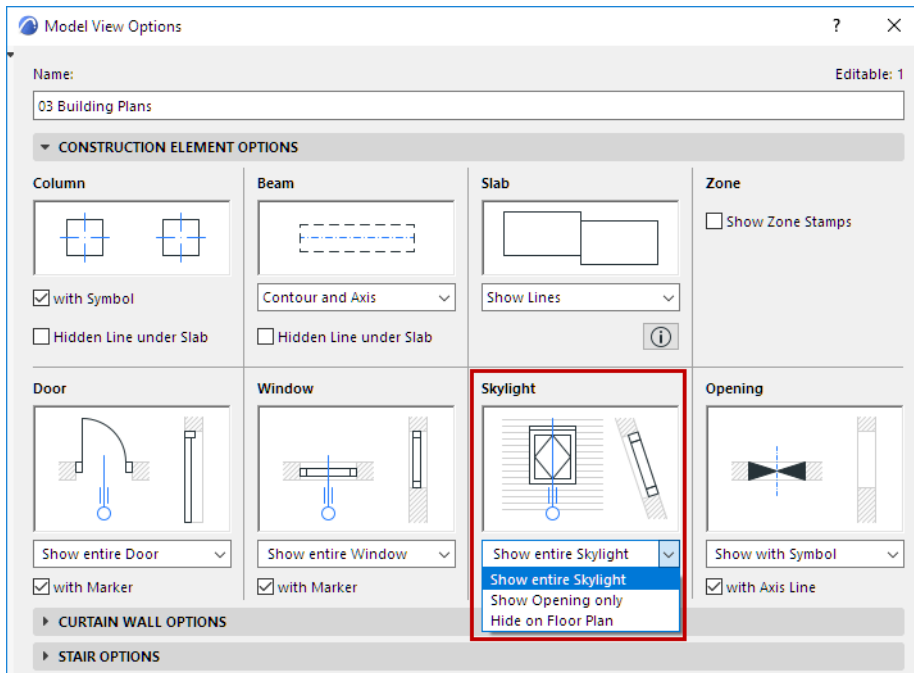
The panels of Skylight Settings are similar to those of the Window/Door, Object and Lamp tools.

[See *Skylight Tool Settings*.](#)

Skylight Display on Floor Plan

- The **Floor Plan display** of Skylights placed in Archicad 15 and later is based on a 3D projection, not a symbolic depiction.
Exception: Skylights migrated together with older-version projects to Archicad 15 and later will be displayed, by default, using “symbolic” attributes for their Floor Plan display.
- Skylights are displayed using the **Floor Plan Display settings** (e.g. Show on Story) of the host Roof or Shell.
- **Pens and lines** are set for each Skylight, as for Doors/Windows, in the Floor Plan and Section panel of Skylight Settings.

- **Model View Options** (Construction Element Options) include options for Skylights, analogous to those of Doors/Windows.



Related Topics:

[Placing and Moving a Skylight](#)

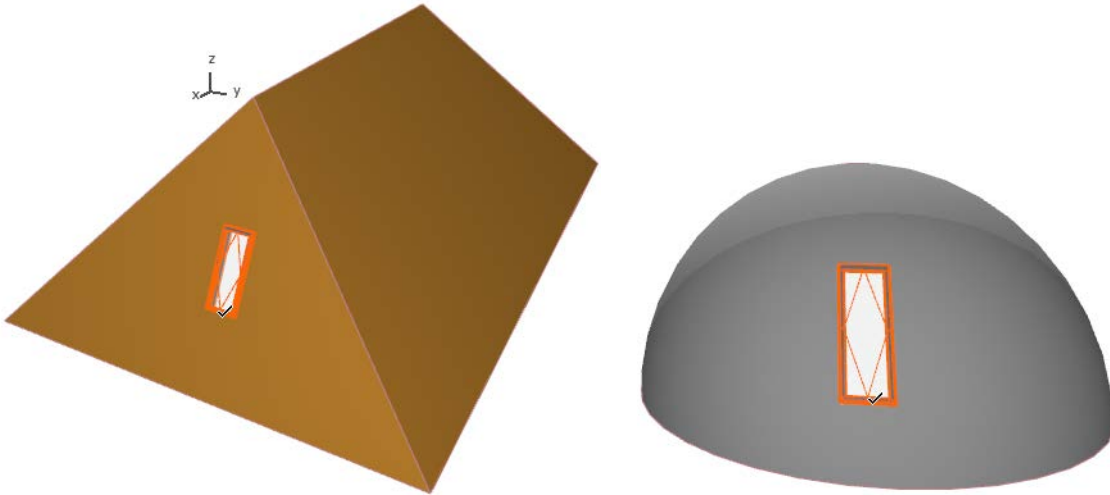
[Skylight Constraint Relative to Roof](#)

[Skylight Tool Settings](#)

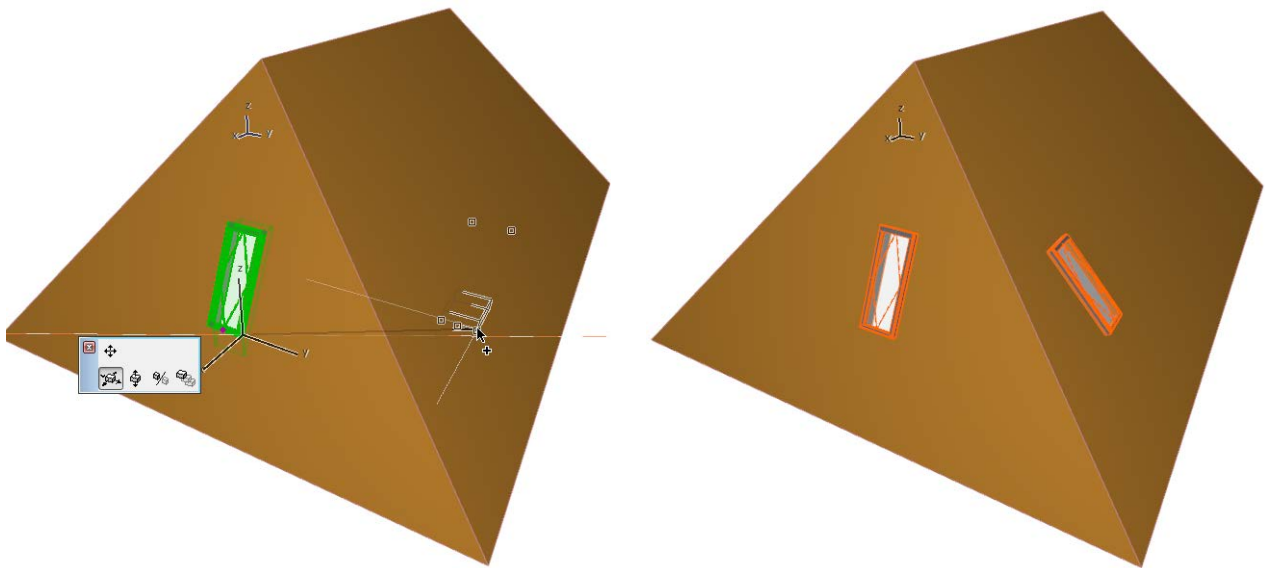
Placing and Moving a Skylight

Skylights can be inserted into Roofs or Shells in the Floor Plan or in 3D view.

Select the Skylight tool and click to place it on either a Roof or a Shell. It will be placed by its anchor point as defined in the Skylight Info Box.



When you drag (or drag a copy of) a Skylight within a Shell, or from one Roof plane to another, the Skylight will automatically adjust itself to the new location.



Position Skylight to Curved Roof Plane

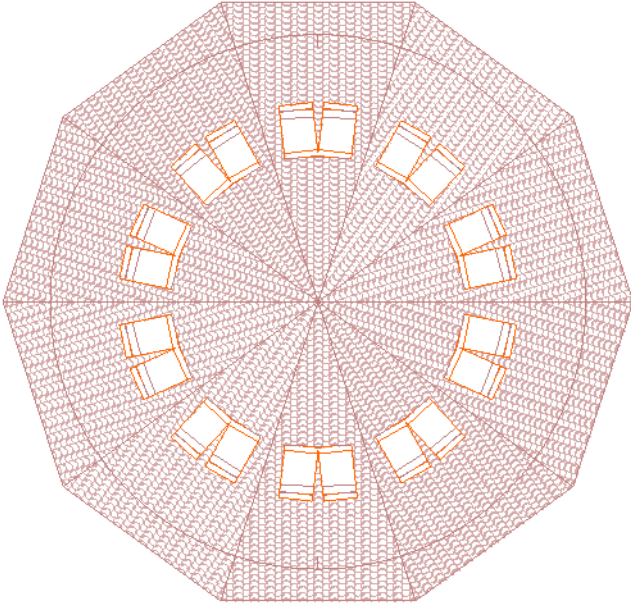
When you place or move a Skylight within a curved Roof segment (having a curved pivot line), Archicad can adjust the Skylight placement using one of two logical approaches:

- Assume that the Roof plane you modeled really represents a curved Roof plane
- Assume that the Roof plane you modeled really represents one of several adjacent flat segments

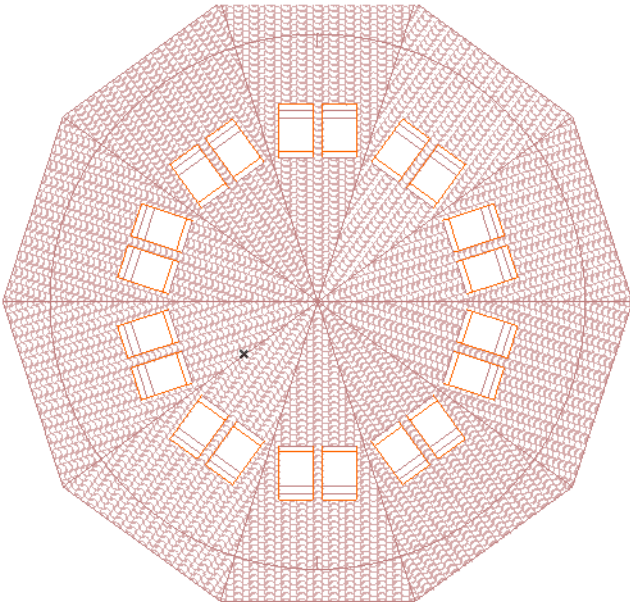
Depending on the assumption, the Skylight may be placed in a slightly different way.

The option, in Roof Settings (or the Info Box), is **Skylights treat segments as curves**. By default, it is checked. This means that Archicad assumes that a Roof plane with a curved pivot line represents a curved Roof plane.

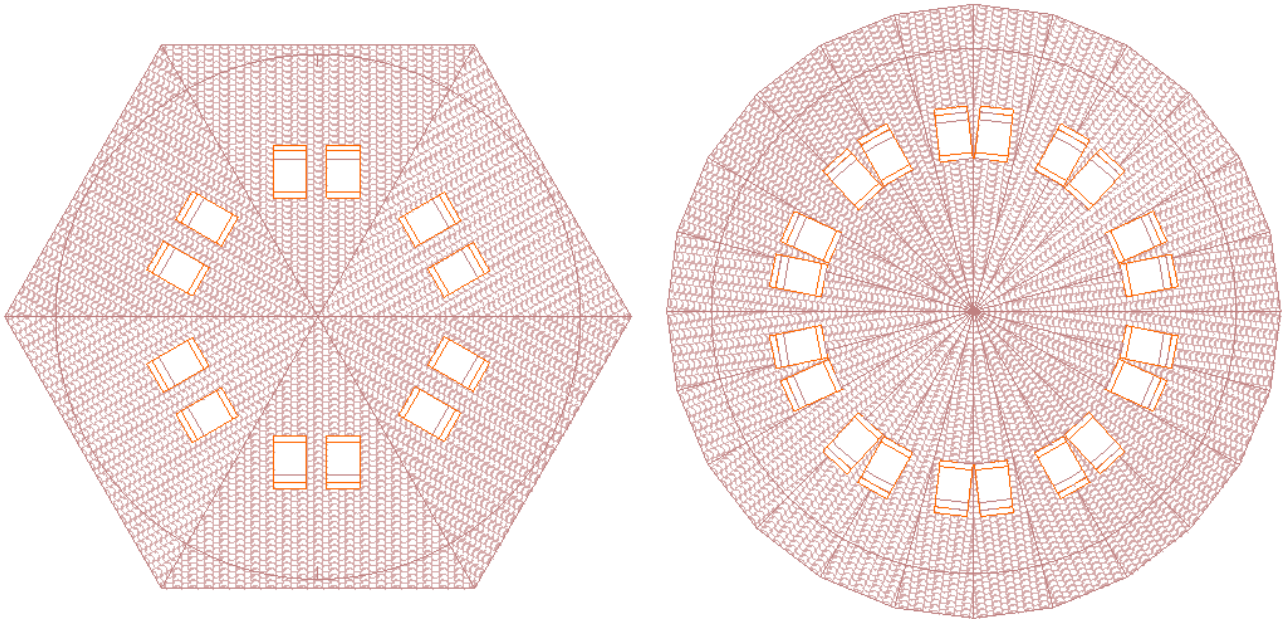
- When you place or move a Skylight in a curved segment of a Roof, the Skylight geometry will be adjusted to reflect a curved surface.



- If you uncheck this option, the Skylight position will reflect that of a Skylight on a flat plane.



In many cases, this option is not relevant. If your curved Roof segment consists of just a few large flat Roof planes, or else many small planes, the Skylight placement will be as expected.



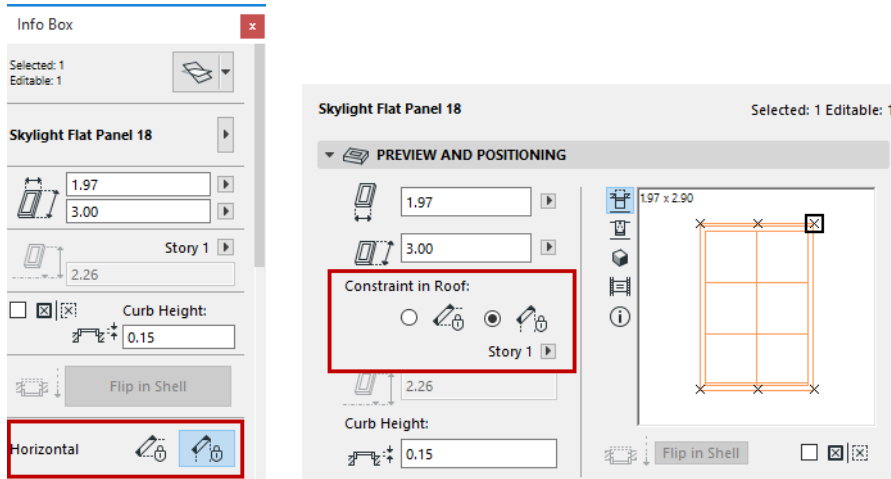
Skylight Constraint Relative to Roof

This feature is applicable to Skylights in Roofs only (not to Skylights in Shells).

If you change the geometry of a Roof that contains Skylight, the Skylight will adjust itself to the new geometry, but you have to decide how to constrain the Skylight position in the modified Roof plane:

- either **Horizontally** (the Skylight will retain its position as seen on the Floor Plan)
- or **Vertically** (the Skylight will retain its vertical elevation regardless of its Floor Plan position.)

Change this **Constraint in Roof** setting in Skylight Settings or the Info Box.



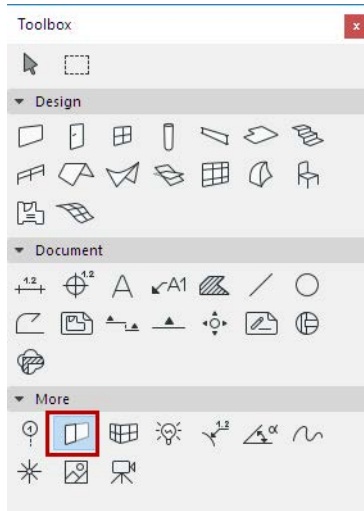
Use the **Horizontal** constraint if you want the Skylight to remain in the same place when viewed from above (that is, on the Floor Plan) even after the host Roof plane geometry changes. This option is useful if your Skylight is the kind that lets in light from the top of the building.

Use the **Vertical** constraint if you want the Skylight to remain at a constant elevation even after the host Roof plane geometry changes. This option is useful if you need a Skylight at a particular vertical position (e.g. as a lookout window in the attic).

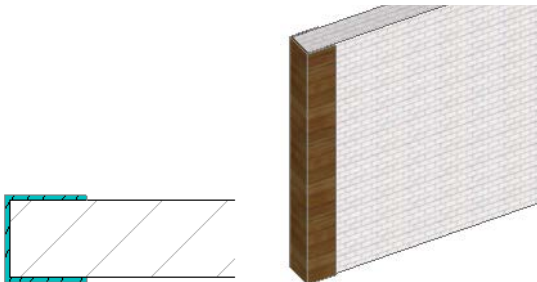
Wall Ends

The Wall End tool allows you to create a correct wall end profile with a single click.

1. Activate the Wall End Tool.



2. From Wall End Settings or the Info Box, choose the desired Wall End object.
3. Click on a hotspot at either end of a wall to place the Wall End.



Wall Ends are parametric GDL objects, similar to window and door objects. Wall End parameters (such as width and height) can be customized using Wall End Settings.

See [Wall End Tool Settings](#).

Wall Ends are counted as objects in list schedules. Depending on its type, a wall end might reduce the wall's surface or volume, but does not add to it. During a **Crop to Roof** operation, Wall Ends are handled as part of the wall, just like windows.

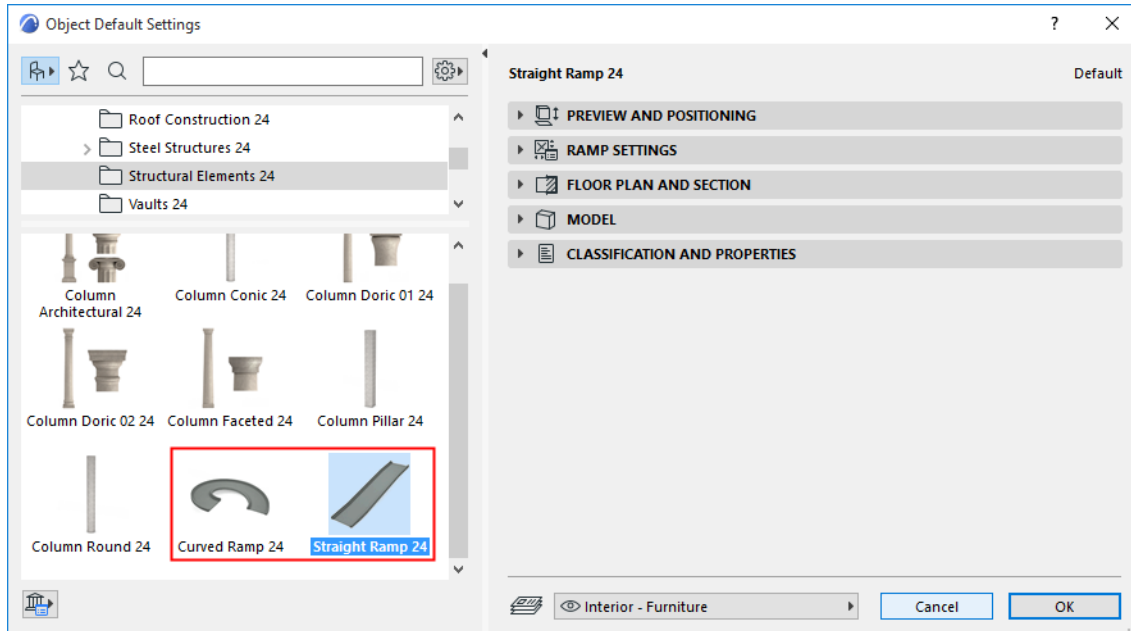
See [Crop Elements to Single-plane Roof](#).

Wall Ends also remain in place if a **Stretch**, **Drag** or **Split** command is executed.

Ramp Object

The Archicad Library includes parametric Ramp objects (Straight and Curved.)

[Watch the Video](#)



Use these to create standard Ramps displayed in the format you need.

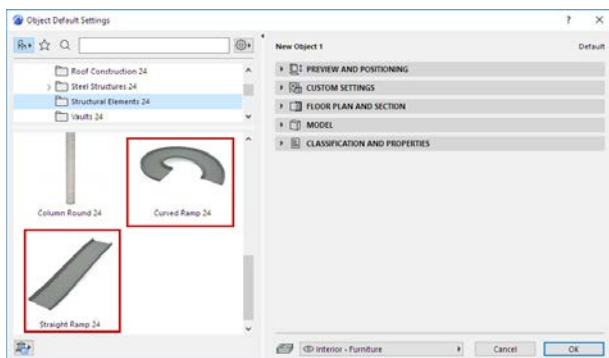
As needed, define separate Transition segments at the top and bottom of the Ramp.

Straight Ramp

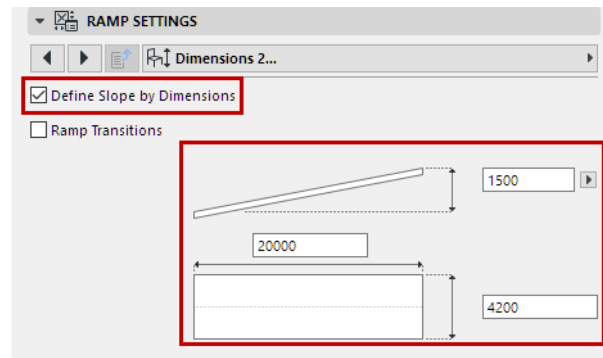
Slope

Use the Ramp Settings panel (Dimensions page) to define the slope of the Ramp.

- Enter the slope using either Gradient, Percentage, or Degree
- Alternatively, check **Define Slope By Dimensions**: then enter Height and Dimension values in the fields below, to automatically calculate the slope.



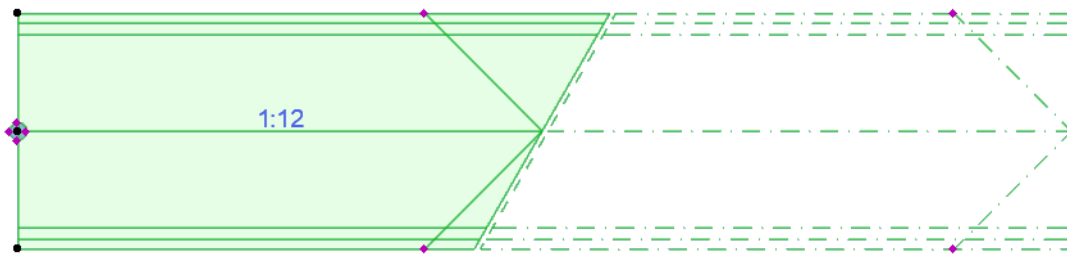
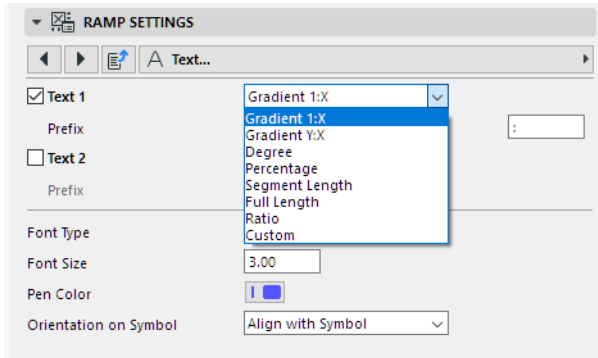
Define Slope Directly



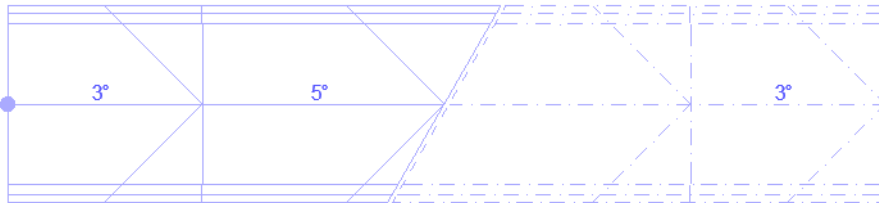
Define Slope by entering Ramp Length/Height

Text

Use the **Text** page of Ramp Settings to optionally display one or two lines of text on the 2D Ramp, in the format you choose. For a slope displayed e.g. as 1:X, you can use any character or text as the separator.



Slope displayed as Gradient

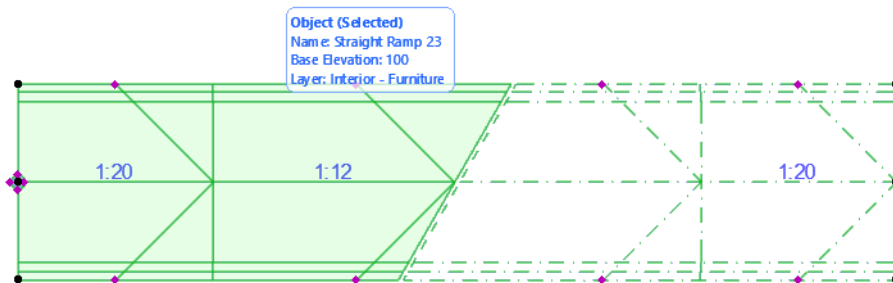
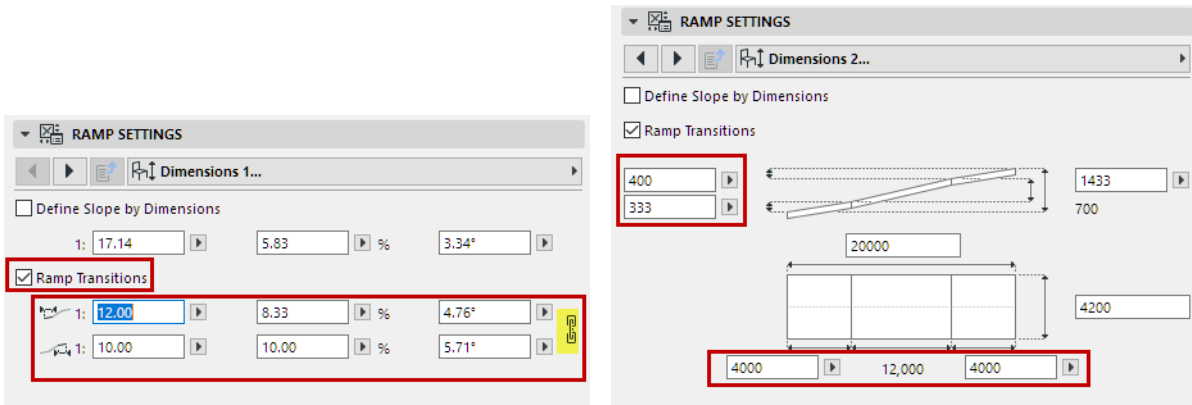


Slope displayed in Degrees

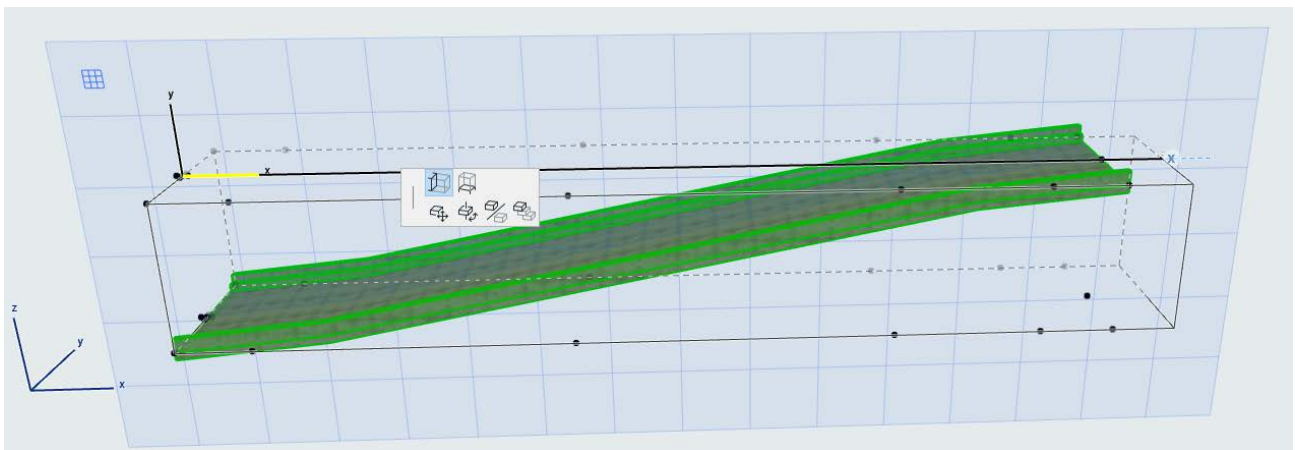
Transitions

On the Ramp Settings panel (Dimensions 1 and 2 pages):

- Check **Ramp Transitions** to create and define transitional segments at the top and bottom of the Ramp.
- For each segment, define its slope (either directly or by Dimensions). (Click the chain icon to use equal values for both top and bottom transition.)
- Additional dimension fields are available on the Dimensions 2 page: enter or calculate length and height values for each transition segment.



Note: Straight Ramps can be stretched graphically, using the Move Node or the Stretch command of the pet palette. When you stretch, the middle segment is modified (not the transitions).

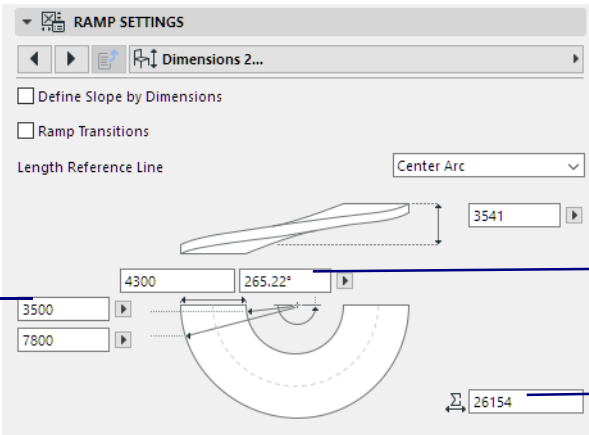


Curved Ramp

For the Curved Ramp object, additional dimension values are available.

- Adjust Ramp width and two radius values (inner and outer)

- The full Ramp length is indicated



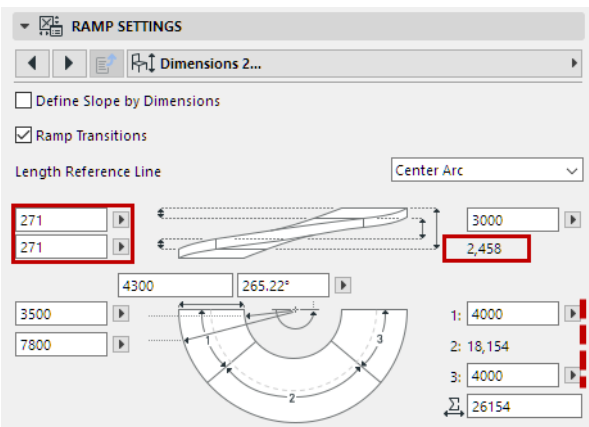
The screenshot shows the 'RAMP SETTINGS' panel with the following values: 'Define Slope by Dimensions' is unchecked; 'Ramp Transitions' is unchecked; 'Length Reference Line' is set to 'Center Arc'; 'Width' is 3541; 'Inner Radius' is 3500; 'Outer Radius' is 7800; 'Angle' is 265.22°; and 'Ramp Total Length' is 26154.

Inner and Outer Radius

Width and Angle

Ramp Total Length

- Check **Ramp Transitions** to create and define transitional segments at the top and bottom of the Ramp

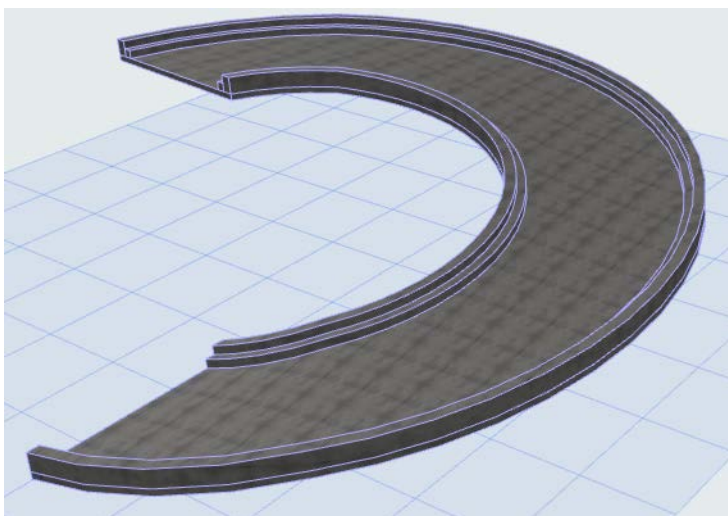


The screenshot shows the 'RAMP SETTINGS' panel with 'Ramp Transitions' checked. The 'Height of Transition Segments' is set to 271. The 'Height of Center Ramp Segment' is 2,458. The 'Ramp Length by Segment' is defined as: 1: 4000, 2: 18,154, 3: 4000. The total length remains 26154.

Height of Transition Segments

Height of Center Ramp Segment

Ramp Length by Segment



TrussMaker

TrussMaker is a built-in tool allowing you to create girder and truss objects for visualization purposes. It is accessed from the **Design > Design Extras** menu. Technically, TrussMaker is an Archicad Add-on and is loaded with Archicad on startup.

Related Topics:

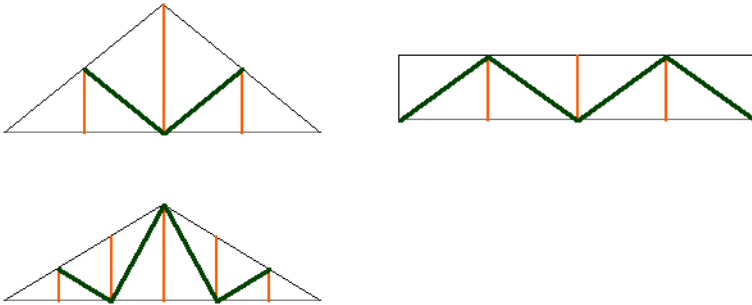
[Creating Trusses in the Floor Plan](#)

[Editing Trusses](#)

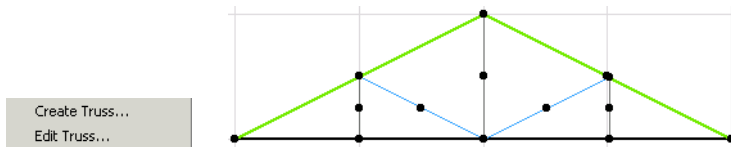
[Creating Trusses in Section/Elevation/IE Windows](#)

Creating Trusses in the Floor Plan

Start by drawing the framework of the truss in the Floor Plan window with the **Line** and **Arc/Circle** tools. Use different pen colors for the different parts of the truss, as seen below. Pen thicknesses have been exaggerated for clarity.



When you are satisfied with your design, select the lines making up the truss and choose the **Create Truss** command from the **TrussMaker** hierarchical menu (**Design > Design Extras**).

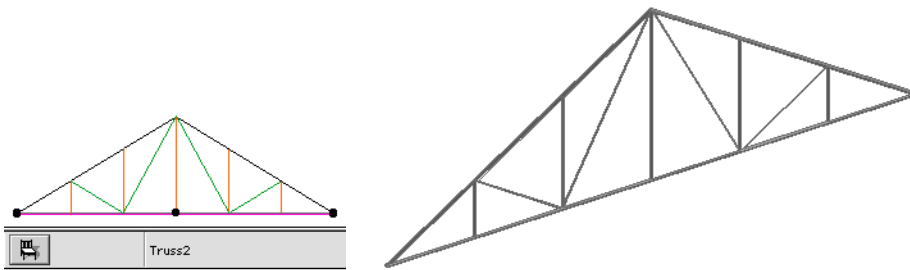


Use TrussMaker Settings to choose a shape and define options for creating the truss.

[See *TrussMaker Settings*.](#)

When you have finished customizing your truss, click the **Save** button at the bottom right of the dialog box. You will then be prompted to name the new truss object and place it in your Embedded library.

The new truss appears on the Floor Plan on the baseline of the contour. Go to the 3D window to visualize it, and then drag it into its final location.



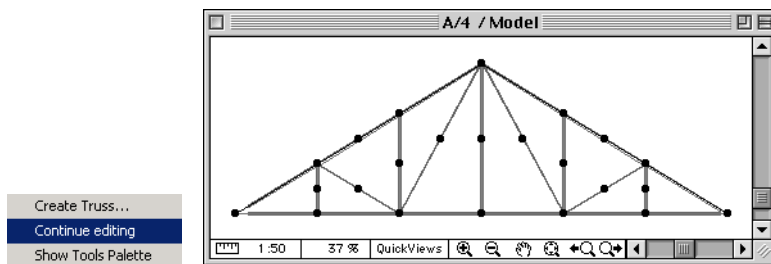
Editing Trusses

The framework drawn on the Floor Plan is not associated to the truss after it has been created and placed (and can only be used to create new trusses). Modifying this outline has no effect on placed trusses.

TrussMaker offers you an additional method for editing placed trusses.

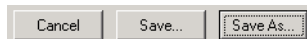
Select a truss and choose the **Edit Truss** command from the **TrussMaker** menu (**Design > Design Extras**) or the **TrussMaker** palette.

An alert will appear, prompting you to create a new Section window. Draw a simple section line with the Section/Elevation/IE tools, open the new Section/Elevation/IE window and then choose **Continue Editing** from the **TrussMaker** menu or palette. The original framework will appear in the window and you will be able to edit it.



When you have finished making your changes, choose the **Create Truss** command again.

The TrussMaker Settings dialog box will appear, with the same options as seen above. The only difference is that in addition to the Save button, a **Save as** button appears.

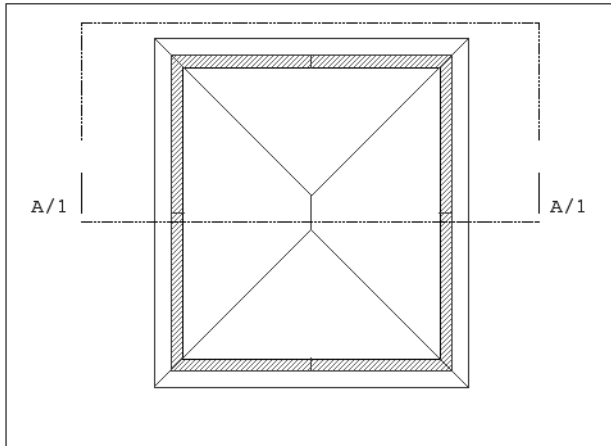


- If you click **Save**, TrussMaker will display a warning telling you that you are about to save a library part under an existing name. Clicking **Overwrite** will replace the previous truss, while clicking **Cancel** will abort the whole process.
- If you click **Save as**, a new truss object will be created. TrussMaker will prompt you to name this object.

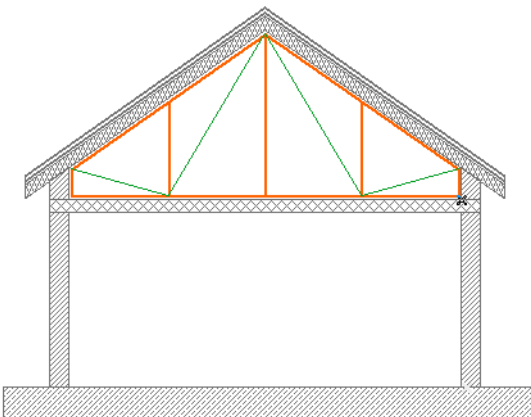
Creating Trusses in Section/Elevation/IE Windows

You can create a truss in a Section/Elevation/IE window:

1. Draw a Roof on the Floor Plan and add a Section Line.



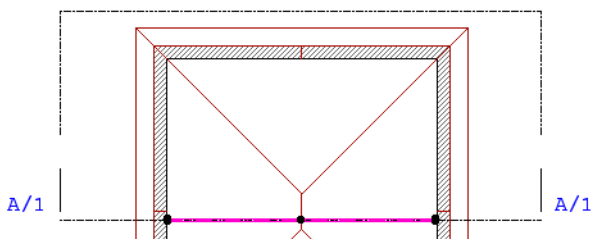
2. Open the corresponding Section/Elevation/IE window.
3. Fit the framework of the truss to the roof. Draw the outline with lines as you would on the Floor Plan.



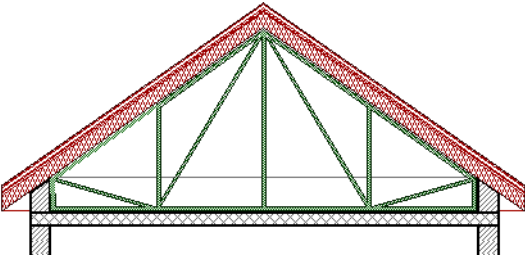
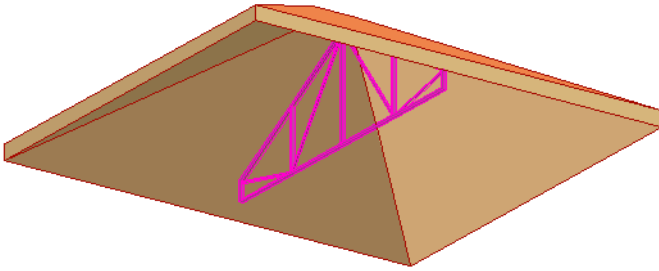
4. Select the framework's elements and choose Create Truss from the TrussMaker hierarchical menu. (**Design > Design Extras > TrussMaker**)

5. Make the appropriate settings in the TrussMaker Settings dialog box and save the object.

An alert will be displayed, informing you that the new truss object has been created on the Floor Plan. The truss is placed on the same line as the section line and will therefore appear in the Section/Elevation/IE window as an element that was cut on its edge.



6. Display the new truss with the roof in 3D or in another Section window.



Solid Element Operations

Solid Element Operations allow you to create complex shapes using Boolean operations, including subtraction, addition and intersection. The result is associative: if you modify the location or the attributes of the elements that participate in the operation, the linked elements will be updated accordingly.

Related Topics:

[About Solid Element Operations](#)

[Solid Operation Types](#)

[Solid Operations: Element Display and Calculation](#)

[Solid Operation Example](#)

[Managing Solid Element Connections](#)

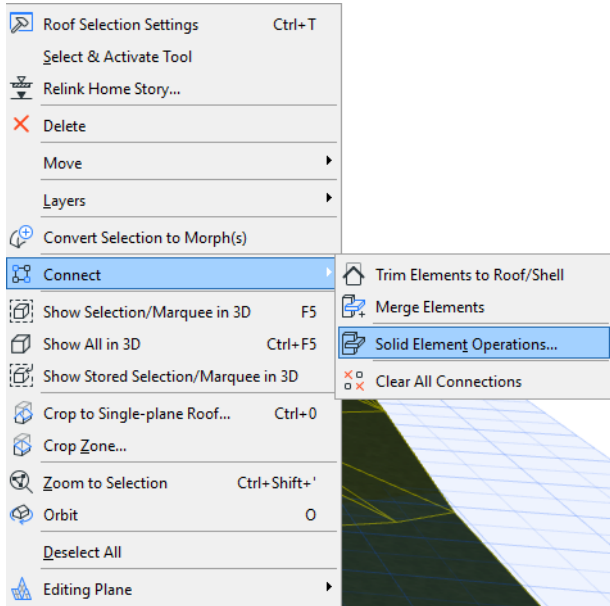
[Solid Element Operations Palette](#)

See also [Boolean Operations with Morphs](#).

About Solid Element Operations

Solid Operations can be performed on elements in any model window by choosing one of the following:

- **Design > Solid Element Operations**
- **Connect > Solid Element Operations** from an eligible selected element's context menu.



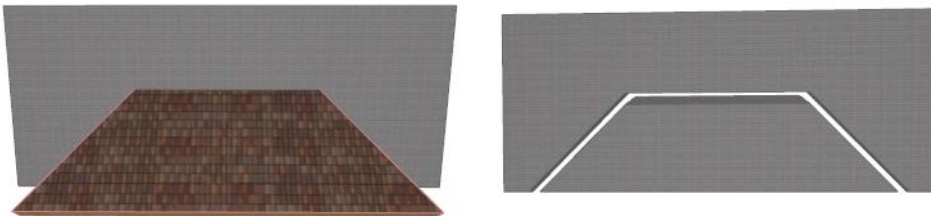
For details on these controls, see [Solid Element Operations Palette](#).

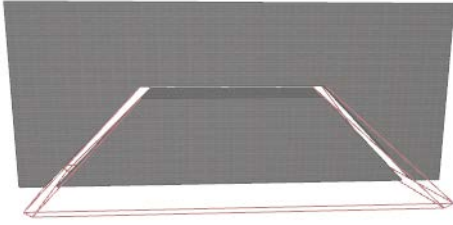
Solid Element Operations distinguish two roles:

- A **Target** is an element whose geometry will be modified by the solid operation.
- An **Operator** is an element that affects the geometry of the element it is linked to.

The operation does not modify or delete the Operator element. You can hide the layers of the Operator elements after the operation has been performed without resetting the modified shape. You can also set the Operator's layer to be shown as Wireframe, which allows you to still see the Operator element's contour and adjust it as necessary.

In the illustration below, a Target Wall's shape was modified by subtracting a Roof type Operator shape. On the left, both elements are shown; in the middle, the Roof's layer has been hidden; on the right, the Roof's layer has been set to Wireframe.





An Operator can work on multiple Targets, and a Target can be affected by multiple Operators. Each element can be a Target and an Operator at the same time, which permits nested Solid Element Operations. The operation remains in effect as long as the involved elements are present. If you wish to keep the obtained shape permanently, save it as a GDL Object from the 3D Window.

[See *Save 3D Elements as Objects*.](#)

Restrictions and Remarks

Bypass automatic intersections: Solid Operations do not affect the results of automatic intersections, if the elements are included in the same Layer intersection group. Before performing a Solid Operation on such elements, change the Layer intersection number of one of them.

[See *Use layers to prevent intersections*.](#)

Beams: If you intend to apply an operation between Beams, the Operator must have at least the same 3D Intersection Priority number as the Target. If the Target's priority number is higher, the operation will appear to have no effect.

[See *Beams and Other Elements*.](#)

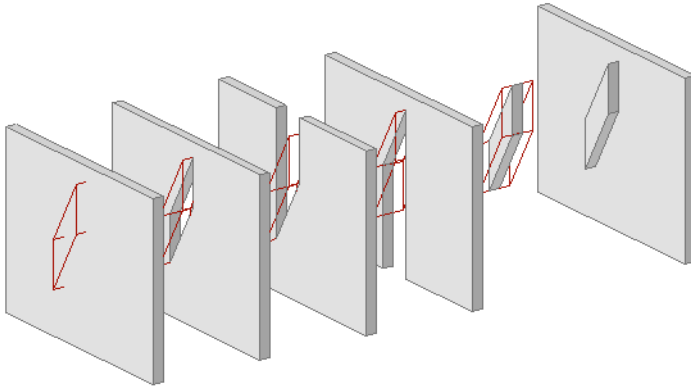
Nested operations: If the desired shape can be only be obtained through a number of nested operations, pay attention to the order in which you apply them. For example, an Operator element may affect a Target that is also an Operator affecting another Target, and so on.

Morphs: There are separate Boolean operation commands for Morphs. Unlike Solid Element Operations, which create connections, Boolean operations on Morphs are not associative - the final result of the operation is permanent.

[See *Boolean Operations with Morphs*.](#)

Solid Operation Types

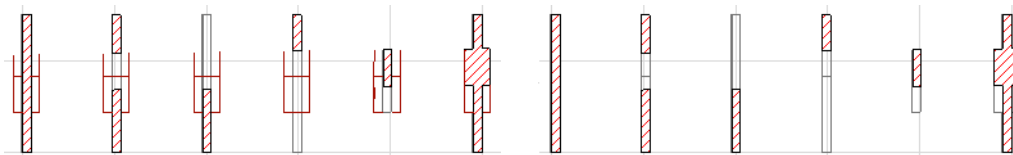
Five types of operations can be performed. The illustration below shows the effect of each: the starting Wall shape is shown at far left, followed by each of the five operations. The Operator element is shown in Wireframe mode.



- **Subtraction** cuts out the shape of the Operator from Target.
- **Subtraction with upward extrusion** cuts out both the shape of the Operator from the Target and its vertical projection from the bottom of the Operator shape to the top of the Target.
- **Subtraction with downward extrusion** cuts out both the shape of the Operator from the Target and its vertical projection from the top of the Operator shape to the bottom of the Target.
- **Intersection** keeps only the common part of the Target and the Operator.
- **Addition** adds the shape of the Target to the shape of the Operator.

Solid Operations: Element Display and Calculation

The solid operation affects the model shape, that is, the 3D view and the generated Section/Elevation/IE model and Detail Drawing view. Below is the Section view of the elements shown above. The Operators are shown in Wireframe, above, and their layers are hidden, below.



The Floor Plan view of both Target and Operator elements is unaffected. Putting the Operator element on a Wireframe layer does not affect its display. If the Operators' layer is hidden, then they will not appear in the Floor Plan, but you will not be able to edit them until you make the layer visible again.

Note: If you select only the Target on the Floor Plan and display the selection in 3D view, the modified shape will be shown, but without displaying the Operator.

If you perform a Solid Operation on a **Section/Elevation/IE** view, you need to rebuild the window's contents to see the results.

Solid Element Operations in Interactive Schedules

Element lists display elements according to the parameters you specify in each Schedule's Scheme Settings. Many elements include separate listing parameters for "Net" and "Gross" values: for example, the Net volume

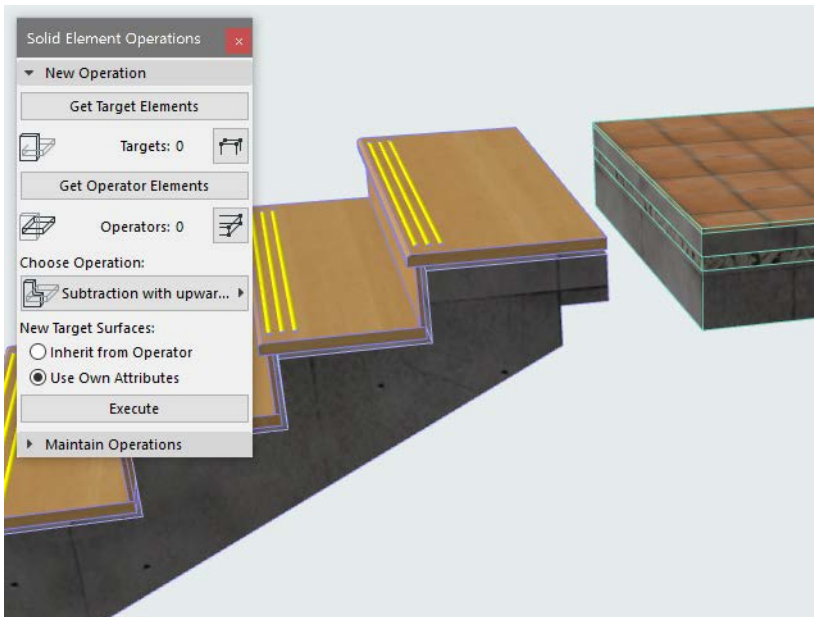
of a column vs. the Gross volume of a column. As a rule, “Net” and “Gross” in these element listing parameters means:

- **Net:** takes any Solid Element Operations into account.
- **Gross:** does not take any Solid Element Operations into account.

See [Element Listing Parameters](#) for more information.

Solid Operation Example

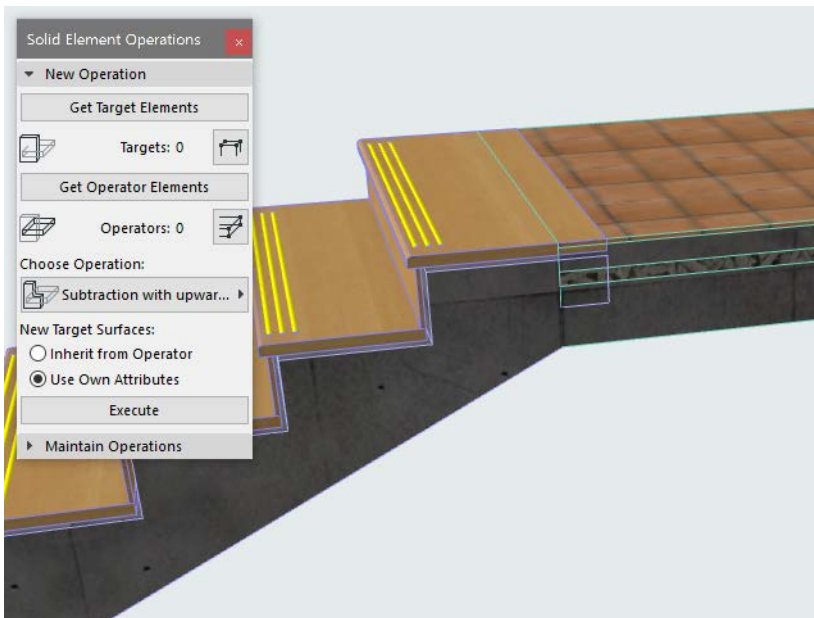
In this example, we will connect a Stair with a Slab.



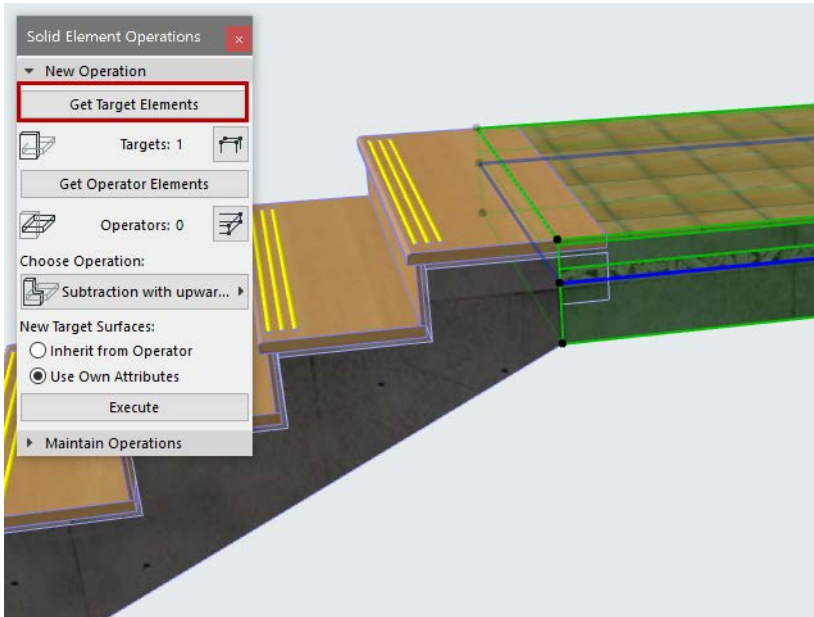
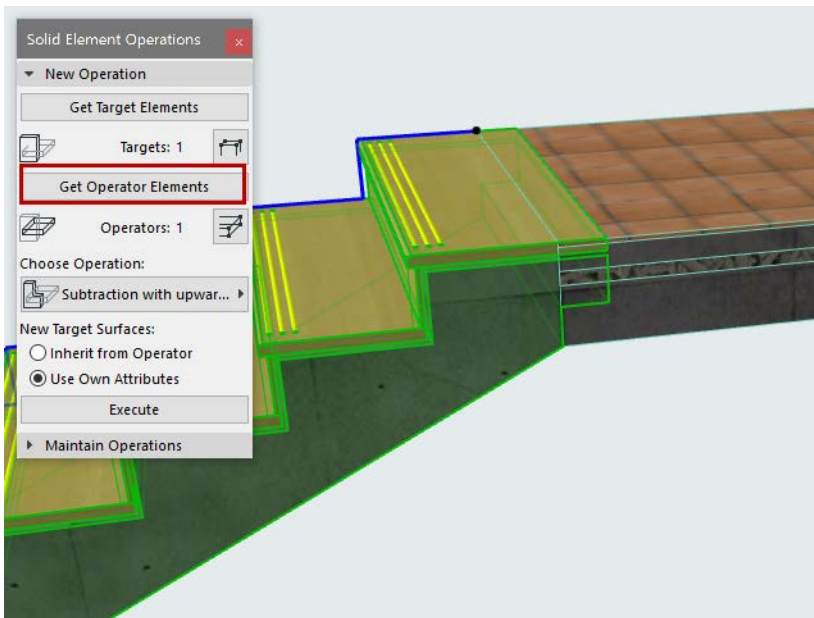
1. Open the dialog box using the **Design > Solid Element Operations** command (also available from the context menu of either selected element).

For details on these controls, see [Solid Element Operations Palette](#).

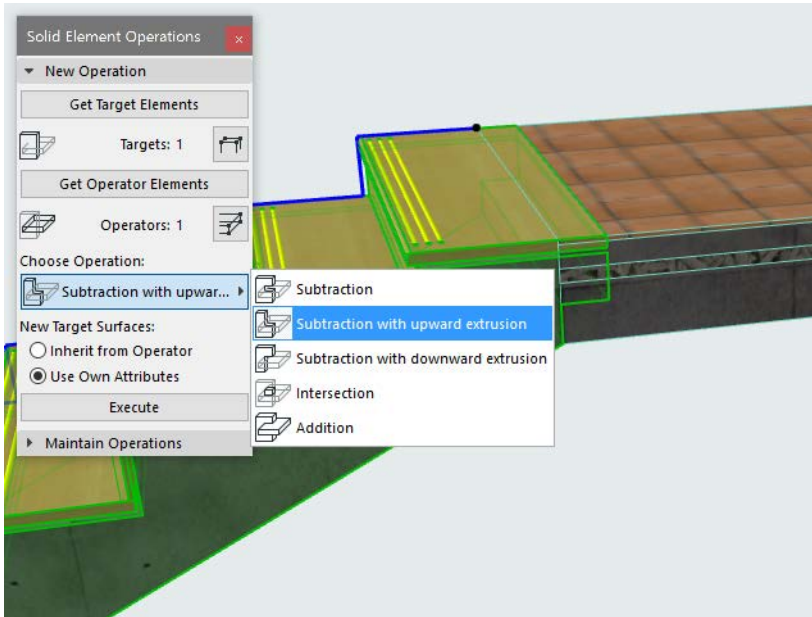
2. Collide the two elements.



3. Select target element: the Slab.

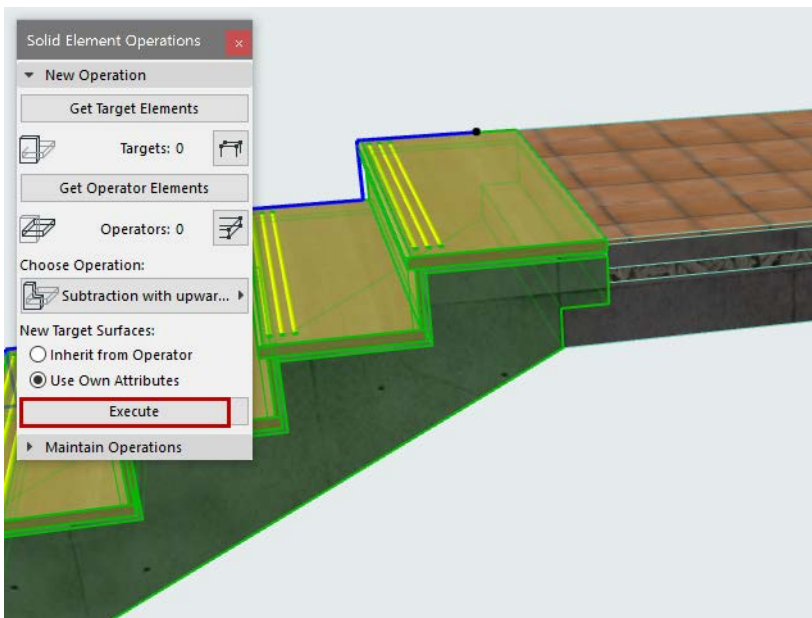
4. Click Get Target Elements.**5. Deselect the Slab.****6. Select the operator element: the Stair.****7. Click Get Operator Elements.**

8. Choose an operation type: Subtraction with upwards extrusion.

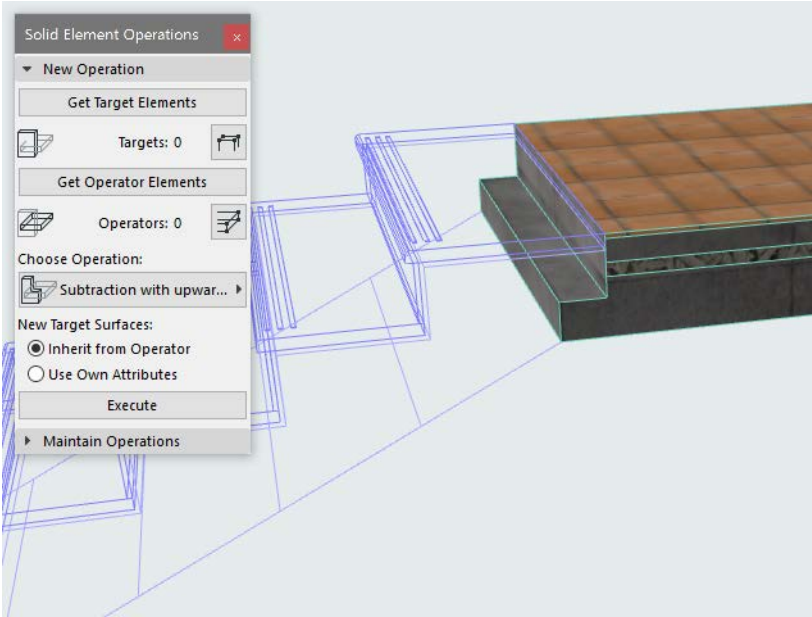


9. Click **Execute**. The Solid Element Operation is complete.

10. View the result in 3D.

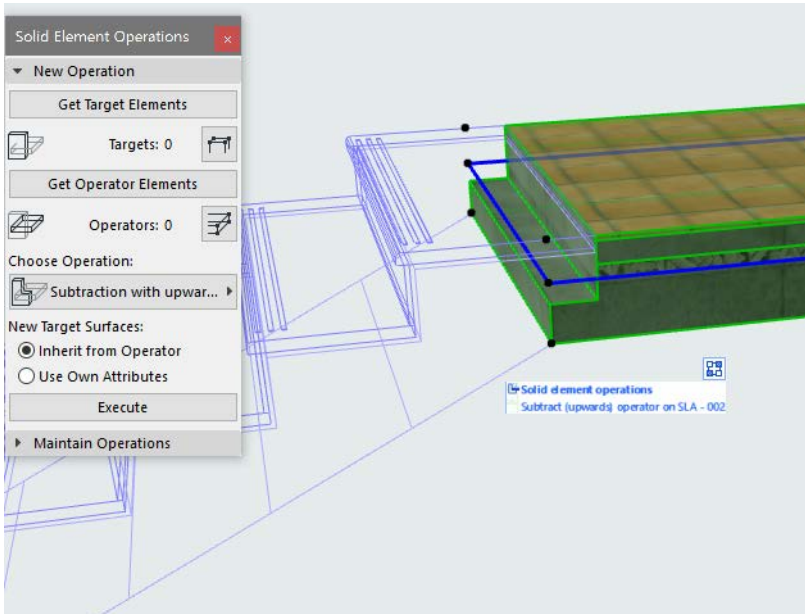


With the Stair shown in wireframe:



Managing Solid Element Connections

On any selected element that is part of a Solid Element Operation, a “connection grip” appears, listing the elements to which it is connected.



You can highlight each connected element and clear any connection as needed.

For details, see [Managing Element Connections](#).

Properties and Classification Systems

What are Properties?

Properties are optional, user-defined data assigned to an element or Building Material to provide additional, searchable information.

Examples of properties:

- General Ratings (e.g. Fire Resistance Rating, Thermal Transmittance)
- Openings (e.g. fire exit, security rating)
- Product Information (e.g. model, serial number, price)

Use the Property Manager (**Options > Property Manager**) to create and delete properties and customize their availability. Property availability depends on the Classification of each element/material.

What you can do with Properties

- List and manage them in the Interactive Schedule
- Display them in Labels, Zone Stamps, Door/Window Markers
- Display elements by property, using Graphic Overrides
- Import and export them among applications
- Export data structure to spreadsheet, then import property data from spreadsheets into Archicad elements or materials

What is Classification?

Classification is used:

- To organize project elements/materials and their related data
- To define which **properties** are available to an element or Building Material (available properties depend on its Classification)
- To provide mandatory deliverables based on international Classification Standards
- To ensure accurate data exchange with other applications, e.g. using the IFC standard
- To **Find & Select** elements or materials by Classification: for example, search for all elements classified as Ramps within the UniClass classification system
- To apply Graphic Override Rules based on Classification

Use the Classification Manager (**Options > Classification Manager**) to manage Classifications and Classifications Systems, and to customize their available properties.

Archicad comes with a default Classification System, but depending on your working methods and local standards, you may want to use different or additional Classification Systems.

The following sections describe Archicad functions related to working with Properties and Classifications:

[Assign Classification](#)

[Assign Classifications/Properties in Element Schedule](#)

[Show Property Data in Labels and Markers](#)

[Default Property Value](#)

[Expression-Defined Element Properties](#)

[Expression-Defined Properties: Examples](#)

[Properties in Hotlinks](#)

Property Manager

Classification Manager

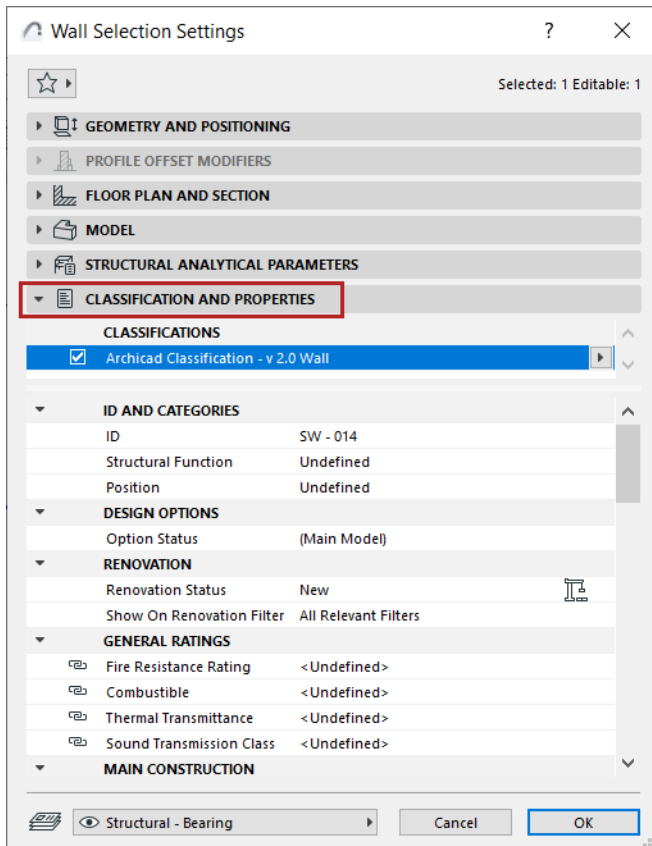
Import/Export Property Data via Spreadsheet

Import BIM Content

Assign Classification

Classifying an element or Building Material determines which properties are available to it.

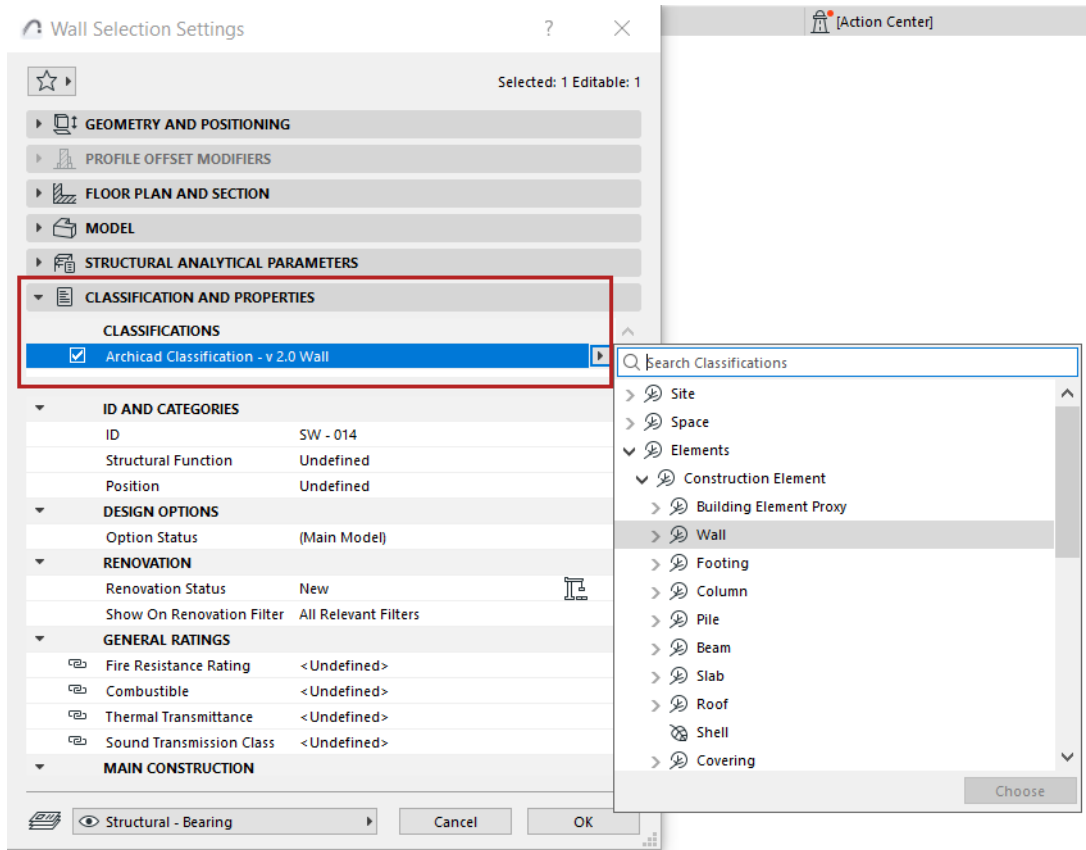
To assign Classification(s), open the **Classification and Properties** panel (Element Settings or Building Materials dialog box).



Select **Classifications** from Classification System(s) at the top of the list:

- If your project contains multiple Classification Systems, they are all listed. Check the name of the System(s) you will use. An element/material can be classified in several classification systems simultaneously.

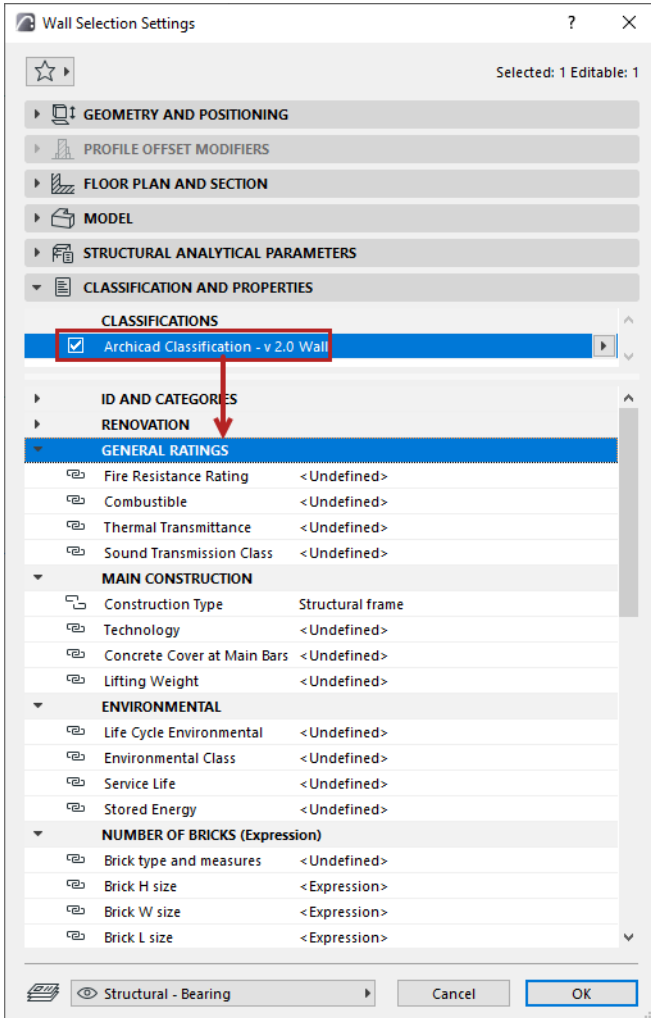
- To change or assign the item’s classification within any of the Systems, click the pop-up arrow at the right:



Use the search field if needed to locate the desired classification(s).

Note: A “branch” classification and a “leaf” classification are each a single classification (the “branch” is a more general grouping than the “leaves” it contains.)

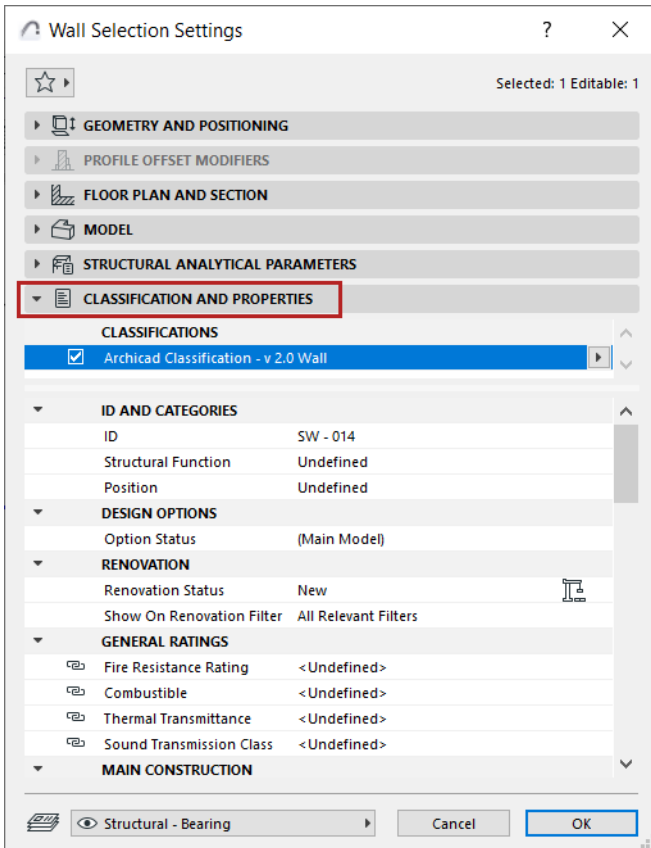
- Available properties (listed below) depend on the classification(s):



Available Properties Depend on Classification

Assign Custom Property Value

To assign custom property values to a selected Archicad element or material, open the **Classification and Properties** panel (Element Settings or Building Materials dialog box).

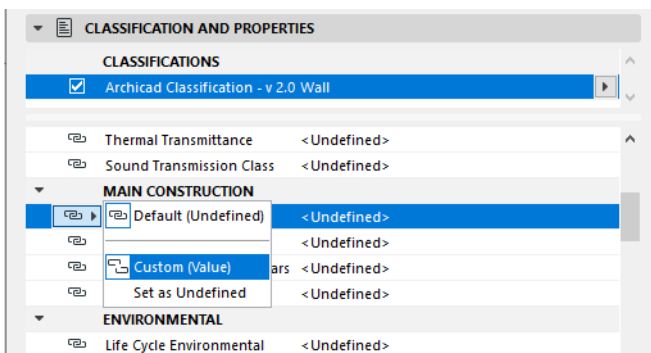


All available properties are listed here. Available properties depend on the Classification(s).

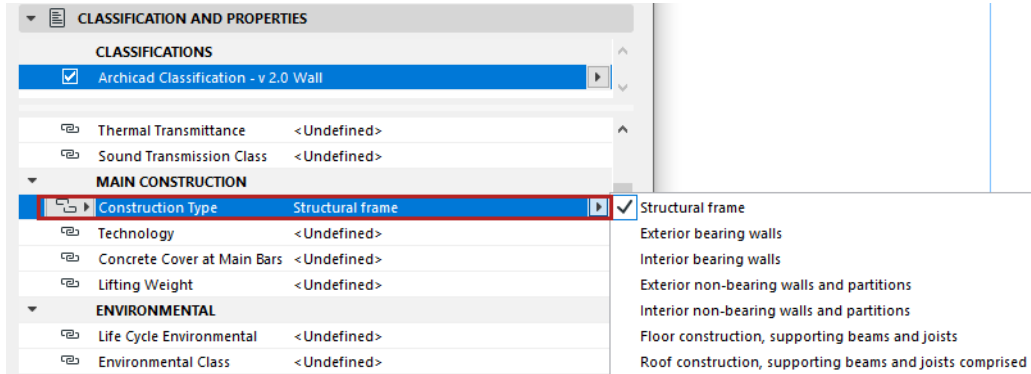
For each property, the chain icon indicates that its value is linked to the default for this property. (Default value of each property is defined in Property Manager.)

Click on any property in the list to assign a custom property value for this item.

If the default value is <Undefined>: click the chain pop-up. Then choose Custom Value to unlink the default value, and enter a custom property value instead.



Depending on the property's data format, you can choose from a list, enter text, or simply type in a value.



The custom value can also be set as “Undefined” for this item.

- Click the chain pop-up.
- Choose **Set Custom as Undefined**.

This undefined value is not linked to the default, and will not be updated in case you change the default value.

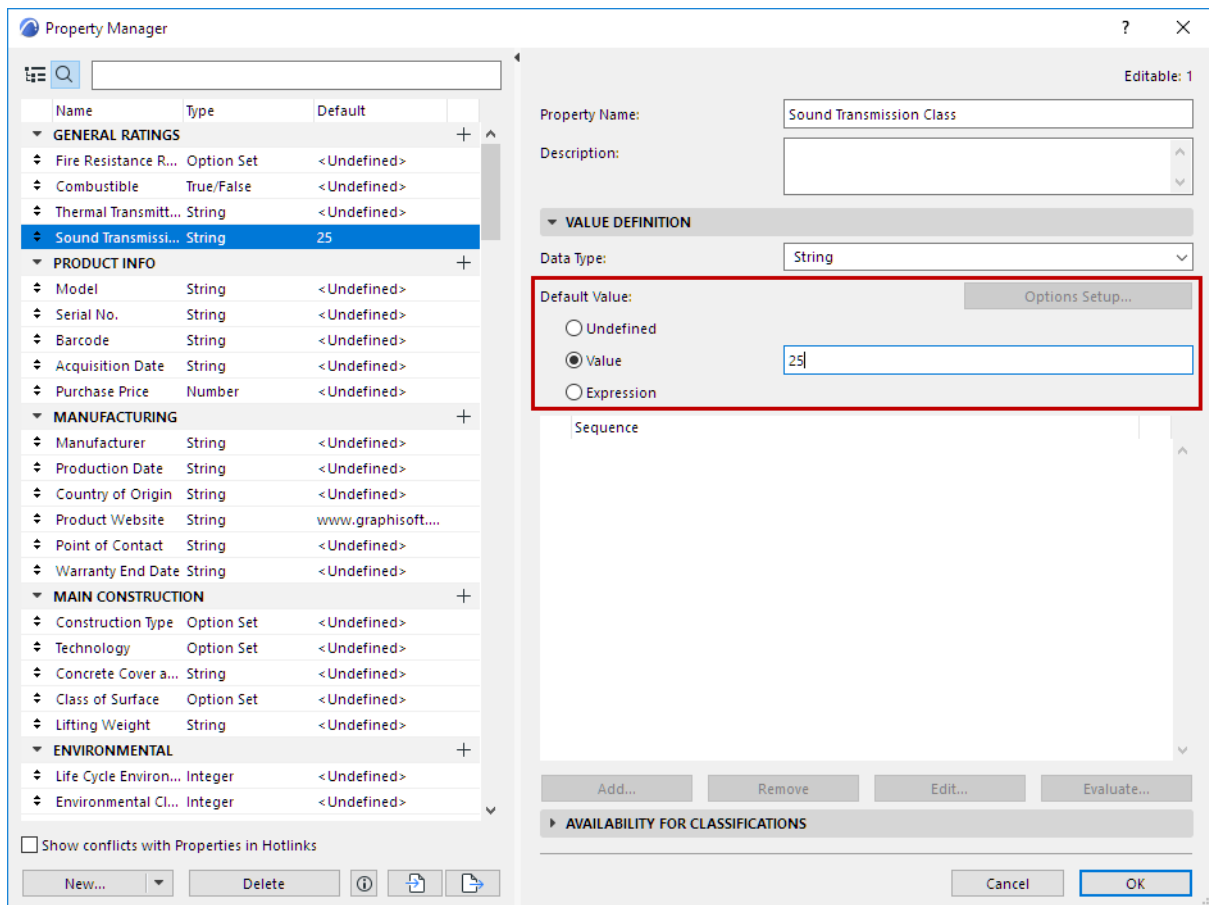
Default Property Value

The default value of a Property is defined in Property Manager, in the Value Definition panel.

The default can be any text or value, or even an expression: a function consisting of operations and variables.

Regardless of the default, you can assign a custom value to any Property for any element or material.

If you edit a property's default value in Property Manager, all linked property values are updated accordingly at the element/material level.



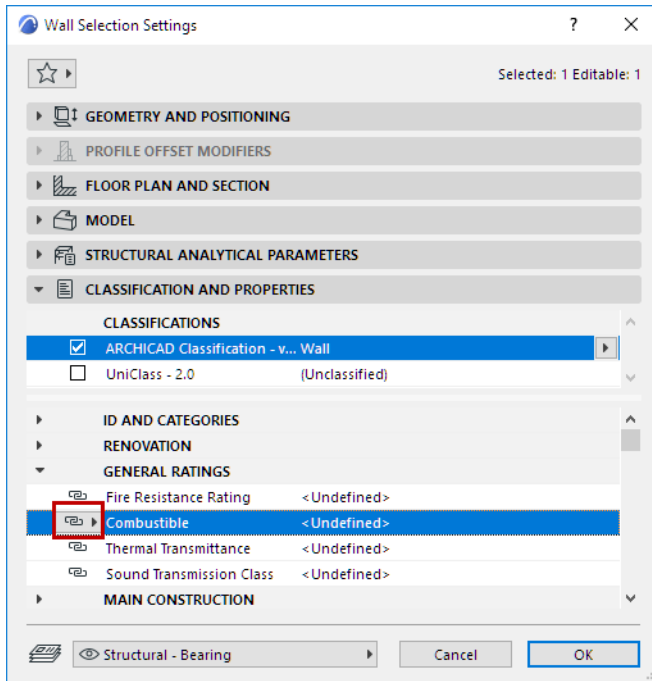
Under the Default Value control, choose an option that defines the Property's Default Value.

- **Undefined:** The Property value is <Undefined> by default. It is up to the user or external consultant to set its custom value for each item.
- **Value:** This is a user-defined, static value that remains fixed until you change it. All items using this Property are assigned this value.
- **Expression:** This Property value is calculated uniquely for each item. An expression-defined value is a function of other data (e.g. element parameters), or a function of other properties (e.g. mapped properties). For example, the value is based on its model parameters (e.g. element Height), or derived from the value of another Property.

See [Expression-Defined Element Properties](#).

Link Property to Default Value

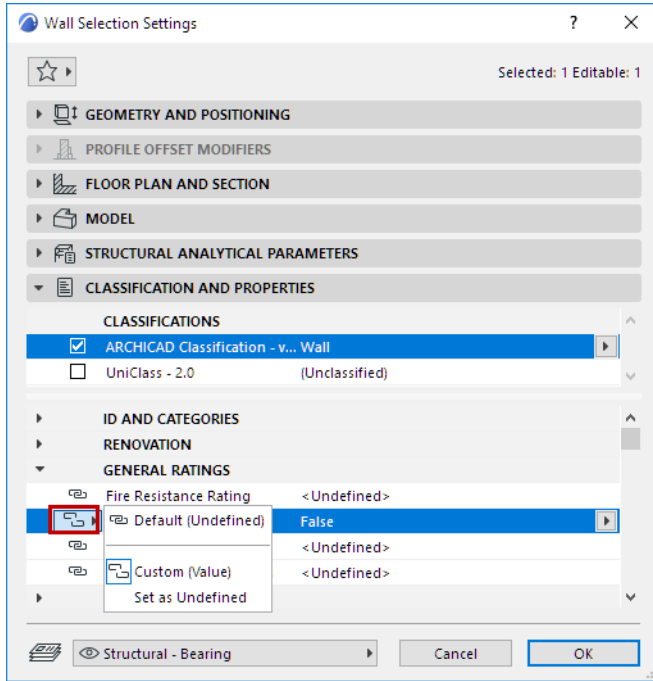
For each property listed in an item's Classification/Properties panel, the chain icon indicates that its value is linked to the default for this property.



Property Linked to Default

Note: Defaults are defined in Property Manager. If you edit a property's default value in Property Manager, all linked property values are updated accordingly at the element/material level.

If you overwrite a value in Element Settings or the Building Material dialog, its link to the default is broken.

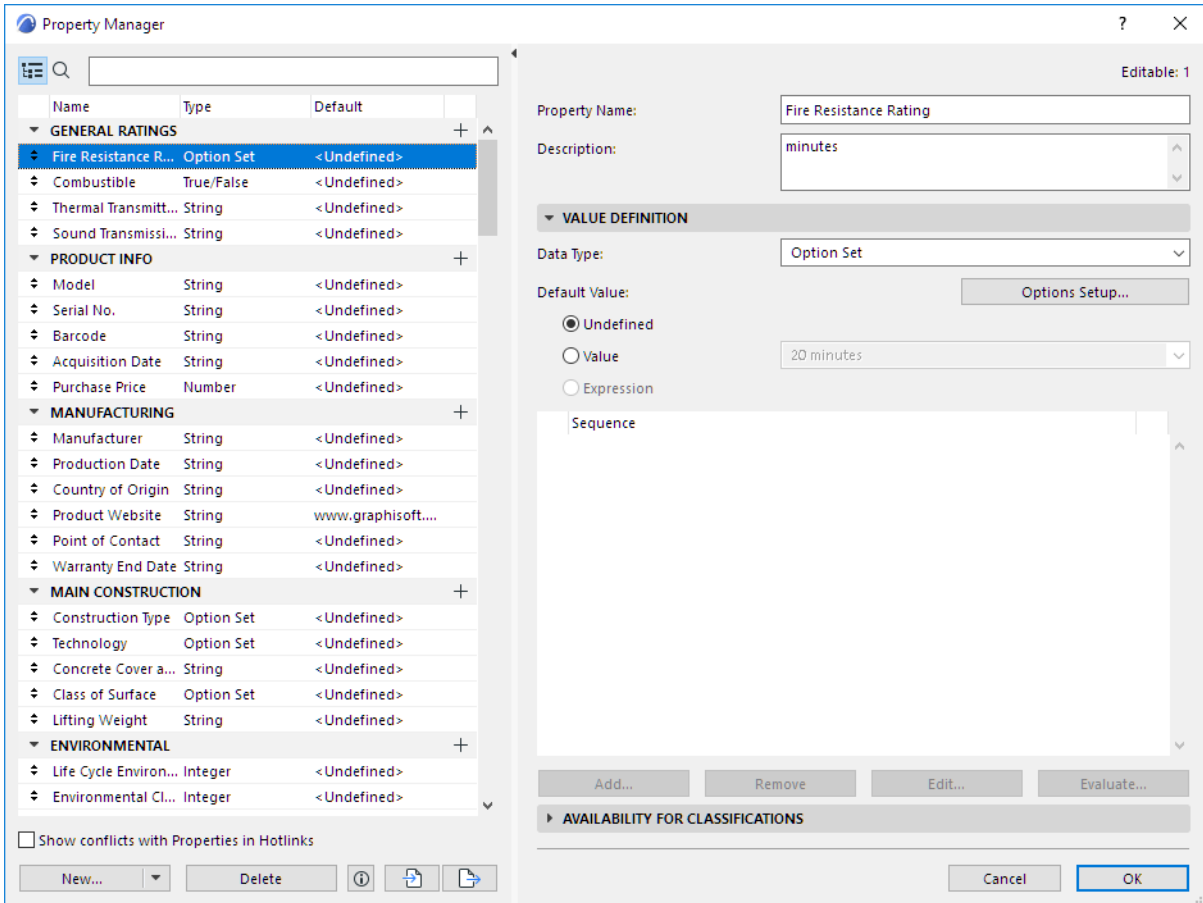


Property Unlinked from Default

To re-link and restore the default value, click the chain icon and choose Default value.

Property Manager

Use the Property Manager (**Options > Property Manager**) to create and manage the set of properties for this project, define Property values, and customize Property availability for the various classifications.

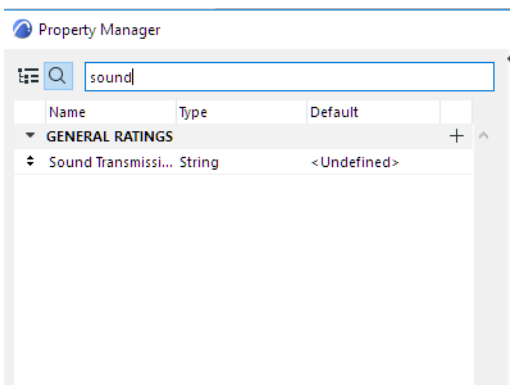


Property Manager can also be opened from the Classification Manager dialog box.

Search Property List

The left side of the Property Manager lists all the properties available in the project.

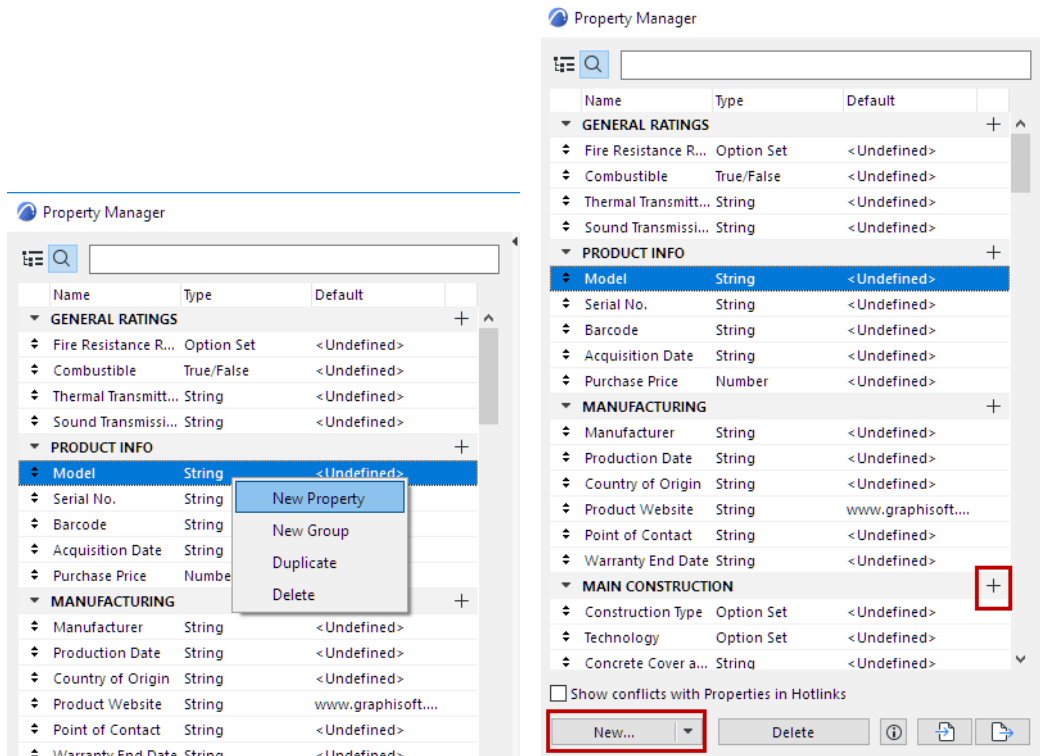
Click the **Search** icon, then start typing text into the search field. All property and group names containing the search term are shown.



To view the full list of properties again, delete the text from the Search field, or click the tree hierarchy icon.

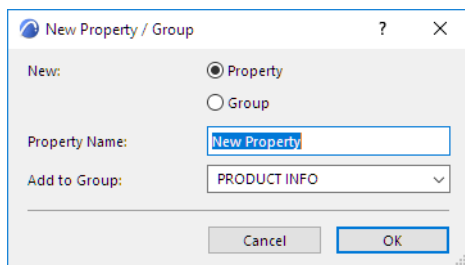
Create New Property or Group

1. Open the Property Manager dialog box at **Options > Property Manager**, or from Classification Manager.
2. Do one of the following:
 - Click anywhere in the Property list and use the context menu
 - Click the **New** button at the bottom of the dialog box
 - Click the **Plus** sign to the right of any Group name



3. In the **New Property/Group** dialog box: Choose a radio button for either a new **Property** or **Group**, then enter its name.

The Property Name must be unique within its Group.

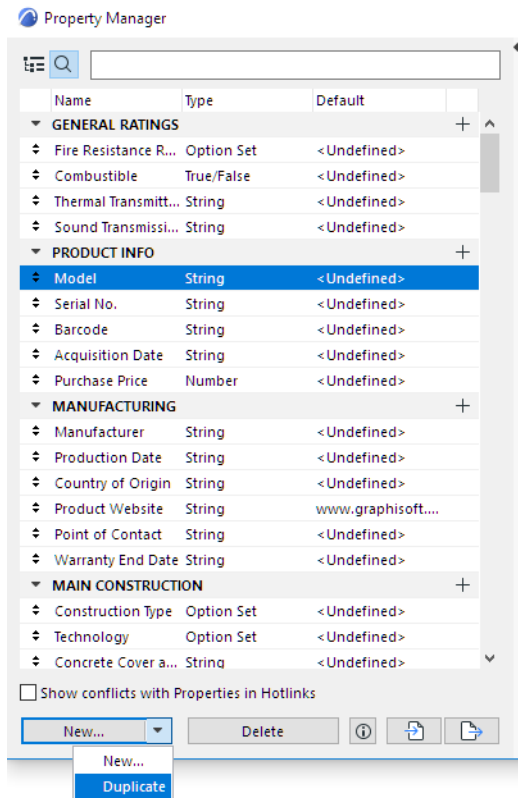


4. **Add to Group:** For a new property, choose the group into which it should be placed.
5. Click **OK**.

Duplicate Property

Select a Property.

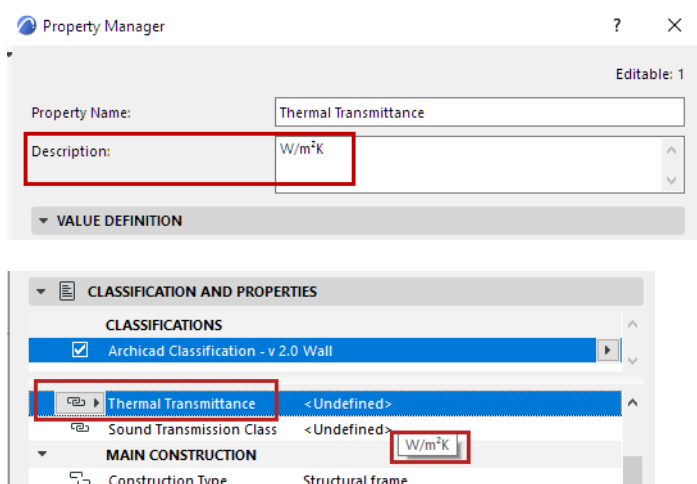
From the context menu or drop-down list under the New button, click **Duplicate** to add a duplicate of that Property to the list.



Property Description

Optionally, enter any further information here to guide the user in assigning a value to this property. For example, its format or measurement unit: “minutes” for the Fire Resistance Rating Property, or “W/m²K” for Thermal Transmittance.

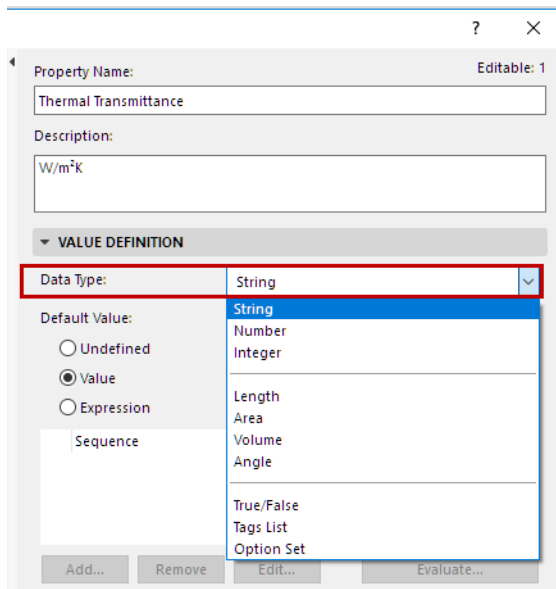
In the item’s Classification and Properties panel, this description is visible when hovering over the property name:



Data Type

The property's Data Type determines the format of the value(s) that can be assigned to it.

1. Select a property from the left side of the Property Manager (e.g. Thermal Transmittance).
2. Choose a **Data Type** from the right side of Property Manager:



A property's Data Type is relevant for the syntax of Expressions, when you define an Expression-based Property.

- **String:** Can use any text or non-unit number (e.g. for the “Manufacturer” property)
- **Number:** Can use any non-unit number expressed in decimals (e.g. for the “Purchase Price” property). The displayed number of decimals depends on the **Numbers without Units** setting in **Options > Working Units**.
- **Integer:** A whole number (e.g. for the “Sound Transmission Class” property)

Data with Unit Values

The next four data types are Unit values:

- **Area**
- **Length**
- **Angle**
- **Volume**
- **Tags List:** Can be a tag or series of tags (e.g. Building A, Floor 2, Common Use, Kitchenette)
 - This data type cannot be used for an Expression-based property.
- **Option Set:** Provides a fixed set of options (e.g. for the “Fire Resistance Rating” property)
 - This data type cannot be used for an Expression-based property. However, Option Set-type properties (without Multi-Choice) can be used within an Expression.

For details, see [Options Setup below](#).

- **True/False** (e.g. for the “Combustible” property)

For information on setting Default Value and Expression-defined Values, see:

Default Property Value

Expression-Defined Element Properties

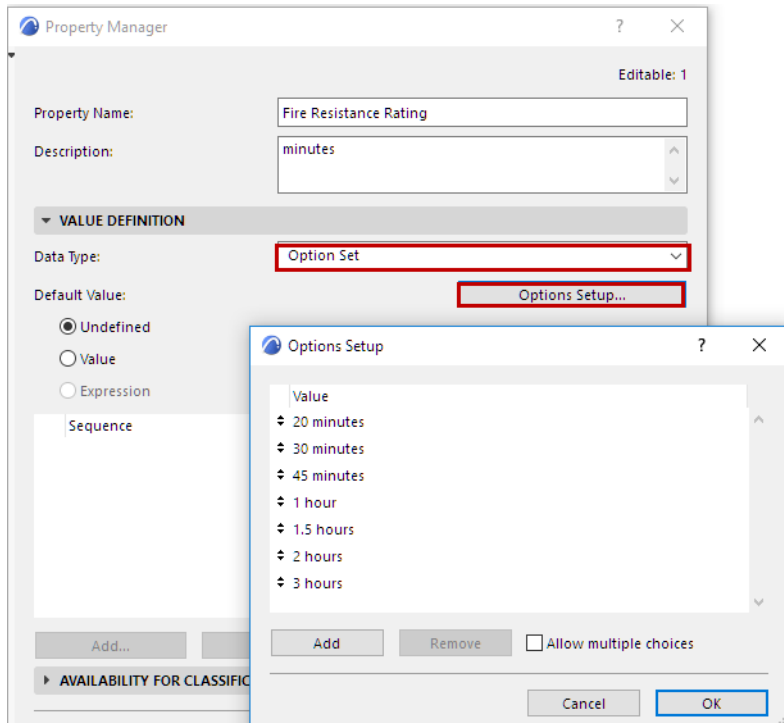
Options Setup

This button is active only if the Data Type is “Option Set”.

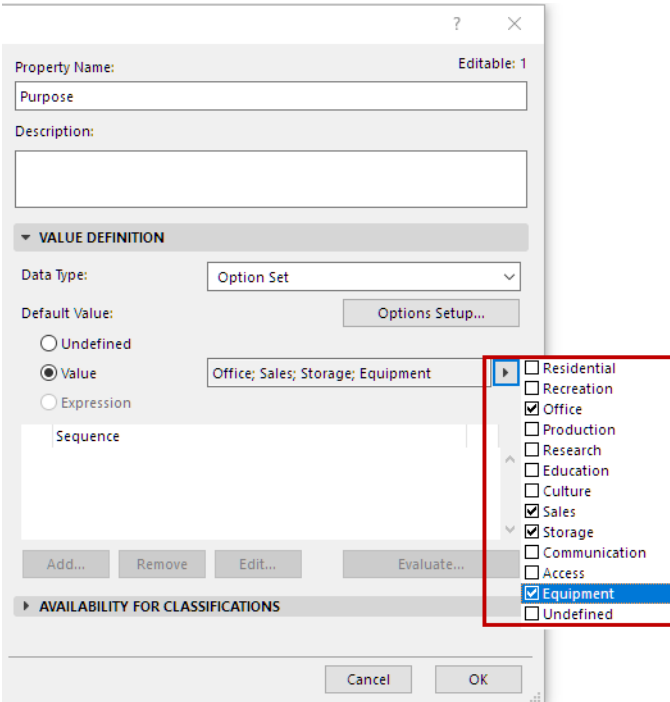
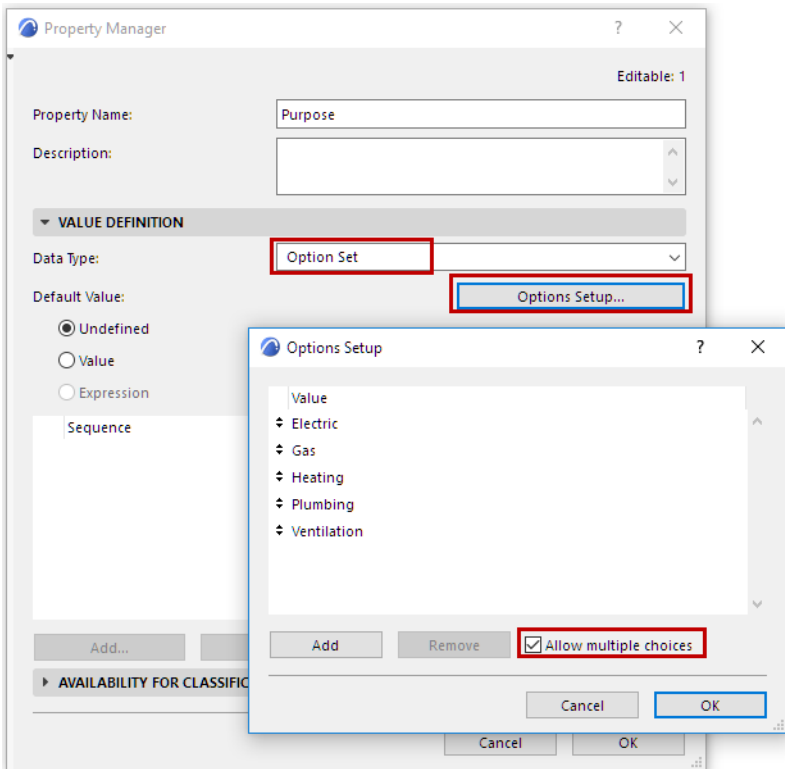
- Not available for Expression-defined Properties. However, Option Set-type properties (without Multi-Choice) can be used within an Expression

Click to open the **Options Setup** dialog box.

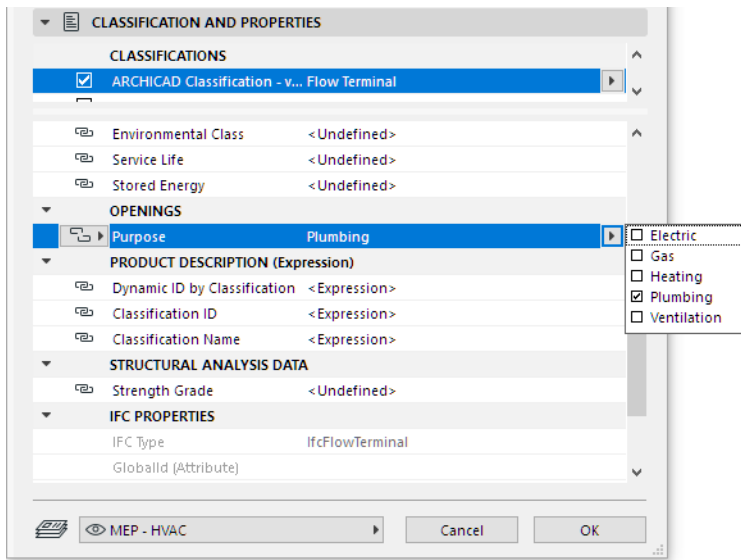
Use **Options Setup** to list a set of specific values that the user can choose from when defining this property for a particular element or material.



- Click **Add** to add a new option (type a value into the **New Option** line).
- To delete an option, select it and click **Remove**.
- Check **Allow multiple choices** to enable the user to choose more than one value for this property (in Property Manager, the Settings dialog, or the Info Box).



Default Value: Multi-Choice Option Set, Defined in Property Manager

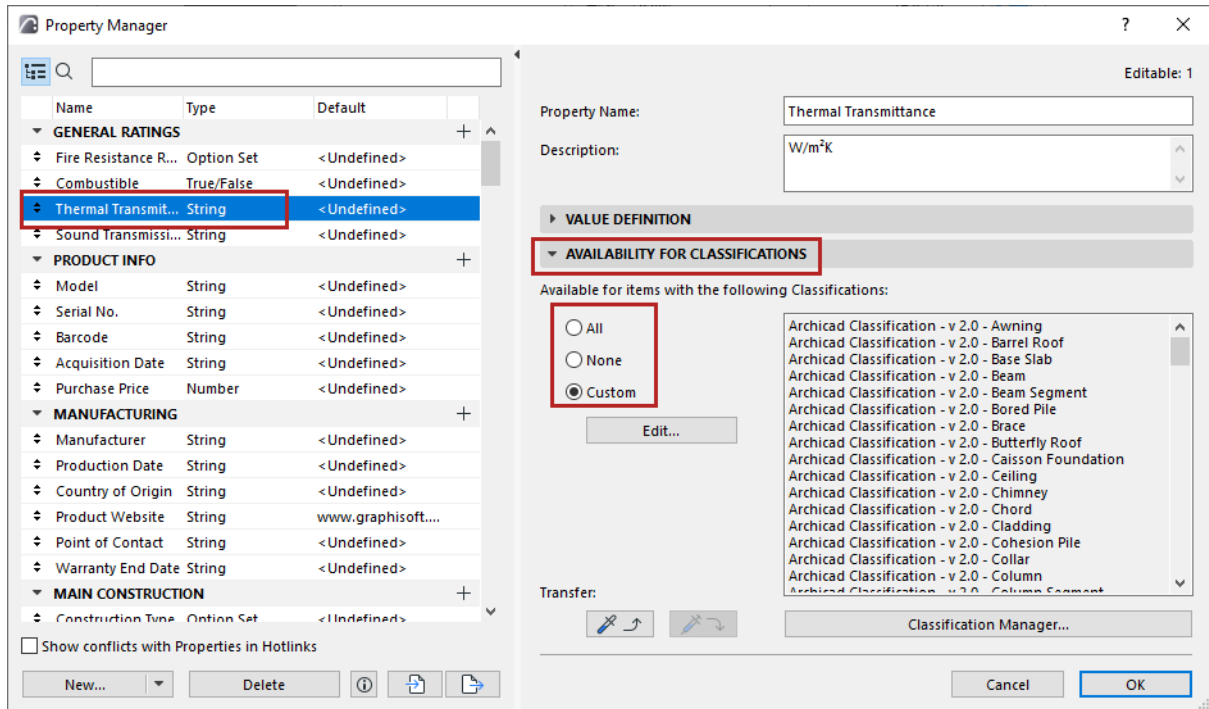


Multi-Choice Property Values in Element Settings

Property Availability for Classifications

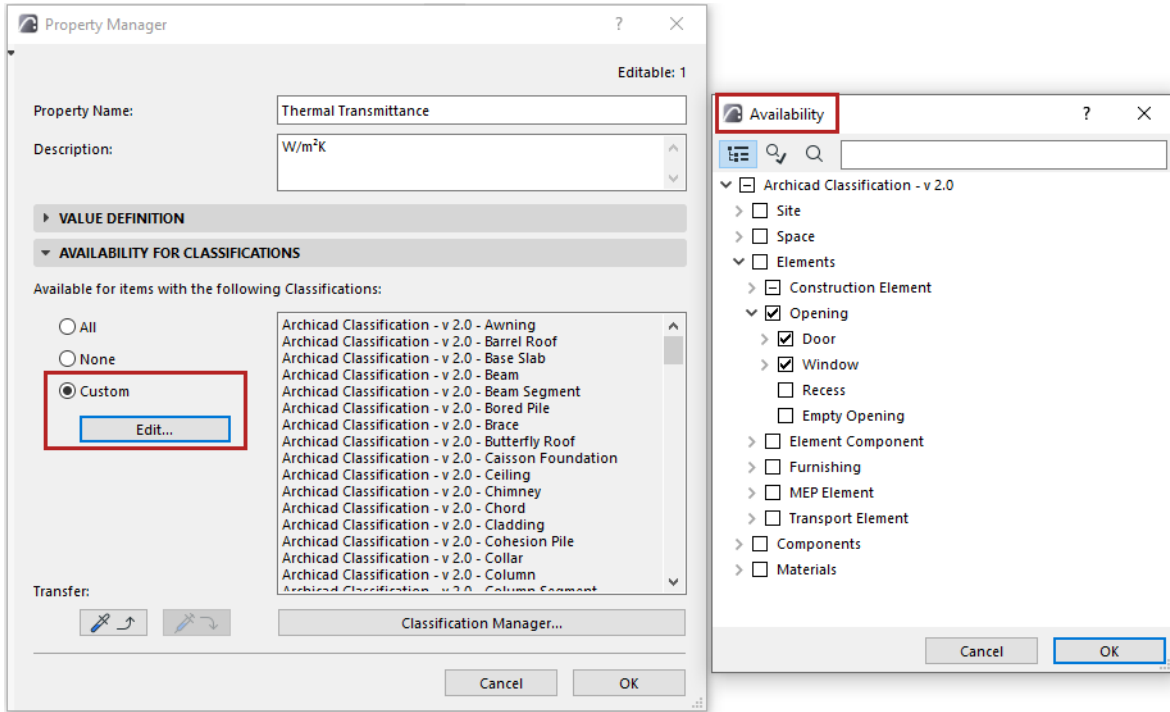
Use these controls in Property Manager to define which properties go with which Classification.

- Select one or more Properties or Groups from the list on the left.
- If you select a group, the Availability definition is applied to all properties in that group.



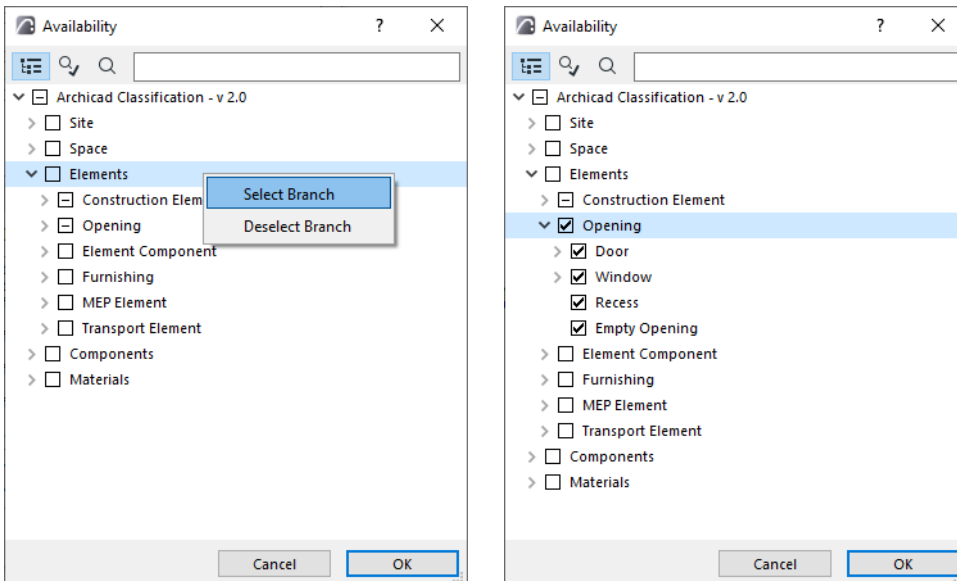
- **All:** The groups or properties selected on the left will be available for all classified items in the project.
- **None:** The groups or properties selected on the left will *not* be available for *any* elements or materials in the project. (You might set certain properties to None if the current project does not use them, yet you still want to use them in other phases of the same project, or other projects, e.g. in a template).
- **Custom:** Click this option, then click **Edit** to select the Classifications that should use this property or group.

For example, the Precast concrete components property (selected at left) is available to items that have one of the Classification groups selected at right.



Apply Availability to All Sub-Items

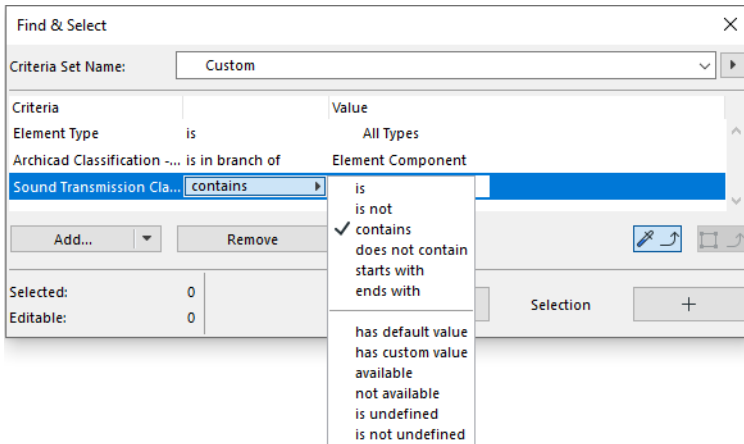
1. In the Availability dialog box, click on a parent item.
2. Right-click to bring up the context menu: click **Select Branch**.



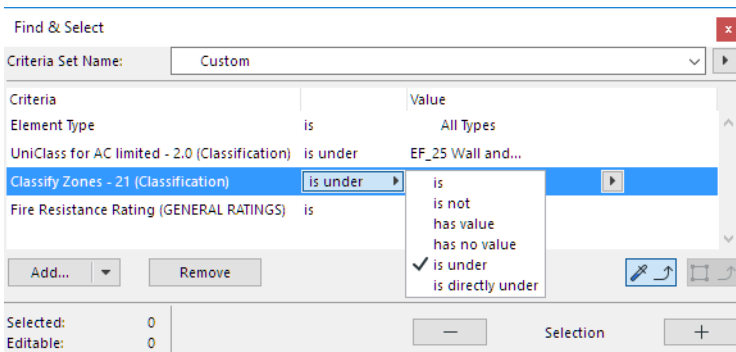
3. All sub-items of the parent item are now enabled.

Find and Select Element or Building Materials: By Property and Classification

- By Property value or Property availability, or Defined/Undefined value:



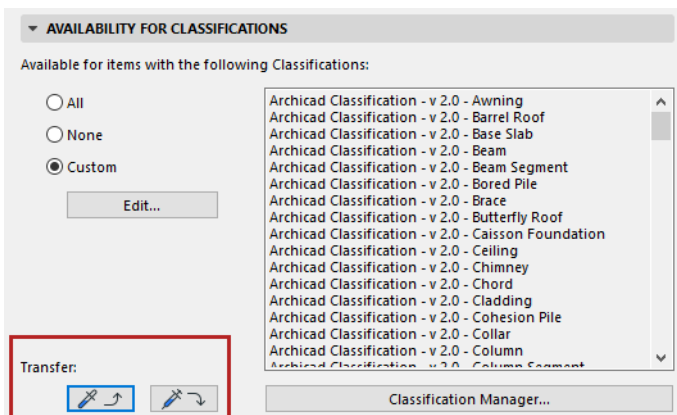
- By Classification:
 - Items classified within (“IS UNDER”) or directly underneath (“IS DIRECTLY UNDER”) a particular Classification branch



- Items having (“IS/IS NOT”) a particular Classification within the given Classification System

Transfer Availability

To transfer a property’s Availability definition to another property or group, use the Transfer buttons in Property Manager:



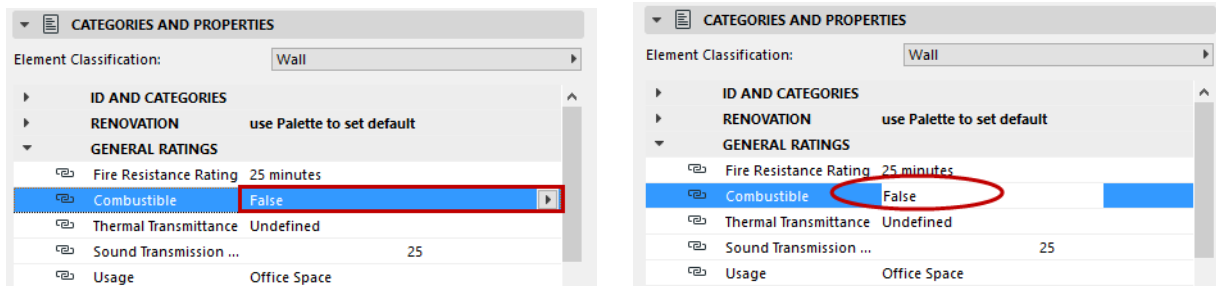
1. From the Property list at left, select the property whose Availability definition you want to transfer.
2. Click the “Pick up” icon.

3. Select a property or group to which to transfer the availability data.
4. Click the “Apply Availability” icon.

The availability data of the selected group (or property) is customized accordingly.

Changes in Property Manager

- After making edits in Property Manager, click **OK** to accept the changes and close the dialog box. At this point, you can use **Undo/Redo** to undo/redo the Property Manager edits.
- If you edit a property's Data Type in Property Manager, all the project items that use this property will retain their assigned values as far as possible. For example, if you change the Combustible property's True/False data type to the General data type, an element's Combustible property value that was set to False will stay that way, except it now becomes editable, as a General data type.

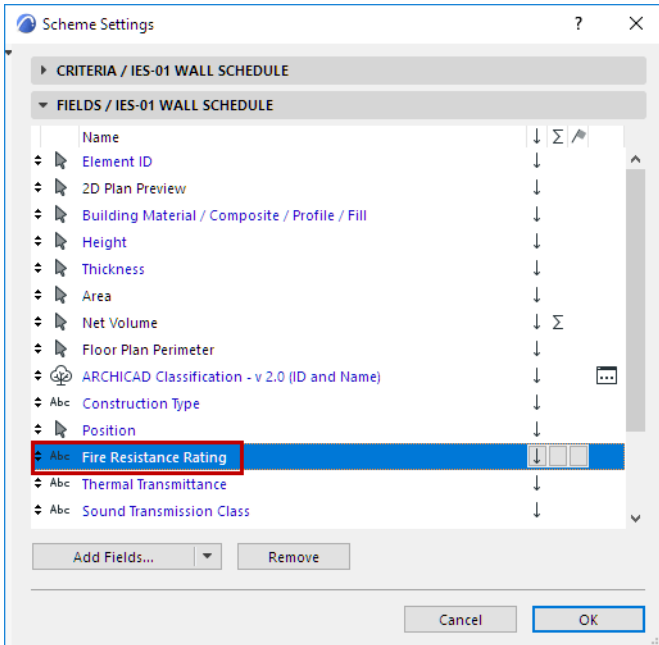


Data Type Changed from True/False to “General”

- From Property Manager, you can open the Classification Manager directly, without losing your modifications in Property Manager: click the Classification Manager button.
Make the changes you need in Classification Manager, then click OK to return to Property Manager. Property Manager is updated with the latest Classification changes.

Assign Classifications/Properties in Element Schedule

Use the Element Schedule as a convenient way to edit Classifications and Properties for a set of items. In the Schedule's Scheme Settings, add the Properties and Classifications you wish to edit (here, Fire Resistance Rating):



See also [Schemes Fields Panel](#).

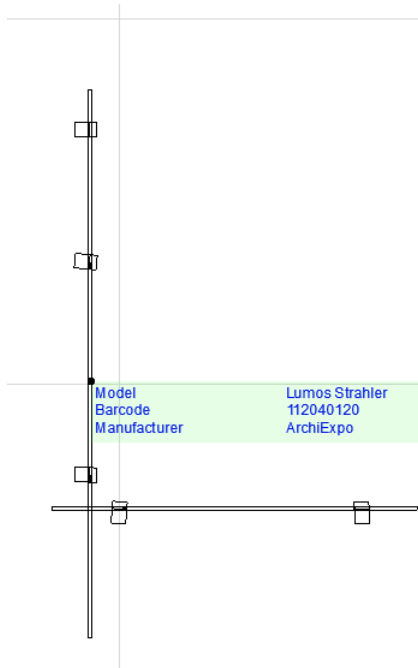
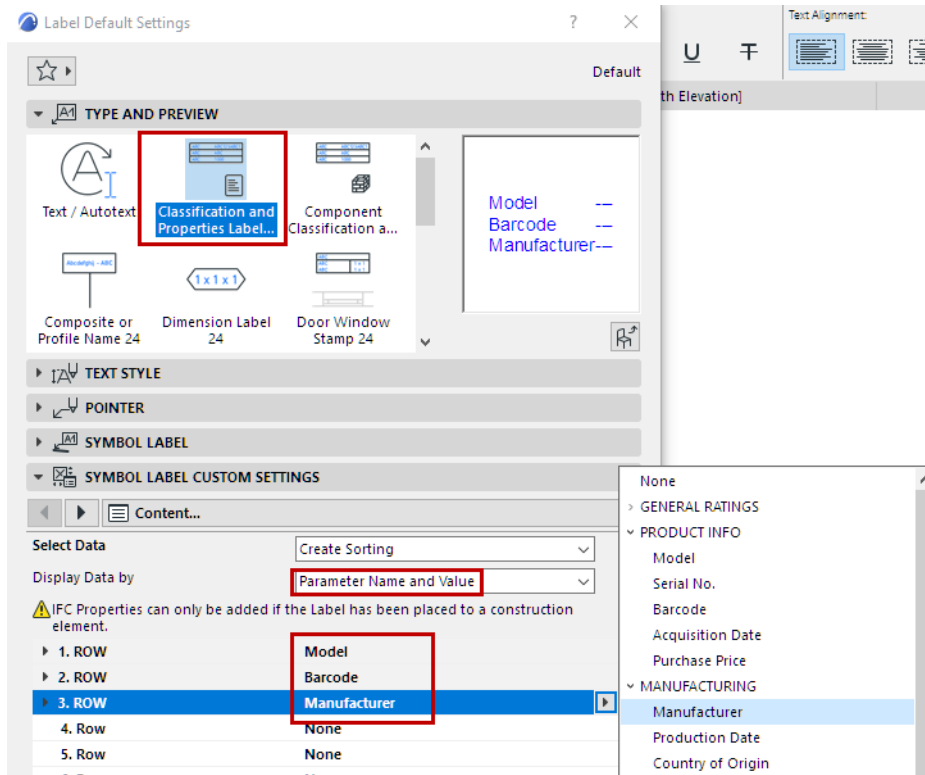
The Properties and Classifications now appear in the Schedule and can be edited.

Freeze Schedule Header

Wall Type	Volume [...]	Thickness [m]	Height [m]	Area [m ²]	Perimeter [m]	Length on Outside Face [m]	Surface Area on Outside Face [...]	Fire Resistance Rating
---	1.33	0.390	0.900	1.48	5.33	2.91	2.62	---
215 Block Insulated Cav...	0.52	0.390	0.900	0.58	3.29	1.11	1.00	---
215 Block Insulated Cav...	0.79	0.390	0.900	0.88	5.28	2.25	2.03	---
215 Block Insulated Cav...	2.01	0.390	1.600	1.28	5.89	2.42	3.87	---
Belt Ledge	0.19	0.442	0.207	1.07	5.72	2.42	0.57	---
Belt Ledge	0.21	0.442	0.207	1.17	6.17	2.64	0.62	---
Concrete Block - Filler	0.75	0.050	0.900	0.83	6.93	3.19	2.87	---
Concrete Block - Filler	1.26	0.390	0.900	1.40	6.53	2.89	2.60	---
Concrete Block - Filler	1.58	0.390	0.900	4.37	9.44	3.46	4.06	---
Custom	0.43	0.390	0.900	0.48	3.25	1.24	1.11	---
Custom	2.12	0.390	0.900	2.31	12.95	6.23	5.42	---
Exterior Wall	0.97	0.470	3.10	0.97	6.05	2.06	1.70	25 minutes
Exterior Wall	1.77	0.470	3.100	0.80	3.53	1.23	3.28	25 minutes
Exterior Wall	31.96	0.470	7.300	9.80	44.38	21.80	72.19	25 minutes
Exterior Wall	21.83	0.470	7.300	6.04	26.38	13.80	51.24	25 minutes
Galleryrailing	0.01	0.050	0.900	0.02	0.87	0.36	0.33	1 hour
Galleryrailing	0.03	0.050	0.900	0.05	2.24	1.10	0.95	1,5 hour
Galleryrailing	0.05	0.050	0.900	0.07	2.86	1.39	1.27	3 hour
Galleryrailing	0.10	0.050	0.900	0.15	6.07	2.96	2.66	Undefined
Galleryrailing	0.33	0.050	0.900	0.50	20.02	9.96	9.00	---
Generic Wall/Shell	1.18	0.300	3.200	0.37	3.06	1.23	3.94	25 minutes
Plywood	0.05	0.050	0.900	0.05	2.09	1.01	0.91	---
Plywood	0.10	0.050	0.900	0.11	4.59	2.26	1.99	---
Reinforced Concrete - S...	0.50	0.800	0.900	0.68	3.47	1.06	0.95	25 minutes
Reinforced Concrete - S...	0.84	0.800	0.900	1.14	5.01	1.89	1.70	25 minutes
Reinforced Concrete - S...	6.01	0.800	0.900	7.80	29.40	14.20	12.24	25 minutes
Reinforced Concrete - S...	8.82	0.800	0.900	12.60	45.40	22.20	16.72	25 minutes

Show Property Data in Labels and Markers

As needed, set up Labels to display any element or Building Material properties (generally the **Property Label** or the **Classification and Properties Label**):



[Watch the Video](#)

Expression-Defined Element Properties

Create functional Expressions which automatically evaluate element Property values, based on one or more of the following:

- Built-in and custom properties and parameters
- An element's model environment (e.g. data from the surrounding Zone, or that of its parent Hotlink Module)
- Global settings (e.g. Project Info).

Use a wide range of function types (many will be familiar from Excel) to generate the property values you need. Display the property values in schedules, labels, and Zone Stamps.

Note: Expressions cannot be evaluated for properties assigned to Building Materials.

Related Topics:

[Create a New Expression-Defined Property: Step By Step](#)

[Using Expression Editor: Context-Sensitive Help](#)

[Evaluate Expression-Defined Property](#)

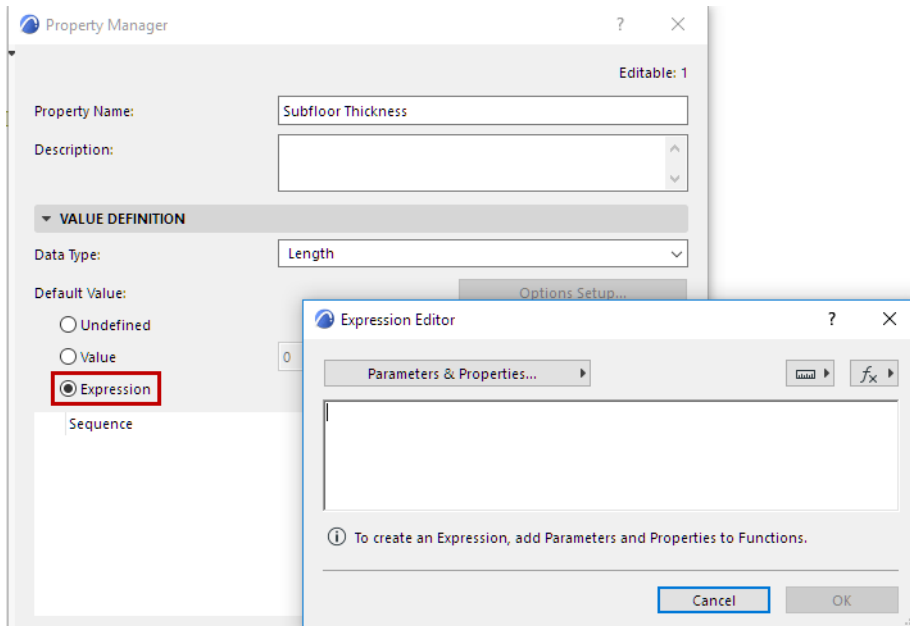
[Errors in Expressions and How to Fix Them](#)

[Expression-Defined Properties: Examples](#)

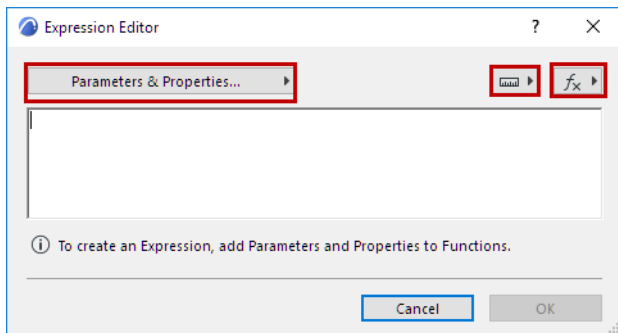
[Read More](#)

Create a New Expression-Defined Property: Step By Step

1. Open Property Manager (Options > Property Manager).
2. Create a New Property.
3. In the Value Definition panel, activate **Expression**. The Expression Editor dialog appears.

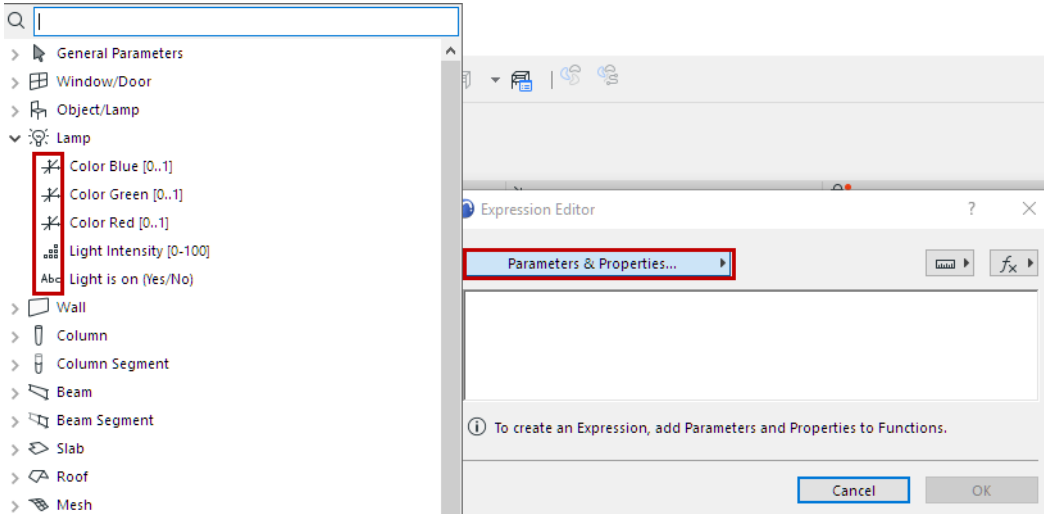


4. Define the Expression with the help of the pop-up lists: Parameters & Properties; Units; Functions.



5. Double-click any item in these lists to add it to the Expressions Editor
For example, start with a Function: use CONCAT to “concatenate” (or join) multiple string-type data items into a single item.

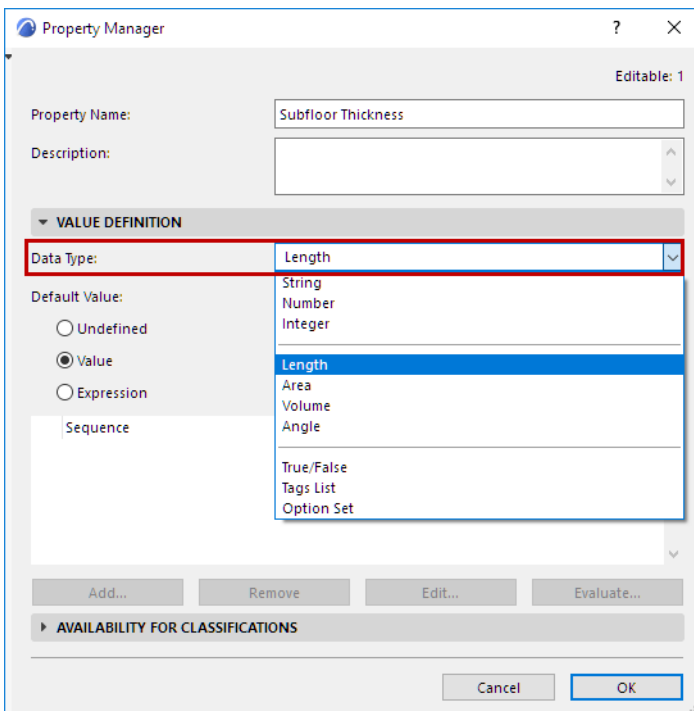
Data types of Parameters and Properties are indicated by their icon:



See the Examples below to get the idea: [Expression-Defined Properties: Examples.](#)

For information on how each parameter works, see: [Element Listing Parameters.](#)

- In the Value Definition panel, choose a Data Type that corresponds to the property expression. Note the Data Types that have unit values: Length, Area, Volume, Angle. (Archicad will help you choose the right ones – see the following section on Context-Sensitive Help.)



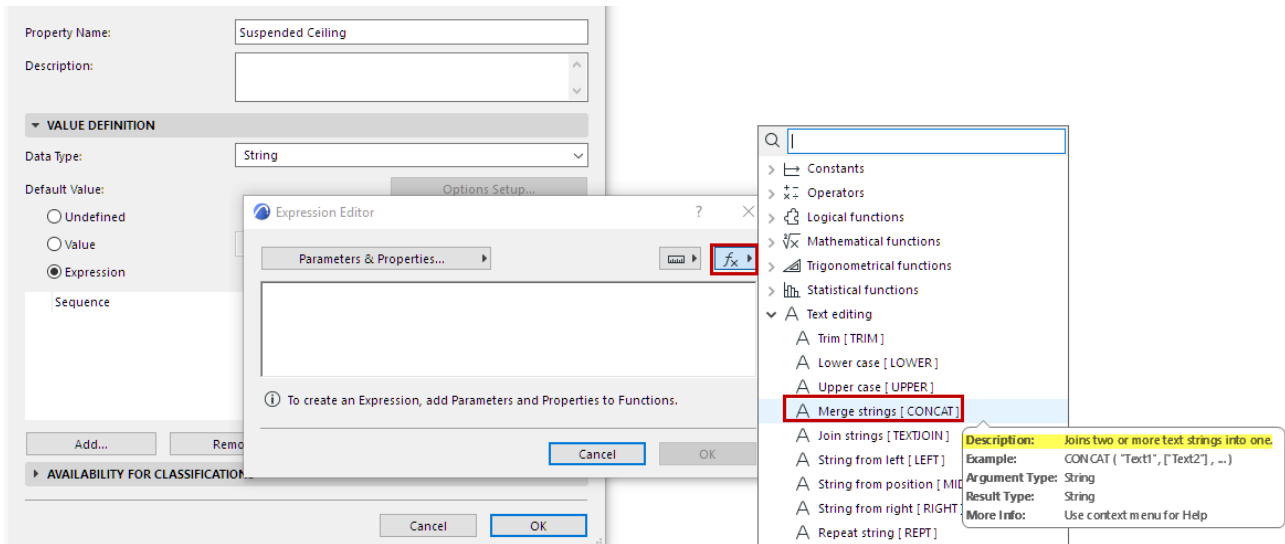
- Don't forget to set Availability for Classifications when you define a new Property!

- Check the generated Property value as needed.

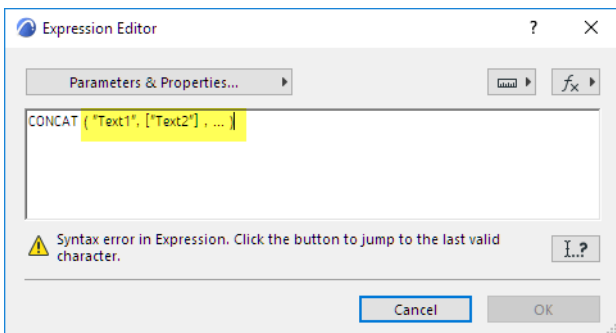
See [Evaluate Expression-Defined Property.](#)

Using Expression Editor: Context-Sensitive Help

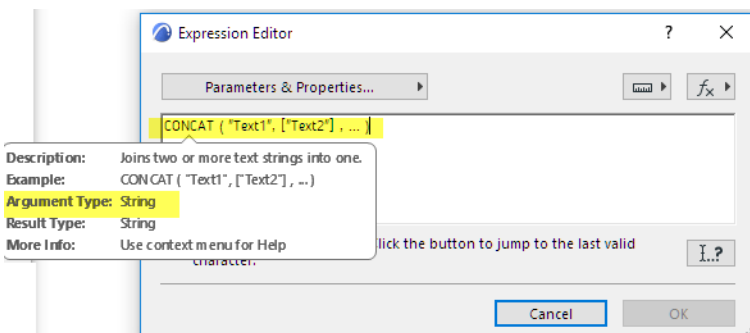
- Hover over any item in the Functions pop-up to see its description and syntax.



- When you add a Function to the Expression Editor, an example of the syntax appears, as a guide. (Delete this part when you no longer need it.)



- The required separator (comma or semi-colon) is also shown – this automatically corresponds to your computer’s location settings.
- When you add a Function to Expression Editor, you can hover it to see its description, syntax, and the Data Type required for each Argument. For example, in this CONCAT function, the “text1” and “text2” arguments must be of the String type.



For more help on how each Function works, see:

Functions in Expressions

For explanations of Error messages, see:

Errors in Expressions and How to Fix Them

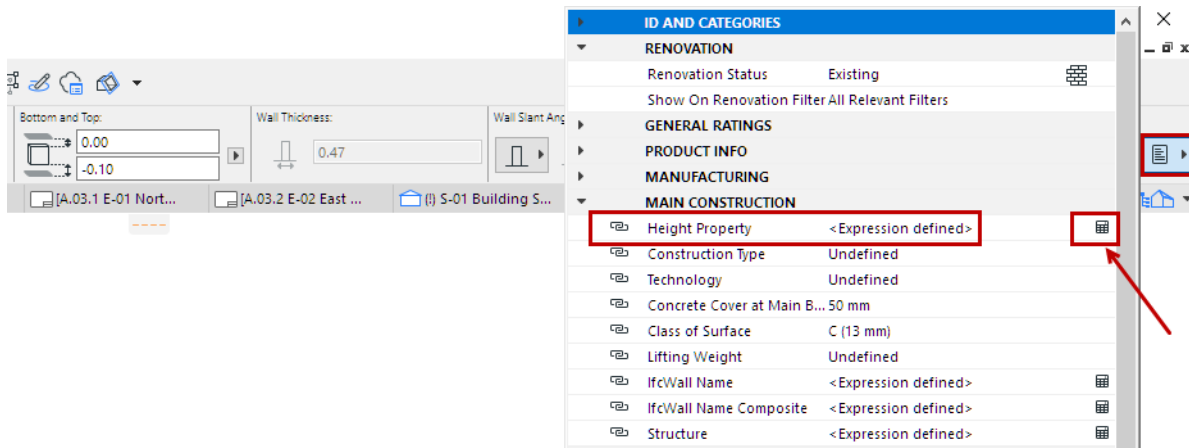
Evaluate Expression-Defined Property

Expression-defined properties are evaluated for model elements in schedules, labels and Zone Stamps, for example.

You can also check the property value for any selected element in the following ways:

Check Property Value in Info Box

To evaluate a selected element's expression-defined Property: Open the Property list from the Info box, and click the calculator icon for that property.



(Expression-defined properties are also listed in Element Settings, but the actual values cannot be shown.)

Evaluate a Property in Property Manager

If it is a new Property: You must first click OK to close Property Manager and save the new Property.

1. In the model, select one or more elements
2. Open Property Manager to the new Property.

3. Click Evaluate.

The screenshot shows a software dialog box for defining a property named 'Height Property'. The 'Data Type' is set to 'Length'. Under 'VALUE DEFINITION', the 'Expression' radio button is selected. A tree view shows 'Height' selected. The 'Evaluate...' button is highlighted with a red box. To the right, a list of 9 elements is displayed, each with its ID, expression, and the resulting height value.

Element ID	Expression	Value of the property
SLA - 001	Height = 0.30	0.30
SLA - 002	Height = 0.27	0.27
SW - 001	Height = 7.30	7.30
SW - 002	Height = 7.30	7.30
SW - 003	Height = 7.30	7.30
SW - 004	Height = 7.30	7.30
SW - 005	Height = 4.00	4.00
SW - 005	Height = 3.10	3.10
SW - 006	Height = 3.10	3.10

Expression-Defined Properties: Examples

Example 1: Simple Mapping

Example 2: Modify Element Parameter, Using an Operator

Example 3: Expand Property with Descriptive Text String

Example 4: Calculate a Quantity and Round Up

Example 5: Calculate a Length and Add Overhang

Example 6: Round Up to Standard Beam Length

Example 7: Display Regulation-Compliance Data for an Element

Example 8: Generate Expanded ID for Openings

Example 9: Calculate Number of Tiles to Purchase

Example 10: List the Total Cost of a Structure in Dollars

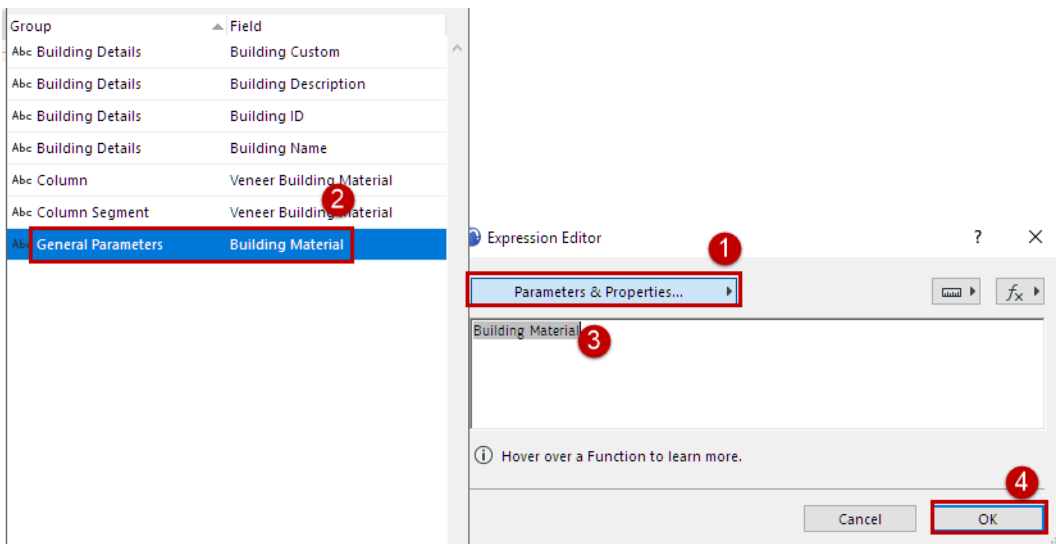
Example 1: Simple Mapping

Create a new property that automatically maps existing element parameter values. No functions are needed for this; just identify the set of existing parameters that you want to evaluate for model elements.

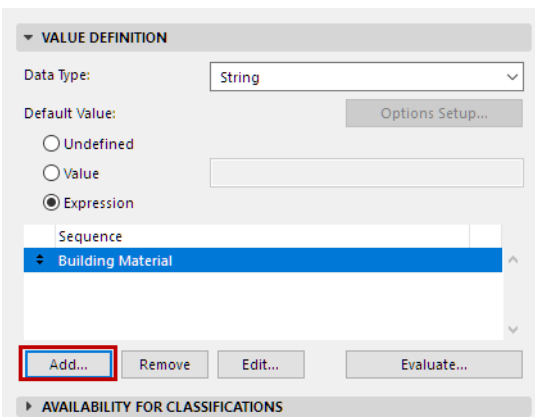
1. In Property Manager, create a **New Property** called Structure.
2. Set its **Data Type** to String.
3. Select the **Expression** option. The Expression Editor appears.

Use the **Parameters & Properties** pop-up to add a sequence of Expressions, each one returning a Structure-type parameter (Building Material, Fill, Composite Structure, Complex Profile). Each Expression in the sequence must have the same Data Type (in this case, String.)

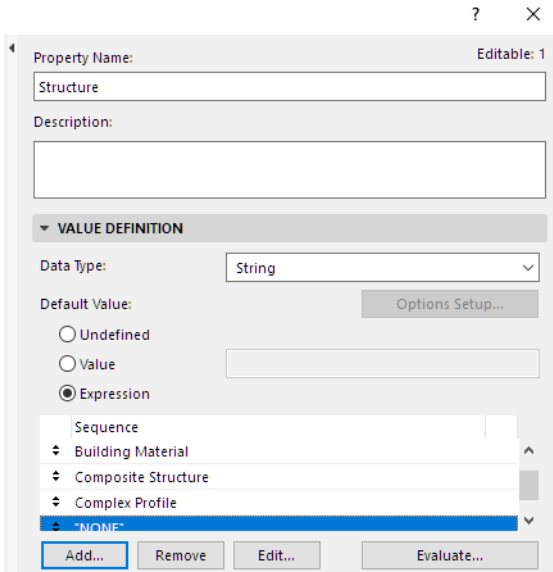
- Click **Parameters & Properties** to search for the “Building Material” parameter
- Double-click the parameter, or click Add to add it to the Expression Editor.
- Click OK to close Expression Editor.



- Click **Add** to add the next Expression (the Fill Type parameter), and so on for all parameters in the sequence.



- Finally, add an expression consisting of the string “NONE”



Archicad will evaluate these Expressions in sequence for each element, returning the first valid value for each element. Once a value is returned, the subsequent expressions are not considered for that element. If none of the definitions apply to an element, that element will list “NONE”.

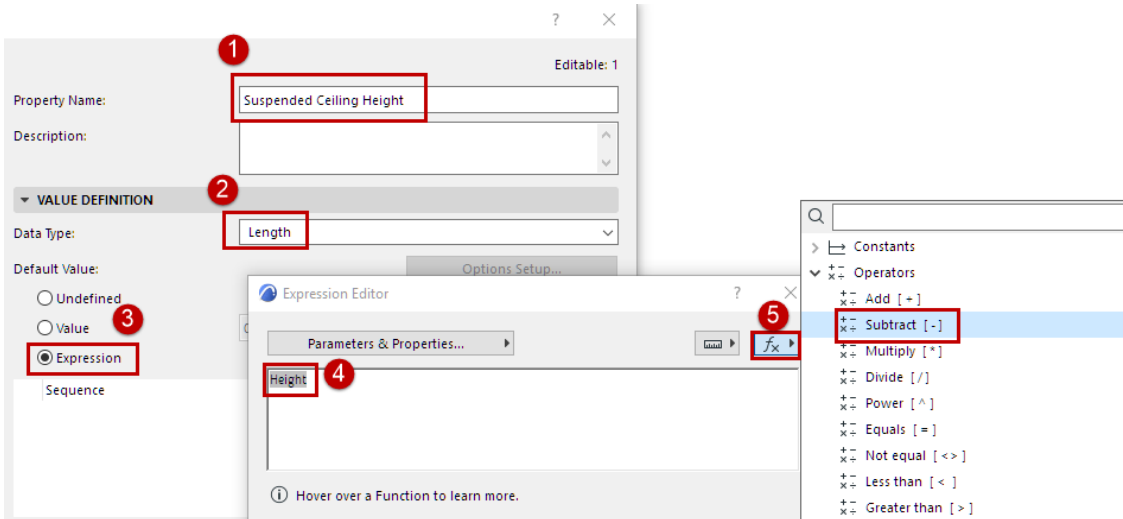
4. Make sure you set the property’s **Availability for Classifications**.
5. Click **OK** to close Property Manager and save the new property.
6. List the property in Interactive Schedule to see the results for model elements:

Element List	
Element ID	Structure
BMR - 001	Timber - Structural
CRE - 001	Timber - Structural
CRE - 002	Reinforced Concrete - Structural
CRE - 003	Reinforced Concrete - Structural
CRE - 004	Reinforced Concrete - Structural
CRE - 005	Reinforced Concrete - Structural
DOO - 001	NONE
DOO - 002	NONE
DOO - 003	NONE
DOO - 004	NONE
DOO - 005	NONE
DOO - 006	NONE
DOO - 007	NONE
FU - 003	NONE
FU - 004	NONE
LI - 001	NONE
LI - 002	NONE
RT - 001	Roof Aluminium
SKY - 001	NONE
SKY - 002	NONE
SKY - 003	NONE
SLA - 001	Concrete Floor Insulated with 10mm Tile
SLA - 002	Exterior Pavement
SLA - 003	Flat Roof
SLA - 004	Concrete Floor with Parquet
SW - 001	Exterior Wall
SW - 002	Exterior Wall
SW - 003	Exterior Wall
SW - 003	Reinforced Concrete - Structural
SW - 003	Stud Partition
SW - 004	Exterior Wall

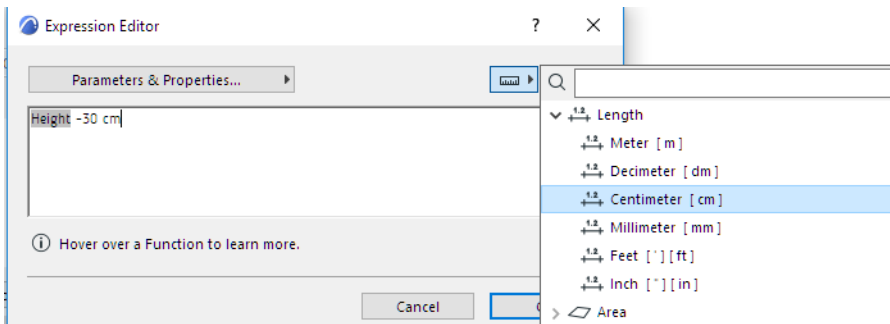
Example 2: Modify Element Parameter, Using an Operator

Define a new property for Zones that returns a value of 30 cm below the Zone height.

1. In Property Manager, create a **New Property** called Suspended Ceiling Height.
2. Set its **Data Type** to Length.
3. Select the **Expression** option. The Expression Editor appears.
4. Use the **Parameters & Properties** pop-up to add HEIGHT as the first expression.
5. Use the **Function** pop-up to choose the Subtract operator (or just type a minus sign), then type 30.

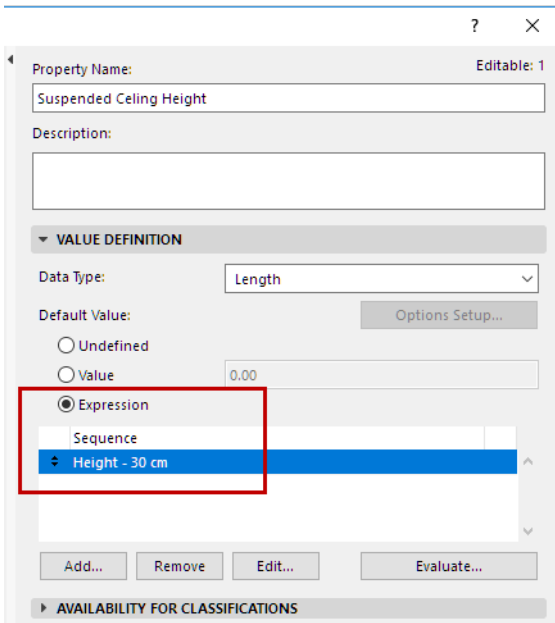


6. Use the **Unit** pop-up to choose cm as the unit (or type it in).



Your function can be run using any of the units listed here. (However, the calculated property value will be shown on the Archicad interface according to the Calculation Units in effect).

7. The resulting Expression:



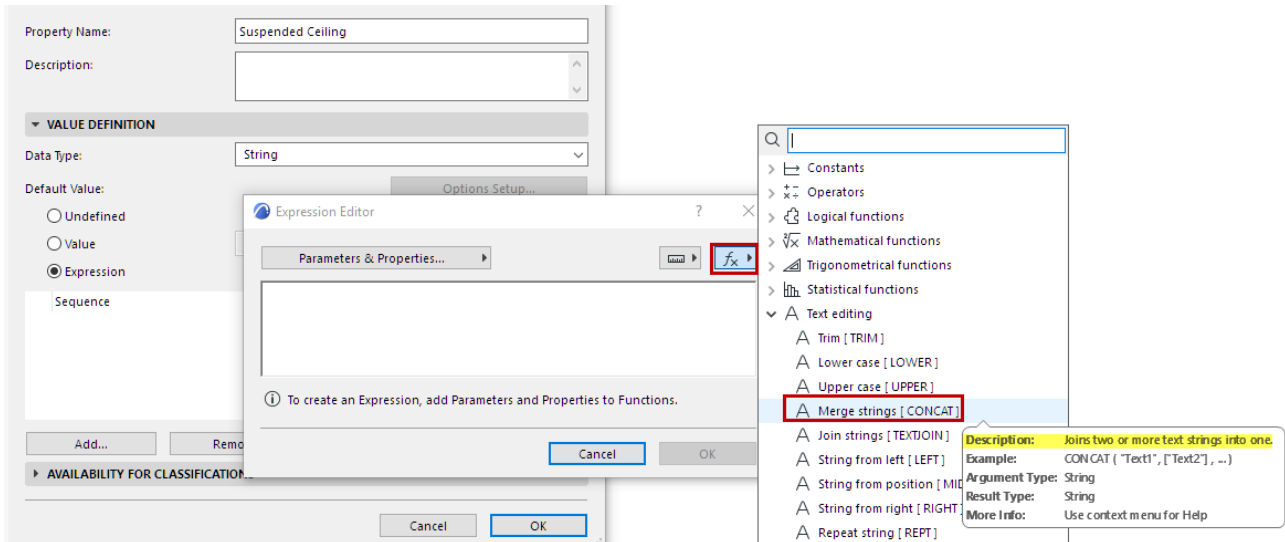
8. Define the **Property Availability** for all Space-classified elements.
9. Click **OK** to close Property Manager and save the new property.
10. Display the “Suspended Ceiling Height” property value in the Zone Stamp.



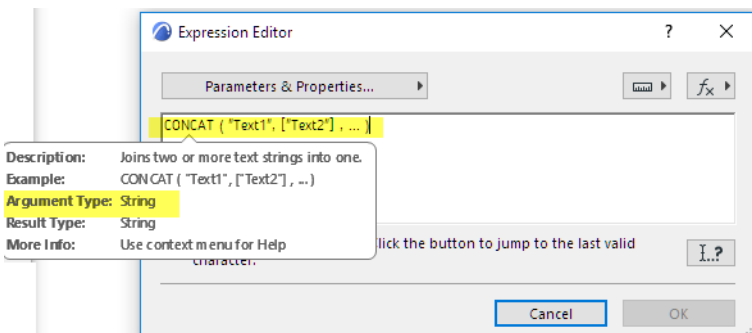
Example 3: Expand Property with Descriptive Text String

To the Suspended Ceiling Height property value (created in [Example 2: Modify Element Parameter, Using an Operator](#)), add a text string that explains this property wherever it is displayed (e.g. on a Zone Stamp).

1. In Property Manager, create a **New Property** called Suspended Ceiling.
2. Set its **Data Type** to String.
3. Select the **Expression** option. The Expression Editor appears.
4. Start by adding the CONCAT function. As indicated in the Function description, CONCAT is used to join two strings together.



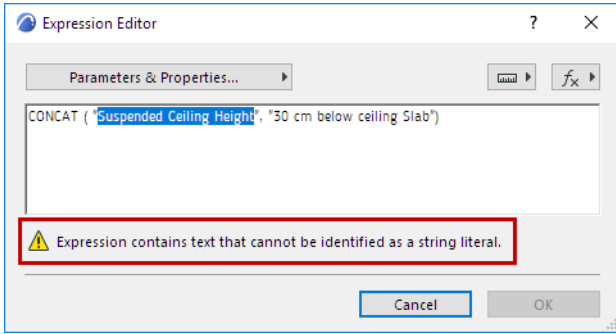
5. When you add the CONCAT function, you can hover over it to see an example of the syntax, as a guide.



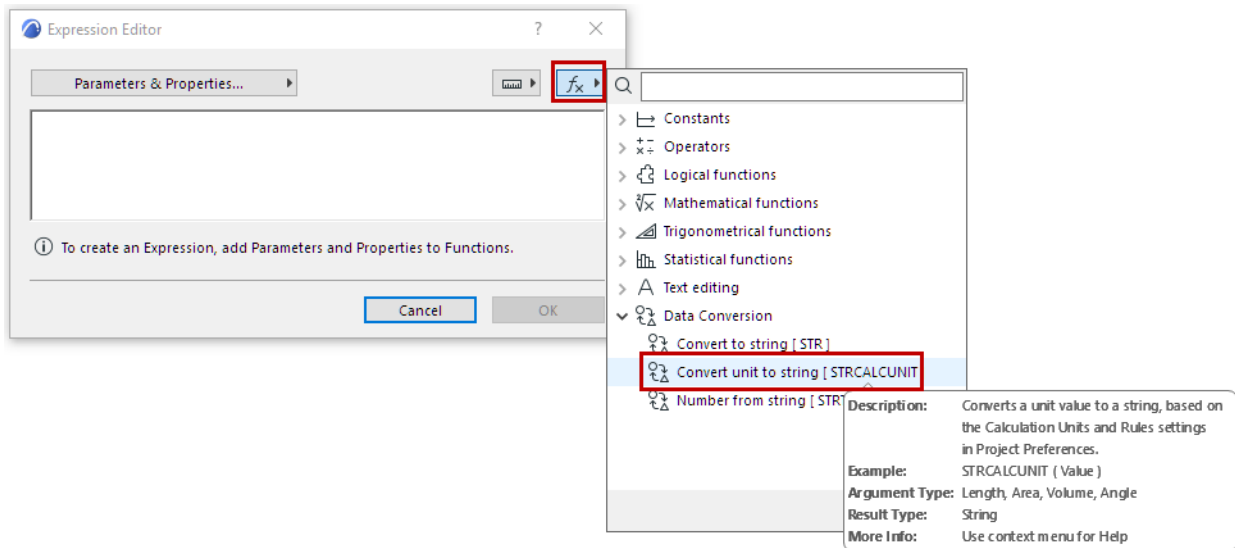
In this case, the CONCAT function requires a series of string-type arguments, with parentheses on the outside.

6. For the first argument, add the **Suspended Ceiling Height** property. (Click **Parameters and Properties**, then search for this property and add it.)
7. For the second argument, enter a Text string in quotes.

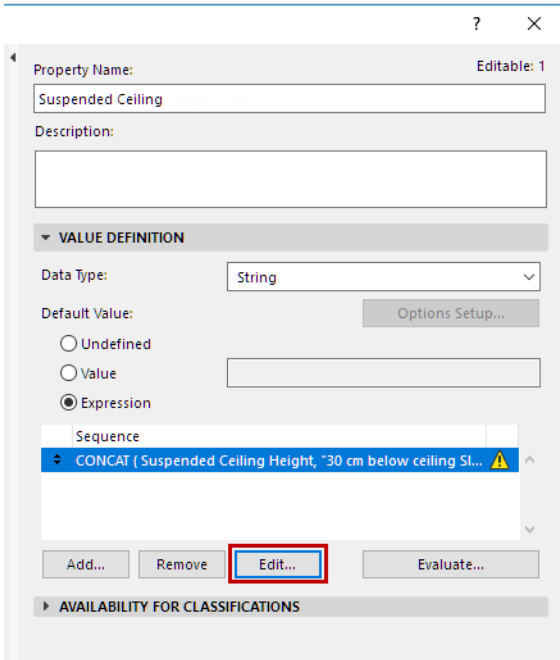
8. Archicad warns you that there is a problem:



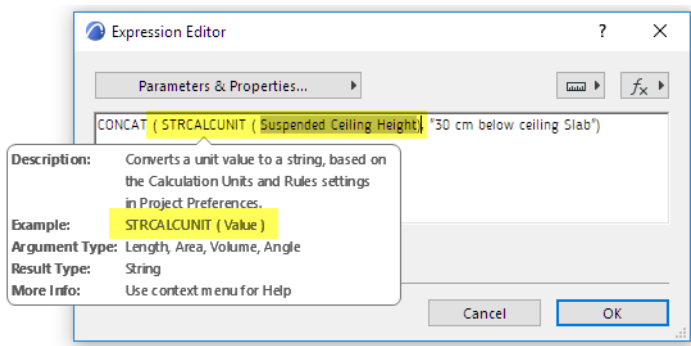
The Data Type of one of your arguments is wrong. The CONCAT function requires two string-type arguments, yet your first argument – the Suspended Ceiling Height property – is a Length-type argument. The solution is to convert the Suspended Ceiling Height value from a Length-type to a String-type value. For this purpose, use the **STRCALCUNIT** function (see its description in the Function list pop-up):



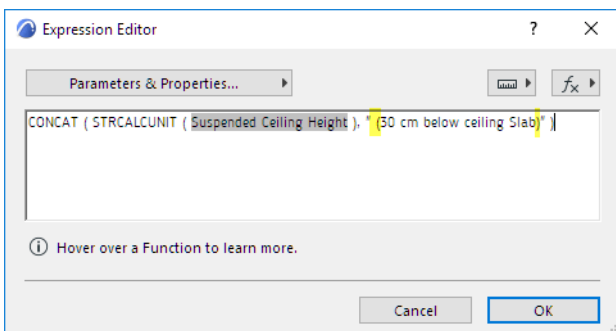
- Now you will edit the CONCAT expression in Expression Editor. (If the Editor is not open, select the expression in Property Manager and click Edit; or double-click the expression in the Sequence window).



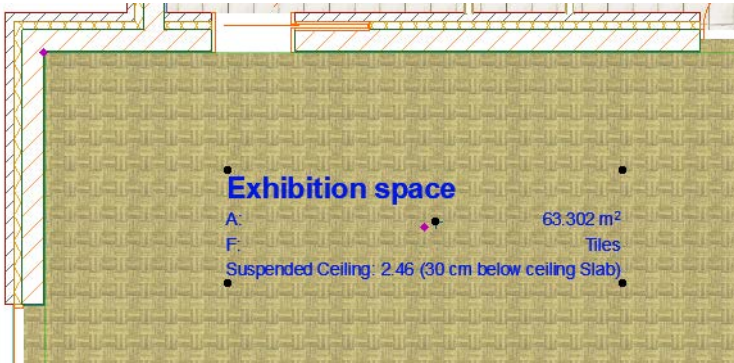
- Within the existing expression, apply the STRCALCUNIT function to the Suspended Ceiling Height property.
Make sure to enclose the STRCALCUNIT function, together with its value, in parentheses. (Hover over STRCALCUNIT function to see the example in the pop-up).



- You can add spaces and parentheses to the second string, within the quotation marks, to set it off from the first argument.



12. Click **OK** to close Expression Editor.
13. Define the **Property Availability** for all Space-classified elements.
14. Click **OK** to close Property Manager and save the new property.
15. Display the “Suspended Ceiling” property value in the Zone Stamp.



Example 4: Calculate a Quantity and Round Up

Round up a calculated quantity to conform to standard material (e.g. brick) size

- Determine the number of bricks you need to order (by rounding up to the next whole number)
- The standard size of a brick (IS Standard) is 190 mm ×90 mm ×90 mm

1. Create a Property called “Number of Bricks”.
2. Use an expression to find the number of bricks by dividing the element’s Net Volume by the standard brick volume. (A volume divided by a volume results in a Number.)

The screenshot shows a property definition window for a property named "Number of Bricks". The "Property Name" field contains "Number of Bricks" and is marked as "Editable: 1". The "Description" field is empty. Under the "VALUE DEFINITION" section, the "Data Type" is set to "Number". The "Default Value" is set to "0.0". The "Expression" radio button is selected, and the expression field contains the formula: $\text{Net Volume} / (190 \text{ mm} * 90 \text{ mm} * 90 \text{ mm})$. At the bottom, there are buttons for "Add...", "Remove", "Edit...", and "Evaluate...".

3. Create another property called “Number of Bricks to order.”
4. This property rounds up the “Number of Bricks” property (created above) to a whole number. Use the `MROUNDUP` function for this purpose.

The screenshot shows a property definition window for a property named "Number of Bricks to order". The "Property Name" field contains "Number of Bricks to order" and is marked as "Editable: 1". The "Description" field is empty. Under the "VALUE DEFINITION" section, the "Data Type" is set to "Number". The "Default Value" is set to "0.0". The "Expression" radio button is selected, and the expression field contains the formula: `MROUNDUP (Number of Bricks, 1)`. At the bottom, there are buttons for "Add...", "Remove", "Edit...", and "Evaluate...".

See [Round Up to Multiple \[MROUNDUP\]](#) for information on this function’s arguments and data types.

Example 5: Calculate a Length and Add Overhang

Calculate required Beam length, taking into account the extra ledge length (20 cm) needed for each. To do this, create a property that expands the existing “Length Right” Beam parameter, by adding 20 cm.

The screenshot shows the 'Properties' dialog box for a beam element. The 'Property Name' is 'Beam Length plus overhang' and it is 'Editable: 1'. The 'Description' field is empty. Under the 'VALUE DEFINITION' section, the 'Data Type' is set to 'Length'. The 'Default Value' is set to 'Expression', and the expression is 'Length Right + 20 cm'. The 'AVAILABILITY FOR CLASSIFICATIONS' section is set to 'Custom' and includes the classification 'ARCHICAD Classification - 21 - Beam'. The 'Transfer' section has icons for copy and paste, and a 'Classification Manager...' button.

Example 7: Display Regulation-Compliance Data for an Element

Given a regulation that a zone area must include 40% glazed area: Calculate the glazed area of a given Zone and display it in the Zone stamp.

1. Create a property (“Glazed Area”) that shows total glazed area (door plus window). Here, you add two existing element properties together.
 - Adding two areas produces a Data Type which is also an Area.

Property Name: Editable: 1
 Glazed Area
 Description:
 [Empty text box]
VALUE DEFINITION
 Data Type: Area
 Default Value: [Options Setup...]
 Undefined
 Value: 0.00
 Expression
 Sequence
 ↕ Doors Surface Area + Windows Surface Area
 [Add...] [Remove] [Edit...] [Evaluate...]

2. Create another property (“Glazed Area Ratio to Zone”) that shows the ratio of the glazed area to the Zone’s Measured Net Area.
 - Here, you divide an area by an area: this produces a Data Type which is a Number.

Property Name: Editable: 1
 Glazed Area Ratio to Zone
 Description:
 [Empty text box]
VALUE DEFINITION
 Data Type: Number
 Default Value: [Options Setup...]
 Undefined
 Value: 0.00
 Expression
 Sequence
 ↕ Glazed Area / Measured Net Area
 [Add...] [Remove] [Edit...] [Evaluate...]

Zones					
Element ID	Net Area	Windows Surface Area	Doors Surface Area	Glazed Area	Glazed Area Ratio to Zone
ZON - 001	63.30	0.00	39.48	39.48	0.62
ZON - 002	7.24	0.90	5.70	6.60	0.91
ZON - 003	3.50	0.36	3.15	3.51	1.00
ZON - 004	1.94	0.36	1.58	1.94	1.00
ZON - 005	24.99	3.52	3.60	7.12	0.28

3. Create another property (“Mandatory Glazed Area”) that defines the glazing area regulation (40 percent). This is a fixed value (not an expression.)

The screenshot shows a 'Property Definition' dialog box. The 'Property Name' field contains 'Mandatory Glazed Area'. Below it is a 'Description' field which is empty. Under the 'VALUE DEFINITION' section, the 'Data Type' is set to 'Number'. The 'Default Value' is set to '0.40' with the 'Value' radio button selected. There are also 'Options Setup...', 'Add...', 'Remove', 'Edit...', and 'Evaluate...' buttons at the bottom.

4. Create another property (“Display Glazing Compliance”) to return one of two strings for each zone: meets/ does not meet minimum glazed area regulation.
 - Use the IF function for this purpose.

See [If Multiple \[IFS\]](#) for information on this function's arguments and data types.

Property Name: Editable: 1
 Display Glazing Compliance

Description:

VALUE DEFINITION

Data Type: String

Default Value: Options Setup...

Undefined
 Value
 Expression

Sequence
 IF ((Glazed Area Ratio to Zone >= Mandatory Glazed Area), 'Meets Regulation', 'Does Not Meet Regulation')

Add... Remove Edit... Evaluate...

Zones						
Element ID	Net Area	Windows Surface Area	Doors Surface Area	Glazed Area	Glazed Area Ratio to Zone	Display Glazing Compliance
ZON - 001	63.30	0.00	39.48	39.48	0.62	Meets Regulation
ZON - 002	7.24	0.90	5.70	6.60	0.91	Meets Regulation
ZON - 003	3.50	0.36	3.15	3.51	1.00	Meets Regulation
ZON - 004	1.94	0.36	1.58	1.94	1.00	Meets Regulation
ZON - 005	24.99	3.52	3.60	7.12	0.28	Does Not Meet Regulation

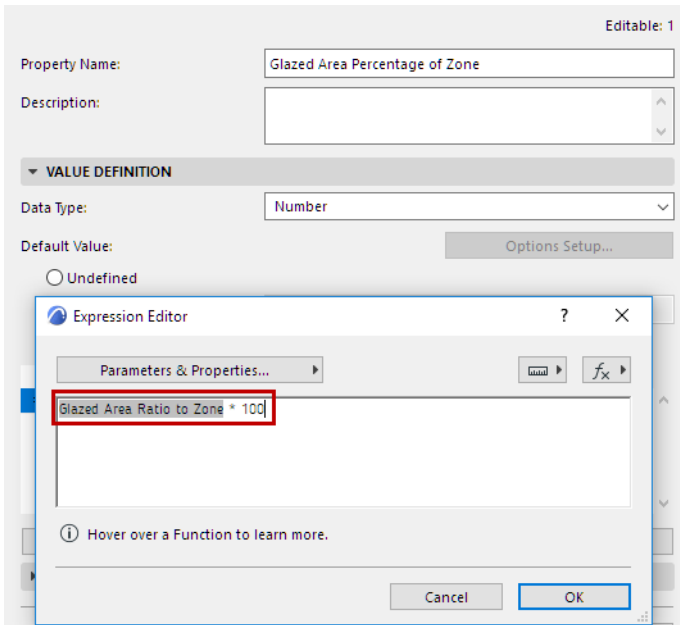
5. Add the text from this property (“Display Glazing Compliance”) to the Zone Stamp.



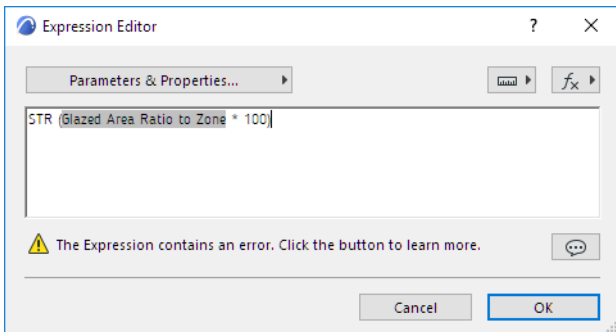
- 6. You can also create and display another property: Glazed Area Percentage of Zone. The resulting value will be a String which includes the percentage symbol.

In this new Expression:

- Start with the existing “Glazed Area Ratio to Zone” property. This property expresses the resulting number as a decimal (with Data Type of Number). To convert this decimal to a percentage, multiply it by 100:



- Now convert this Number-type value to a String, using the STR function:



- Change the property’s Data type to String.
- Finally, use CONCAT to join this String to another String (the second String is the percentage sign, which you must put into quotation marks):

Editable: 1

Property Name:

Description:

VALUE DEFINITION

Data Type:

Default Value:

Undefined
 Value
 Expression

Sequence

← CONCAT (STR (Glazed Area Ratio to Zone * 100), "%")

See [Convert to String \[STR\]](#) and [Merge Strings \[CONCAT\]](#) for information on these functions' arguments and data types.

- Add this property to the Zone Stamp



Example 8: Generate Expanded ID for Openings

Automatically generate an expanded ID for all Doors and Windows that includes:

- Element ID
- Home Story Number
- Opening Direction (Orientation)

In the Expression shown below, the CONCAT function joins these three data into a single String. This will be the new ID property (called “Door/Window Full Element ID”).

Note: Since Home Story Number is not a String-type data, you must convert it to a String using the STR function.

The screenshot shows a configuration window for a property. At the top, the 'Property Name' is 'Door/Window Full Element ID' and it is marked as 'Editable: 1'. Below this is a 'Description' field. The main section is titled 'VALUE DEFINITION'. It shows 'Data Type' set to 'String'. Under 'Default Value', the 'Expression' radio button is selected. The expression field contains: `CONCAT (Element ID, "- ", STR (Home Story Number, 0), "- ", Orientation)`. At the bottom, there are buttons for 'Add...', 'Remove', 'Edit...', and 'Evaluate...'.

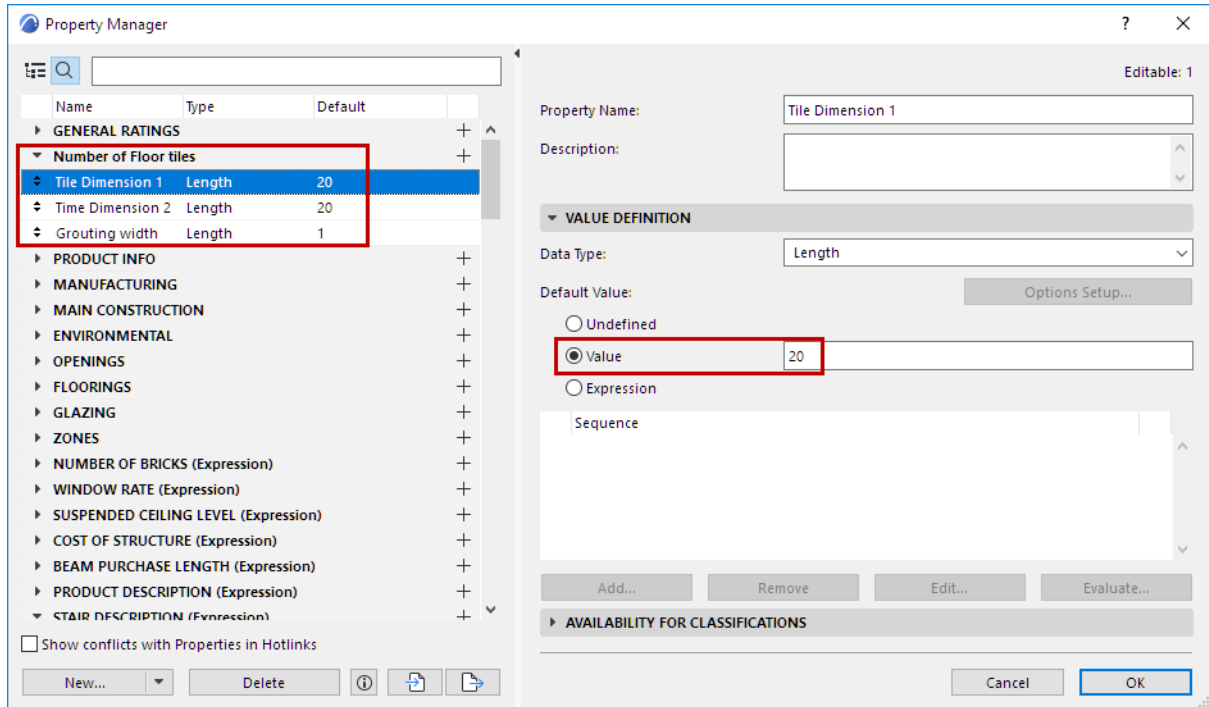
Doors-Windows with Expanded ID			
Element ID	Home Story Number	Orientation	Door/Window Full Element ID
DOO - 001	0	L	DOO - 001 - 0 - L
DOO - 002	0	L	DOO - 002 - 0 - L
DOO - 003	2	R	DOO - 003 - 2 - R
DOO - 004	1	R	DOO - 004 - 1 - R
WD - 001	0	L	WD - 001 - 0 - L
WD - 001	1	L	WD - 001 - 1 - L
WD - 001	2	L	WD - 001 - 2 - L
WD - 002	0	R	WD - 002 - 0 - R
WD - 002	1	R	WD - 002 - 1 - R
WD - 002	2	R	WD - 002 - 2 - R

See [Convert to String \[STR\]](#) and [Merge Strings \[CONCAT\]](#) for information on these functions' arguments and data types.

Example 9: Calculate Number of Tiles to Purchase

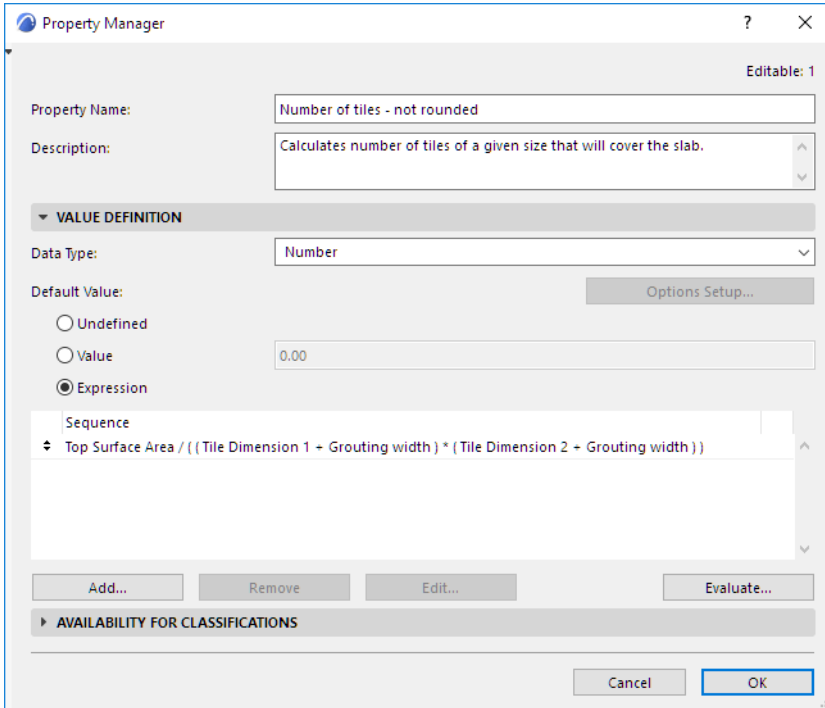
For a given Slab, how many tiles are needed to cover it?

1. Define three Properties for Tile dimensions: Tile Dimension 1 and Tile Dimension 2, and the Grouting dimension.
 - These are fixed Length-type values (not Expressions).

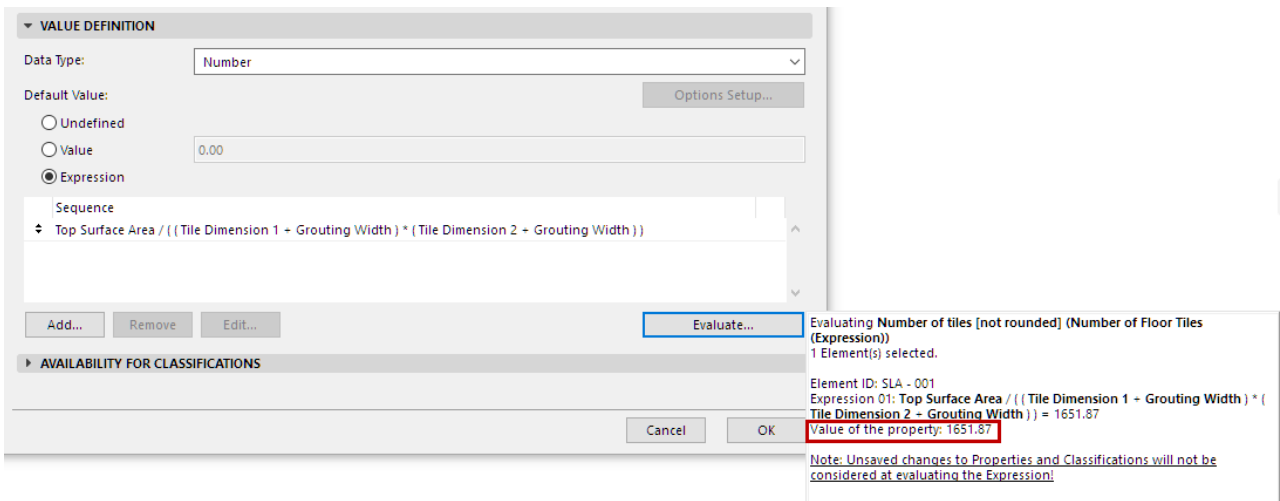


2. Define an expression called "Number of tiles - not rounded", using the three dimensions you defined in Step 1, to calculate the number of tiles needed to cover the slab:

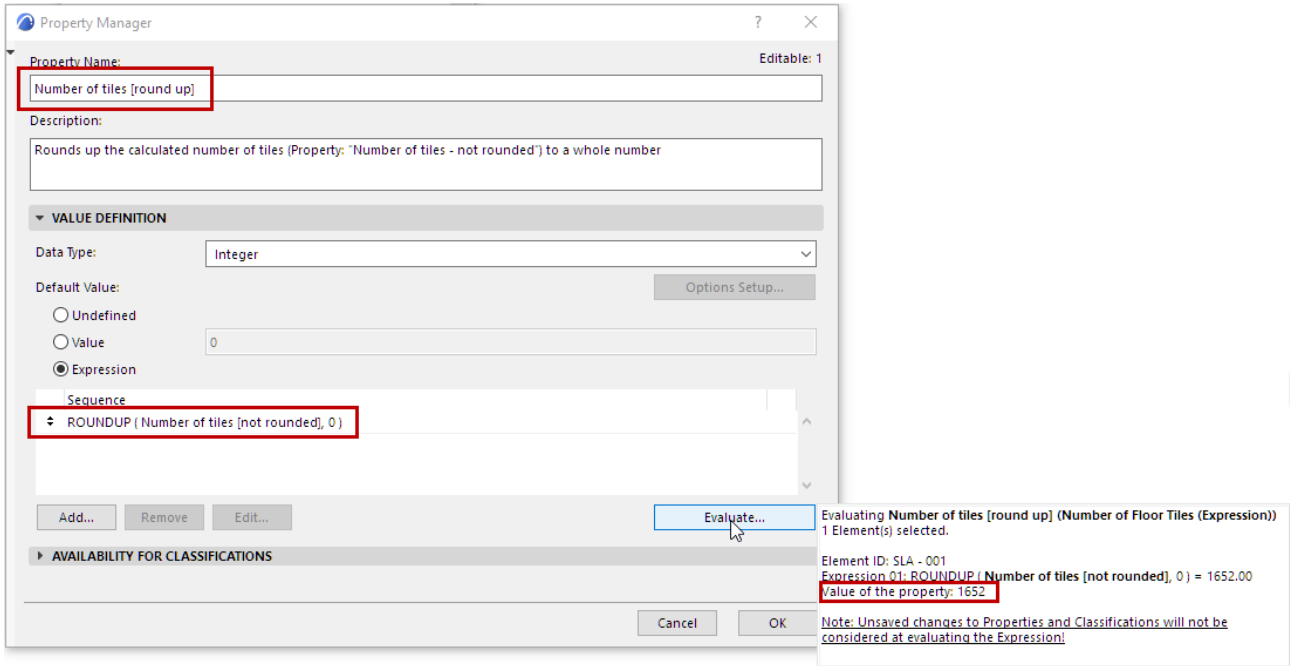
Take the **Top Surface Area** of the Slab and divide it by the area of the tile, including grouting. (For tile area: Add the grouting length to each tile dimension, then multiply them.)



Evaluate for the selected Slab to see the resulting Property value:



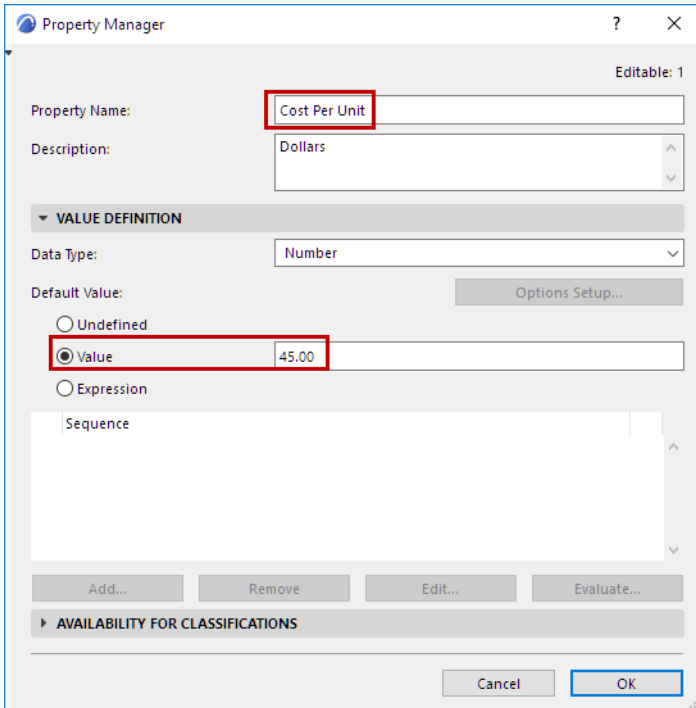
- Define another expression (using the expression you defined in Step 2) that rounds up the number of tiles to a whole number.



See [Round Up \[ROUNDUP\]](#) for information on this function's arguments and data types.

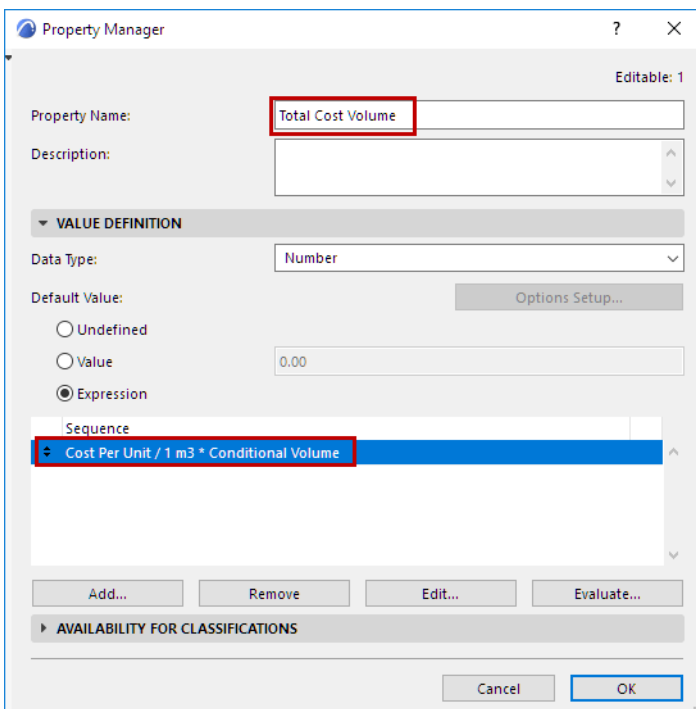
Example 10: List the Total Cost of a Structure in Dollars

1. Define a Property that defines the cost of a particular structure type. This is a fixed value - here, it is 45.



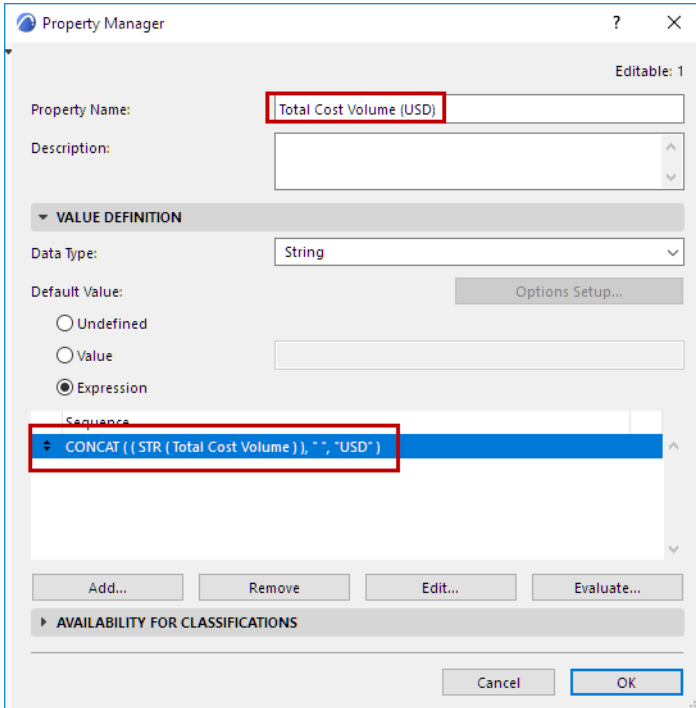
2. Define another Property to do the following:

- Express the Cost Per Unit property (defined above) as a volume unit (divide the Cost value by cubic meters): $\text{Cost Per Unit} / 1 \text{ m}^3$
- Multiply this cost by the actual volume of the structure (its Conditional Volume)



3. Define another property which adds the string “USD” to the result.
4. To do this, use the CONCAT function to join String-type data into a single String. Add a space before the “USD” string.

Note: Since the Total Cost Volume property is not a String-type data, you must convert it to a String using the STR function, as shown here.

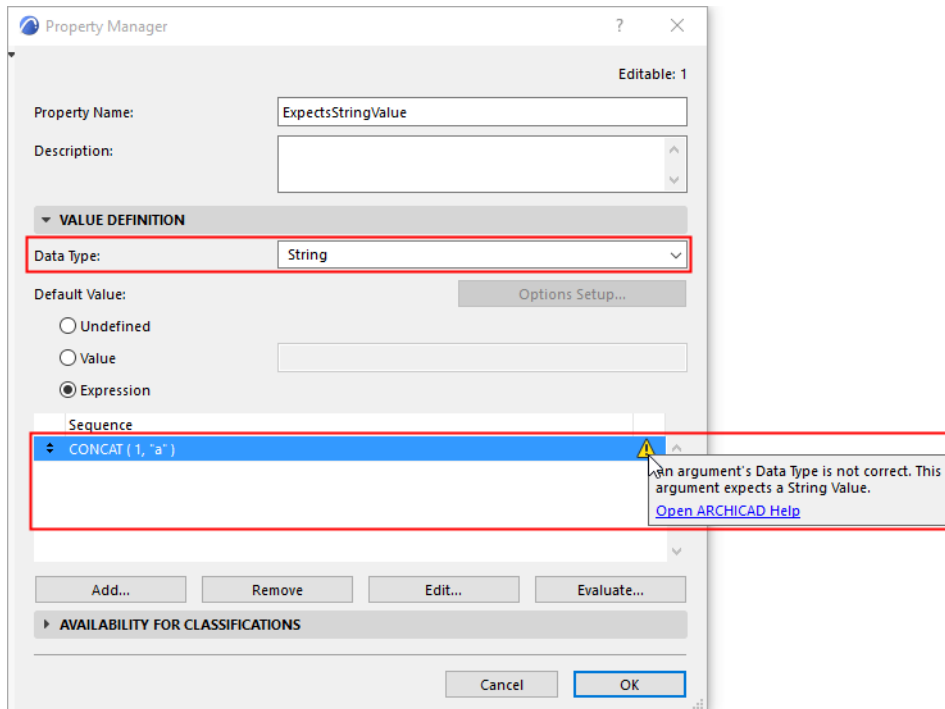


Wall Schedule	
Element ID	SW - 001
2D Plan Preview	
Wall Type	Generic Wall/Shell
Height [m]	3.000
Thickness [m]	0.300
Element Classification	Wall
Construction Type	Exterior bearing walls
Position	Exterior
Fire Resistance Rating	30 minutes
Total Cost Volume (USD)	196.21 USD

Errors in Expressions and How to Fix Them

When you work with Expressions in Property Manager, Archicad automatically compares the Property's defined Data Type with the result of the Expression. These must match.

If there is a discrepancy, Archicad automatically gives you a Warning. Click it to see the error message.



The following are the possible error messages and what you can do to fix them.

Result Conflicts with Data Type

All Arguments Must Use the Same Data Type and Compatible Units

Argument's Data Type is Not Correct

Data Type Not Available in Archicad

Mathematical Error

Incorrect Sign and/or Number Type

Data Type Cannot be Changed

Does Not Contain Numeric Characters

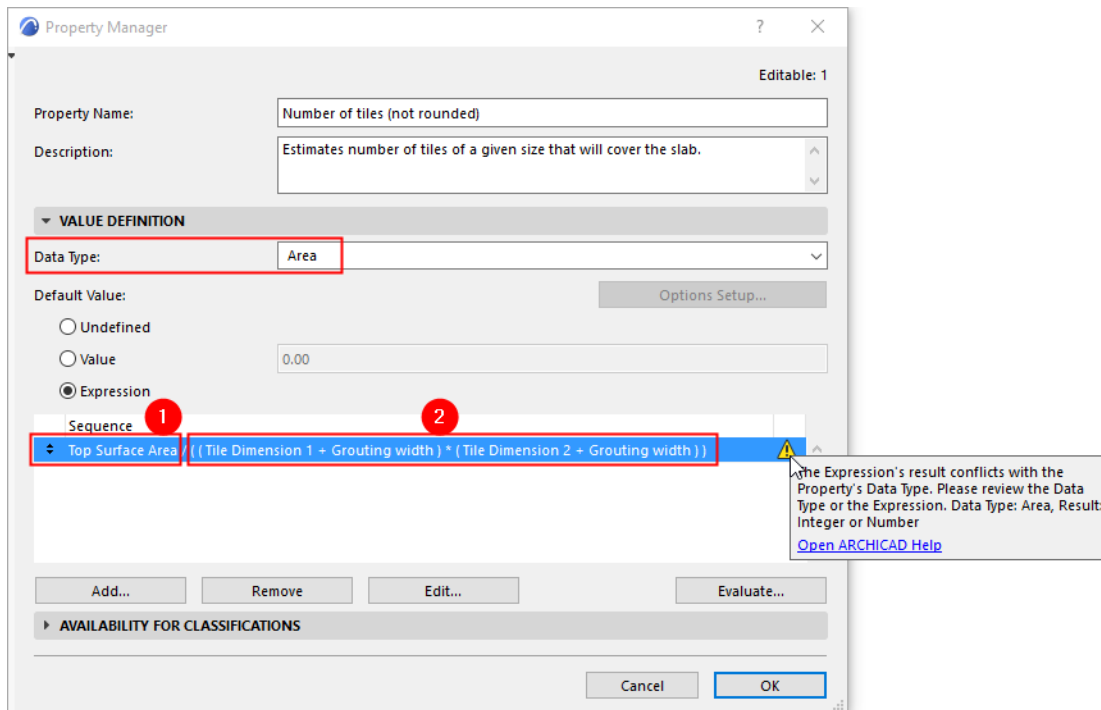
Result Conflicts with Data Type

If you get this error message, you must change the Data Type so that it matches the result of your expression.

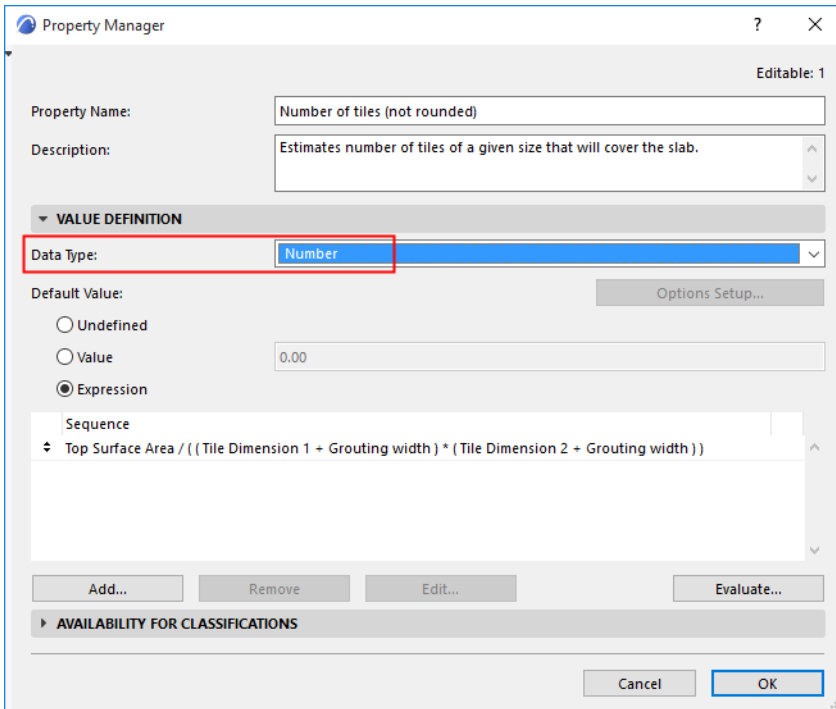
Example 1: Area Divided by Area: Requires Data Type of “Number”

This Expression-based property divides an **area-type property** (no. 1) with another **area** (the product of two lengths, no. 2).

The Property’s Data Type is set to Area, but this is incorrect, because an area divided by an area is a Number.

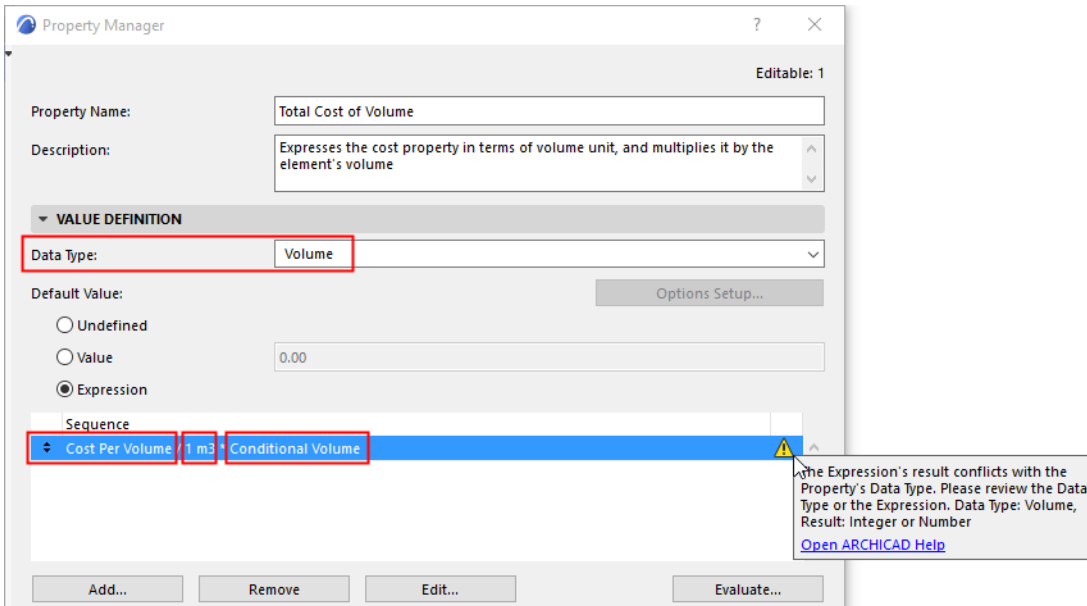


Set the Data Type to Number. The error is corrected.



Example 2: Cost-Per-Volume Multiplied by Volume: Requires Data Type of “Number”

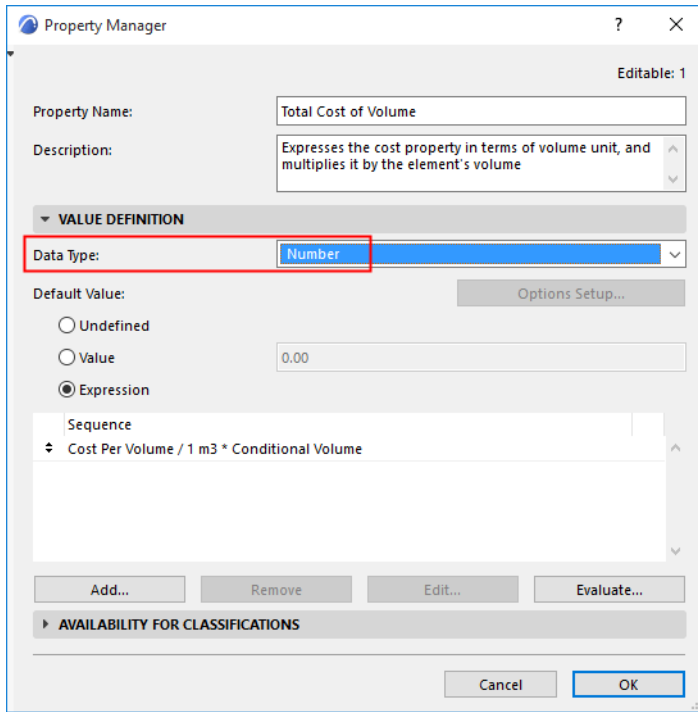
In this Expression-based property:



- The first argument (no. 1) is a number-type property (Cost per Volume).
- To express this as a volume unit: divide the first argument (no. 1) by 1 cubic meter (no. 2): Cost Per Volume / 1 m3.
- Now you can multiply this cost-per-volume by the actual volume of the structure (its Conditional Volume, argument no. 3).

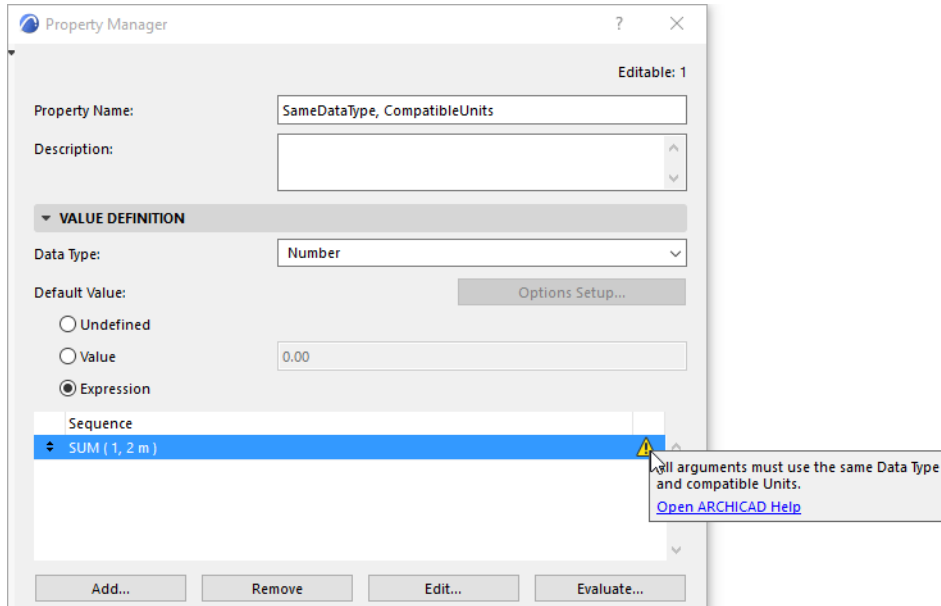
The Property's Data Type is set to Volume, but this is incorrect, because a Cost-per-volume multiplied by a volume is a Cost - that is, a non-unit Number.

Set the Data Type to Number. The error is corrected.



All Arguments Must Use the Same Data Type and Compatible Units

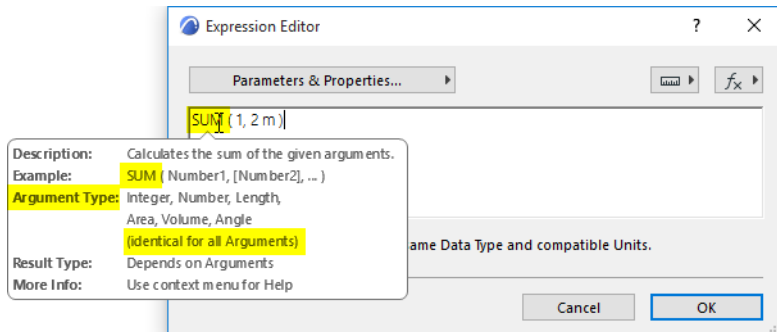
If you get this error, make sure all arguments in the Expression have the same Data Type.



Example: Add a Number to a Length?

SUM (1, 2m)

This is incorrect, because the two arguments use two different data types: 1 is a Number, 2m is a Length. Yet in the **SUM** Function, all arguments must use identical data types. (You can see this by opening Expression Editor and hovering over the **SUM** item.)



Solution

Change the expression so that both arguments use an identical Data Type (here, Length in meters).

SUM (1m, 2m)

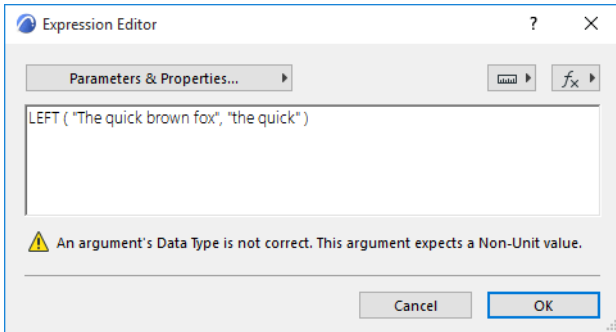
The resulting Property is also a Length. Make sure the Property's Data Type is set to Length.

Argument's Data Type is Not Correct

If you get this error, check to see which data types are required by each argument in the Function.

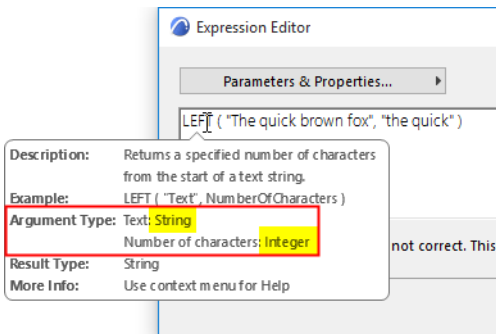
Example 1: Check Required Data Types for Each Argument

This Expression uses the `LEFT` function with two Arguments. However, the error message indicates a problem with the Arguments' Data Type(s).



In Expression Editor, hover over the `LEFT` function name.

Check the pop-up to see the required data types:



You see that the first (Text) argument must be a string, and the second (NumberOfCharacters) argument must be an integer.

But your expression contains two strings, which is incorrect:

```
LEFT ("The quick brown fox", "The quick")
```

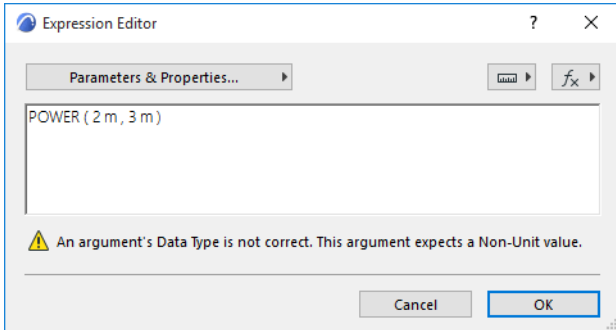
Solution

Change the second argument to an integer.

```
LEFT ("The quick brown fox", 9)
```

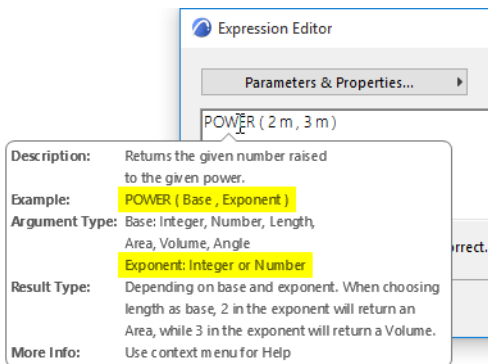
Example 2: Check Required Data Types for Each Argument

This Expression uses the `POWER` function with two Arguments. However, the error message indicates a problem with the Arguments' Data Type(s).



In Expression Editor, hover over the `POWER` function name.

Check the pop-up to see what data types are required by each argument in this Function:



You see that the second (Exponent) argument must be an integer or number.

Your expression contains two Lengths, which is incorrect:

```
POWER (2m, 3m)
```

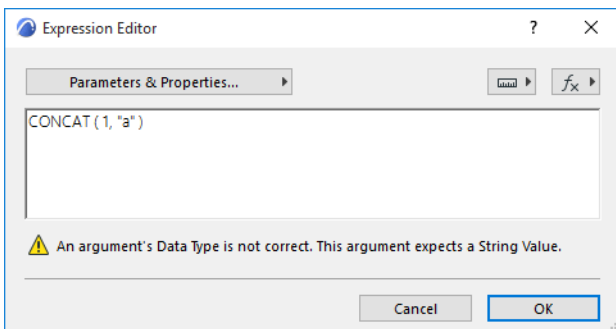
The second argument (Exponent) must be an integer or a number, without a unit.

Solution

```
POWER (2m, 3)
```

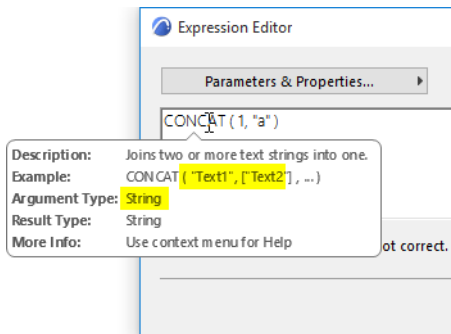
Example 3: Check Required Data Types for Each Argument

This Expression uses the `CONCAT` function with two Arguments. However, the error message indicates a problem with the Arguments' Data Type(s).



In Expression Editor, hover over the `CONCAT` function name.

Check the pop-up to see what data types are required by each argument in this Function:



In the `CONCAT` function, all arguments must be strings. A text is a string if it is in quotation marks.

In your expression, the first argument is a number, which is incorrect:

```
CONCAT (1, "a")
```

Solution

Change the first argument to a string by doing one of the following:

- by enclosing text within quotation marks:

```
CONCAT ("1", "a")
```

- or by using the `STR` function:

```
CONCAT (STR (1), "a")
```


Data Type Not Available in Archicad

This error appears if the function produces a result whose Data Type does not exist or is not supported in Archicad.

Solution

Correct the function so that it produces a result that corresponds to an existing Data Type.

Mathematical Error

Problem

This error appears if a mathematical function uses arguments that are mathematically impossible to calculate - such as division by zero, or the square root of a negative number.

Solution

Review the arguments and adjust their values to make the function mathematically correct.

Incorrect Sign and/or Number Type

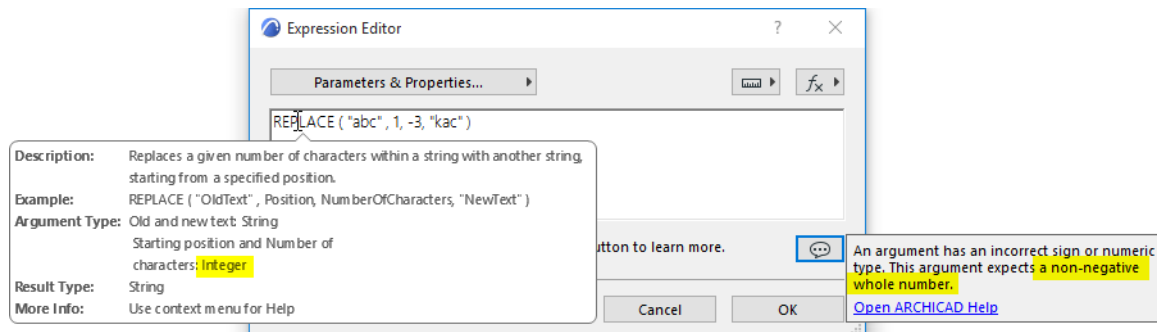
Problem

This error appears if

- you entered a value with the wrong sign - e.g. you have a negative number for an argument that requires a positive one
- you entered a value with the wrong number type - e.g. you used a decimal number for an argument that requires a whole number (integer).

Solution

Review the arguments and adjust their values.



Data Type Cannot be Changed

For this Property, you cannot change the Data Types to one that uses model units (Length, Area, Volume, Angle). These Data Types are greyed out in the list.

This can occur in the following scenarios:

- you have assigned a custom Property value to one or more elements or materials that use this Property.
- In a Teamwork project, this Property was created by another user.

Reason: When you change a property's Data Type, Archicad attempts to convert the existing property values. However, a Property value that has a model unit (e.g. a Length value) cannot be converted to a different unit (e.g. an Area value).

In this situation, switching Data Types is not allowed.

Similarly, you cannot edit the unit-based Data Type of a Property created by another user (in Teamwork).

Solution

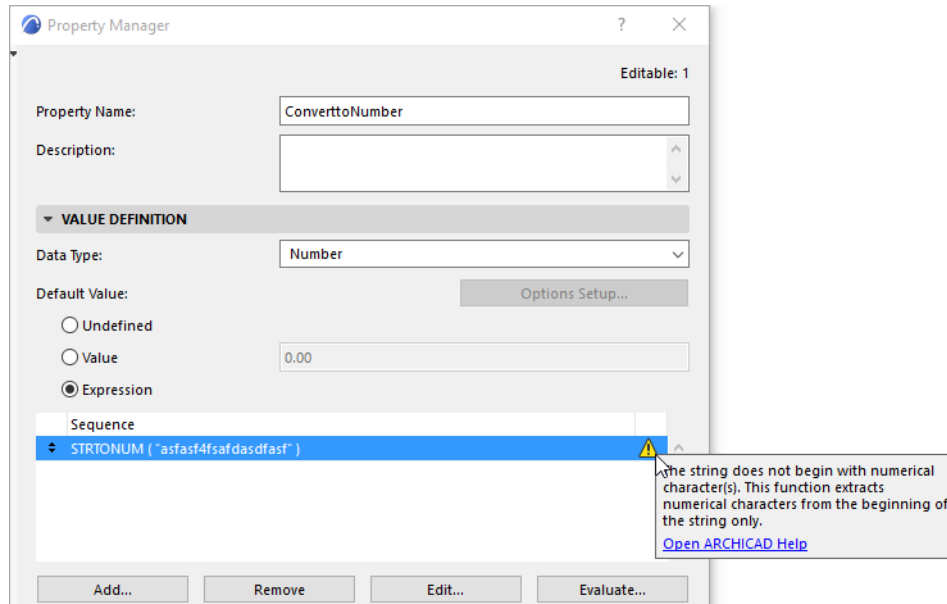
- For item(s) with a custom value assigned for this Property, revert the Property value to its default
 - Hint: Use Find & Select to locate such items: as the criterion, set "Property name", with "custom value"
- Alternatively: In Property Manager, create a duplicate Property and set its Data Type as needed
- In Teamwork, you cannot change a Property's data type if it was created by a different user.

Does Not Contain Numeric Characters

The `STRTONUM` function extracts numeric data from the beginning of a string, and returns the numbers as Number-type data.

This error message occurs if:

- the string provided contains no numeric characters, or
- the string's numeric characters are not at the beginning of the string



SOLUTION

Make sure the string provided contains numerical character(s) at the beginning of the string.

Properties in Hotlinks

It is recommended that everyone working on a particular project use the same set of Properties, and that one person be in charge of managing them.

Use Import/Export Properties to ensure that everyone uses the same Property set.

[See *Export/Import Properties Between Projects*.](#)

However, if a Hotlink Module does contain different properties than the host file, then you can manage them based on the following use cases:

Hotlink Module Contains New Properties

Hotlink Module Contains Identical Property, With Conflicting Definition

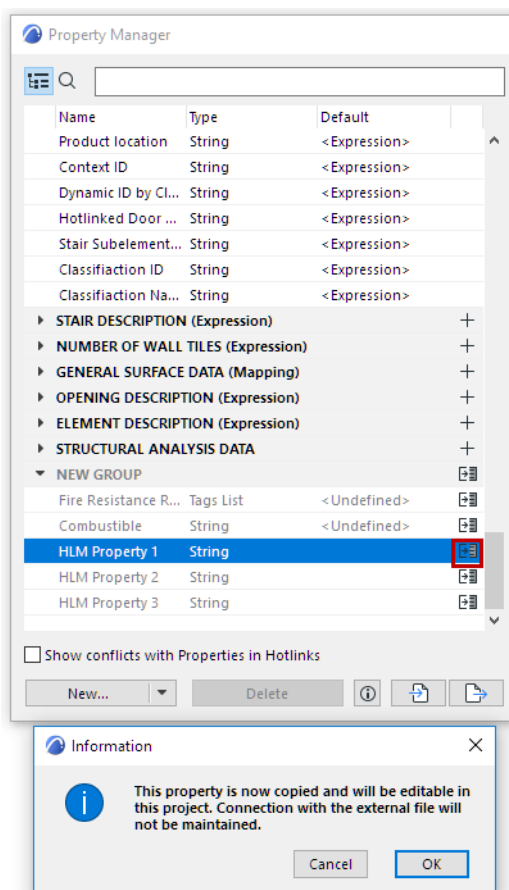
Hotlink Module Contains New Properties

If the Hotlink Module contains new properties (or groups) which don't exist in the host project, these are appended to the list in Property Manager, and are shown in gray.

Properties from Hotlink Modules

- Are searchable, like any property
- Can be scheduled, like any property
- **Cannot** be edited or deleted in the host project; they can be edited only in the external file.

For each such property or group, you have the option to merge it into the host project: click the Merge icon at the end of the row in Property Manager.



Once the property is merged, it becomes a fully editable part of the host project.

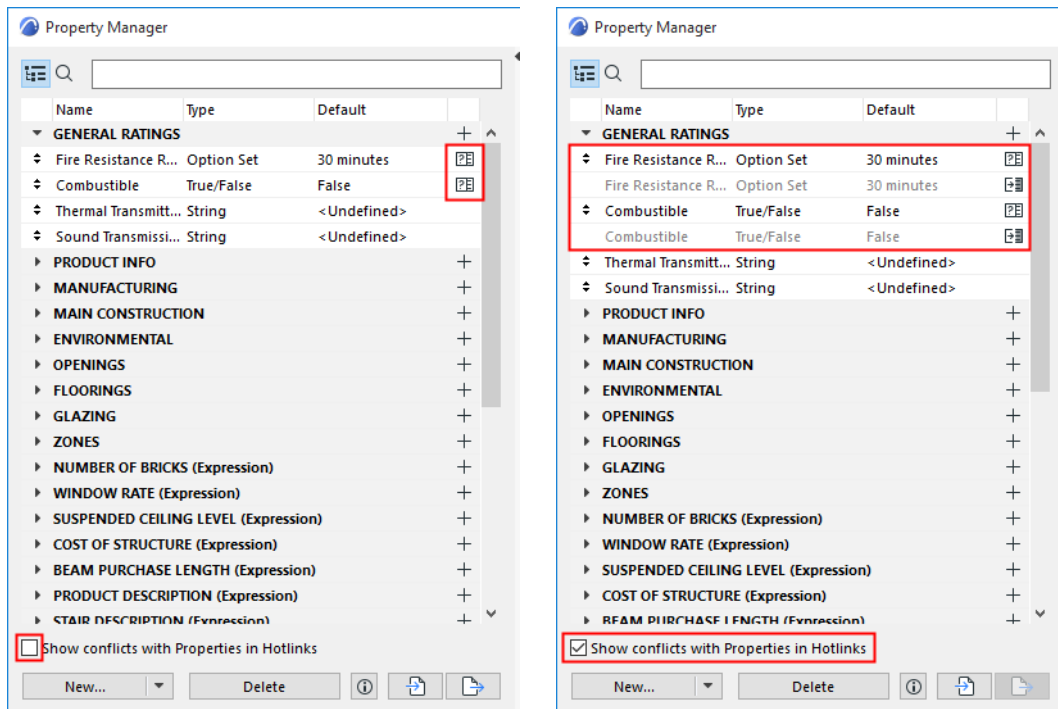
A merged property loses its connection to the external file: any changes to this property in the external file will not be reflected in the host file.

Hotlink Module Contains Identical Property, With Conflicting Definition

Properties are considered identical if they have the same name and are in the same group.

However, a Hotlink Module might contain an identical property as the host file, but with a conflicting property definition. For example, the two properties have the same name (e.g. “Fire Resistance Rating”), but with a different data type (e.g. Tag List instead of Option Set) or availability.

In this case, the Hotlink Module’s property is in conflict with that of the host file: the conflicting property is appended to the list in Property Manager, but is shown only if you check **Show conflicts with Properties in Hotlinks**.



For each such property or group, you can do one of the following:

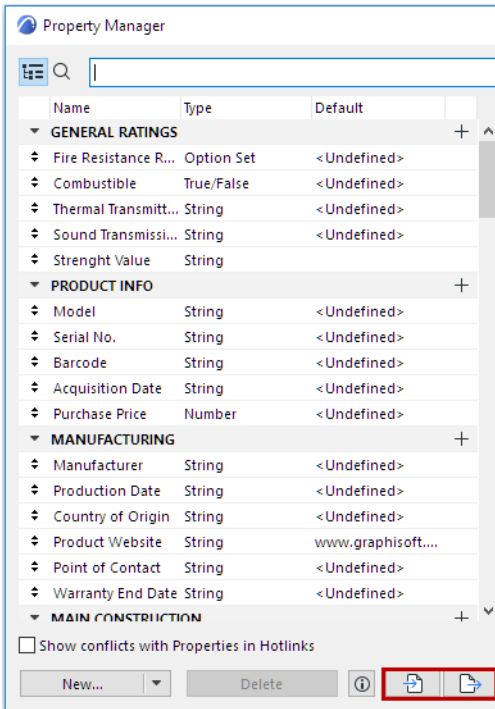
- Click the **Merge** icon at the end of the row in Property Manager. The property from the Hotlink Module, with its definitions, will *override* the corresponding property in the host file.
- Open the hotlink’s source file, and edit its property definition so that it matches that of the host file. Then, in the host file, update the Hotlink Module to eliminate the conflict.
- Do nothing. In this case, the hotlink property availability and value will be used in the hotlinked elements, and those of the source file will be used in the source file’s elements.
 - Exception: If the properties differ only in their data type, then the host project property’s data type will prevail. (The hotlinked element’s property will be converted to use the data type of the host project property).

[Watch the Video](#)

Export/Import Properties Between Projects

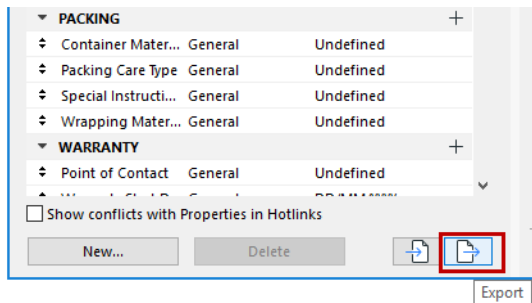
Use the buttons at the bottom of **Options > Property Manager** to export and import a project's Property set (either all properties or just selected ones) between Archicad Projects, from either an XML file or an Archicad file.

Exported data includes the Property Name, Group name, type, default value, and availability. A property's defined Option Set, if any, is also exported.

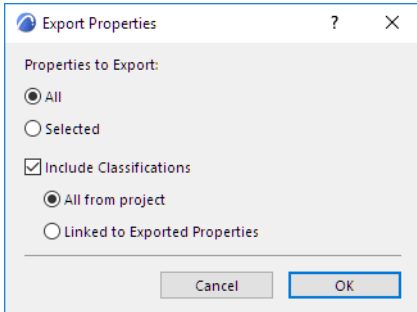


Export Properties

1. Select any properties or groups in Property Manager.
2. Click the **Export** button.



3. Set preferred options in the dialog box:



Include Classifications

Check this to export Classifications as well as Properties.

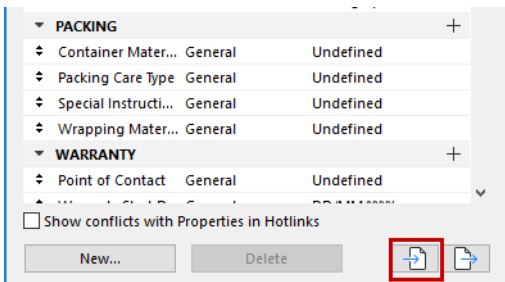
Choose a preferred export option:

- **All from Project:** Export all Classification Systems.
- **Linked to Exported Properties:** Export only those Systems having available properties among the properties you are exporting.

4. Click OK.

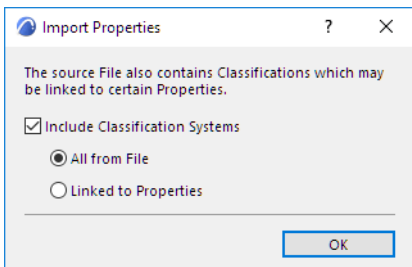
Import Properties

1. In Property Manager, click the **Import** button.



2. In the Open file dialog box, browse for the XML file or Archicad file that contains the properties you need, then click OK.

3. Set preferred options in the dialog box:

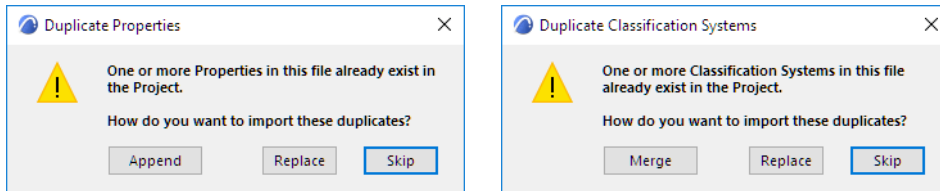


Include Classifications

- **All from Project:** Import all Classification Systems.
- **Linked to Exported Properties:** Import only those Systems having available properties among the properties you are importing.

4. Click OK.

If the imported properties, Classifications and/or Classification Systems already exist in the target project, a dialog box gives you a choice for how to handle them:



Duplicate Properties: If you choose **Append**, the duplicate properties are imported with a suffix added to their name.

Duplicate Classifications Systems: If you choose **Merge**, then incoming Classifications that do not yet exist in the project are added to the project. In case of duplicate Classifications, a dialog box again gives you a choice for how to handle them.

Import BIM Content

Property definitions are also imported to your project if you use the **Import BIM Content** or **Download BIM Content** commands.

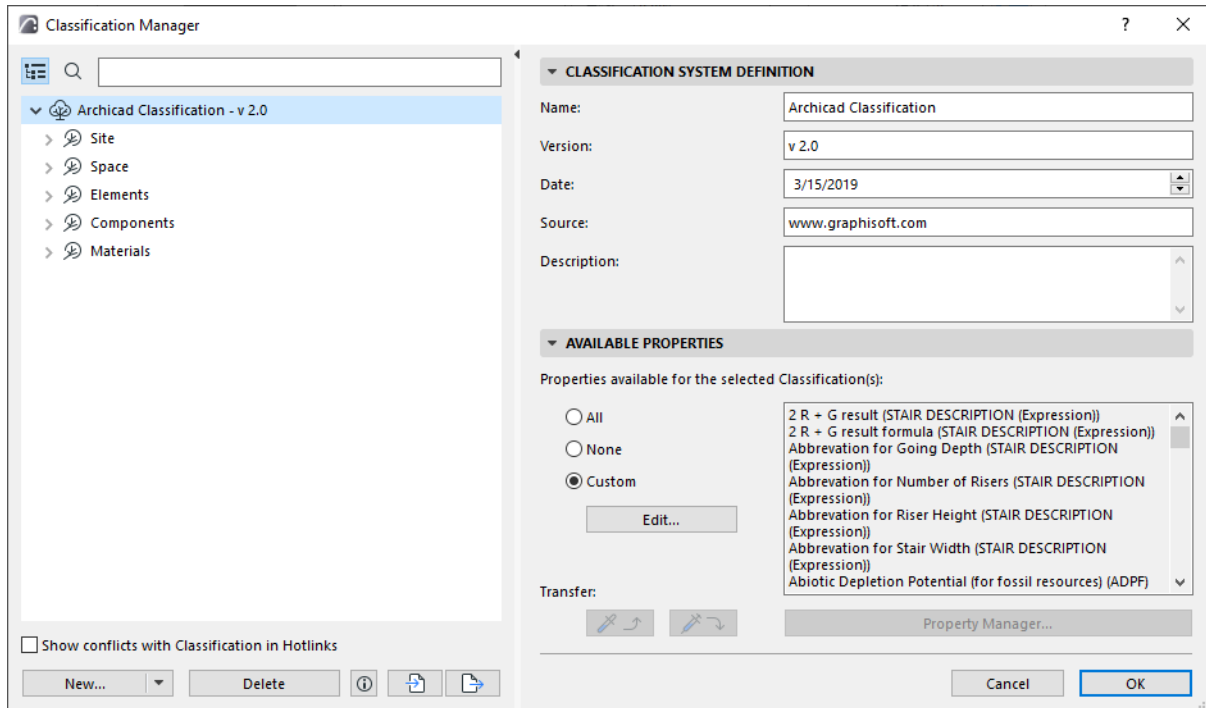
See [Import BIM Content](#).

Classification Manager

Use Classification Manager (**Options > Classification Manager**)

- To create and manage the Classification Systems and Classifications for this project
- To customize the set of properties available to each Classification

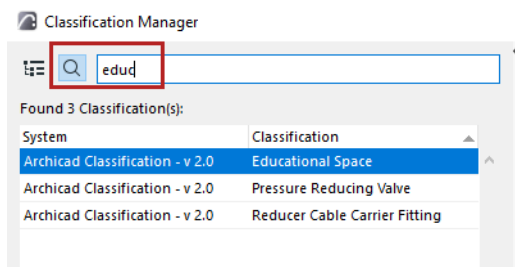
Classification Manager is also available from Property Manager.



The left side of the Classification Manager lists all the Classifications available in the project.

Search List

Click the **Search** icon, then start typing text into the search field. All classifications containing the search term are shown.

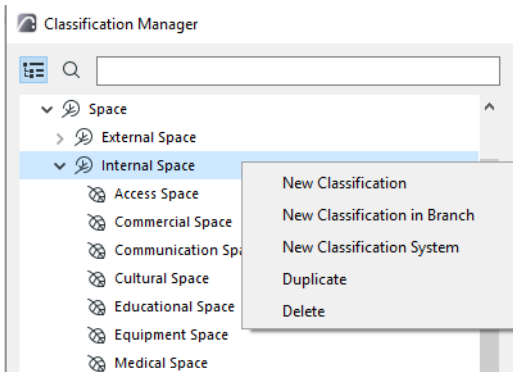


To view the full list again, delete the text from the Search field, or click the tree hierarchy icon.

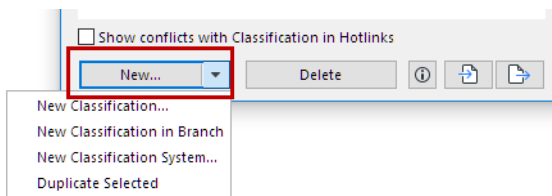
Create New Classification

1. Open the Classification Manager dialog box at **Options > Classification Manager**, or from Property Manager.
2. Select an item in the Classification list: the new classification will be created subordinate to the selected item.

3. Do one of the following:
 - Click anywhere in the Classification list and use the context menu

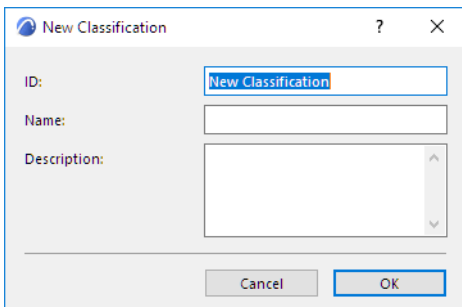


- Click the **New** button at the bottom of the dialog box



- Choose the hierarchical level of the new Classification, relative to what is selected in the tree:
 - **New Classification** adds it to the bottom of the tree list within the current System
 - **New Classification in Branch** adds it to the currently selected level of the tree
 - **New Classification System** adds the new System to the bottom of the tree

4. In the **New Classification** dialog box:
 - Define an ID, Name, and optional Description for the new Classification.



5. Click **OK** to create the new Classification.

[Watch the Video](#)

Duplicate Classification or System

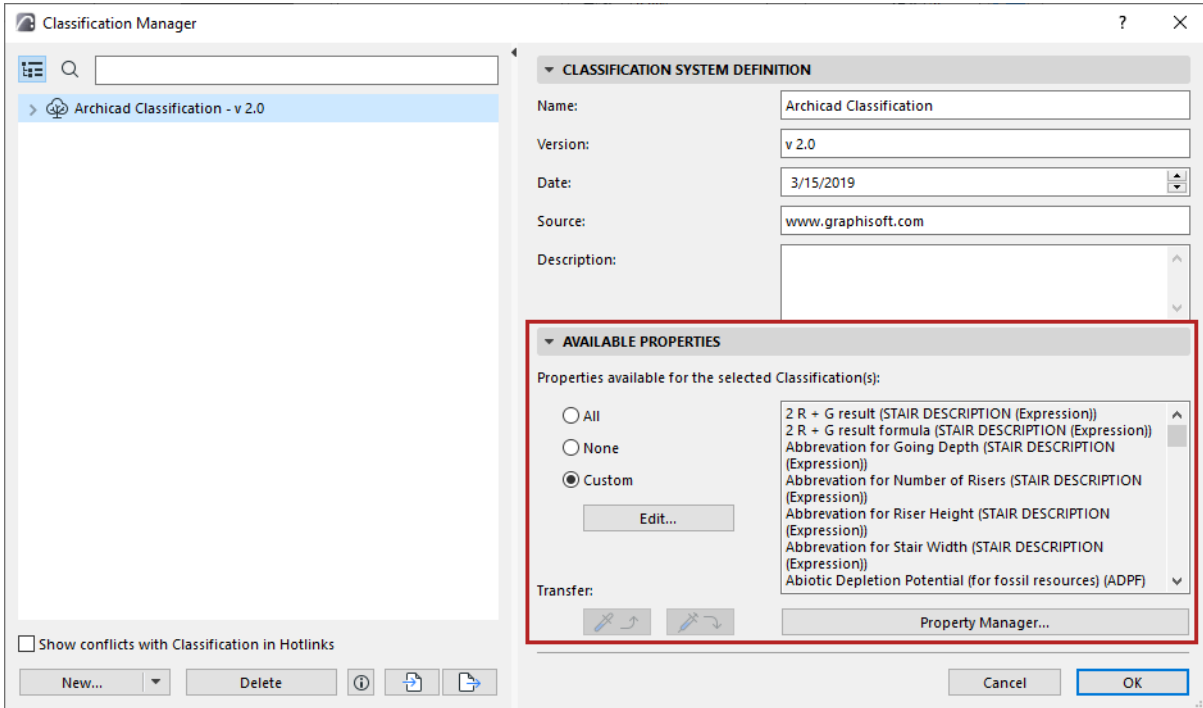
Select a Classification (or System) from the list in Classification Manager.

From the context menu or drop-down list under the New button, click **Duplicate** to create a duplicate, then adjust its settings.

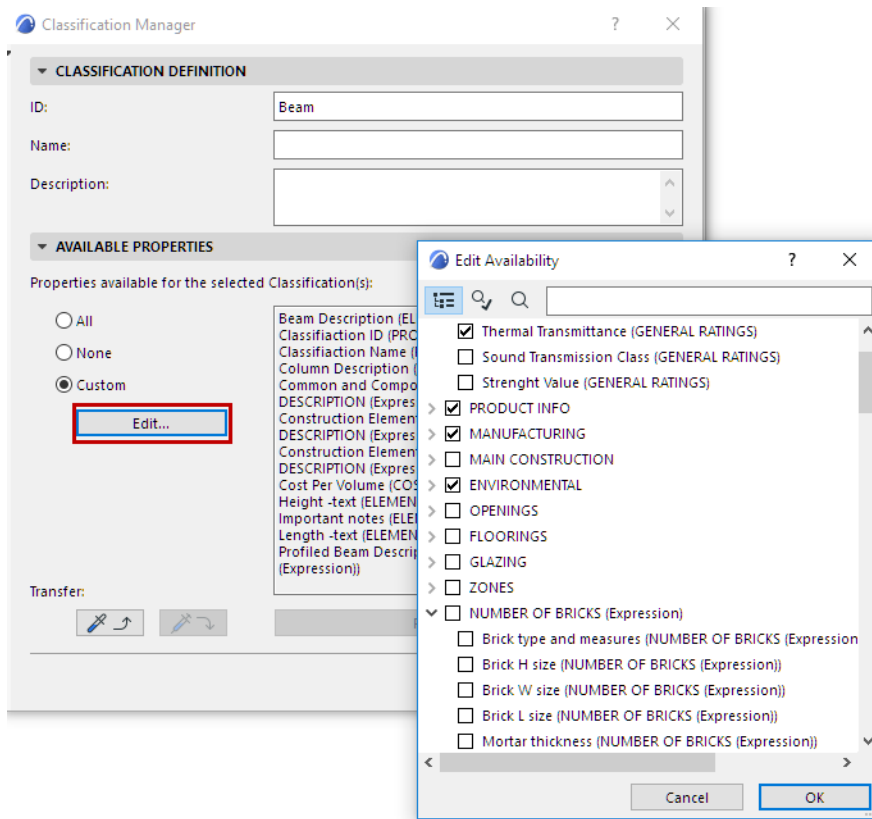
Available Properties

Use these controls to define which properties should be available for a given Classification.

- Select one or more Classifications from the list on the left.



- **All:** All properties in the project are available for the selected classification(s).
- **None:** No properties are available for the selected classification(s).
- **Custom:** Click Custom, then click **Edit** to select the Properties to be available for the selected classification(s).



See also [Transfer Availability](#).

Import/Export Classifications

Use the buttons at the bottom of **Options > Classification Manager** to export and import a project’s set of Classifications (either all classifications or just selected ones) between Archicad Projects, from either an XML file or an Archicad file.

The process and options are analogous to Import/Export Properties.

See [Export/Import Properties Between Projects](#).

Import BIM Content

Classifications, with associated Property and IFC Mapping definitions, are also imported to your project if you use the **Import BIM Content** or **Download BIM Content** commands.

See [Import BIM Content](#).

Library Part Maker

Note: Library Part Maker is available for SSA/Forward customers only.

Library Part Maker (LPM) is an Archicad add-on that simplifies and streamlines the creation of custom GDL-based Library Parts.

Install Library Part Maker from the [Downloads](#) page at graphisoft.com

Open the Library Part Maker palette using one of these commands:

- Design > Design Extras > Library Part Maker
- Window > Palettes > Library Part Maker

Introduction

The Library Part Maker workflow is based on the use of standard 2D and 3D Archicad tools and does not require any GDL programming knowledge. The LPM approach focuses on 'real-life' manufactured products, in which geometry and information are firmly tied together (e.g. a certain type of product has always a unique bar code number). Yet LPM allows for parametric 2D/3D (LOD-based) and parametric CAD (Archicad Attribute-based) 2D symbols and 3D models. Also, LPM greatly simplifies creation and design of complex doors and windows in Archicad.

Scope

LPM provides tools for creating the following GDL Library Part types:

- Generic Object
- Window
- Door
- Skylight
- Lamp
- MEP Library Parts (fully functional with MEP Library and the MEP Modeler Add-on).

Migration

LPM for Archicad 25 removed the layer-based functionality of previous LPM releases and replaced it with the Assignment-based concept. This concept removes the rigid organization of elements by layers, a process that could be error prone.

The new workflow is based on the upgraded and integrated Library Part Palette.

Any content created by LPM release prior to Archicad 25 will be automatically migrated to appropriate assignments. These elements can be freely moved to any layer.

The creation of Handles and Shutter accessories as separate types has been discontinued. These Library Parts are covered by a standard Archicad functionality in File > Libraries and Objects > Save Selection As.

Related Topics:

[Library Part Maker Palette](#)

[Library Part Maker: Detailed workflow](#)

[Library Part Assignments](#)

[Customizing Data Properties](#)

Library Part Maker Palette

Note: Part of the Library Part Maker add-on. See [Library Part Maker](#).

All commands needed for creating a Library Part are available on a single Library Part Maker palette.

Open Library Part Maker Palette

Open the palette using one of these commands:

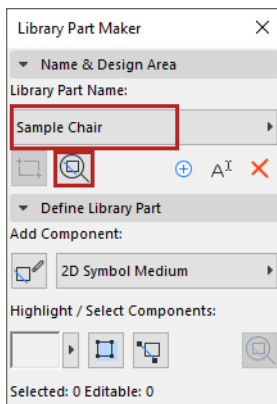
- Design > Design Extras > Library Part Maker
- Window > Palettes > Library Part Maker

Selection and Highlight

Use the Palette icons to help you identify and select to particular Library Parts and components:

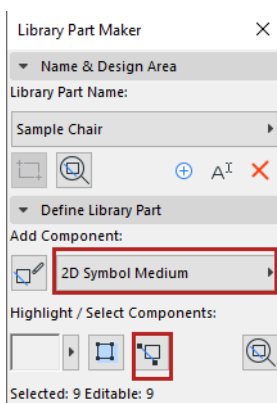
Zoom to a Design Area

1. Select a Library Part Name from the pop-up.
2. Click the Zoom button to zoom to its Design Area on the Floor Plan



Select Components by Type

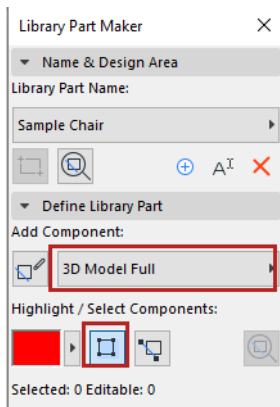
1. Select a Component Type from the pop-up (e.g. 2D Symbol Medium)
2. Click the **Select Components** button.



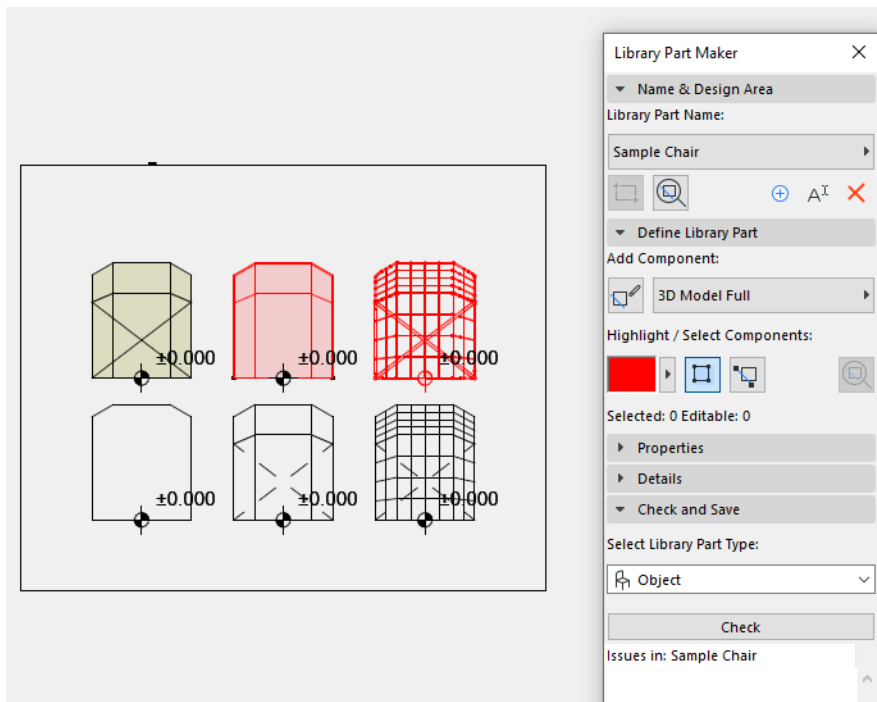
All Components of this type are now selected on the plan.

Highlight Components by Type

1. Select a Component Type from the pop-up (e.g. 3D Model Full)
2. Click the **Highlight Components** toggle button.



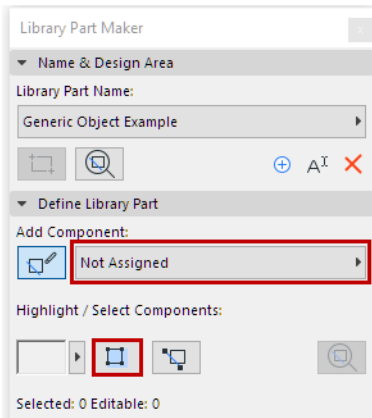
All Components of the selected type are now highlighted on the plan.



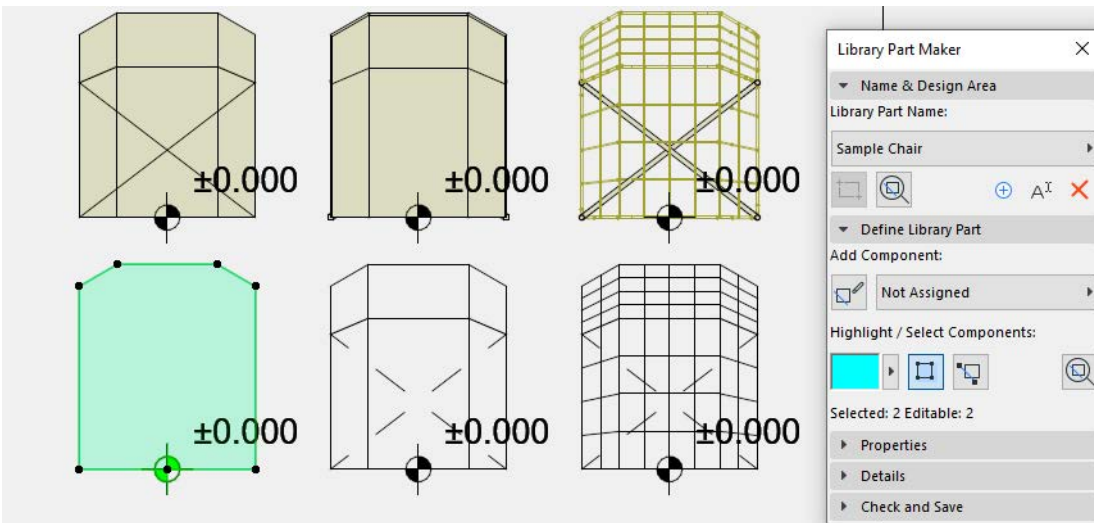
To remove the highlight, click the **Highlight Components** button again.

Highlight Unassigned Components

- Select the “Not Assigned” Component Type from the pop-up
- Click the **Highlight Components** toggle button.



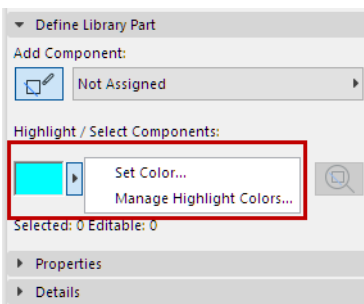
All Components on the plan that have not been assigned a type are now highlighted.



To remove the highlight, click the **Highlight Components** button again.

Set Highlight Colors

Click the pop-up next to the Highlight preview to access its color setting interface. You can set a separate color for not assigned components, as well as Transparency (this is useful in the 3D window).



Library Part Maker: Detailed workflow

Use the controls of Library Part Palette from top to bottom to design your new Library Part.

The screenshot shows the Library Part Maker interface with the following steps highlighted by numbered callouts:

- 1. Add new Library Part Name**: Points to the 'Library Part Name' input field containing 'My Sample Object'.
- 2. Define the Design Area for your Library Part**
Mark the workspace in which you will design the Library Part Components: Points to the design area selection tools (rectangle, circle, and zoom).
- 3. Start adding components for each Detail Level**
Draw the 2D and 3D model symbols: Points to the 'Add Component' dropdown menu showing '2D Symbol Full'.
- 4. Select Property Scheme**
Which listing parameters should the Library Part use?: Points to the 'Import properties from' radio buttons, with 'Property Scheme' selected.
- 5. Add Details**
Define a Preview, add other info... don't forget the author!: Points to the 'Details' section, specifically the 'Library Part Name' and 'Author' (Renata) fields.
- 6. Specify Library Part Type**
Is it a Door, a Window, an Object?: Points to the 'Select Library Part Type' dropdown menu, which is set to 'Object'.
- 7. Check your Library Part**
Verify your creation, and fix any issues.: Points to the 'Check' button and the 'Issues in: My Sample Object' section, which shows 'No issues found'.
- 8. Customize Parameter and Attributes**
Open the pop-up, where you can refine parameters and attributes (names, values, defaults): Points to the 'Parameters' button at the bottom of the interface.
- 9. Save it (Finally!)**
And it's ready to place.: Points to the 'Save As' button at the very bottom.

Each step is explained in the sections that follow.

Define Name & Design Area

Add Library Part Components

Component Assignments

Select Property Scheme

Add Library Part Details

Select Library Part Type

Check Library Part

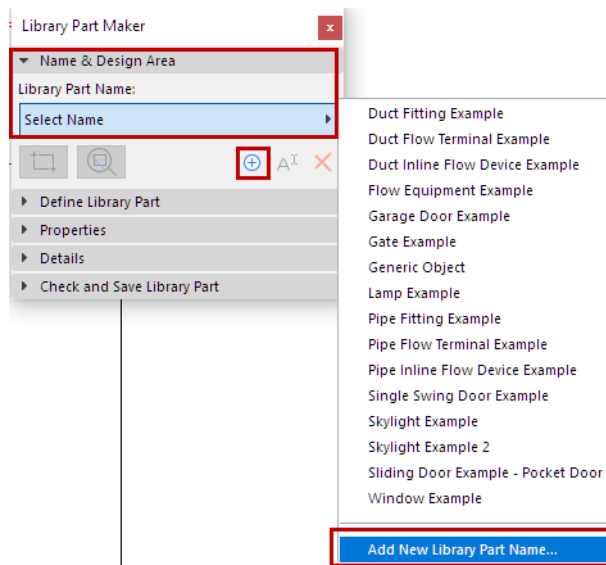
Save Library Part

Define Name & Design Area

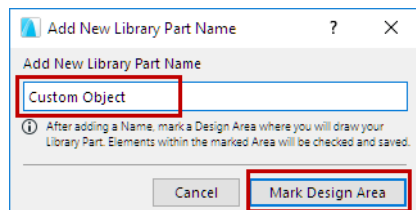
Note: Part of the Library Part Maker add-on. See [Library Part Maker](#).

The first step is to create a Library Part Name and draw its Design Area on the Floor Plan.

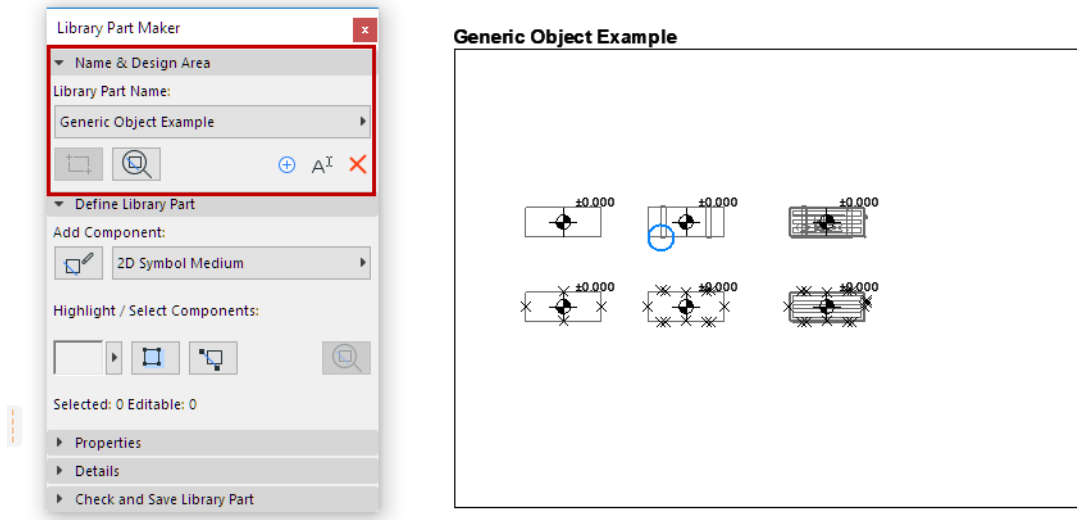
1. In the Library Part Maker palette, do one of the following:
 - Click the Plus sign.
 - From the LibPart Name pop-up, choose **Add New Library Part Name**.



2. In the appearing dialog, assign a name.
3. Click **Mark Design Area**.



- Click on the Floor Plan to draw a Design Area frame. The name is displayed next to the frame automatically.



You can also create new Design Areas by copy-pasting an existing Design Area (e.g. in a matrix). Each copy is automatically given a new name and listed separately in the Palette's pop-up.

What is a Design Area?

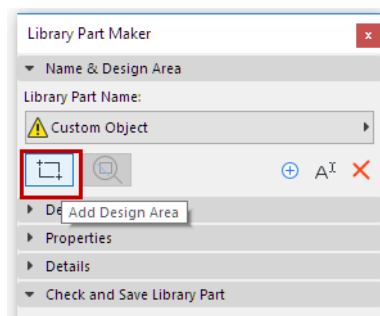
- The Design Area contains the 2D and 3D elements (components) which constitute the Library Part. The Design Area groups these components and their attributes, and preserves any modifications you make along the way.
- The Design Area can be added before or after creating the current Library Part.
- The Check and Save function considers the current Design Area content when checking the validity of the library part.
- Deselecting the Design Area or any of its parts will not result in lost changes.

See also [Zoom to a Design Area](#).

Add New Design Area

If the Design Area frame is deleted, the Library Part Name is still available in the palette, but you will not be able to save it: when you run a check, you will get a warning.

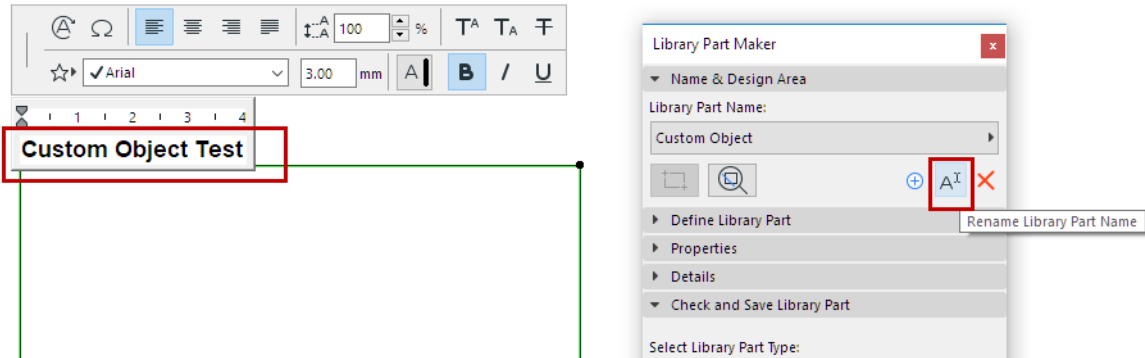
To draw a new Design Area, click the **Add Design Area** icon and draw it again.



Rename Library Part Name

Do one of the following:

- Edit the Library Part name directly on the Floor Plan
- Click the **Rename** button on the palette.



Add Library Part Components

Note: Part of the Library Part Maker add-on. See [Library Part Maker](#).

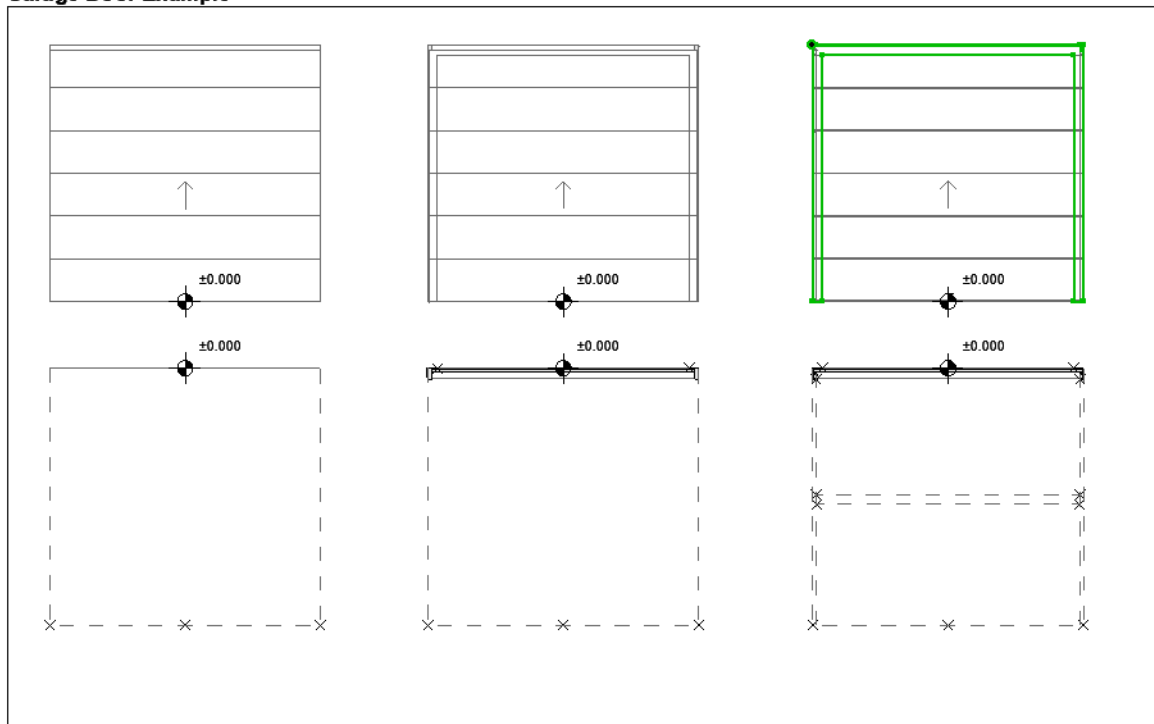
Inside the Design Area, use 2D and 3D tools to draw the components of the Library Part.

Note: It is recommended to consolidate and check the Archicad tools' attribute settings before you draw or model Library Part definitions. This will help you avoid unwanted attributes as parameters in the Library Part user interface. Setting up your tool attribute settings in the Favorites palette is the best way to minimize errors and avoid the laborious process of checking the tool attribute settings every time.

The following example shows the 3D model components in the top row (with LOD increasing from left to right), and the 2D symbols in the bottom row.

Note: To focus on a particular Design Area on the Floor Plan, use the Zoom function from the LPM Palette: [see Zoom to a Design Area](#).

Garage Door Example



Design Principles for 2D Symbols

- Draw 2D symbol(s) using any 2D Archicad tool (except Text tool).
For a door/window, the 2D symbol orientation must be aligned with horizontal axis (x). The external face of opening should face up (see [Examples of 2D Symbol Level of Detail](#)).
- Use Hotspot Tool to define custom 2D hotspots (effective for 2D Symbols only)
- Assign a content origin to each component (see [Define Component Origins \(Place Level Dimensions\)](#))
- Any Library Parts used in your 2D symbols should be exploded to 2D primitives (lines, arcs, polylines etc.)

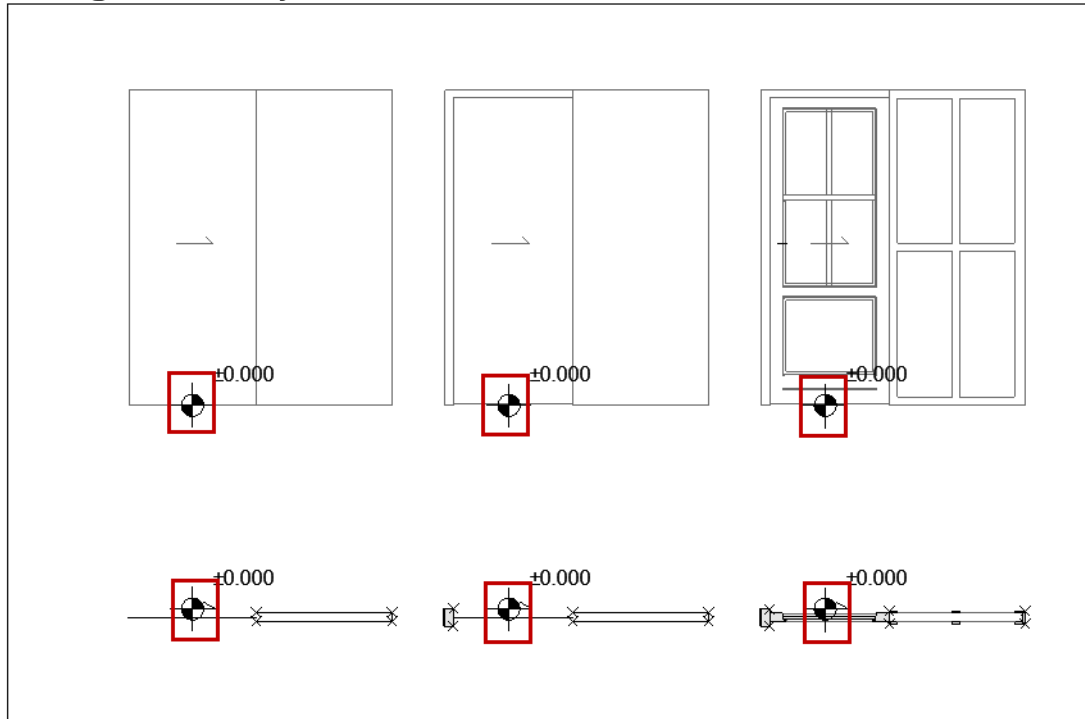
Define Component Origins (Place Level Dimensions)

For each component (both 2D Symbol and 3D Model), use the Level Dimension Tool to place a Content Origin. This is used to align the 2D Symbol and 3D Model content when the Library Part is saved.

The origin can be placed at any position as long as it is aligned with other component origins. The recommended approach is to place the origin in the center of an object, or (in case of irregular 2D symbols and 3D models) its bounding box.

The origin must be assigned a component type for each used 2D Symbol and 3D model.

Sliding Door Example - Pocket Door

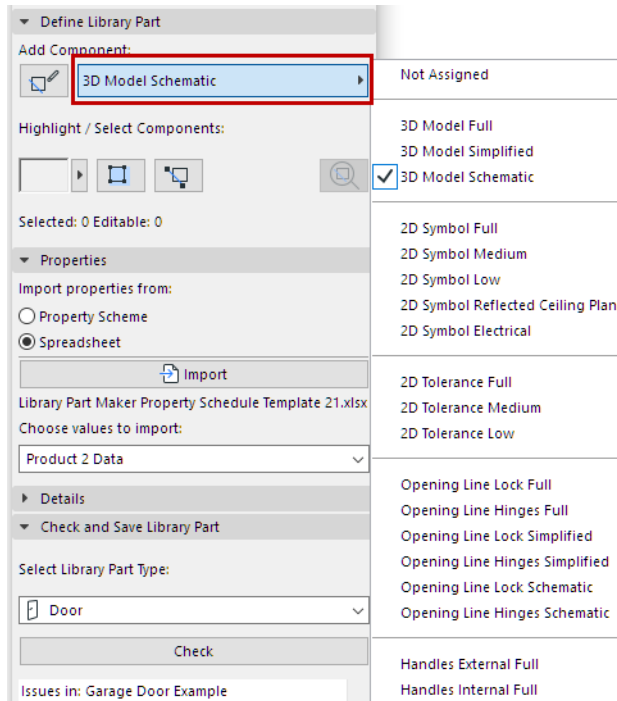


See [Door/Window Component Origins](#).

Component Assignments

Note: Part of the Library Part Maker add-on. See [Library Part Maker](#).

For each component, assign its Level of Detail (LOD) using the Component Type pop-up on the LPM Palette. Each type represents a 2D or 3D representation at a particular detail level:



See also [Highlight Components by Type](#).

Once the Library Part is saved and placed, these assigned LOD representations can be controlled for each placed instance – either from its Settings dialog, or from Archicad’s Model View Options.

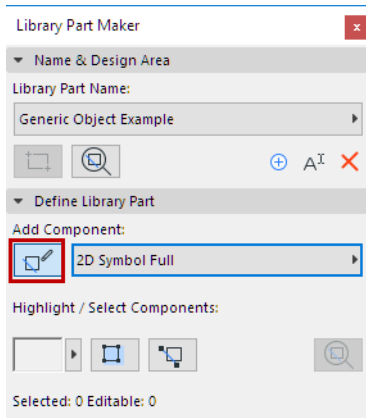
It is possible to skip the LOD definition for some components; these components have a definition of “Not Assigned”. (When you check the Library Part, you get a warning of unassigned components, but you can still save it.) When placed, such Library Parts will use only the defined representation. However, this practice is not recommended as it can lead to unexpected 2D/3D representations of Library Part in documentation and 3D model views with certain scale or Model View Options settings.

See also [Highlight Unassigned Components](#).

Add Components Using Current Type

To create multiple components with a particular Component type:

1. Choose the desired Component type from the pop-up (e.g. 2D Symbol Full)
2. Click the **Start/Stop Adding Components** toggle button.



All the components you add now will automatically be assigned the current Component type, until you click the toggle again.

Assignments for 3D Model Components

LPM Assignment	Content shown with corresponding Model View Option (MVO) at:
	<ul style="list-style-type: none"> • Detail Level of Door, Window and Skylight Symbols • Miscellaneous Settings for Library Parts
3D Model Full	3D and Section/Elevation: Full
3D Model Simplified	3D and Section/Elevation: Simplified
3D Model Schematic	3D and Section/Elevation: Schematic

Assignments for 2D Symbol Components

LPM Assignment	Content shown with corresponding Model View Option (MVO) at:
	Miscellaneous Settings for Library Parts
2D Symbol Full	Floor Plan Realistic Full
2D Symbol Medium	Floor Plan Realistic Medium
2D Symbol Low	Floor Plan Realistic Low
2D Symbol Electrical	Lighting and HVAC Floor Plan Symbol: Electrical
2D Symbol Reflected Ceiling Plan	Lighting and HVAC Floor Plan Symbol: Reflected Ceiling Plan

Please note that 2D Electrical and Reflected Ceiling Plan assignments are ignored for MEP objects.

Select Property Scheme

Note: Part of the Library Part Maker add-on. See [Library Part Maker](#).

The new Library Part must include a set of listing properties. These listing properties will be available in the Descriptions area (Custom Settings panel) in the Settings dialog of the saved Library Part.

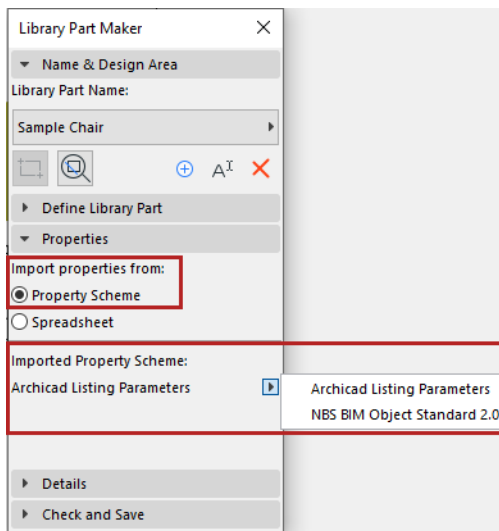
To define the Library Part's listing properties:

In the Properties panel, select an option to define the Library Part's listing properties:

- Property Scheme
- Spreadsheet

Default Property Scheme

To populate the list for newly created objects, Library Part Maker uses the “Archicad Listing Parameters scheme” by default.



- You can create any custom parameter scheme instead.
- A custom scheme together with defined values can be imported directly from an Excel spreadsheet.

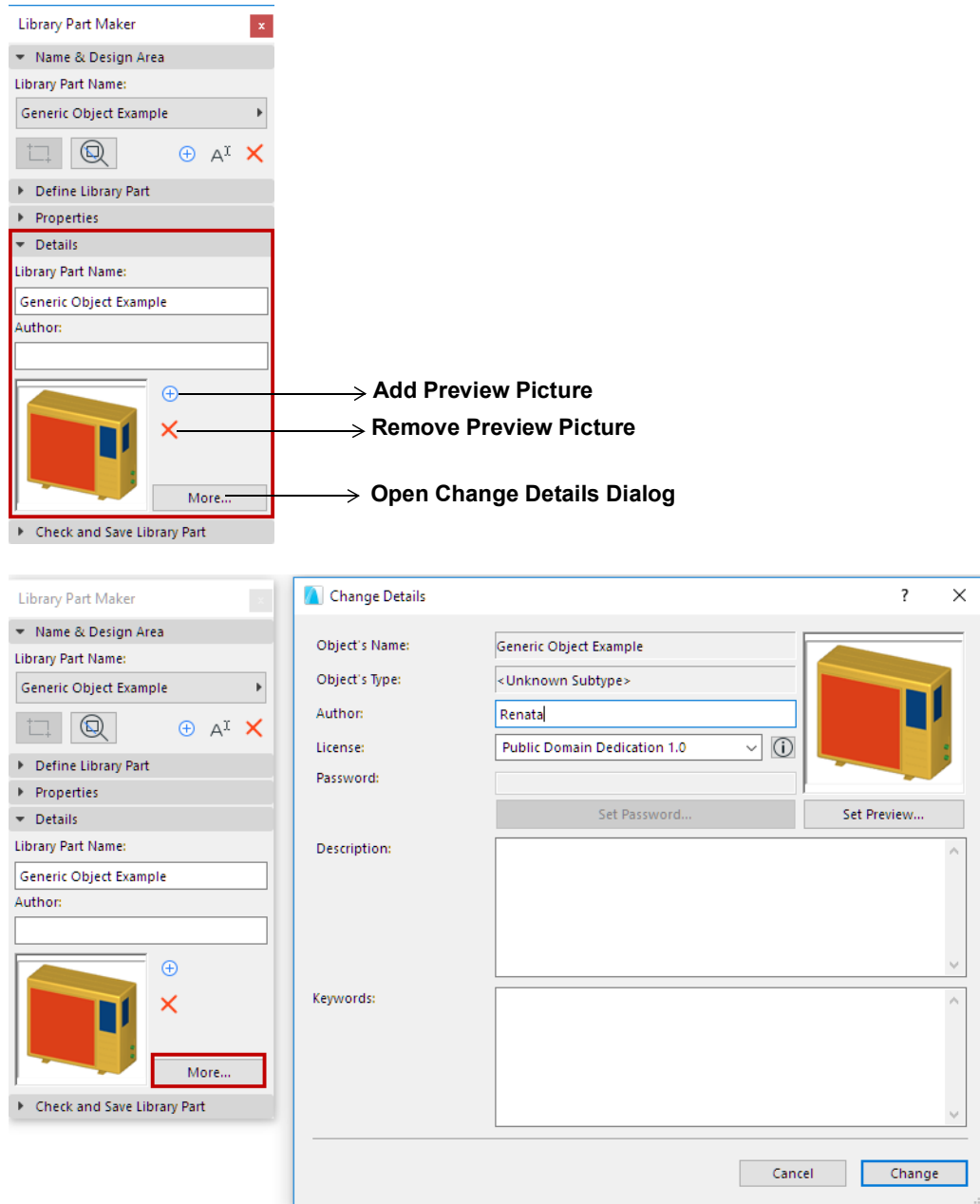
See [Customizing Data Properties](#) for details on creating or importing a custom scheme.

Add Library Part Details

Note: Part of the Library Part Maker add-on. See [Library Part Maker](#).

In the Details panel, add optional authoring and other metadata (Author, License, Password for script locking, Preview image, Description and Keywords).

Click **More** to open the Change Details dialog, with all available options.

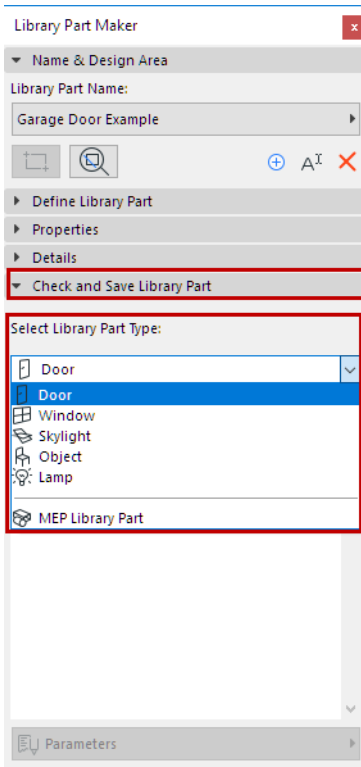


Select Library Part Type

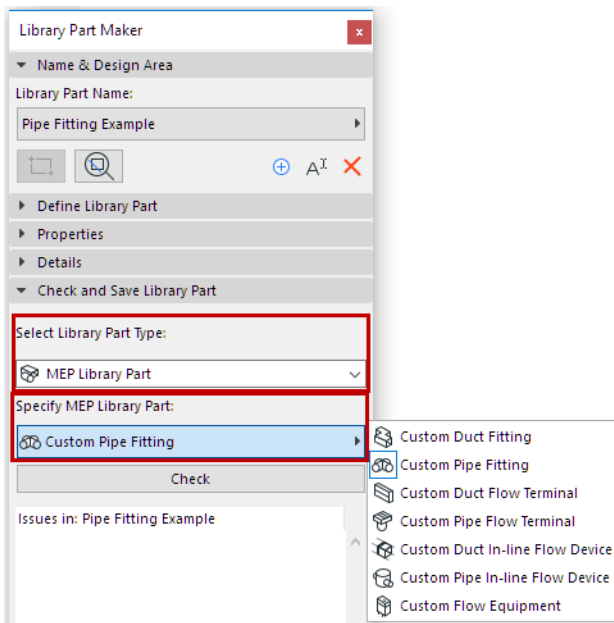
Note: Part of the Library Part Maker add-on. See [Library Part Maker](#).

In the Check and Save Library Part panel:

Choose an option from the **Select Library Part Type** drop-down.



If you choose MEP Library Part, an additional pop-up appears: You must further specify the MEP type.



Check Library Part

Note: Part of the Library Part Maker add-on. See [Library Part Maker](#).

You cannot save the current Library Part until you check it.

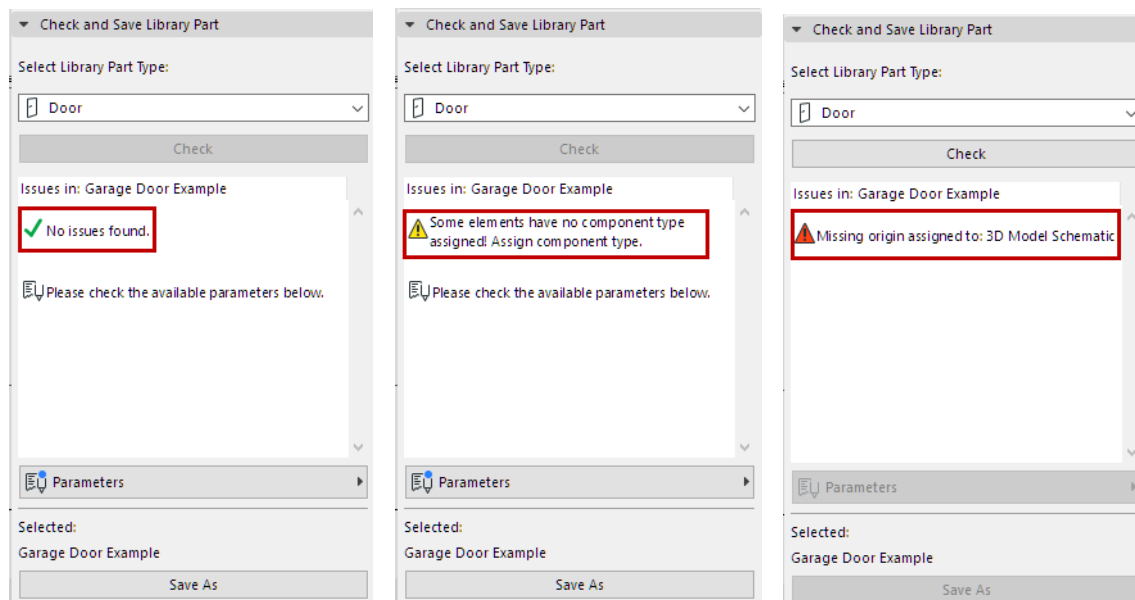
In the Check and Save Library Part panel, press **Check**.

The check is run on the Design Area of the current Library Part Name.

Review Issues

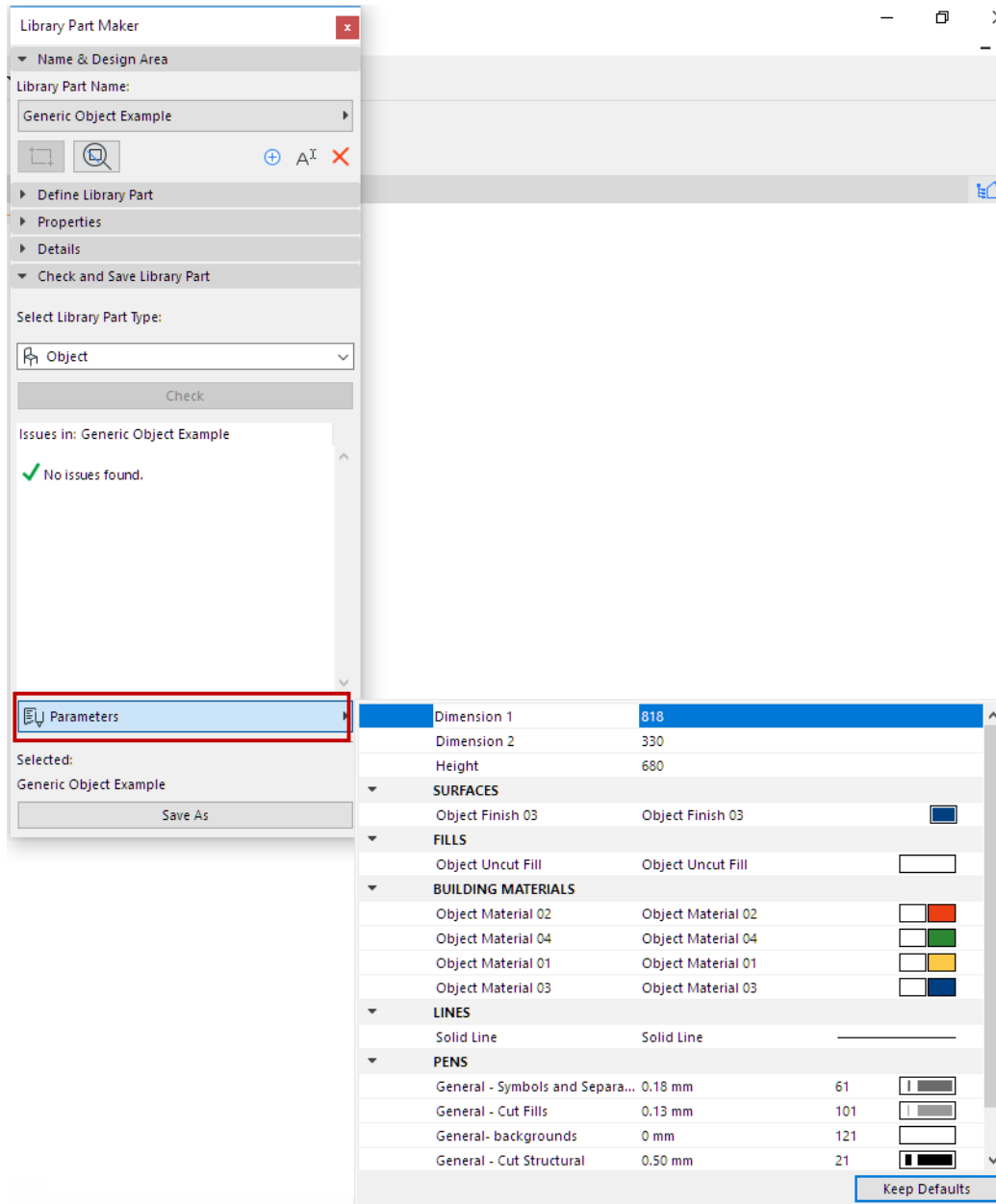
Review the displayed issues (alerts, warnings) that could affect or prevent saving of Library Part. Correct any issues that require it.

- **Green checkmark:** No issues found. Review the parameters, then Save.
- **Yellow warning:** This calls your attention to an anomaly, but this does not prevent you from saving the Library Part. Review the parameters, then Save.
- **Red warning:** This is an error which must be fixed before you can continue.



Adjust Parameters and Attributes

Once the Check produces no warnings that require fixing, the Parameters pop-up is available. Here, you see the parameters and attributes that will be saved with the current Library Part.



The attributes (line types, pens, fills, surfaces, building materials) are those used in the 2D symbols and 3D models of the Design Area. The parameter names are automatically derived from the name of the current project Attributes.

You can edit the names and values here. The Design Area will remember the parameter changes you make.

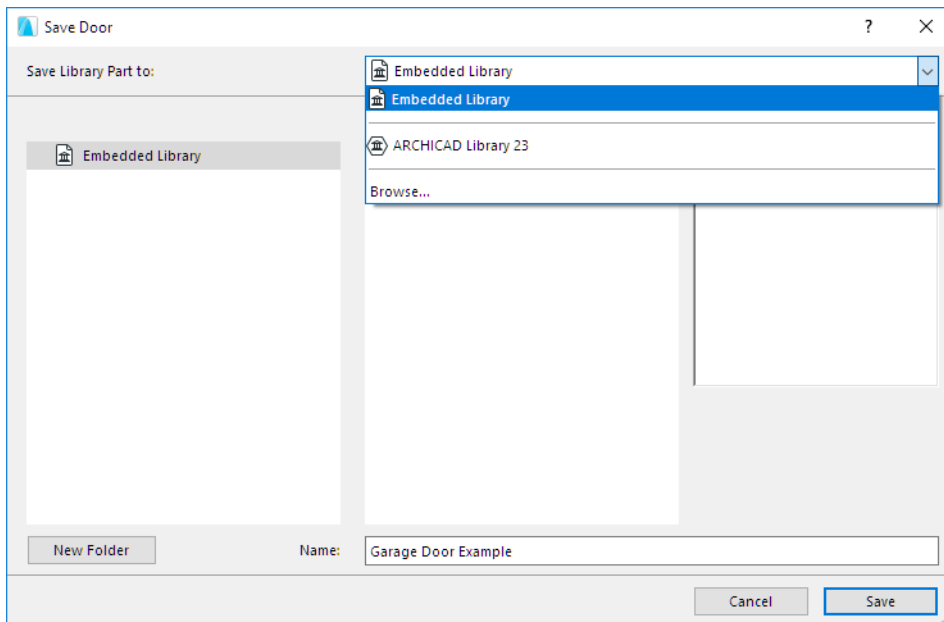
Save Library Part

Note: Part of the Library Part Maker add-on. See [Library Part Maker](#).

Click **Save as**.

Note: This command is only available after you have run the Check, and provided that the Check produced no warnings that require fixing. (See previous section.)

In the Save dialog which appears, enter a name and choose the desired folder location for the object:



- to the Embedded Library (default choice. It is recommended to save customized, project-specific objects to your Embedded Library.)
- browse for a linked library or a BIMcloud library that has been added to the project

Note: In Teamwork, you must reserve the BIMcloud Library folder before saving to it.

Library Part Assignments

Note: Part of the Library Part Maker add-on. See [Library Part Maker](#).

The following sections list the relevant Assignments for each Library Part type. In addition, they provide more details on what to consider when creating Library Parts of a particular type.

[Creating Objects](#)

[Creating Lamps](#)

[Creating MEP Objects](#)

[Creating Doors and Windows](#)

[Creating Skylights](#)

[Overview of LOD Assignments](#)

[LOD Recommendations](#)

Creating Objects

Note: Part of the Library Part Maker add-on. See [Library Part Maker](#).

Level of Detail Definition for Objects (similar to Lamps)

These tables specify how the placed object will be displayed depending on its Object Settings (Symbol Type, Detail Level, etc.)

The Default LOD is taken from the Library Part Maker assignments. If a certain assignment is undefined, the “Substitute” value is used.

Selected Symbol Type	Default LOD	Substitute LOD 1
Realistic	Selected 2D Detail Level	---
Electrical	Electrical Symbol	Selected 2D Detail Level
Reflected Ceiling Plan	Reflected Ceiling Plan Symbol	Selected 2D Detail Level

Selected 2D Detail Level	Default LOD	Substitute LOD 1	Substitute LOD 2	Substitute LOD 3
Full	Full	Medium	Low	System 2D bounding box
Medium	Medium	Low	Full	System 2D bounding box
Low	Low	Medium	Full	System 2D bounding box

Selected 3D Detail Level	Default LOD	Substitute LOD 1	Substitute LOD 2	Substitute LOD 3
Full	Full	Simplified	Schematic	3D not shown
Simplified	Simplified	Schematic	Full	3D not shown
Schematic	Schematic	Simplified	Full	3D not shown

System predefined parameters for Objects

Dimension 1 – Overall object size along X axis

Dimension 2 – Overall object size along Y axis

Height – Overall object size along Z axis

Creating Lamps

Note: Part of the Library Part Maker add-on. See [Library Part Maker](#).

Level of Detail definition of Lamps (similar to Objects)

Selected Symbol Type	Default LOD	Substitute LOD 1
Realistic	Selected 2D Detail Level	---
Electrical	Electrical Symbol	Selected 2D Detail Level
Reflected Ceiling Plan	Reflected Ceiling Plan Symbol	Selected 2D Detail Level

Selected 2D Detail Level	Default LOD	Substitute LOD 1	Substitute LOD 2	Substitute LOD 3
Full	Full	Medium	Low	System 2D bounding box
Medium	Medium	Low	Full	System 2D bounding box
Low	Low	Medium	Full	System 2D bounding box

Selected 3D Detail Level	Default LOD	Substitute LOD 1	Substitute LOD 2	Substitute LOD 3
Full	Full	Simplified	Schematic	3D not shown
Simplified	Simplified	Schematic	Full	3D not shown
Schematic	Schematic	Simplified	Full	3D not shown

Additional components/assignments

At least one light source, with an assignment of 3D Model Full, must be defined for each Lamp-type Library Part.

Light source Library Parts are located at **Archicad Library > General Light Sources**.

Creating MEP Objects

Note: Part of the Library Part Maker add-on. See [Library Part Maker](#).

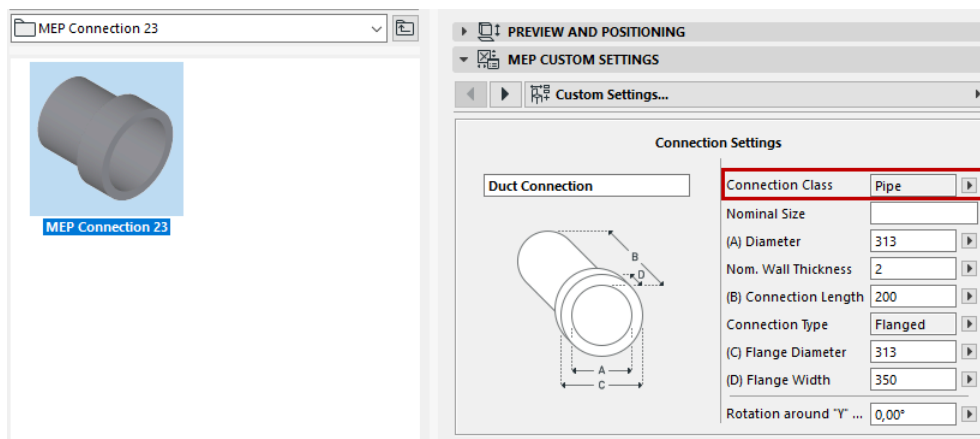
Level of Detail definition of MEP Objects

Selected 2D Detail Level	Default LOD	Substitute LOD 1	Substitute LOD 2	Substitute LOD 3
Full	Full	Medium	Low	System 2D bounding box
Medium	Medium	Low	Full	System 2D bounding box
Low	Low	Medium	Full	System 2D bounding box

Selected 3D Detail Level	Default LOD	Substitute LOD 1	Substitute LOD 2	Substitute LOD 3
Full	Full	Simplified	Schematic	3D not shown
Simplified	Simplified	Schematic	Full	3D not shown
Schematic	Schematic	Simplified	Full	3D not shown

Additional components/assignments

- For MEP objects, 3D Representation requires the use of the MEP Connection object (which can be found in MEP Library > MEP Connection), in order to facilitate the automatic connectivity of MEP objects.
- This MEP Connection component must be assigned to 3D Model Full.
- For Duct/Pipe Inline Flow Library Parts, there must be at least 2 MEP Connections, assigned to 3D Model Full. These must be placed on a single axis facing in opposite directions.
- When creating a Duct or Pipe MEP Library Part, make sure that the MEP Connection component is assigned the corresponding Connection Class: Duct or Pipe.



Creating Doors and Windows

Note: Part of the Library Part Maker add-on. See [Library Part Maker](#).

Level of Detail definition of Doors and Windows

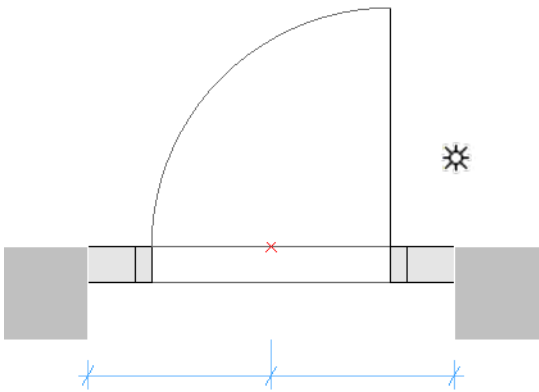
Selected 2D Detail Level	Default LOD	Substitute LOD 1	Substitute LOD 2	Substitute LOD 3
Full	Full	Medium	Low	System empty opening
Medium	Medium	Low	Full	System empty opening
Low	Low	Medium	Full	System empty opening

Selected 3D Detail Level	Default LOD	Substitute LOD 1	Substitute LOD 2	Substitute LOD 3
Full	Full	Simplified	Schematic	System empty opening
Simplified	Simplified	Schematic	Full	System empty opening
Schematic	Schematic	Simplified	Full	System empty opening

Door/Window Component Origins

A special methodology applies in case of Door and Window Library Parts.

- Component origins for 2D door/window symbols must be aligned with the center of door/window wall opening and the external edge of wall they will be placed in.
- Component origins for 3D door/window models must be aligned with the bottom center of door/window wall opening.



Door Symbol with Component Origin at Hotspot

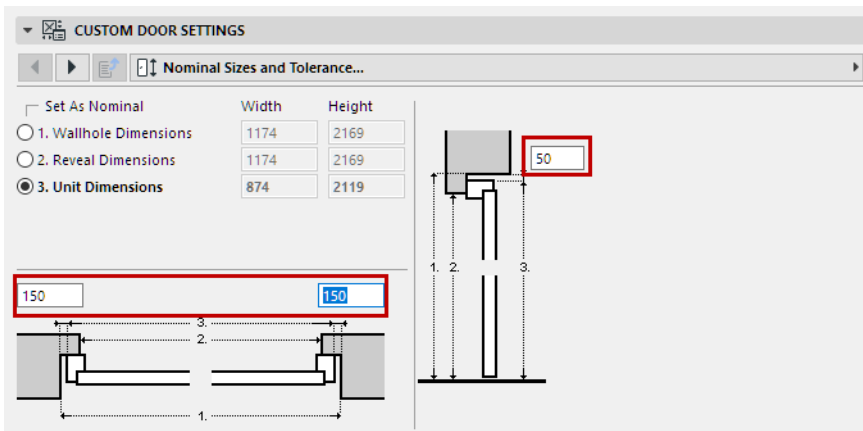
Definition of Door/Window Tolerance

LPM Assignment	Content shown with corresponding Model View Option (MVO): Detail Level of Door, Window and Skylight Symbols
2D Tolerance Full	Floor Plan Symbol Full
2D Tolerance Medium	Floor Plan Symbol Medium
2D Tolerance Low	Floor Plan Symbol Low

The Wallhole is the part cut out of the wall to accommodate the whole door/window structure.

Wallhole Width is equal to Unit Dimension Width plus any Horizontal Tolerance.

Wallhole Height is equal to Unit Dimension Height plus any Vertical Tolerance.



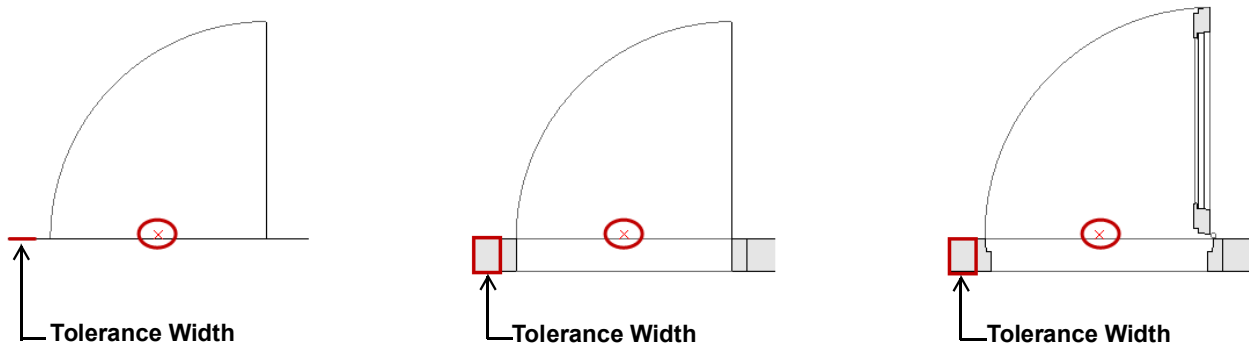
Tolerance Values in Door Custom Settings

Design Principles of Tolerance

The tolerance 2D symbol must always be drawn (using Archicad 2D tools) on the left-hand side of the door/window 2D symbols, with the same width for all used 2D symbols in a door/window Library Part.

The 2D tolerance symbols will be automatically added to right, top and bottom door/window edges, and they will be parametrically adjusted to Wallhole edges. The 3D model will be created automatically, based on the

tolerance width defined in 2D Symbol and the door/window frame thickness. The tolerance width and tolerance surfaces will be automatically added to the door/window user interface as parametric values.



Tolerance Displayed at Low, Medium and Full LOD Assignment

Opening Lines, Hinges and Handles

LPM Assignment	Content shown with corresponding Model View Option (MVO): Miscellaneous Settings for Library Parts
Opening Line Hinges Full	Show Opening Line (Hinges orientation, Full 3D)
Opening Line Hinges Simplified	Show Opening Line (Hinges orientation, Simplified 3D)
Opening Line Hinges Schematic	Show Opening Line (Hinges orientation, Schematic 3D)
Opening Line Handle Full	Show Opening Line (Handle orientation, Full 3D)
Opening Line Handle Simplified	Show Opening Line (Handle orientation, Simplified 3D)
Opening Line Handle Schematic	Show Opening Line (Handle orientation, Schematic 3D)

Design Principles of Opening Lines, Hinges and Handles

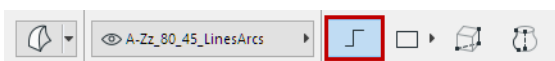


Handle Opening Line orientation



Hinges Opening Line orientation

LPM allows to define any number and directions of Opening Lines and/or Opening Operation symbols in 3D Models by using 3D Archicad tools. The 3D opening lines can be defined by Morph tool (set to the Line Geometry Method).



It is also possible to use e.g. the 3D Text Library Part (from standard Archicad library) to define any fixed text annotations.

Considering the varying standards for the direction of opening lines, LPM provides 2 sets of assignments that define opening line orientation as either “Hinges” or “Handle”.

The **Opening Line Orientation** control (available in Model View Options and the Library Part user interface) allow the user to flip their positioning as needed.

Positioning of Door/Window Handle Assembly

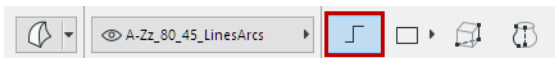
Note that the option for 3D Handle assembly is available for “3D Model Full” LOD only.

LPM Assignment	Content use
Handles Internal Full	Used for Handle assembly on internal face of opening
Handles External Full	Used for Handle assembly on external face of opening

LPM allows definition of any number and directions of Handle assembly types, separate for interior and exterior faces of an opening. The type of Handle assembly can be selected from standard Archicad types or any Custom Component Handle Library Parts (in loaded libraries).

Design Principles of Handle Positioning

Use the 3D lines that can be defined by Morph tool (set to the Line Geometry Method)

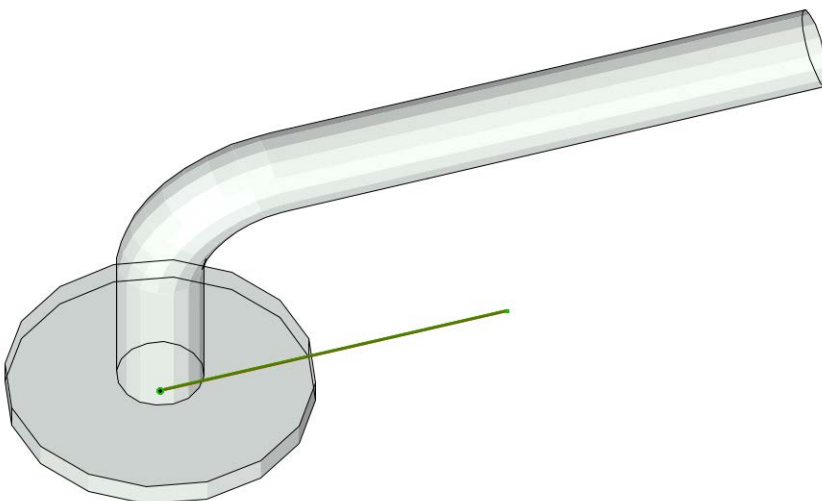


Definition (Morph line)



Morph Origin

The morph origin will be used as the anchor point (in 3D space, so the morph line must be positioned on door/window surface), and the direction of the line will be used as the Handle assembly’s orientation. The length of the line does not affect the final handle position or shape. The relationship of a morph line used for design and the final Handle model is illustrated below:



The assignment type will define handle orientation with regards to Internal or External face of a door or window.

See [Door/Window Component Origins](#).

Door and Window Standard Fixtures and Fittings

The following standard Archicad library door or window parametric elements and functionality will be added to each door or window automatically:

Windows

- Wall Opening – Reveal
- Wall Opening – Wall Closure
- Wall Opening – Wall Inset
- Wall Opening – Masonry Arch
- Wall Opening – Attributes
- Fixtures & Fittings – Sill
- Fixtures & Fittings – Board
- Fixtures & Fittings – Casing Inside/outside
- Fixtures & Fittings – Sunshade
- Fixtures & Fittings – Custom Corner
- Fixtures & Fittings – Attributes

Door

- Wall Opening – Reveal
- Wall Opening – Wall Closure
- Wall Opening – Masonry Arch
- Fixtures & Fittings – Threshold
- Fixtures & Fittings – Casing Inside/outside
- Fixtures & Fittings – Sunshade
- Fixtures & Fittings – Attributes

System predefined parameters for Doors and Windows

- Width – width of Door/Window set
- Height – height of Door/Window set
- Nominal Frame Thickness – frame thickness of Door/Window set
- Orientation Displaying – If set to 'Automatic', Label Default Position and Label Reversed Position values will be used for Door/Window labels and in Interactive Schedules. If set to 'Custom', the value of Custom Orientation parameter will be used instead.
- Custom Orientation – used in conjunction with Orientation Displaying parameter
- Label Default Position – (set to 'L' for 'Left' by default) used in conjunction with Orientation Displaying parameter
- Label Reversed Position – (set to 'R' for 'Right' by default) used in conjunction with Orientation Displaying parameter
- IFC Operation – IFC operation definition

- Surfaces > Door/Window Tolerance Outer Surface – outer surface of tolerance infill
- Surfaces > Door/Window Tolerance Inner Surface – inner surface of tolerance infill
- Surfaces > Door/Window Handle Surface – surface of Door/Window handle(s)
- Pens > Opening Line Pen – pen of Door/Window opening lines

Creating Skylights

Note: Part of the Library Part Maker add-on. See [Library Part Maker](#).

The same methodology applies as in case of Door and Window for 3D Representation. The 2D Symbol will be automatically created by the system as 2D projection of 3D Representation.

Level of Detail definition of Skylights

Selected 2D Detail Level	Default LOD	Substitute LOD 1	Substitute LOD 2
Full	System 3D projected Full	System 3D projected Medium	System 3D projected Low
Medium	System 3D projected Medium	System 3D projected Low	System 3D projected Full
Low	System 3D projected Low	System 3D projected Medium	System 3D projected Full

Selected 3D Detail Level	Default LOD	Substitute LOD 1	Substitute LOD 2
Full	Full	Simplified	Schematic
Simplified	Simplified	Schematic	Full
Schematic	Schematic	Simplified	Full

Skylight Opening Lines

These are the same as Door/Window Opening Lines.

LPM Assignment	Content shown with corresponding Model View Option (MVO): Miscellaneous Settings for Library Parts
Opening Line Hinges Full	Show Opening Line (Hinges orientation, Full 3D)
Opening Line Hinges Simplified	Show Opening Line (Hinges orientation, Simplified 3D)
Opening Line Hinges Schematic	Show Opening Line (Hinges orientation, Schematic 3D)
Opening Line Handle Full	Show Opening Line (Handle orientation, Full 3D)
Opening Line Handle Simplified	Show Opening Line (Handle orientation, Simplified 3D)
Opening Line Handle Schematic	Show Opening Line (Handle orientation, Schematic 3D)

Overview of LOD Assignments

Note: Part of the Library Part Maker add-on. See [Library Part Maker](#).

Objects, Lamps and MEP Objects

Assignment	Object	Lamp	Duct Fitting	Pipe Fitting	Duct Flow Terminal	Pipe Flow Terminal	Duct In-line Flow Device	Pipe In-line Flow Device	Flow Equipment
2D Symbol Full	+	+	+ ¹	+ ¹	+	+	+ ¹	+ ¹	+
2D Symbol Medium	+	+	+ ²	+ ²	+	+	+ ²	+ ²	+
2D Symbol Low	+	+	+	+	+	+	+	+	+
2D Symbol Electrical	+	+							
2D Symbol Reflected Ceiling Plan	+	+							
3D Model Full	+	+	+	+	+	+	+	+	+
3D Model Simplified	+	+	+	+	+	+	+	+	+
3D Model Schematic	+	+	+	+	+	+	+	+	+

1. Projected from 3D Model Full (or Simplified, if no Full is defined)
2. Projected from 3D Model Simplified (or Schematic, if no Simplified defined)

Doors, Windows and Skylights

Assignment	Door	Window	Skylight
2D Symbol Full	+	+	
2D Symbol Medium	+	+	
2D Symbol Low	+	+	
3D Model Full	+	+	+
3D Model Simplified	+	+	+
3D Model Schematic	+	+	+
2D Tolerance Full	+	+	
2D Tolerance Medium	+	+	
2D Tolerance Low	+	+	
Opening Line Hinges Full	+	+	+
Opening Line Hinges Simplified	+	+	+
Opening Line Hinges Schematic	+	+	+

Assignment	Door	Window	Skylight
Opening Line Handle Full	+	+	+
Opening Line Handle Simplified	+	+	+
Opening Line Handle Schematic	+	+	+
Handles Internal Full	+	+	+
Handles External Full	+	+	+

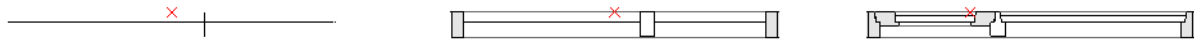
LOD Recommendations

Note: Part of the Library Part Maker add-on. See [Library Part Maker](#).

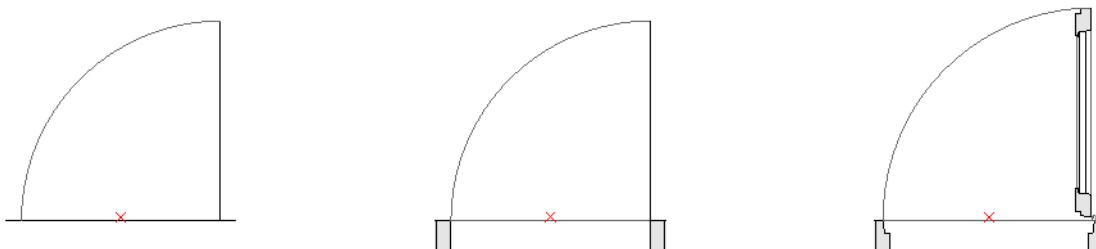
Recommended 2D Symbol Level of Detail

Consider the pen thickness and detail appearance at scale – too detailed 2D representations for larger scales might be difficult to read.

Examples of 2D Symbol Level of Detail



Window: Low, Medium, Full LOD Assignments



Door: Low, Medium, Full LOD Assignments



Object: Low, Medium, Full LOD Assignments

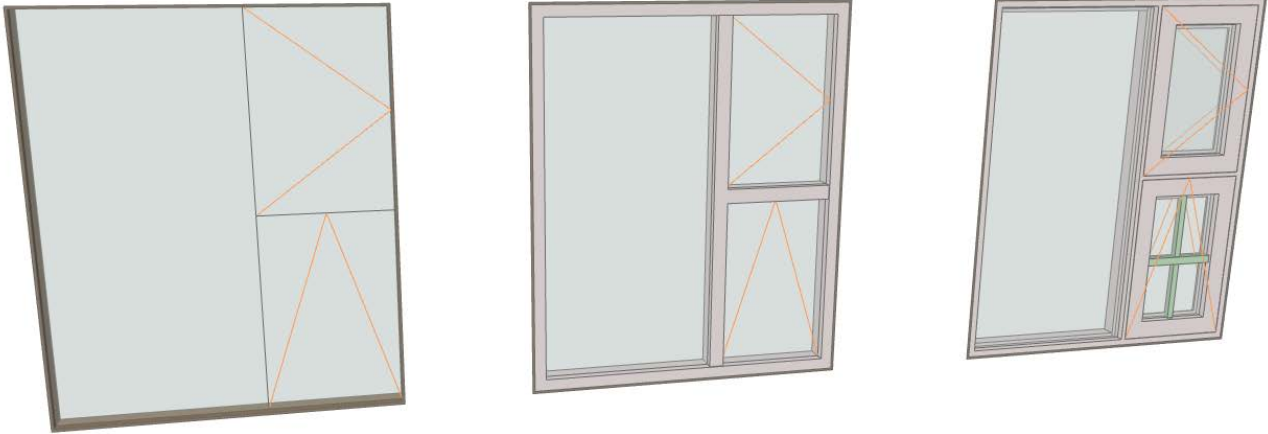
Recommended 3D Model Level of Detail

Creating very detailed 3D models can lead to a heavy polygon count that can slow down Archicad. Here are some hints to help you create Library Parts without compromising performance:

- Avoid modeling details legible at scales less than 1:20 in general (e.g. 1:10 or 1:5 etc.)
- Simplify shapes of frame/sashes sections or assembly profiles
- Do not model unnecessary elements (if possible) such as door/window seals, detailed door/window ironmongery, fasteners and similar.
- Avoid creating an excessive number of curved surfaces.
- Creating curved surfaces with very small radii will result in faceted faces; the best approach is to simplify them. (A typical example for the simplification use would be extruded steel/aluminum profiles of door or window frames and sashes).

- Use the Morph tool (plane with zero thickness) for elements that should appear as a single line in Sections (e.g. glazing panes or door leaf/window sash in “Schematic” 3D LOD).

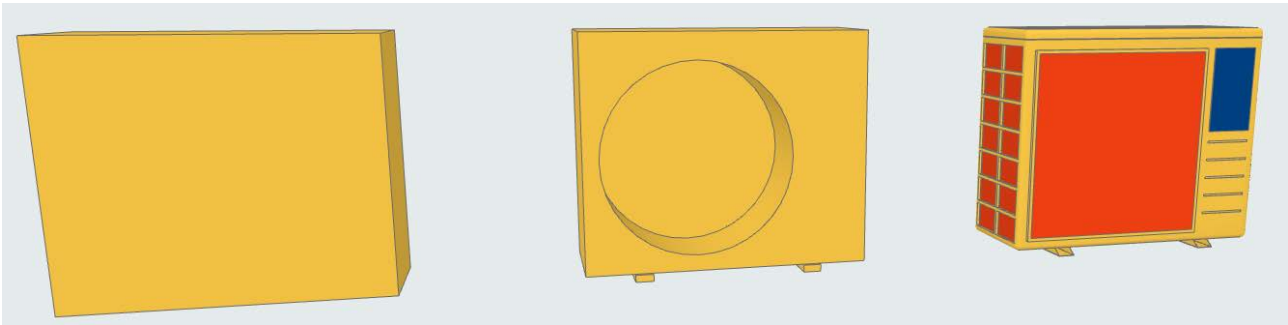
Examples of 3D Model Level of Detail



Window: Schematic, Simplified, Full LOD Assignments



Door: Schematic, Simplified, Full LOD Assignments



Object: Schematic, Simplified, Full LOD Assignments

Customizing Data Properties

Note: Part of the Library Part Maker add-on. See [Library Part Maker](#).

The new Library Part must include a set of listing properties. These listing properties will be available in the Descriptions area (Custom Settings panel) in the Settings dialog of the saved Library Part.

Use these as additional, Libpart-specific data or descriptors that can be listed in schedules. These will be non-editable GDL parameters of the resulting Library Part.

To define the Library Part's listing properties:

In the Properties panel, select an option to define the Library Part's listing properties:

[See also Select Property Scheme](#).

By default, Library Part Maker uses the "Archicad Listing Parameters scheme".

- You can create any custom parameter scheme instead.
- A custom scheme together with defined values can be imported directly from an Excel spreadsheet.

Related topics:

[Create Custom Property Scheme](#)

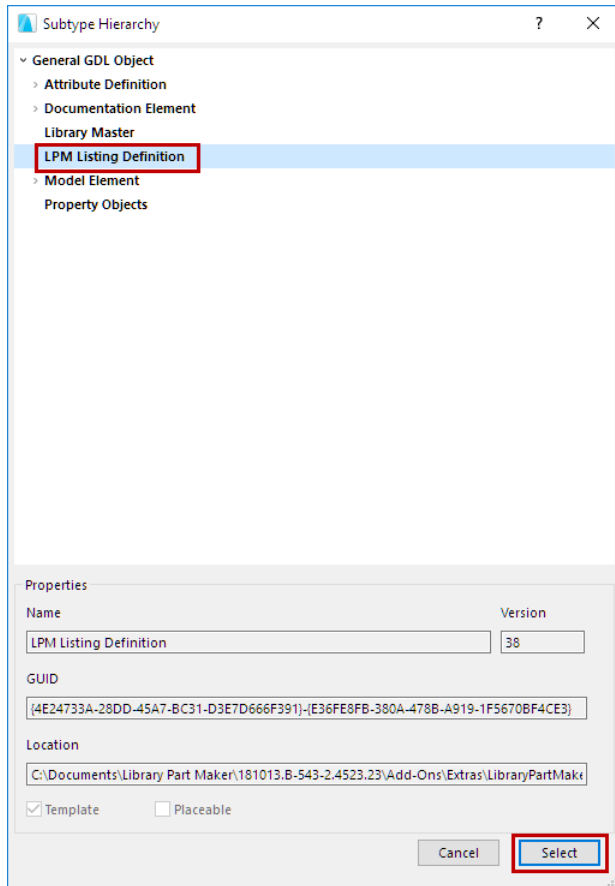
[Import Property Data from Excel](#)

Create Custom Property Scheme

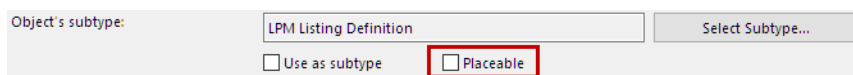
Create a custom property scheme (a library object, whose subtype is “LPM Listing Definition”) and save it to one of the loaded libraries in your Archicad project.

Follow these steps:

1. Create a new Library Part: File > Libraries and Objects > New Object...
2. In the appearing Object Editor window, click Select Subtype.
3. From the Subtype Hierarchy dialog, choose “LPM Listing Definition”, and click Select.



4. Back in the Object Editor window, uncheck **Placeable**.



5. In the Object Editor, use the Parameters control to define parameter names and their types.
6. Use **File > Save** to save the Library Part (your new Property Scheme) to a project library.
7. In Library Manager, reload your project libraries (File > Libraries and Objects > Library Manager...)

The new Listing Definition scheme will appear in the drop-down menu of Property Schemes, in the Library Part Maker palette.

Import Property Data from Excel

Note: Part of the Library Part Maker add-on. See [Library Part Maker](#).

The new Library Part must include a set of listing properties. These listing properties will be available in the Descriptions area (Custom Settings panel) in the Settings dialog of the saved Library Part.

In the Properties panel, select an option to define the Library Part's listing properties:

[See also Select Property Scheme.](#)

By default, Library Part Maker uses the "Archicad Listing Parameters scheme".

Alternatively, you can import a custom scheme together with defined values from an Excel spreadsheet.

The spreadsheet must be formatted to the columnar format with standard column header names as follows:

Property Group

Used for grouping parameter headers in Library Part parameter definitions

Property Name

Used for parameter names, truncating the name to 28 characters

Archicad Data Type

Values in this column define Archicad GDL parameter types. The definition values are limited to the following types:

- Boolean
- Real
- Integer
- Text
- Length
- Angle

The property value sets are defined by the column and will be used in the LPM's Choose values to Import drop-down list.

Views of the Virtual Building

This section explains the different model views of the Virtual Building: e.g. Sections, 3D views, schedule views. The guide explains how to instantly create the desired view, and how to control which elements will appear in the view and with which effects.

On-Screen View options let you toggle certain screen-only aids on and off.

The Renovation feature lets you control element display from the point of view of their status in a renovation project. Finally, use the Graphic Overrides feature to convey view-by-view information at a glance.

[Archicad Model Views](#)

[Model View Options](#)

[On-Screen View Options](#)

[Partial Structure Display](#)

[Renovation](#)

[Graphic Overrides](#)

For information on managing the Navigator View Map:

See [Navigator View Map](#).

Archicad Model Views

You can approach the Virtual Building model from several different model views. Model views have two main uses. First, you will work in model views to modify the Virtual Building. Second, saved model views are the basis for creating architectural drawings (placed onto Layouts) for the final documentation of the Virtual Building.

Related Topics:

[Model View Options](#)

[Floor Plan Window](#)

[Stories](#)

[Floor Plan Cut Plane \(Global Setting\)](#)

[Element Display in Floor Plan and Section](#)

[3D Window](#)

[Sections](#)

[Elevations](#)

[Interior Elevations \(IE\)](#)

[3D Document](#)

[Details](#)

[Worksheets](#)

[Managing Markers in Archicad](#)

[Interactive Schedule](#)

[Project Indexes](#)

For information on managing the Navigator View Map:

See [Navigator View Map](#).

Floor Plan Window

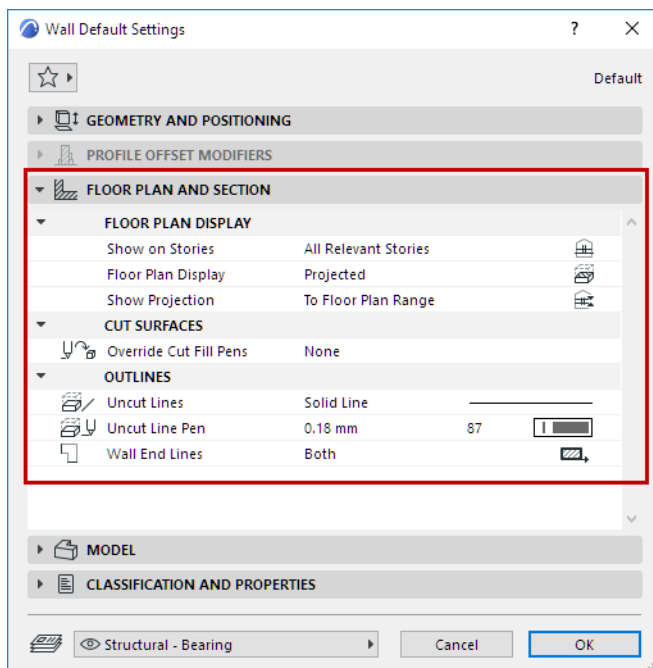
The **Floor Plan** Window is the basic construction area where most editing operations take place. It shows a representation of the current project as a traditional architectural drawing. When you start Archicad, the first window you see is the Floor Plan. Each story of the Virtual Building has its own Floor Plan.

Archicad gives you the freedom to replicate 3D-like aspects of construction elements in the Floor Plan window, using Cut Plane settings and element-level display and projection options. These options let you visualize parts of multi-story elements that are located on a different floor plan, or to project parts of the element that are above or below the cut plane.

See the sections below for how these options work.

The visibility and accessibility of elements on the Floor Plan depends on:

- chosen options in the element's Tool Settings dialog box, including multi-story display and projection settings;



- the Floor Plan Cut Plane;
- Layer Settings (elements on invisible layers will not be displayed; elements on locked layers cannot be edited).

Modifications to the model in the Floor Plan window will be updated accordingly in the 3D, Detail/Worksheet and Section/Elevation/IE and 3D Document windows. The converse is also true: changes made in other windows are automatically displayed on the Floor Plan when activated. The updates can take place either manually or automatically depending on the Detail or Section/Elevation/IE update settings you choose.

For more information, see [Updating Sections and Updating the Detail Window](#).

Stories

Stories in Archicad, just as in real buildings, serve to divide space vertically and replicate the story structure of your real building.

Related Topics:

[About Stories](#)

[Home Story](#)

[Define Toplink for Wall, Column, Zone or Stair](#)

[Show On Stories](#)

[Navigate Among Stories](#)

[Defining Story Display in 3D](#)

[Defining Story Display in Section/Elevation/IE](#)

[Create or Delete or Rename Stories](#)

[Story Settings Dialog Box](#)

[Story Level Lines](#)

[Edit Elements by Stories Dialog Box](#)

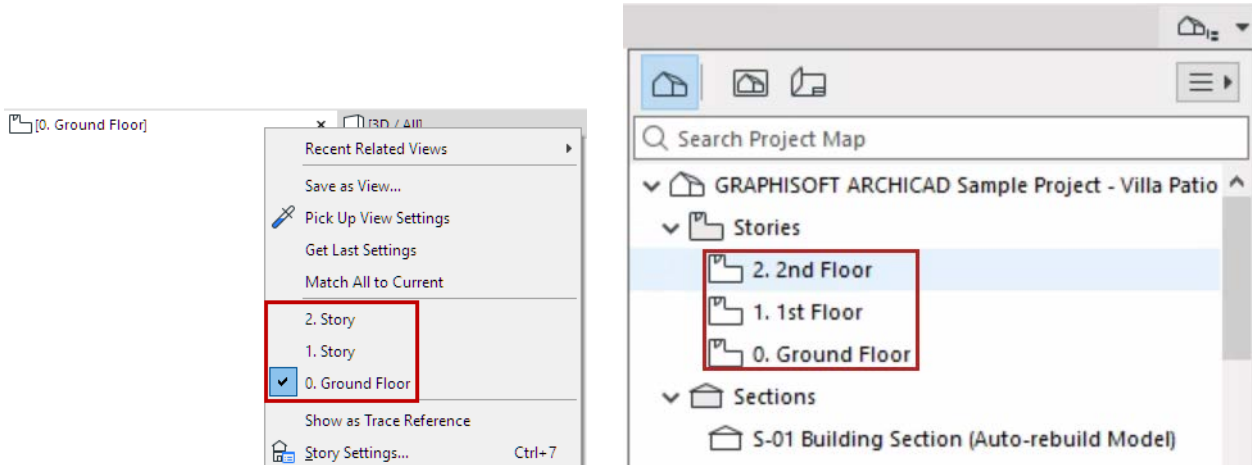
About Stories

Story names and levels are defined in Story Settings.

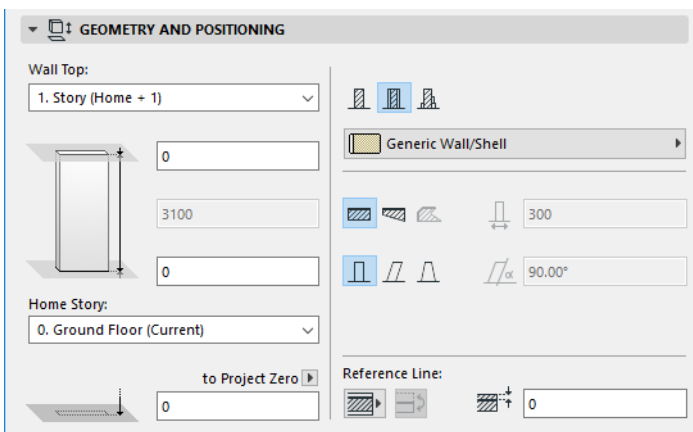
[See Story Settings Dialog Box.](#)

Each story in the project is listed:

- in the Navigator's Project Map
- in the Tab bar's pop-up Navigator
- in the Tab bar's Floor Plan context menu



The elevation of new construction elements placed on the Floor Plan are calculated relative to the element's Home Story. Some elements (Wall, Column, Zone) can also be top linked to any story in the project.

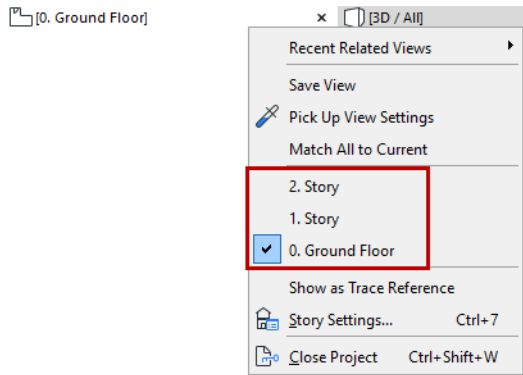


[For more information, see Home Story and Define Toplink for Wall, Column, Zone or Stair.](#)

Navigate Among Stories

To navigate among stories, do one of the following:

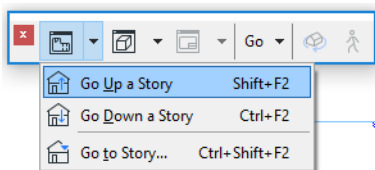
- In the Tab bar, click the context menu of the Floor Plan to access a list of stories



- In the Tab bar, use the Project Map of the pop-up Navigator (at the right edge of the Tab bar)
- In the Navigator Project Map, double-click the Story you wish to open
- Activate a story from the Story Settings dialog box (**Design > Story Settings**, or right-click any story from the Project Map): click on its name, then click OK to close the dialog box and display the selected story on the Floor Plan
- Use predefined keyboard shortcuts to jump among stories
- Use the Mini-Navigator Toolbar

Story Shortcuts in Mini-Navigator Toolbar

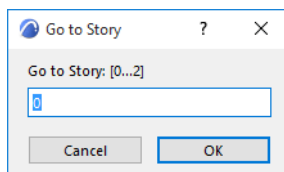
Use the story shortcuts located in the predefined “Mini-Navigator” toolbar.



If you attempt to go to a story that does not exist, the **Create New Story** dialog box appears.

[See *Create or Delete or Rename Stories*.](#)

Go To Story: Jump directly to an existing Story by entering a Story number.



Defining Story Display in 3D

In the 3D window, all Stories are projected by default, but you can limit the projection using the **Stories to Show in 3D** controls at **View > Elements in 3D View > Filter and Cut Elements in 3D**.

See [Filter and Cut Elements in 3D Dialog Box](#).

Defining Story Display in Section/Elevation/IE

In Section/Elevation/IE views, you define a vertical range that can include entire Stories or only parts of them. Also, Story Level Lines can be optionally displayed in these views.

[Horizontal/Vertical Range of Section Viewpoint](#)

[Show/Hide Story Level Lines in Section/Elevation/IE](#)

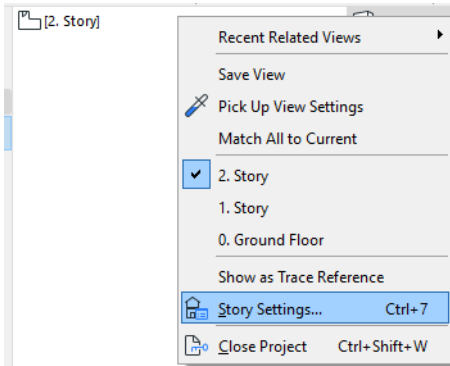
Create or Delete or Rename Stories

Use Story Settings (**Design > Story Settings**) to create and delete stories.

[See Story Settings Dialog Box.](#)

Story Settings (and some story-related commands) are also available:

- in the Tab bar's Floor Plan context menu



- from the context menu if you have selected a Story in the Navigator's Project Map or View Map
- from the Mini-Navigator Toolbar

[See Story Shortcuts in Mini-Navigator Toolbar.](#)

Delete a Story

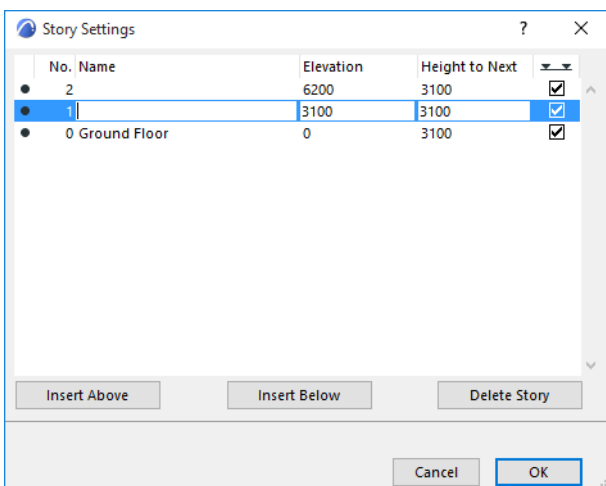
Use the Navigator or Story Settings (as described above) to delete one or more stories.

Warning: If you delete a Story, all elements that had that story as their Home Story will also be deleted!

Warning: The undo queue is cleared entirely whenever you delete or add a story, or change story height.

Create a New Story

1. Go to **Design > Story Settings**.



2. Click "Insert Above" or "Insert Below" to insert a new story above or below the story name selected in the dialog box list.

3. Click **OK**. The new story appears above or below the selected story. The numbering is automatically updated.

See also [Update of Story Level Dimensions](#).

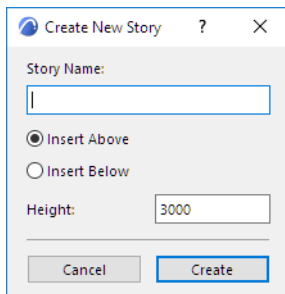
There is no limit to the number of stories.

See also [Story Settings Dialog Box](#).

Warning: The undo queue is cleared entirely whenever you delete or add a story, or change story height.

Alternatively:

1. Select a Story in the Navigator's Project Map.
2. Use the context menu's **"Create New Story"** command.



Rename a Story

1. Go to **Design > Story Settings**.
2. Click the name of the story you wish to rename, and type the new name.

Alternatively:

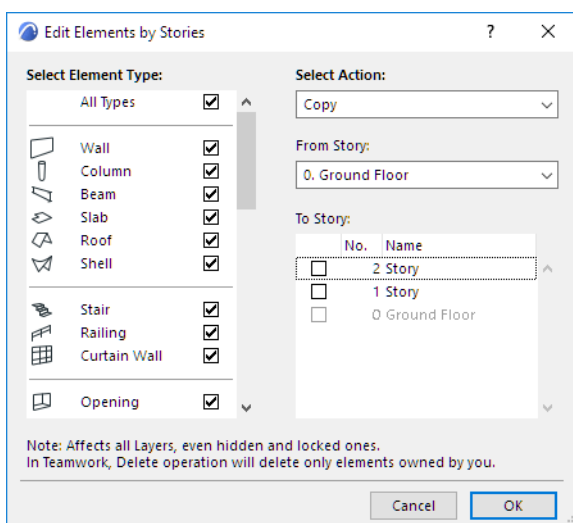
1. Right-click the story name in the Navigator, and choose the Rename Story command.

Copy/Move Elements Between Stories

When creating a new Story, you may wish to copy elements onto it which already exist on another Story.

Go to

- Design > Edit Elements by Stories, or
- Edit > Move > Edit Elements by Stories



For more information, see [Edit Elements by Stories Dialog Box](#).

Story Level Lines

Story Level Lines indicate the vertical location of stories in Sections, Elevations and Interior Elevations. Their values are defined at **Design > Story Settings**.

See [Story Settings Dialog Box](#).

Story Levels can be set to appear in Section/Elevation/IE views. In this case, each story level line included in the Section/Elevation/IE range will appear in the window.

You can place associative dimensions onto Story Level Lines.

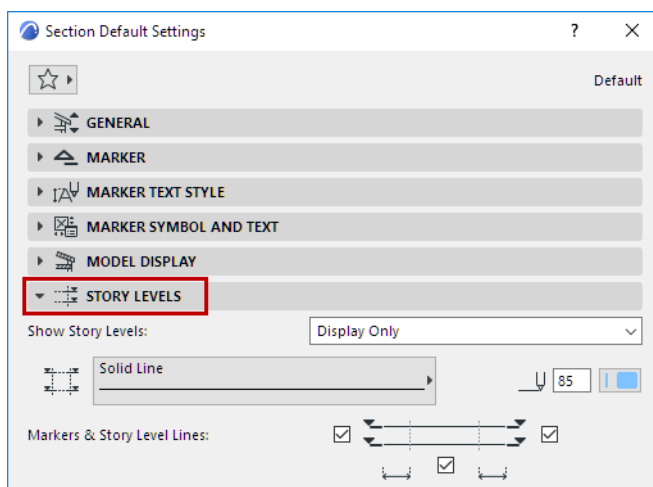
The story line markers are library parts. The Story Marker parameters and position can be edited in Section/Elevation/IE Settings dialog boxes.

For more information, see [Story Levels Panel](#).

Show/Hide Story Level Lines in Section/Elevation/IE

For each individual Section/Elevation/Interior Elevation, you can hide, show and/or output Story Level Lines and their markers.

To set these options, open the Section/Elevation/IE Settings Dialog box, go to the Story Levels panel, and use the “Show Story Levels” pop-up.



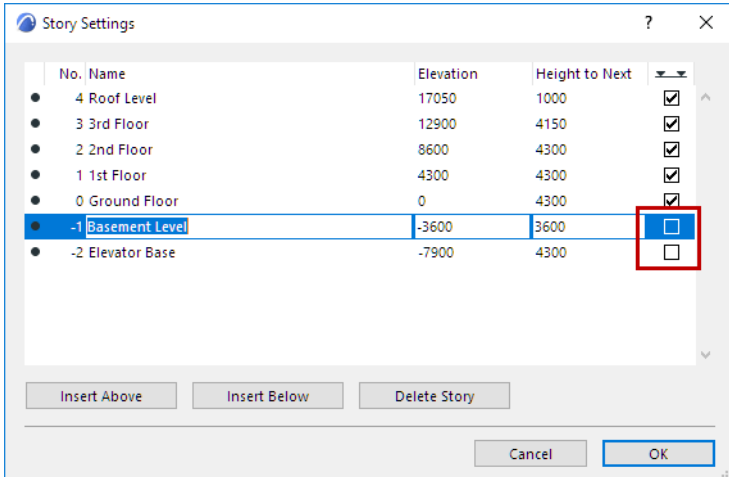
See [Story Levels Panel](#).

Show/Hide Story Level Lines Throughout the Project

For any story, it is possible to turn off level line display throughout the project (in every Section/Elevation/IE view).

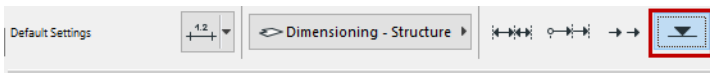
Go to **Design > Story Settings**, and for any selected story, uncheck the “Story Level Line” box. This story’s level line will henceforth not appear in any model window in the project.

Any associative dimensions of the Story Level Line will be deleted.

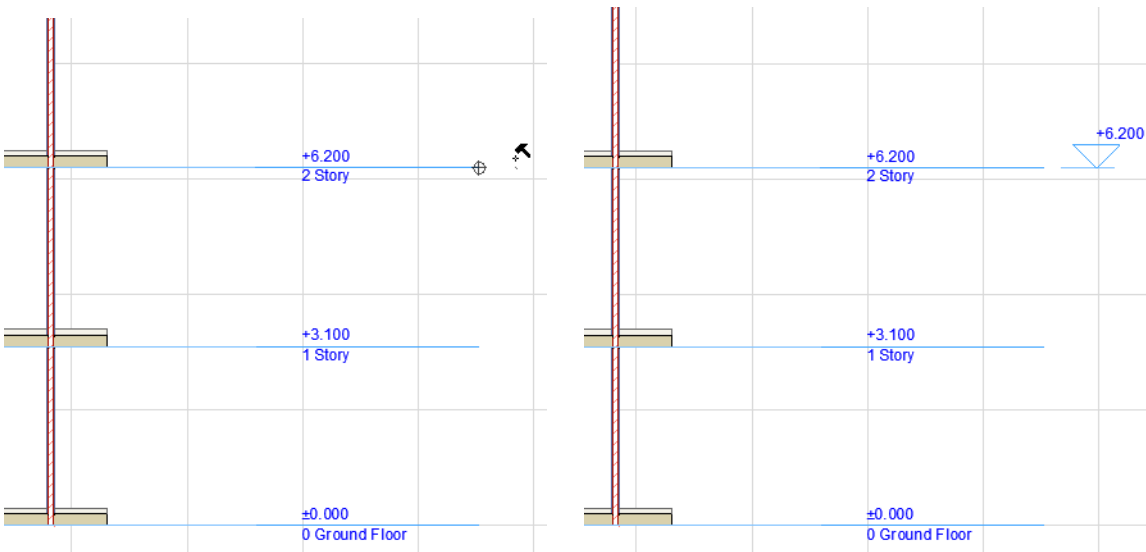


Dimension Story Level Lines

You can place associative Dimensions on Story Level Lines.
For example, use the Elevation type of the Linear Dimension tool.

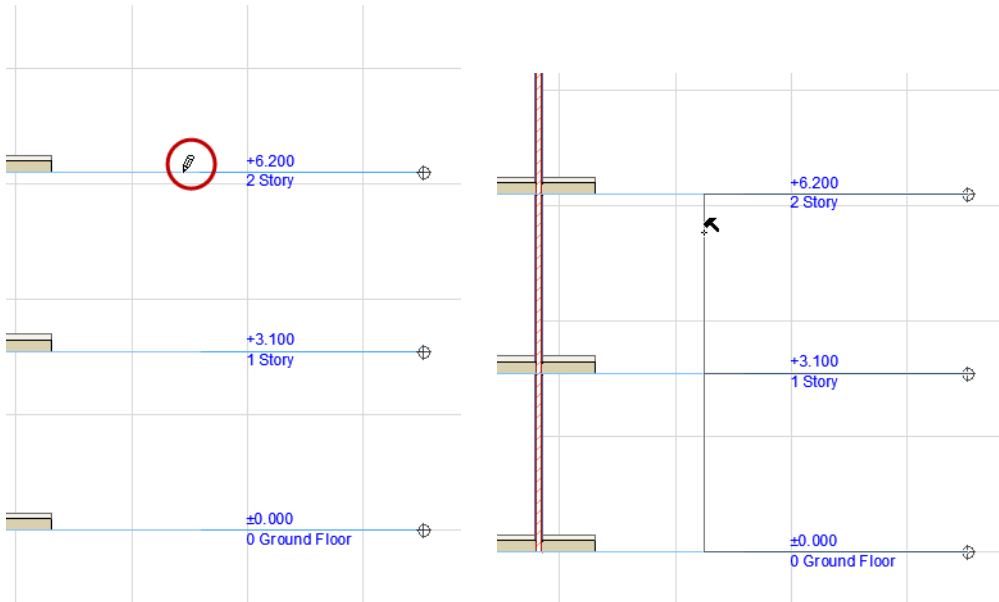


- Dimension a Single Story Level Line
Click on a Story Level Line *endpoint*, then double-click to place the dimensions.



- Dimension All Story Level Lines

Click on the Story Level Line itself, then double-click to place the dimensions.



Display of Story Level Dimensions

If you turn off Story Level display (in Section/Elevation Settings), the Dimensions remain in place.

Update of Story Level Dimensions

If you delete a Story, its Story Level Dimension is also deleted. The rest of the Story Level dimensions are updated automatically.

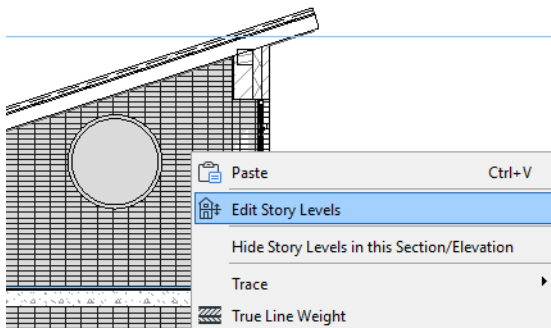
If you modify a Story height, the rest of the Story Level dimensions are updated automatically.

However, If you insert a new Story, you must place its Story Level dimension manually.

Edit Story Levels

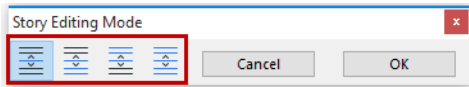
You can edit the elevation of any given story or set of stories using Story Editing Mode:

1. Choose the **Design > Edit Story Levels** command, or the same command from the context menu that appears by right-clicking inside the Section window.

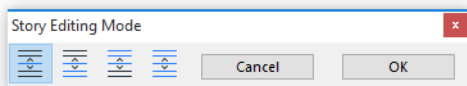


2. The Story Editing Mode palette appears. As long as the palette remains open on screen, you are in Story Editing mode and can move the story level lines.

3. Choose a mode from this palette by clicking one of the four buttons:



- Adjust only the selected story
 - Adjust the selected story and all stories below
 - Adjust the selected story and all stories above
 - Adjust all stories
4. Move the cursor onto the Story Level Line you wish to move. The cursor will assume the Mercedes shape. Click and drag the story level line to edit its elevation.



5. Click **OK** to apply changes and exit Story Editing Mode. Associative elevation markers will be updated too. The elements that are on the given stories will retain their elevation respective to their Home Story, as you will see when the view is updated after you click **OK**.

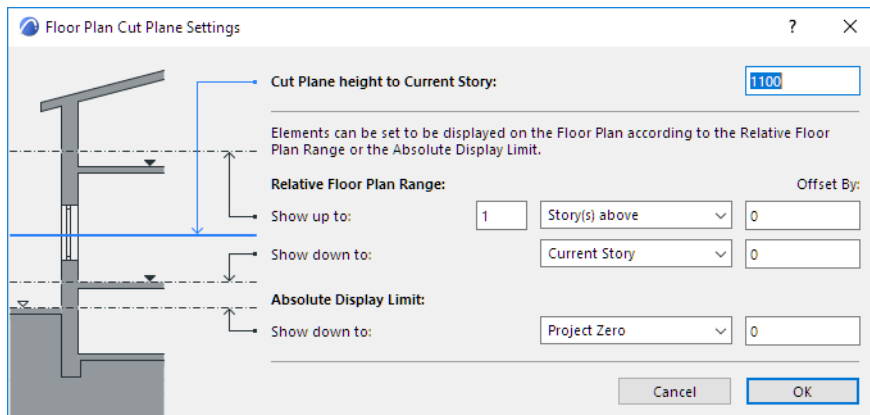
Floor Plan Cut Plane (Global Setting)

3D construction elements on the Floor Plan (Walls, Curtain Walls, Columns, Beams and Roofs) are displayed as if they were cut horizontally along a theoretical plane; this is the Floor Plan Cut Plane.

The default height of the Cut Plane, measured from the base of the current story, depends on local architectural conventions, but you can set any Floor Plan Cut Plane for the current window using the Floor Plan Cut Plane dialog box.

To set the Cut Plane, go to **Document > Floor Plan Cut Plane**

[See also Floor Plan Cut Plane Settings Dialog Box.](#)



Note: The Cut Plane does not affect the display of Mesh, Slab or Object elements. However, GDL Objects can be programmed to display themselves according to the Floor Plan Cut Plane Settings.

Enter a value for “Cut Plane Height to Current Story.”

If you wish to display all construction elements in their entirety, on all stories (the default settings for construction elements), the rest of the settings in the Floor Plan Cut Plane dialog box are not relevant.

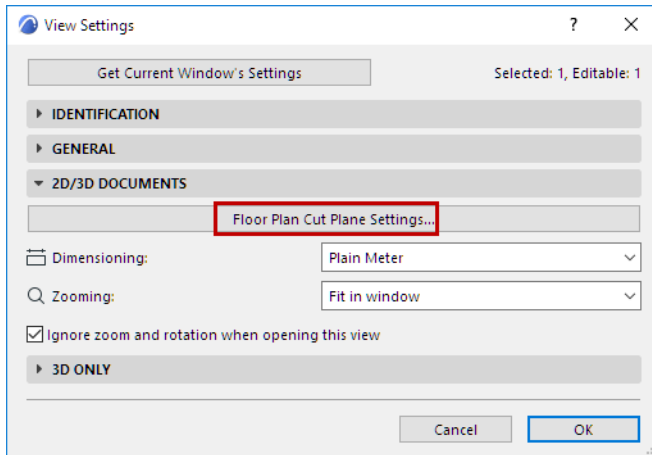
The Relative Floor Plan Range and Absolute Display Limit - the other Floor Plan Cut Plane Settings - are only relevant if any of your construction elements are defined as having a limited projection.

The Floor Plan Cut Plane settings are global; the current settings apply to all the stories of the project. Once you set the global Floor Plan Cut Plane, you can further fine-tune the Floor Plan display of *individual* construction elements (Walls, Columns, Beams and Roofs only) in their element Settings dialog boxes.

[For more information, see Element Display in Floor Plan and Section.](#)

Set Separate Floor Plan Cut Planes in Different Views

For every view, you can define a unique Floor Plan Cut Plane, if required, using the Floor Plan Cut Plane controls accessible from View Settings.



Element Display in Floor Plan and Section

Relevant for Wall, Beam, Column, Slab, Roof, Shell, Morph, Mesh.

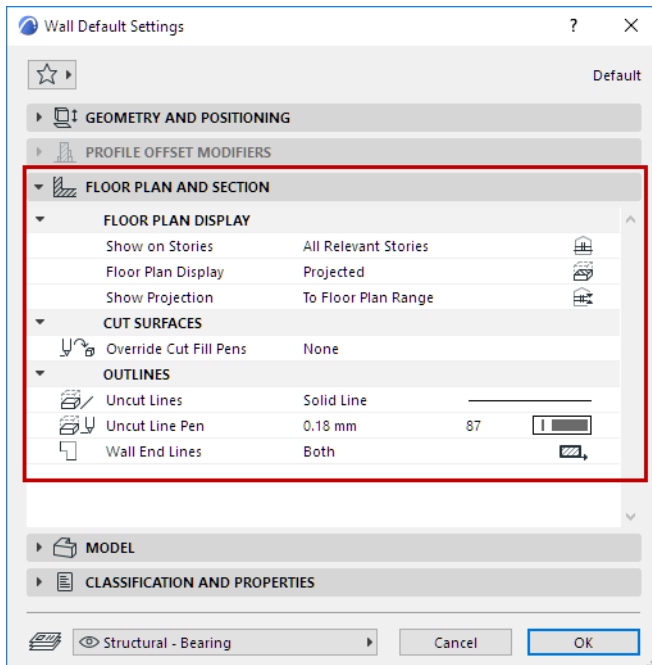
Use the Floor Plan Display settings in each element's Tool Settings Dialog Box.

Show On Stories

Floor Plan Display (Element Settings)

Show Projection

Cut Surfaces and Outlines (Element Settings)



For specifics on other elements, see:

Floor Plan Display of Doors/Windows

Floor Plan/Reflected Ceiling Plan Display (Stair Settings)

Show On Stories

Relevant for Wall, Slab, Roof, Shell, Morph, Mesh, Beam, Column.

Use the Show on Stories pop-up in the **Floor Plan and Section Panel** of Tool Settings to control the display of construction elements by story.

The available options differ by element.

All Relevant Stories

The element will be shown and editable on all stories which it physically intersects. When available, this will be your preferred option most of the time.

All Stories

The element outline - regardless of its actual location - will be shown on every story of the project.

Home Story Only

The element will be shown only on its Home Story. (If the element is drawn so that it does not physically intersect its Home Story, then the only way the element will be shown on the Floor Plan is if you set its Floor Plan Display to one of the symbolic options: Symbolic Cut, Outlines only, Overhead all.)

[See *Home Story*.](#)

Home story plus one story up and/or down

One Story Up/One Story Down

Beams, Columns and Objects have a unique display option: **One Story up** and **One Story Down**. This way, you can opt to *not* display these elements on their Home Story at all.

Custom

Choose this option to set separate display combinations for the element's outline and fill in **Show on Stories: Custom Settings**. Once you set these Custom Settings, use the Edit Custom option to edit them.

Floor Plan Display (Element Settings)

Relevant for Wall, Beam, Column, Roof, Shell, Morph.

Use the **Floor Plan Display** pop-up from the Floor Plan & Section panel of the element's Tool Settings dialog box.

Options vary depending on the element.



Projected with Overhead

Shows cut part of element's 3D model (e.g., as cut at the level of the Floor Plan Cut Plane), plus the element's overhead part (i.e. the part of the element that is above the Floor Plan Cut Plane), plus its uncut (downward) part.

Note: "Projected with Overhead" is the default display setting for slanted or complex columns and walls, and for all roofs and beams.

Projected

Shows cut part of element's 3D model, plus its uncut (downward) part.

If you choose either of these "projected" options (Projected with Overhead or Projected), you can then further define the extent of the projection, using the **Show Projection** options.

For more information, see [Show Projection](#).

Cut Only

Displays only the cut part, as cut with the Floor Plan Cut Plane. (Available for construction elements; not available for windows or doors.)

Some additional abstract display options are available:

Symbolic Cut

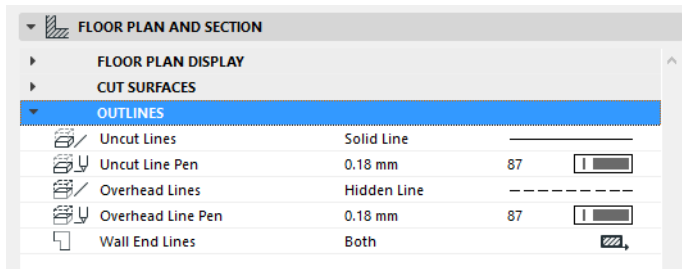
(Available only for simple straight walls, Openings, and for vertical or complex columns.)

The whole floor plan projection will be displayed as cut, using their cut line and cut fill attributes, regardless of the elements' vertical position.

This option is available only if the Show on Story control is set to **Home Story only**. The Floor Plan Cut Plane settings do not affect the display of these elements.

Outlines Only

The entire element's outline is shown using its uncut attributes. (Uncut attributes are set for each element using the "Outline" controls of the Floor Plan and Section panel of its Settings dialog box.)



Overhead All

The entire element's outline is shown using its overhead attributes. (Overhead attributes are set for each element using the "Outline" controls of the Floor Plan and Section panel of its Settings dialog box.)

Show Projection

Relevant for Wall, Beam, Column, Roof, Shell, Morph.

The **Show Projection** pop-up provides options for setting the range within which an element should be displayed.

This control is only available if the element's Floor Plan Display is set to either "Projected" or "Projected with Overhead."

Exception: For Morph elements, Show Projection options are available regardless of the Floor Plan Display.

Entire Element

The entire projection will be displayed on the applicable stories.

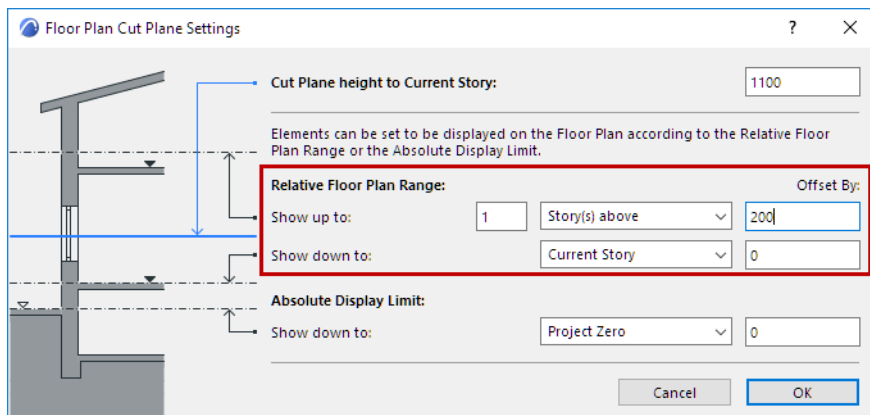
However, you may prefer not to show the entire element.

In this case, you have two other ways to set its display in the **Show Projection** pop-up:

To Floor Plan Range

Show the element on a range of stories (the current story, plus a given number of stories above and below it, and an optional offset.)

This element will be shown on the stories defined as the Relative Floor Plan Range (i.e. the number of stories on which to show this element in either direction) in **Document > Floor Plan Cut Plane**.



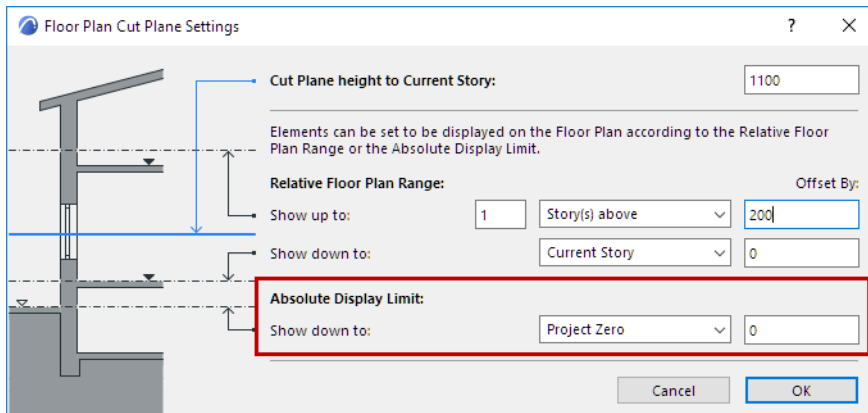
By Absolute Display Limit

Set a fixed lower limit (by default, this is Project Zero), then show all parts of the element above this limit.

The Absolute Display Limit option enables you to ensure that the Floor Plan will never display any part of the model that falls below the level you define here in absolute terms (independent of story levels).

For example, you can enter the level of the terrain or the water level, so that anything below this level is not shown on the Floor Plan. Or, if you want to show a rooftop terrace without displaying the garage below it, enter the terrace level as the Absolute Display Limit.

If you choose this option, then the Absolute Display Limit set in Floor Plan Cut Plane Settings determines this element's lower display limit.



In certain situations, the current Floor Plan Cut Plane and Relative Floor Plan Range settings may conflict:

- The Cut Plane level may be outside the current Floor Plan Range (e.g. an extremely small story height as compared to the default Cut Plane level). In this case, the Cut Plane will be automatically relocated to the upper limit of the Floor Plan Range (or to its lower limit, if the Cut Plane would fall below the lower limit).
- The lower Floor Plan Range limit may be higher than the upper limit. In this case, the lower limit will be considered as both the lower and upper limit of the Floor Plan Range (which will also equals the Cut Plane level, as described above.)

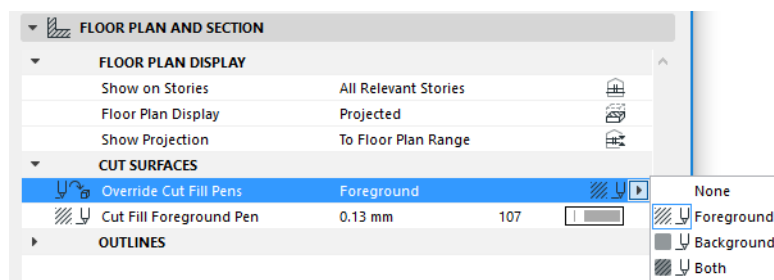
Cut Surfaces and Outlines (Element Settings)

Relevant for Wall, Slab, Roof, Shell, Morph, Mesh, Beam, Column.

Cut Surfaces

Set the attributes displayed for cut surfaces both in Section and on the Floor Plan.

- For a **basic** element:
 - Define the cut line and pen
- For a **basic, composite or complex profile** element:
 - If needed, override the fill foreground/background pens of this element (defined by default in the Building Material). To do this, choose Foreground, Background or Both from the **Override Cut Fill Pens** pop-up to access the respective controls.



Outlines

Use these settings to define lines and pen colors to element outlines that fall above (**Overhead**) and/or below (**Uncut**) the Floor Plan Cut Plane.

- **Overhead** Lines will be displayed if the Floor Plan Display pop-up is set to “Projected with Overhead” or “Overhead All”.
- **Uncut** Lines will be displayed if the Floor Plan Display pop-up is set to “Projected” or “Outlines Only.”

3D Window

The 3D window displays your model in real perspective or axonometry: this gives the best overall view of what the final building will really look like, inside and out.

In the 3D window, you can directly edit your model, and create new construction elements, in either perspective or parallel view, using any of the construction tools.

The **3D** window is directly linked to the **Floor Plan** and to the **Section/Elevation/IE** windows: any changes made on the Floor Plan or in a Section/Elevation/IE window will be visible in the 3D window and vice versa.

3D Documents can be derived from either a 3D Window source or the Floor Plan. If you change the projection, the cutting planes, the zoom level or the selected/marqueed/filtered items of the 3D source window, you can redefine the 3D Document accordingly.

Special **navigation** techniques in the 3D Window let you explore the model much more freely than in other windows.

The **3D display** of individual construction elements is controlled by the options of the Model panel in their Settings Dialog box.

General settings affecting the parameters for displaying the 3D window are in **3D Styles**.

The commands that affect the set of elements displayed in 3D are in the **View > Elements in 3D View** and the **View > 3D View Options** hierarchical menus.

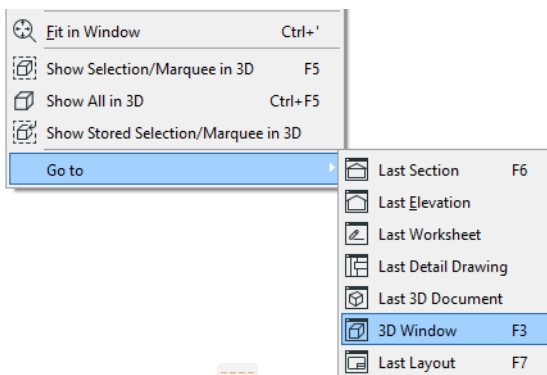
These commands are also accessible in the **3D Visualization** toolbar.



Open the 3D Window

To open or activate the 3D window, do one of the following:

- press the **F3** key
- use the **Window > 3D Window** command
- press the 3D window control on the **3D Visualization toolbar** or the **Mini-Navigator toolbar**
- use context menu commands from right-clicking on blank space in the current window:



Show All in 3D

To show the entire model in 3D, regardless of any current selection, use the **View > Elements in 3D View > Show All** option; or use the same command from the context menu.

This command's shortcut is **Ctrl + F5**.

The full model display resulting from the **Show All** command may be limited by filtering criteria in the Filter and Cut Elements in 3D dialog box.

See [Filter and Cut Elements in 3D Dialog Box](#).

Default Display in 3D

If you simply switch to the 3D window, without using any specific 3D display command, the 3D window will display the contents last shown in the 3D window, irrespective of what might be selected in any other window.

Related Topics

[Show Selection/Marquee in 3D](#)

[Filter and Cut Elements in 3D Dialog Box](#)

[Redefine Image Settings \(3D View\)](#)

[3D Engines](#)

[3D Styles](#)

[3D Projections](#)

[Navigation in the 3D Window](#)

[3D Navigation Extras](#)

[3D Cutting Planes](#)

[3D Window Size](#)

[Physically based renderer in 3D Window - Experimental feature](#)

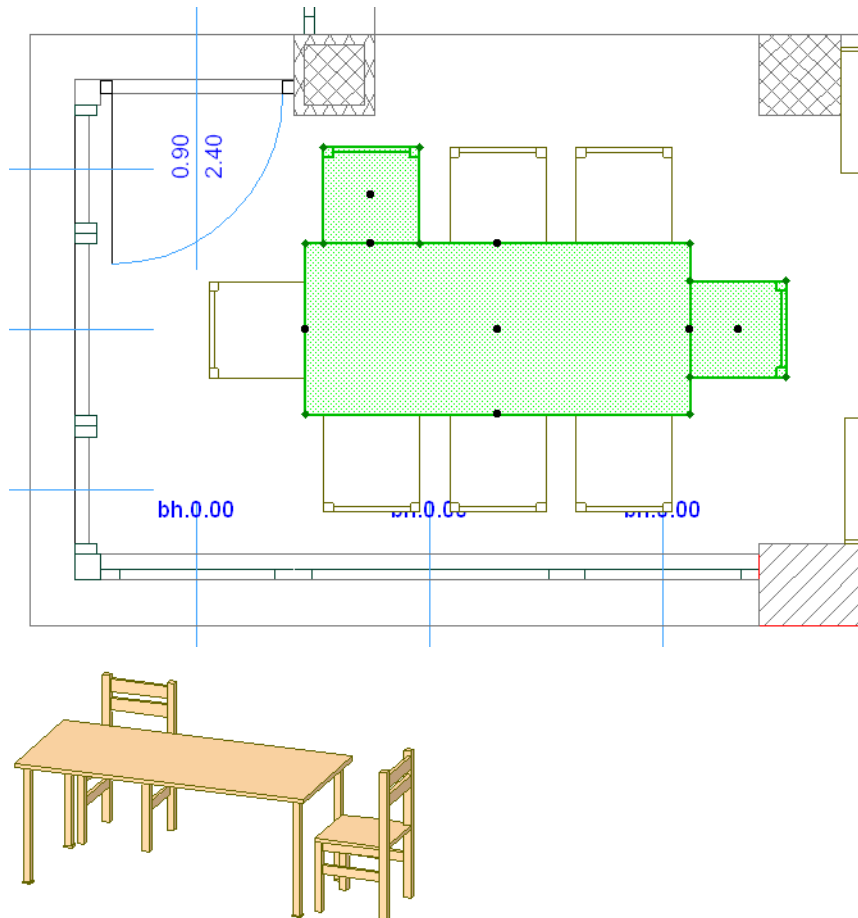
[Editing Plane in 3D Window](#)

Show Selection/Marquee in 3D

For a 3D display of only the items selected explicitly or by Marquee:

- Right-click to access the **Show Selection/Marquee in 3D** command; or
- Use **View > Elements in 3D View > Show Selection/Marquee in 3D**

(Or use the 3D Visualization toolbar control for this command.)

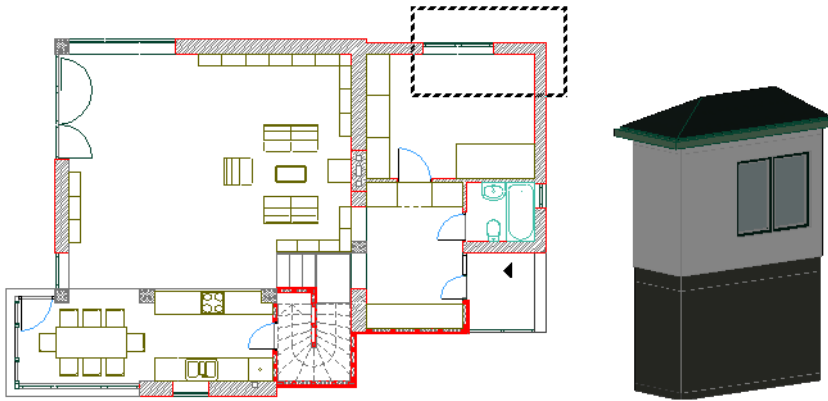


If your plan includes both explicitly selected elements and a marquee selection, this command will disregard the marquee and show only the explicitly selected elements.

Marquee Area in 3D

If the selection is a Marquee area, the caption of the 3D Window will include the “Marquee” mention.

For more information on defining a marquee area, see [Marquee Area](#).



By default, elements inside the Marquee are displayed in 3D.

To change this, use the Marquee Effect controls in **View > Elements in 3D View > Filter and Cut Elements in 3D**.

See [Filter and Cut Elements in 3D Dialog Box](#).

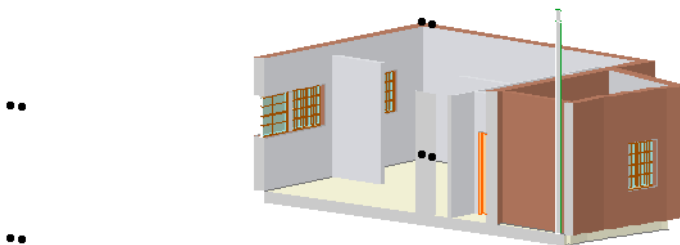
Elements in a Marqureed 3D Window

If you create an element in the 3D Window which is delimited by a marquee, the new element will only appear in the 3D Window if you draw it *inside* the marquee area.

If you place a new element outside the boundaries of a Marquee-based 3D view, or drag an existing element outside the Marquee boundaries, that element will disappear immediately from the 3D Window.

When you go to the Floor Plan, however, you will see the elements. Change your marquee area (or remove it) and return to the 3D Window to check it.

If you select one or more elements in a trimmed 3D view, resulting from either a Marquee area or a 3D Cutaway, some elements will be only partially visible, but selection dots of the whole element will be displayed and all the nodes and edges can be found with the cursor.



Redefine Image Settings (3D View)

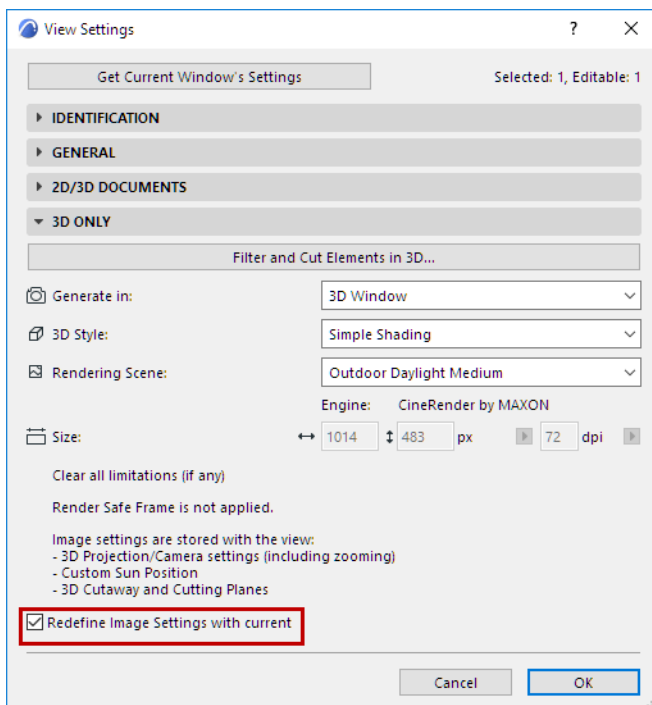
If you save the contents of your 3D window as a view, the **3D Only** panel of its View Settings dialog box will provide feedback on whether that view is a result of a filter or a marquee selection.

3D view settings include: 3D projection settings (including zooming), filter elements in 3D, 3D Styles, 3D cutting planes, 3D cutaway and photo rendering settings.

If you change any of these settings in the 3D window, then use the 3D Only panel of View Settings to redefine the view accordingly: check **Redefine Image Settings with current**.

This affects the listed “image”-type view settings only.

[See also *Redefine View with Current Settings*.](#)

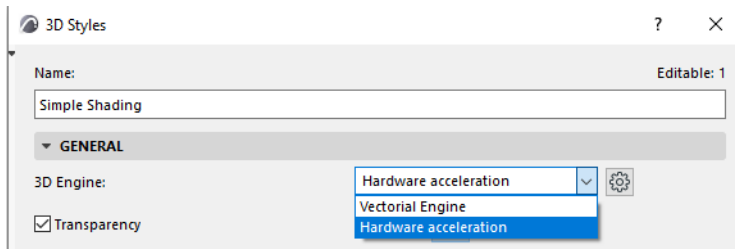


3D Engines

3D Engines are built-in configurations which control the generation of the 3D model in Archicad. By default, two engines are offered:

- Vectorial Engine
- Hardware acceleration Engine

To choose an engine, go to **View > 3D View Options > 3D Styles**.



The choice of engine affects the 3D display effects available, and thus the appearance of the model in 3D. Set up your 3D display preferences in the 3D Styles dialog box.

[See 3D Styles.](#)

3D Projections

Archicad can display parallel (axonometric) projections and perspectives.

Axonometric projections automatically show the entire model in the center of your view. Perspectives are defined by a viewpoint and a specific target.

When the 3D window is active, the **3D Navigator Preview Palette** contains a miniature representation of the entire project view. This allows you a quick way to modify your 3D Projection Settings, whether perspective or parallel (axonometric); you can adjust the view cone or change the axonometry while retaining an overview of the whole project.

[See Navigator Preview \(3D\).](#)

3D Projection Settings

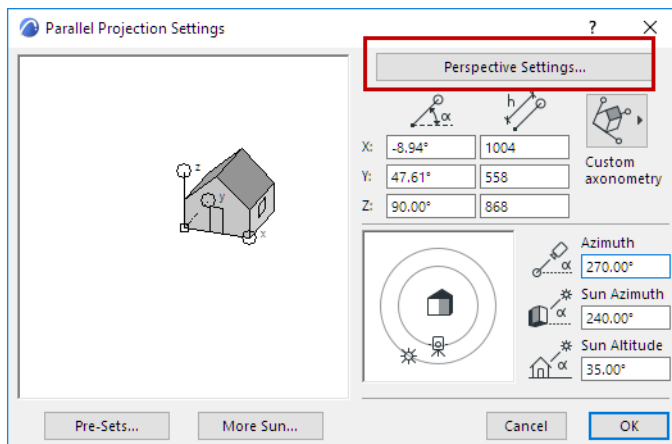
To open these settings, do one of the following:

- Use the context menu from the 3D window or the tab bar
- Use the 3D Visualization or Simple 3D toolbars
- Use the commands of the **View > 3D View Options** menu

The name and contents of the dialog box depend on the projection type:

- [Parallel Projection Settings](#) or
- [Perspective Projection Settings](#).

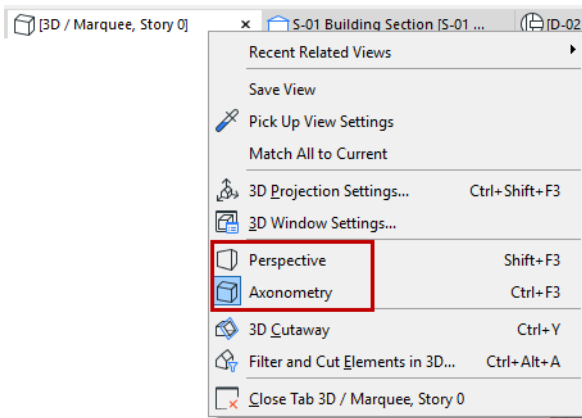
You can easily switch from one settings dialog to the other with the button at the top right corner of Projection Settings.



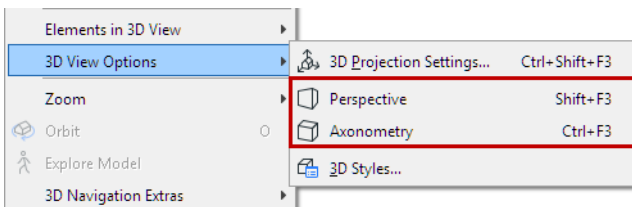
How to Switch Between Axonometric and Perspective Views

Do one of the following:

- Use the tab bar of the 3D window



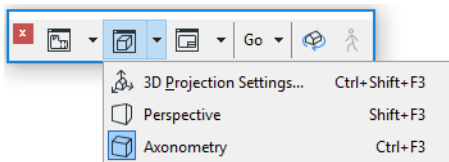
- Predefined shortcuts:
Ctrl+F3 = Axonometry
Shift+F3 = Perspective
- Commands available from **View > 3D View Options**:



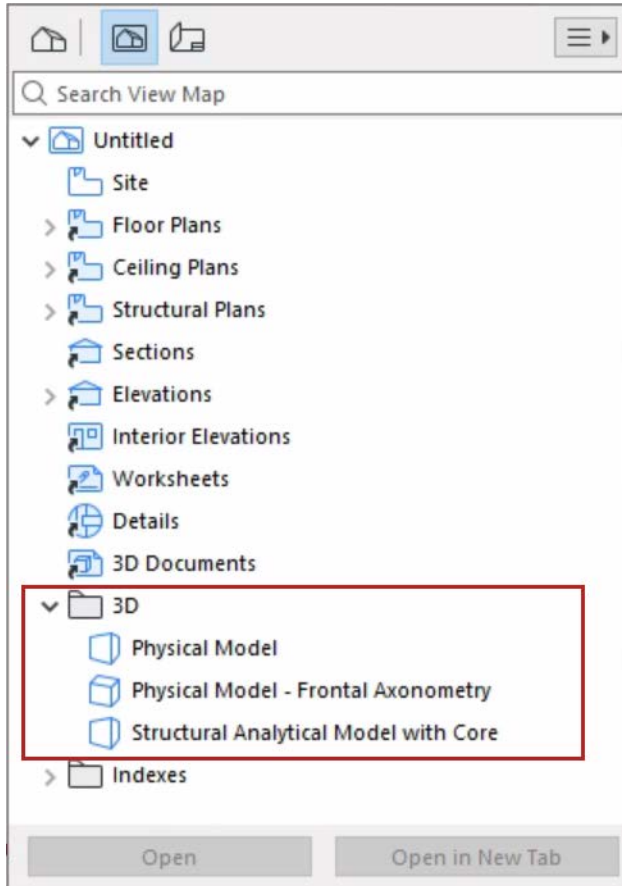
- 3D Visualization toolbar:



- Mini-Navigator toolbar



- Navigator:



How to Store a 3D Projection

To store projections that you set up in the 3D Window, you can:

- Use the usual **Save View** commands from the context menu of the Navigator or tab bar. This will create a view (perspective or axonometric) based on the currently set up projection in the 3D window. The new view will be added to your Navigator View Map.
- Add the current perspective to your Project Map, using the **View > 3D Navigation Extras > Place a Camera into the path** command. A camera capturing the current perspective will also appear on the Floor Plan.

[See also Cameras.](#)

- Add the current axonometric (parallel) projection to the series of Pre-Set views, using the **View > 3D Navigation Extras > Add Current Projection** command.

[See also Pre-Set Projections and 3D Navigation Extras.](#)

3D Navigation Extras

Other controls useful in modifying the 3D view are available from the **View > 3D Navigation Extras** (or a toolbar containing 3D window commands). These controls allow you to:

- Interactively set the target point for the perspective (**Look to**).
- Click on a surface to switch to a view perpendicular to the clicked point (**Look to Perpendicular of Clicked Surface**).
Note: When using this command in a parallel projection (axonometry), Archicad defines the new projection based on your click (direction) and the previous view rectangle (extent of projection plane). If this produces unexpected results, redefine the view rectangle (e.g. with Fit in Window), then try again.
- Reset the roll angle to zero to rapidly return from a bizarre view obtained during navigation (**Reset Roll Angle**).
- Return with a single command to a horizontal view of the model (**Horizontal View**).
- **Edit pre-set projections** and **Add current projection**: These commands are available if you are viewing an axonometric (parallel) projection.
- **Put a camera into the path** (for perspectives). This command adds the current perspective to the Project Map, if no camera is selected on the Floor Plan. If you have cameras on the Floor Plan already that define an animation path, the new camera will be added after the active camera in the active path, which is marked by its view cone on the Floor Plan.

If you are in perspective view, with a camera selected, the following commands are also available from **View > 3D Navigation Extras**:

Note: As you use these commands to change the camera selection, the selected item in the Navigator Project Map also changes accordingly.

- **Go to the previous/Go to the next**: Use these commands to navigate to the perspective views of the previous and next cameras in the active path, and to select the camera.
- **Modify the selected**: The selected camera will be updated to reflect the perspective view that you currently see in the 3D window. Use this when you have changed the view in the 3D window.
- **Revert view to the selected**: Discards all changes you made manually and shows the 3D model as the selected camera sees it.
- **Insert a new camera after the selected one**: A new camera is added to the path to define the current perspective view, and will be selected.

Related Topics:

[Navigation in the 3D Window](#)

[Cameras](#)

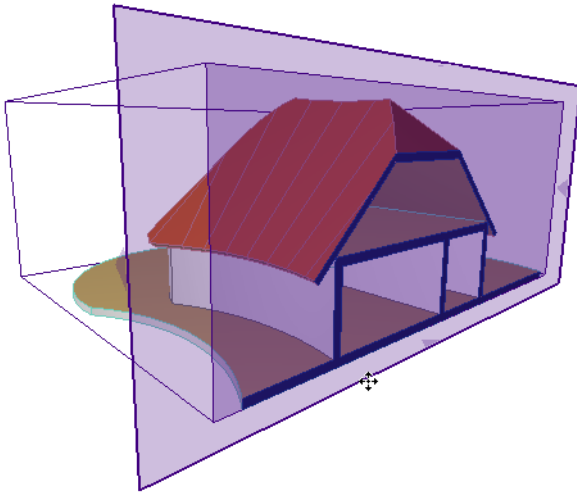
[Pre-Set Projections](#)

[Classic 3D Navigation Toolbar](#)

3D Cutting Planes

3D Cutaway is an imaging mode in Archicad that allows flexible visualization of building spaces.

3D Cutaway works by applying Cutting Planes to the model (Floor Plan, Section, or 3D window).



The 3D Cutaways you define are stored within the project, so you can see them when opening the project in a future session. You can apply up to 9 Cutting Planes at a time.

3D Cutaways are saved with views, so if you wish to keep Cutting Planes over the long term, save a 3D view.

To see the cutaway view of your model, turn on Show 3D Cutaway. To see the full model, turn it off.

Related Topics:

[Show or Hide 3D Cutaway](#)

[Create a 3D Cutaway on the Floor Plan or Section](#)

[Create a 3D Cutaway in the 3D Window](#)

[Edit Cutaway](#)

[3D Cutaway Attributes](#)

[Hide 3D Cutting Planes](#)

[Delete Cutting Planes](#)

Show or Hide 3D Cutaway

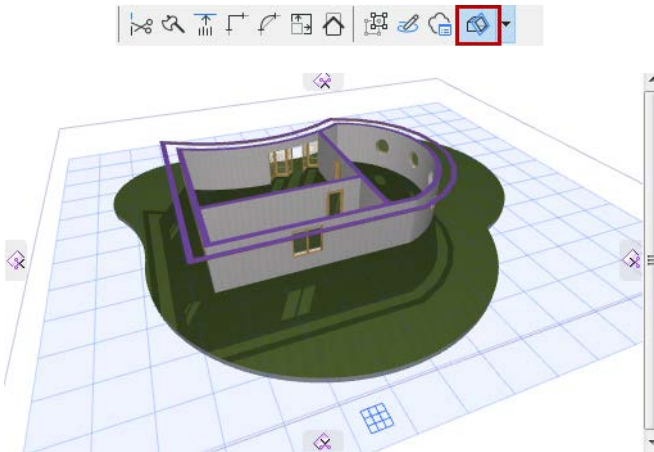
Do one of the following:

- Click the 3D Cutaway toggle on the Standard Toolbar.

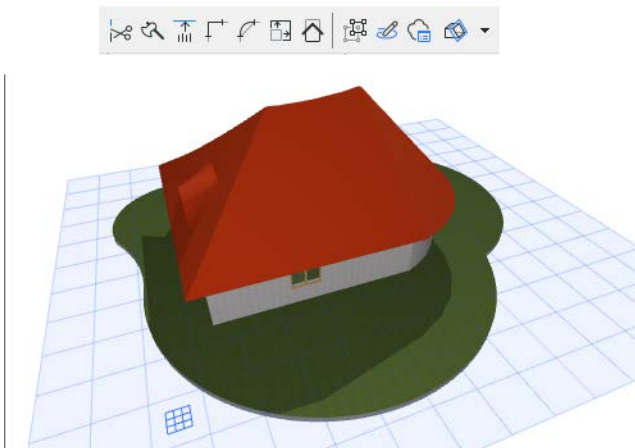


- Use the following toggle command: **View > Elements in 3D View > 3D Cutaway**

When 3D Cutaway is **on** (show), you see the cutaway view of your model, plus the 3D Cutaway interface.



When it is **off**, you see the full model.

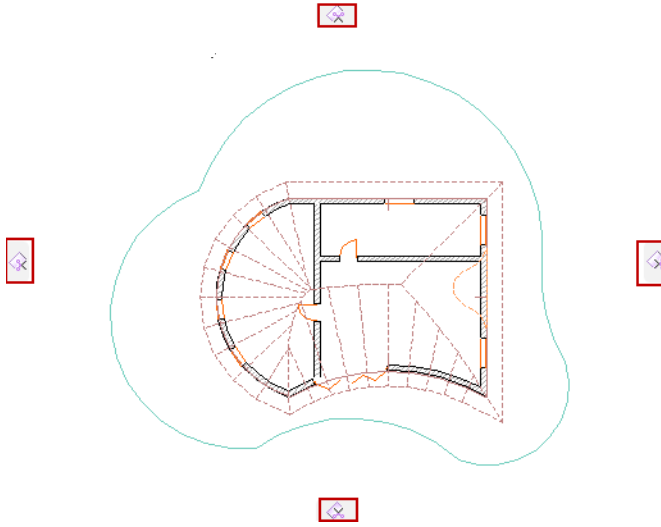


Create a 3D Cutaway on the Floor Plan or Section

1. Make sure 3D Cutaway is On.

See [Show or Hide 3D Cutaway](#).

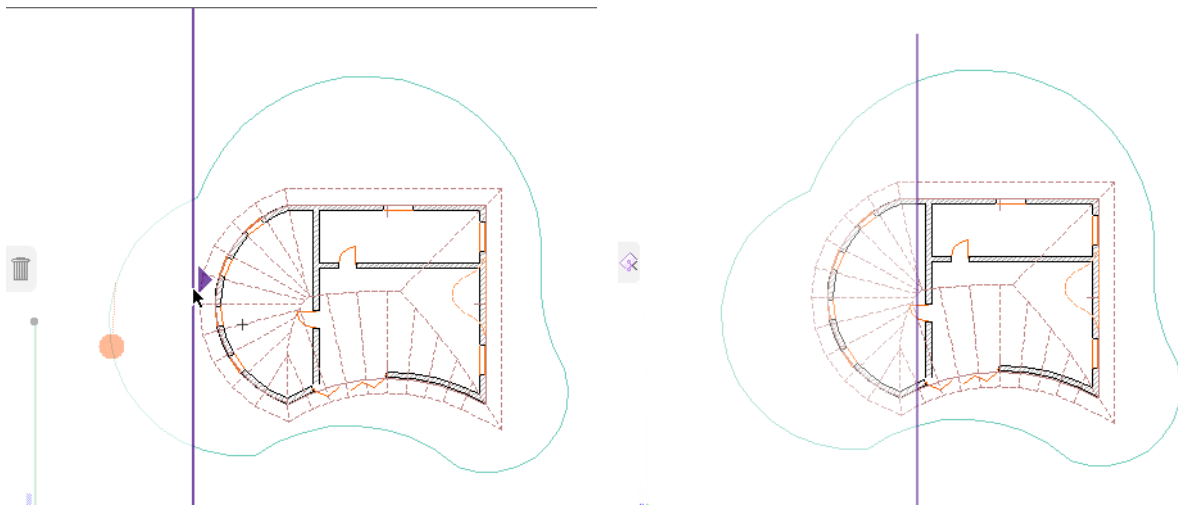
2. The window will display Cutting Planes handles on all four sides of the screen.



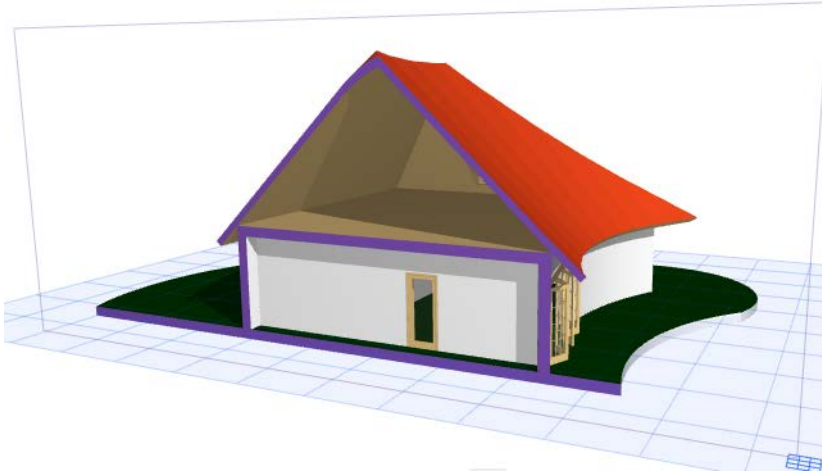
3. Click and drag one or more tabs to the desired location. By default, you will drag the Cutting Plane orthogonally.

4. Click to place the Cutting Plane at the desired location.

To cancel the operation: Note that the tabs on all four sides change into garbage cans: if you change your mind while dragging, just move the Cutting Plane back into one of the garbage cans (or press Esc) to cancel the operation.



5. View the resulting 3D Cutaway.



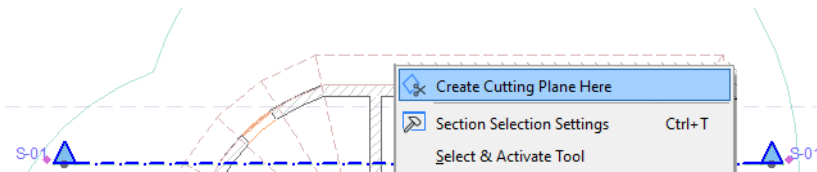
To hide the Cutting Plane:

See [Hide 3D Cutting Planes](#).

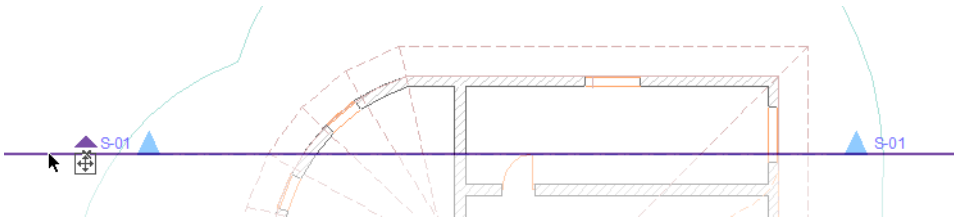
Use Section Line as Cutting Plane

It may be convenient to use an existing Section as the basis for your 3D Cutaway.

Select the Section line, and choose **Create Cutting Plane Here**.



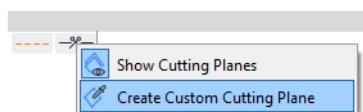
The Cutting Plane coincides with the Section Line, but is independent of it. You can now move and edit the Cutting Plane if needed; the Section remains unchanged.



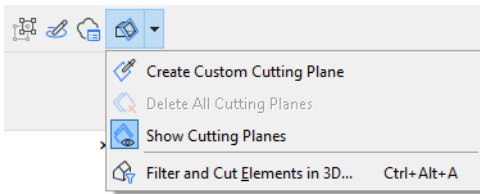
Create Custom Cutting Plane on Floor Plan or Section

If you wish to cut the model from a different direction than the predefined orthogonal directions, create a custom Cutting Plane.

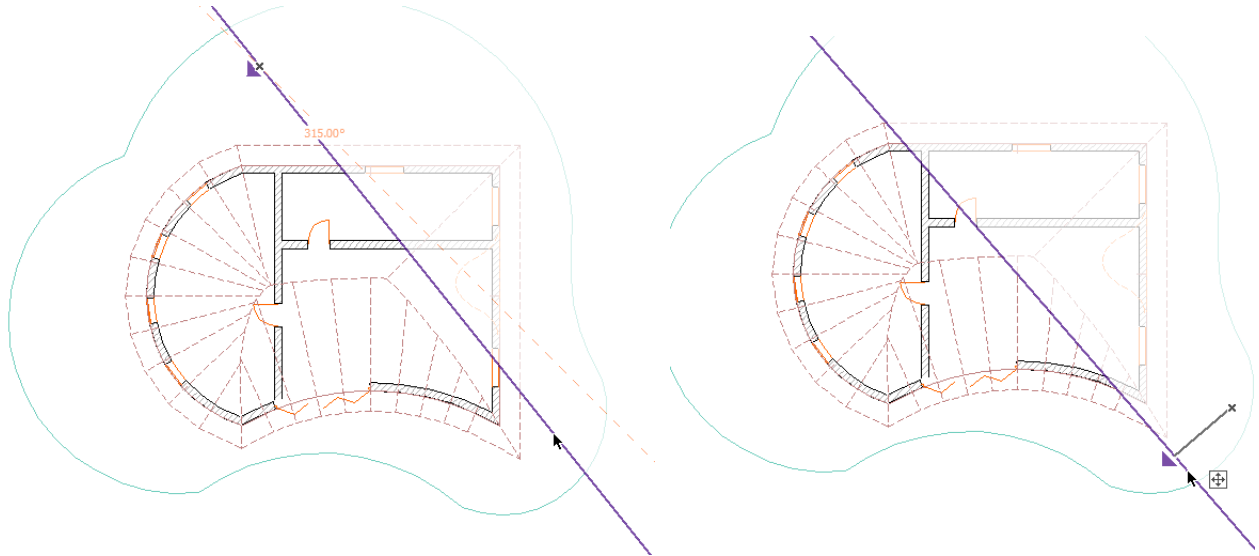
1. From any of the four Cutting Plane tabs, click to bring up the context menu.
2. Choose **Create Custom Cutting Plane**.



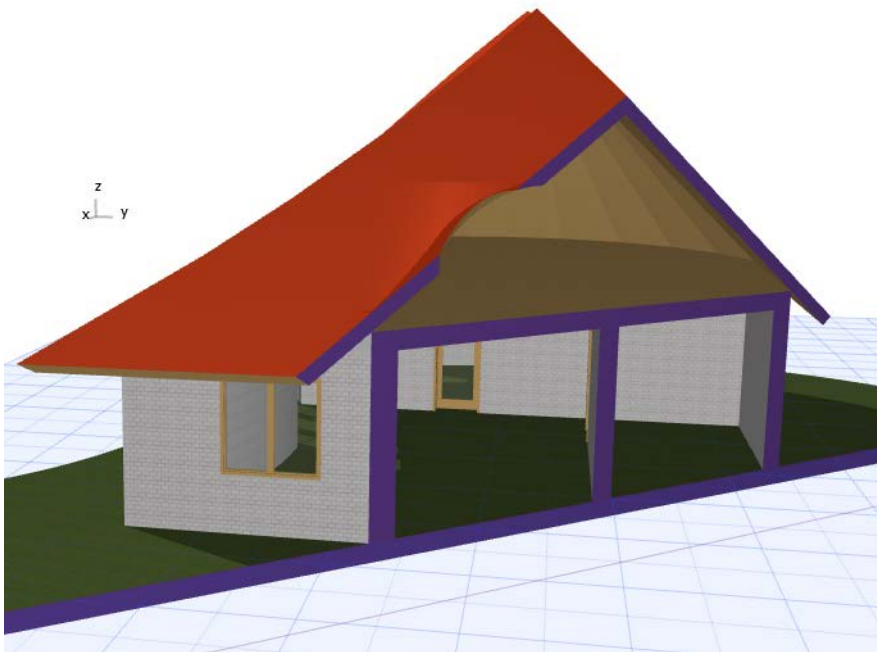
Note: This command is also available from the Standard Toolbar.



3. Click twice to define the cutting line.
4. Click on the cutting line, then drag it to the desired location to create the Cutaway.



5. View the resulting 3D Cutaway.



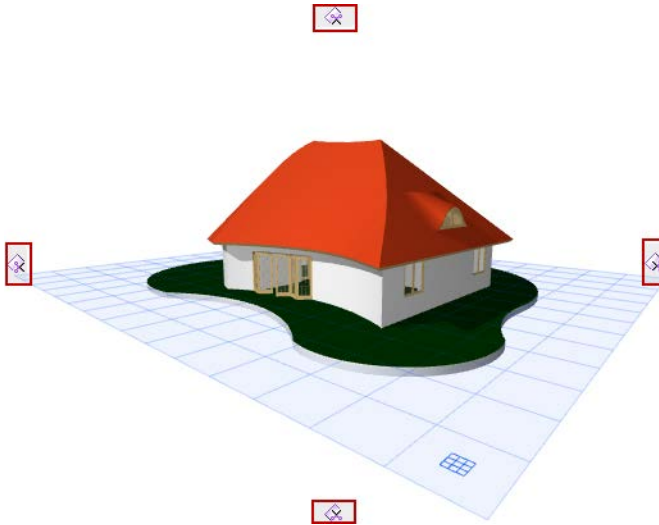
[Watch the Video](#)

Create a 3D Cutaway in the 3D Window

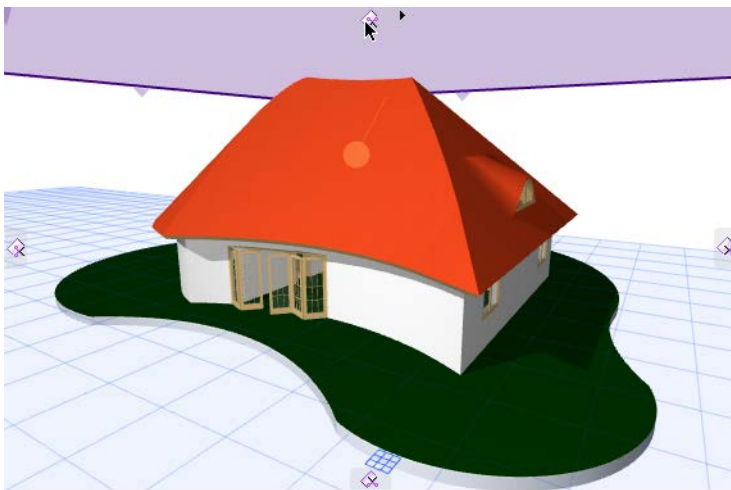
1. Make sure 3D Cutaway is On.

See [Show or Hide 3D Cutaway](#).

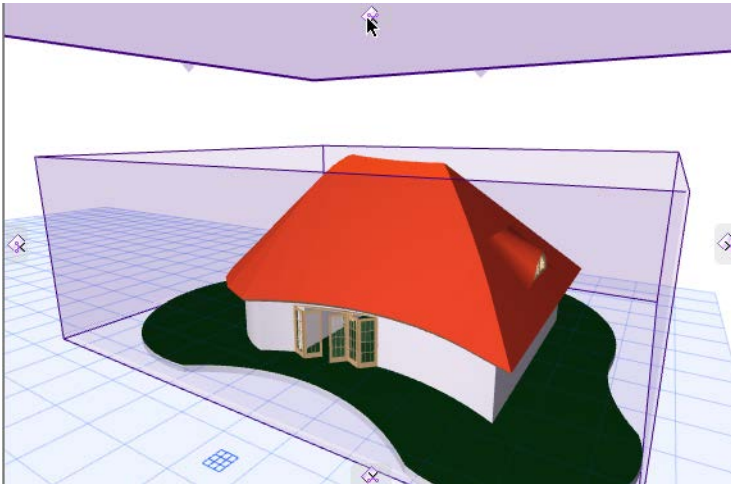
2. The window will display Cutting Planes tabs on all four sides of the screen.



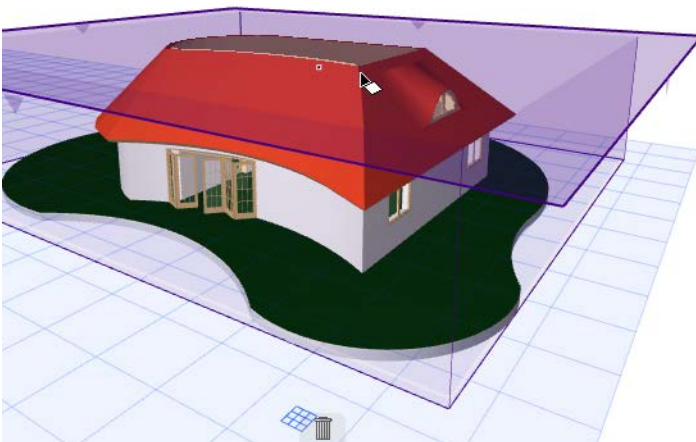
3. Hover over any of the tabs to see the orthogonal Cutting Plane in that position.



4. As soon as you click one of the tabs, you will see the model's bounding box. The predefined Cutting Planes will cut the model on planes that are parallel to this bounding box.

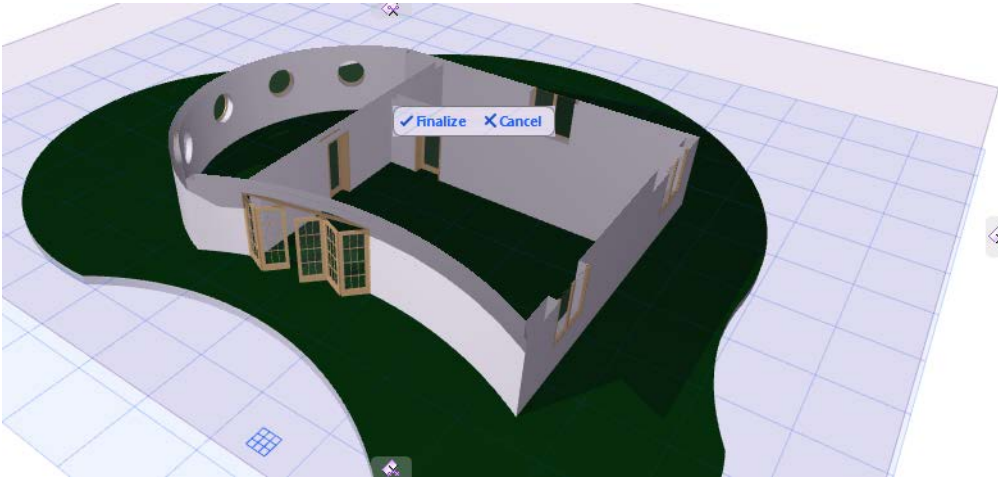


5. Drag the tab to cut the 3D model orthogonally.
[See also *Create Custom Cutting Plane in 3D Window.*](#)

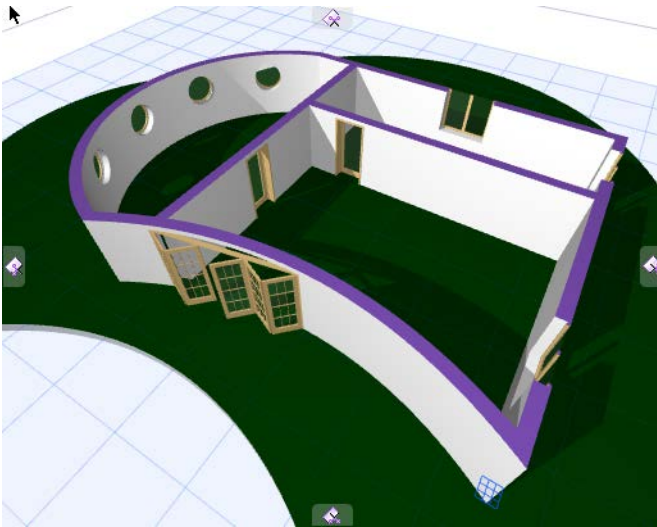


To cancel the operation: Note that the tabs on all four sides change into garbage cans: if you change your mind while dragging, just drag the Cutting Plane into one of the garbage cans (or press Esc) to cancel the operation.

- Click when the Cutting Plane is in the right place. The **Finalize** floating command appears.



- Click **Finalize**, or **Cancel** to undo the operation.
- View the resulting 3D Cutaway.



To hide the Cutting Plane:

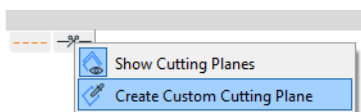
[See *Hide 3D Cutting Planes*.](#)

Create Custom Cutting Plane in 3D Window

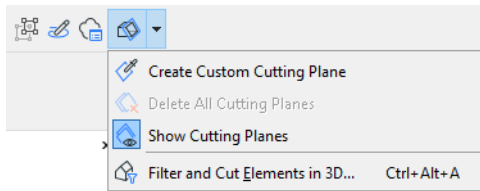
If you wish to cut the model from a different direction than the predefined orthogonal directions, you can create a custom Cutting Plane.

Make sure 3D Cutaway is On.

- From any of the four Cutting Plane tabs, click to bring up the context menu.
- Choose **Create Custom Cutting Plane**.



Note: This command is also available from the Standard Toolbar.



3. Define the custom Cutting Plane orientation by doing one of the following:

- Use an element's plane (e.g. a Roof top plane)
 - Click on the plane surface of an existing element to define the Cutting Plane angle. A rectangle will help you see the plane.
 - Click again to define the Cutting Plane position
- Use an edge as a rotation axis of the plane
 - Click on an existing edge with the Mercedes cursor, to define the rotation axis
 - Click on any point that the Cutting Plane should touch; this defines the Cutting Plane position
- Click on any three points (such as three element hotspots) to define a plane
 - Click on two points to define the rotation axis
 - Click on any point that the Cutting Plane should touch; this defines the Cutting Plane position

4. Click when the Cutting Plane is in the right place. The **Finalize** floating command appears. Click **Finalize** if you are satisfied with the Cutting Plane.

5. View the resulting 3D Cutaway.

[Watch the Video](#)

Edit Cutaway

Once you have created a 3D Cutaway, you can still edit it at any time.

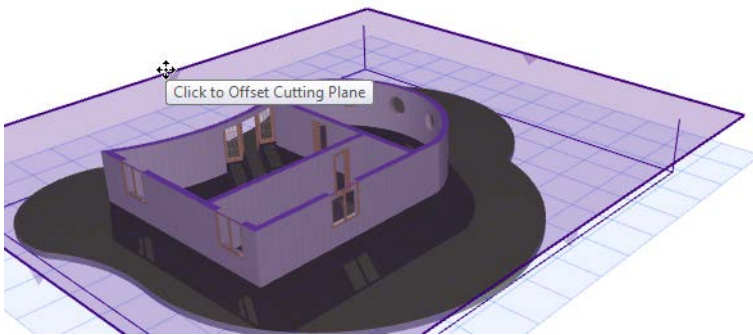
Make sure 3D Cutaway is activated, and Show Cutting Planes is on.

See [Show or Hide 3D Cutaway](#).

Offset Cutting Plane

Grab the plane (click on any of its edges) and offset it to the desired location. Click to place, then click Finalize.

Note: After the initial offset, before you click Finalize, you can click anywhere on the plane surface to adjust the offset further.

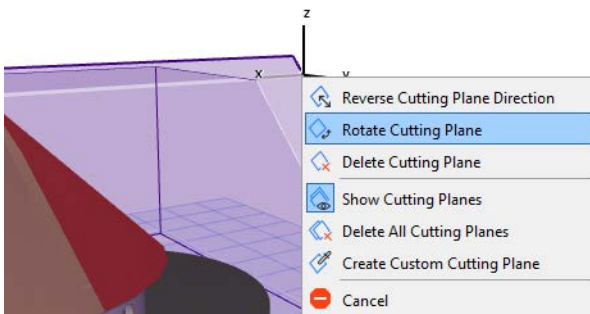


Rotate Cutting Plane

Right-click the edge of a Cutting Plane. Choose **Rotate Cutting Plane** from the context menu.

Then rotate the plane as you would other elements.

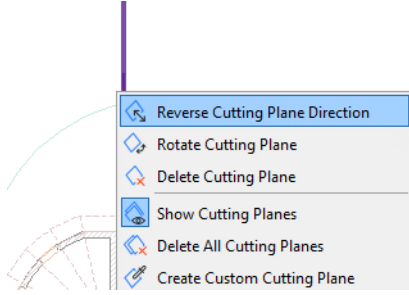
See [Rotating Elements](#).



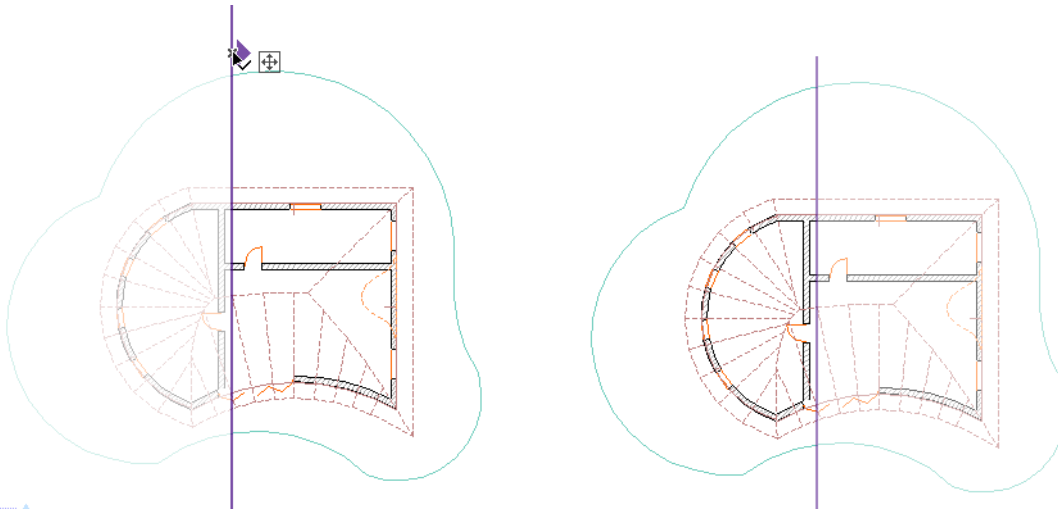
[Watch the Video](#)

Reverse Cutting Plane Direction

Right-click the edge of a Cutting Plane. Choose **Reverse Cutting Plane Direction** from the context menu.



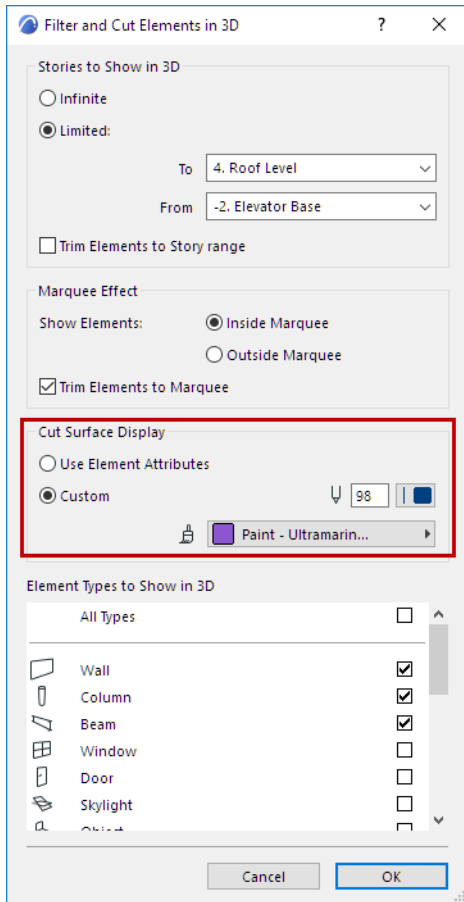
The Cutaway is now created on the opposite side of the selected Cutting Plane.



3D Cutaway Attributes

Use the Cut Surface Display controls in the following dialog box:

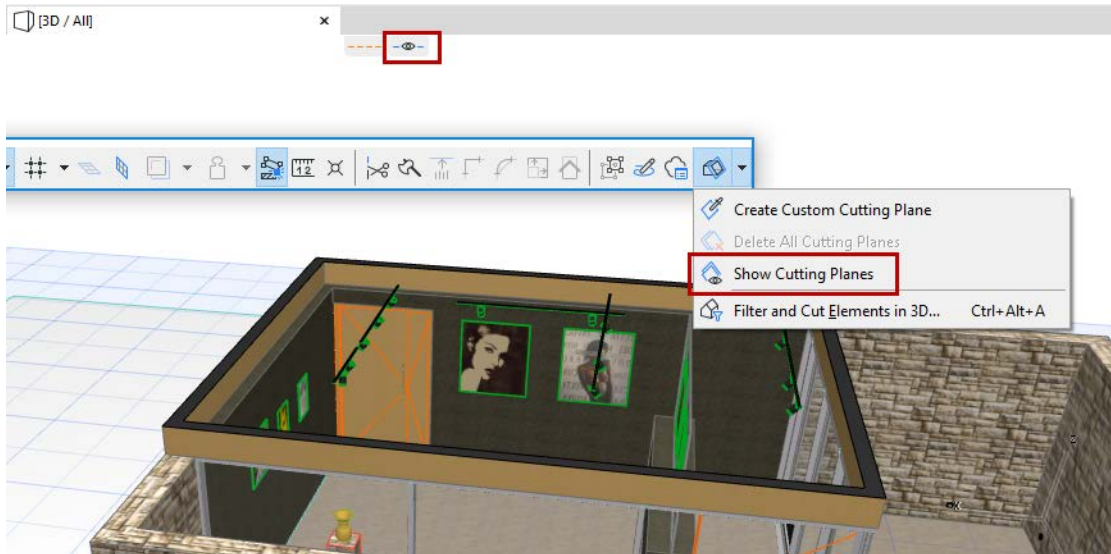
View > Elements in 3D View > Filter and Cut Elements in 3D



Hide 3D Cutting Planes

Do one of the following:

- Right-click the Cutting Plane to bring up its context menu, and deactivate the **Show Cutting Planes** toggle
- Deactivate the **Show Cutting Planes** toggle from the drop-down 3D Cutaway menu of the Standard toolbar.

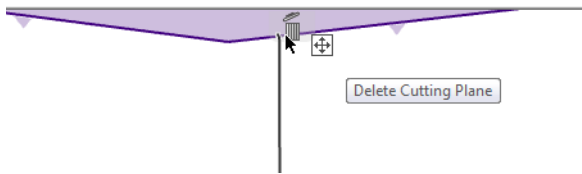


The 3D Cutaway is in effect, but the Cutting Plane and the tabs are hidden, except for the tab at the top of the screen.

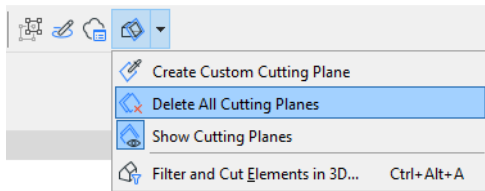
To show the Cutting Planes again, click the toggle command or the eye icon at the top of the screen.

Delete Cutting Planes

At any time, with 3D Cutaway on, grab an existing Cutting Plane or line and drag it into the nearest garbage can to delete it.



To delete all Cutting Planes, use the **Delete All Cutting Planes** command from the Standard toolbar



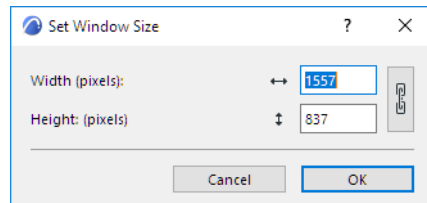
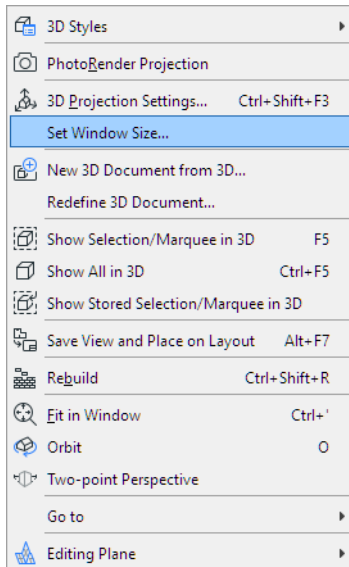
or from **View > Elements in 3D View > 3D Cutting Planes**.

[Watch the Video](#)

3D Window Size

Use the controls at **Set Window Size**:

- from the 3D Window context menu
- from the 3D Window tab context menu



Enter the horizontal and vertical size of the 3D Window in pixels (maximum 16,000)

Click the Chain icon to keep the proportion locked.

Physically based renderer in 3D Window - Experimental feature

Try physically based rendering in the Archicad 3D window - instead of the default renderer.

Physically based rendering delivers a much more vivid and realistic appearance while you explore the 3D model. The feature simulates the soft shadows that naturally occur when indirect or ambient lighting is cast onto a scene. The lighting remains constant while the user navigates in 3D (no headlight effect).

Enable this feature

1. Go to Options > Work Environment > More Options > Experimental Features.
2. Check the box to **Enable Physically Based Rendering in 3D**.

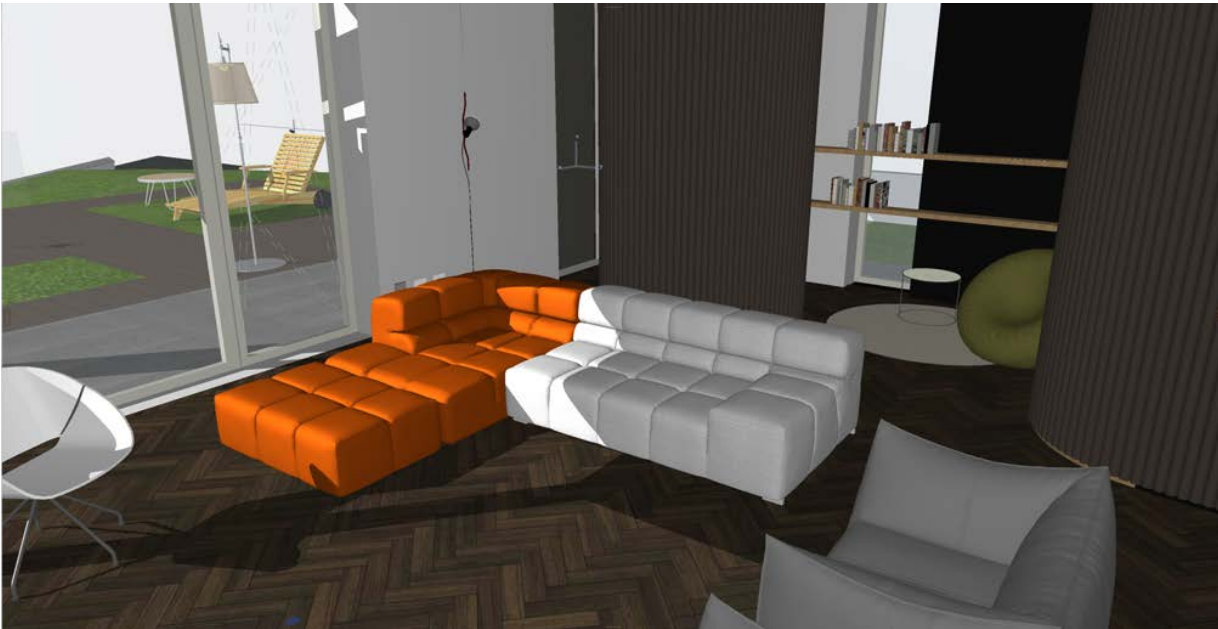
This is an experimental feature, which may change or disappear in a future Archicad release. Please try it out!

See [Experimental Features](#).

After enabling the feature, open the 3D window to explore the design and see the difference.



Physically based rendering OFF



Physically based rendering ON

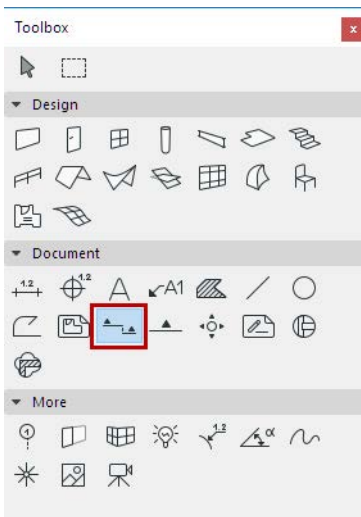
You can fine-tune the 3D visualization using 3D Styles or Sunlight settings.

Limitations:

- Contours (in 3D Styles) are not currently visible with physically based rendering. (See [3D Styles](#))
- Color and intensity settings for Sunlight (in Sunlight settings) do not affect physically based rendering. But you can adjust the color and intensity of Ambient light. (See [Sunlight Settings](#))

Sections

Use the Section tool to generate a sectional viewpoint of the Virtual Building, with an accompanying marker.



The Section marker can take one of three different forms:

1. a **source marker**, which generates a section viewpoint
2. a **linked marker**, which does not generate a section viewpoint, but acts as a reference to any other view, viewpoint or drawing.
3. an **unlinked marker** containing custom text

Note: Another type of Section viewpoint is the Independent Section (with or without a marker), having no model source.

[See *Create an Independent Section Viewpoint*.](#)

Sections can be saved as Views and placed on a Layout as Drawings; the contents of a Section window can also be published directly.

In the Section window, you can view and modify elements but you cannot create new construction elements. (The only exception is if you use the “**Drag a copy**” or **Multiply** command.) If you paste an element into a Section window, it will be reproduced as drawing primitives (points, lines, fills).

Objects can be placed in a Section window, but they are considered as graphic symbols only. (No corresponding 3D model element is generated.)

Related Topics:

[Create a Section Viewpoint](#)

[Open a Section Viewpoint](#)

[Horizontal/Vertical Range of Section Viewpoint](#)

[Define Marker Reference for Source Marker](#)

[Floor Plan Display of Section Markers](#)

[Model Appearance \(Section-Type Views\)](#)

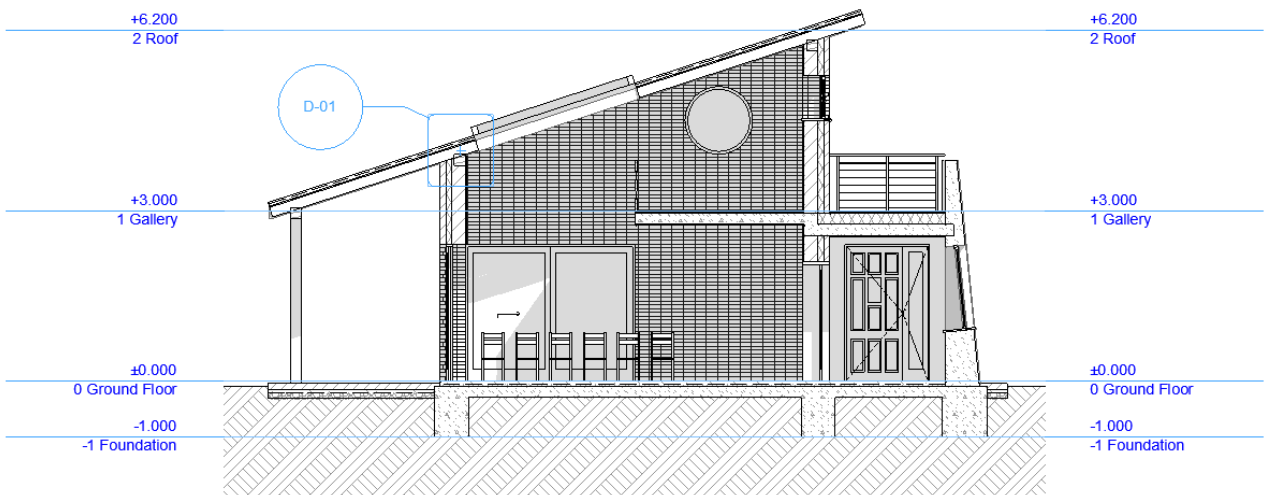
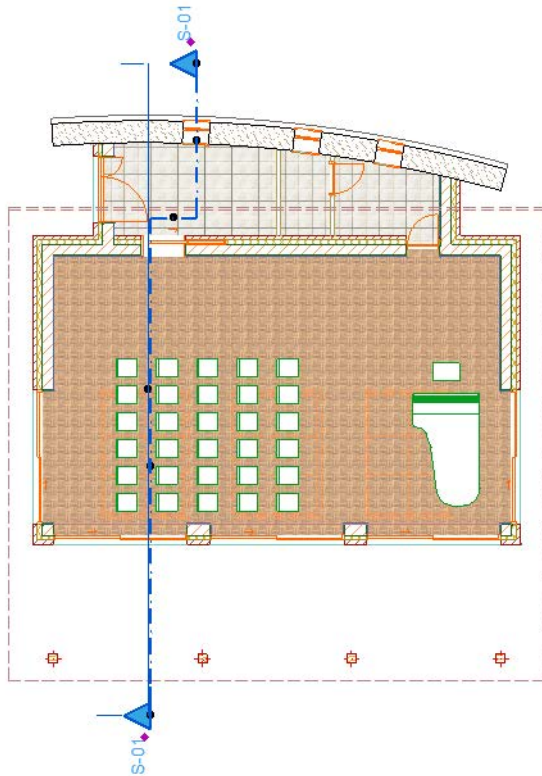
[Editing Section/Elevation/IE Lines on Floor Plan](#)

[Place a Linked Section Marker](#)

[Place Unlinked Marker](#)

[Create an Independent Section Viewpoint](#)

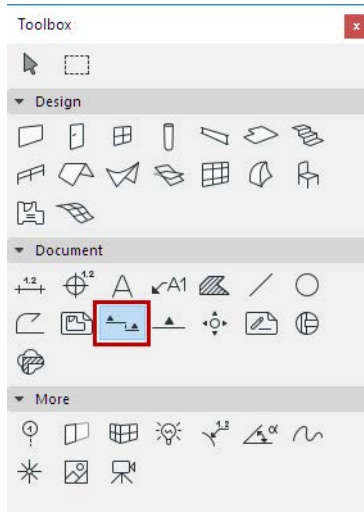
Assign Section Rebuild Status
Updating Sections
Summary of Rebuild Commands



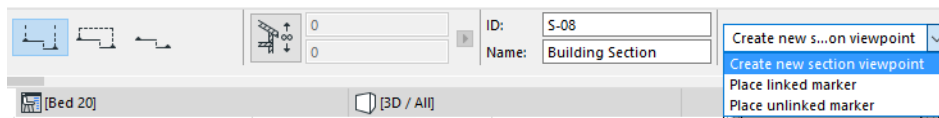
Create a Section Viewpoint

Note: The creation process is identical for Elevations.

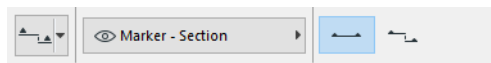
1. Activate the Section tool.



2. In the Info Box or Section Default Settings, make sure that “Create new section viewpoint” is selected.



3. Choose an input method (either Single or Staggered) from the Info Box and draw a Section line on the Floor Plan.



- Single: Click twice to define each end of the line.
- Staggered: Click as many times as needed to define each segment of the Section line. Double-click to complete the input line.

The **Eyeball** cursor appears.

4. With the eyeball cursor, click on either side of the line to set the orientation of the Section. The place where you click also defines the Section’s limit line, if you have chosen a “Limited” horizontal range in Section Tool Settings.
5. The new Section source Marker is automatically placed after the section line is completed. (If you are placing a Section of limited horizontal range, the limit line is also placed automatically.)
6. A new Section viewpoint is created and listed in the Navigator Project Map.

Distant Elements Limit

Check the **Fade Distant Elements** box in Section Settings (Model Appearance Panel) to create an optional “distant” area of your Section viewpoint, with separate colors/effects. The Distant Elements limit line is then displayed as part of your Section, and is editable.

See [Model Appearance \(Section-Type Views\)](#).

Related Topics

[Open a Section Viewpoint](#)

[Horizontal/Vertical Range of Section Viewpoint](#)

[Editing Section/Elevation/IE Lines on Floor Plan](#)

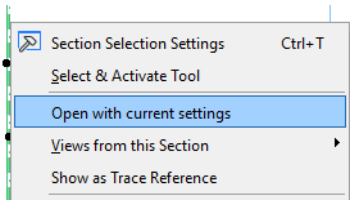
[Define Marker Reference for Source Marker](#)

[Model Appearance \(Section-Type Views\)](#)

Open a Section Viewpoint

To open a Section viewpoint, do one of the following:

- Double-click its name in the Navigator
- Select the Section line on the Floor Plan and use one of the “**Open**” commands from the context menu



- Use a menu command (**Window > Navigate > Sections > Open Section**)

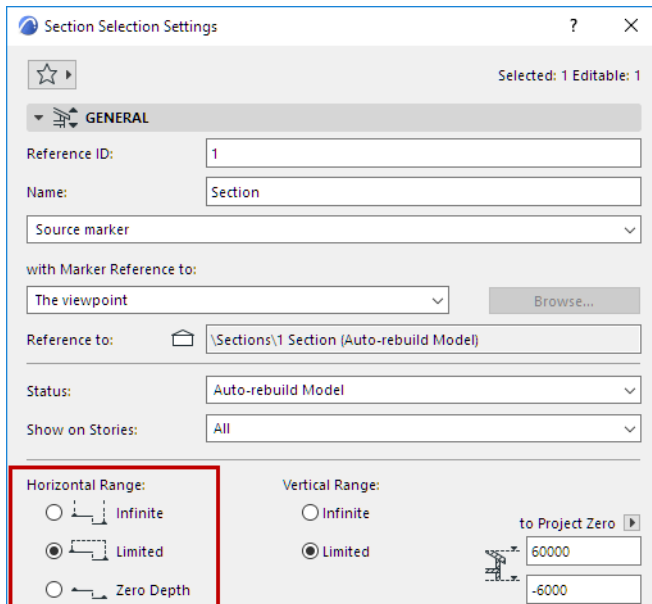
By default, each Section is opened in a new tab of the Tab Bar (or a new window, if you are not using the Tab Bar.)

Horizontal/Vertical Range of Section Viewpoint

Horizontal Range

The horizontal range is the depth of the Floor Plan that will be included in the Section window. (Available for source-type Section markers only.)

Define the horizontal range in the Info Box/Section Settings:



- **Infinite:** All elements behind the Section Line will be shown in the Section window, provided that they are not hidden by other elements.
- **Limited:** Only the elements between the Section Line and the limit (depth) line will be shown in the Section window. (The limit line is defined when you click with the Eyeball cursor after you finish drawing the Section line.) You can select and edit the limit line as necessary
Note: The limit (depth) line is a display-only Marker item, and is not shown on the Layout.

See [Display of Marker Range Lines](#).

- **Zero Depth:** Only the parts of the model actually cut by the Section Line will be shown, at the cut location only. (For Elevations and Interior Elevations, there is no “zero depth” option.)

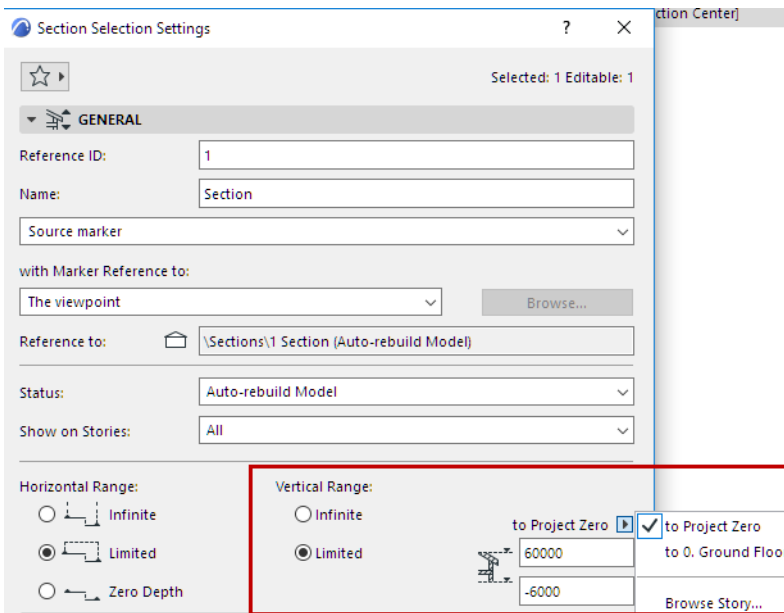
Vertical Range

The vertical range of the project to be included in the Section window. (Available for source-type Section markers only.)

Define the vertical range in the Info Box/Section Settings.

- **Infinite:** Shows all stories of the model.
- **Limited:** The Section is confined to a limited vertical range. Enter the elevation values for the upper and lower limits of the Section. Elements above and below these limits will not appear.

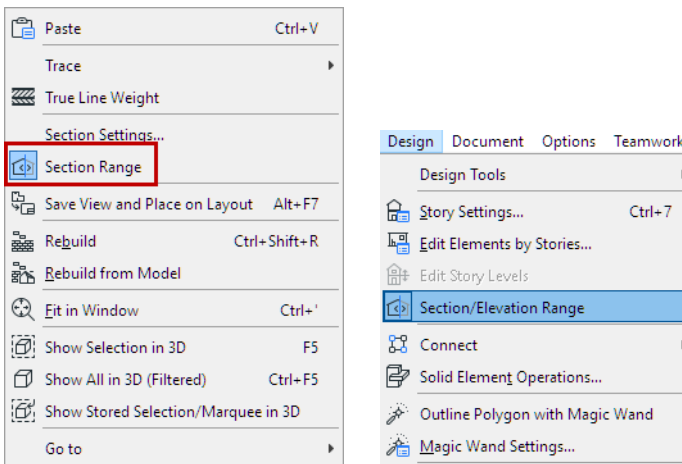
Click the black arrow to choose the reference for the elevation values: to Project Zero, or to a particular story.



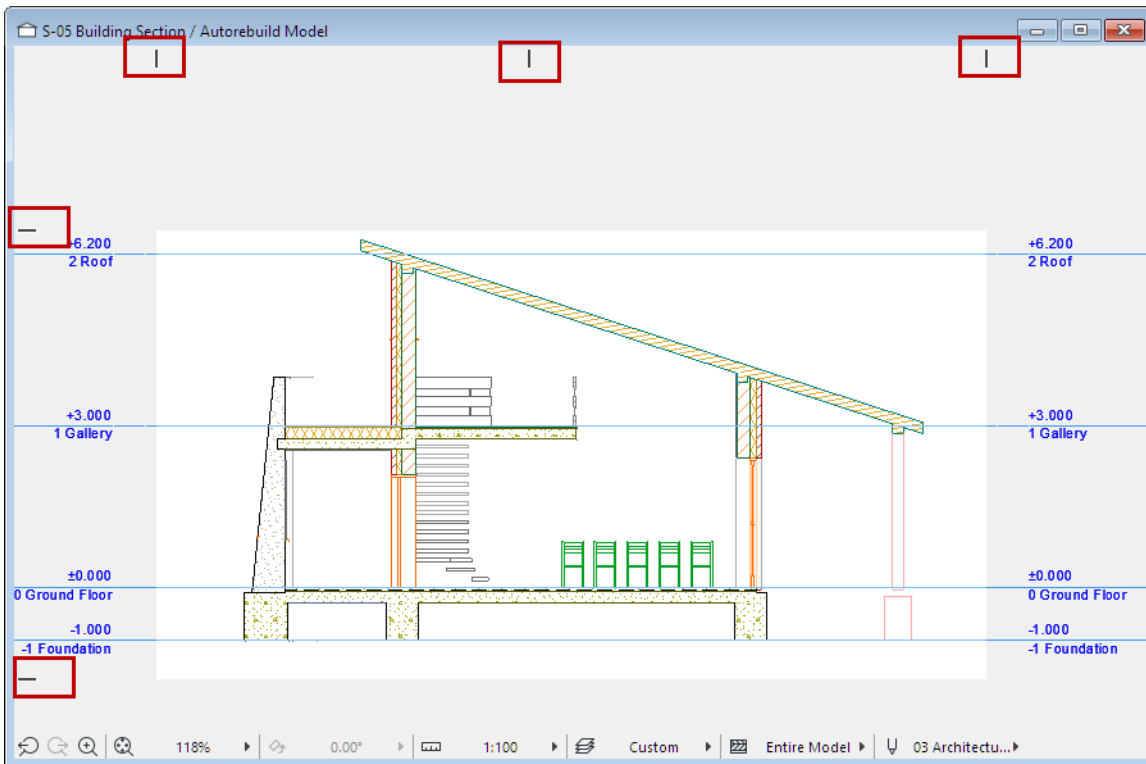
Edit Range or Limit Lines Graphically

With the Section/Elevation/Interior Elevation window active:

1. Turn on the Range feedback from the context menu, or using **Design > Section/Elevation Range**.

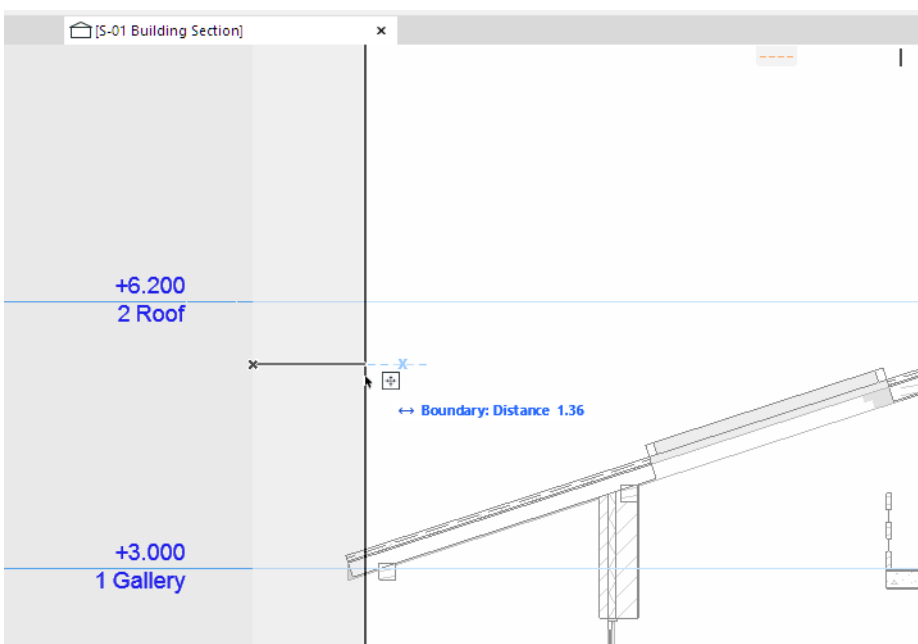


2. Marks appear in the Section window, in shades of dark grey, to indicate the horizontal/vertical boundaries and break lines, if any. The grey fill represents the area of the model outside of the Section range.



3. Hover over any mark to display its tag and editing handle. Drag the line as needed: this will modify the horizontal/vertical range, break mark, or distant elements limit as applicable. To remove a break, drag the break line outside of the horizontal range.

Note: Horizontal range cannot be edited in the Interior Elevation window.

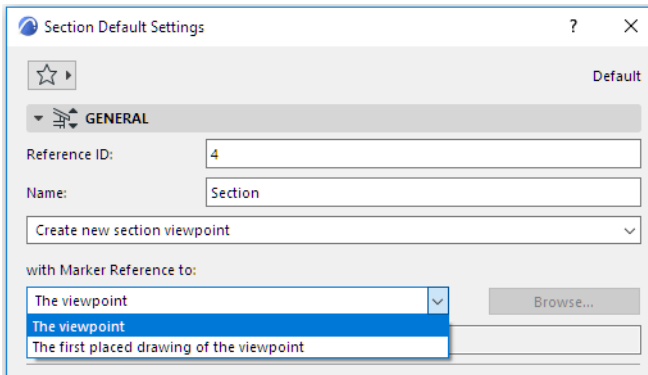


[Watch the Video](#)

Define Marker Reference for Source Marker

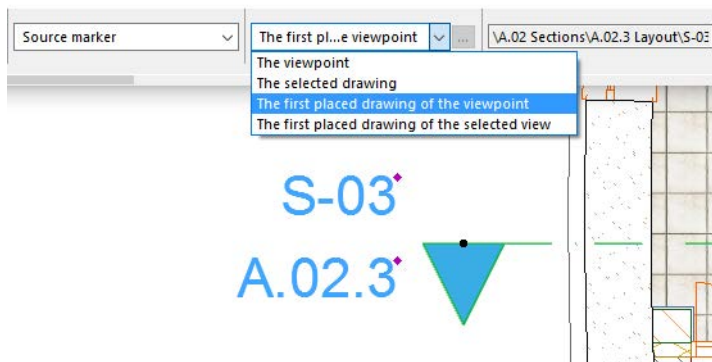
Follow these steps to define reference information for source-type Section, Elevation, Interior Elevation, Detail and Worksheet markers.

When placing a **new source marker** on the Floor Plan to create a viewpoint, the tool's Default Settings gives you two choices for defining the Marker Reference:



- the viewpoint, or
- the first placed drawing of the viewpoint. (“First” means the topmost eligible drawing in the Navigator Layout Book.)

For example, the following Section has been defined to display the information of the first placed drawing created out of this viewpoint.



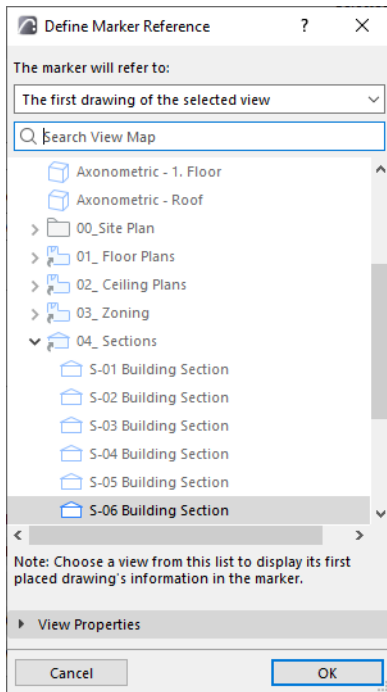
If you choose “first placed drawing,” and no drawing has been placed yet, the marker will display autotext (such as #DrgID), and once the drawing is placed, this autotext will be replaced by the relevant information.

The same marker information is shown in the Marker Panel in the Section Settings dialog box.

If you select an **already placed source marker** and open the tool's Selection Settings or Info box, you have two additional choices, because the viewpoint may already have multiple drawings or views created out of it:

- the selected drawing
- the first placed drawing of the selected view

Choosing either of these two options will bring up a directory dialog box showing the Navigator Layout Book or the Navigator View Map. Eligible drawings/views are available for selection (i.e., those which have been created from the selected Section viewpoint); other items are grayed and cannot be selected.



Note: The option to refer to a view is available in case the desired Drawing is located in a different project's Layout Book (and is therefore not listed in the "Define Marker Reference" dialog box).

See [Place view from external Archicad project \(menu command\)](#).

Since you cannot refer to the actual drawing, you can refer to the View (in the current project) from which the Drawing was created (in the other project). The marker will display the drawing data of the first drawing created out of this view as autotext (e.g. #Drawing Name, #DrgID). Once the Layout Book that contains the reference drawing is opened alongside the project containing the marker, the marker will fill in the correct drawing data.

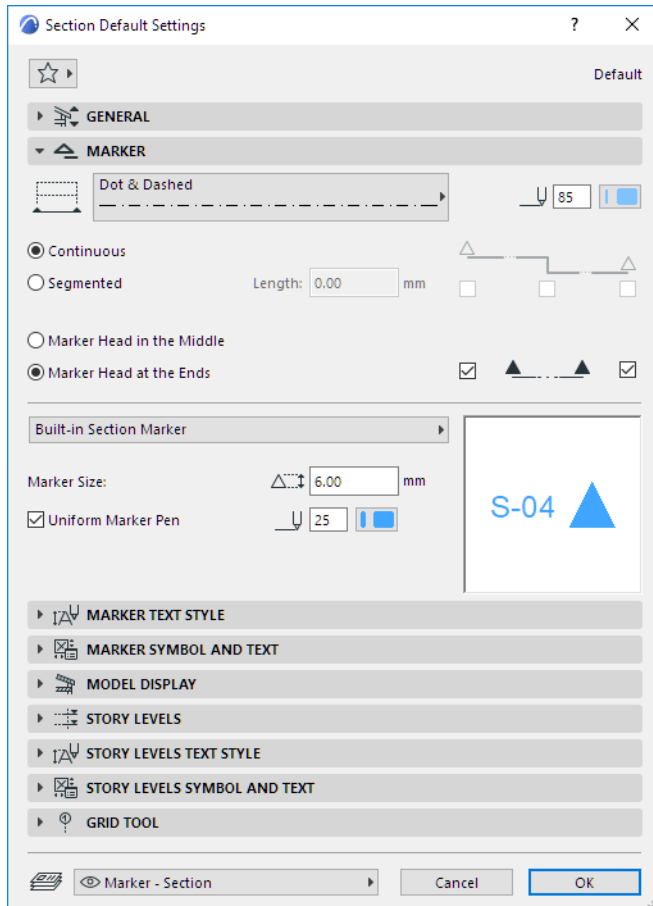
The link information that will appear in the marker is previewed in the **Reference to** text field, as long as the referred item (such as a drawing) already exists in the project.

Floor Plan Display of Section Markers

Display of Section Line and Marker

To customize the attributes of the Section Line and marker, use Section Settings.

See [Marker Panel and Marker Symbol and Text Panel](#).



Highlight Source Markers

Source markers are distinguished on screen by an optional semi-transparent fill. (Use **View > On-screen View Options > Highlight Source Markers** to enable or disable this distinguishing fill for all source markers. The color of this fill can be set in **Options > Work Environment > On-Screen Options**.)

Display of Limit Lines and Distant Elements Lines

Both the Section limit line and the Distant Elements line are on-screen-only elements which indicate “Marker Range”.

- To show or hide the Section limit lines and Distant Elements limit lines on the Floor Plan, use the toggle control in **View > On-Screen View Options > Marker Range**.
- To change these lines/colors, use the control in **Options > Work Environment > On-Screen Options**.

For more information, see [Display of Marker Range Lines](#).

Model Appearance (Section-Type Views)

Use the Model Appearance Panel of Section Settings to define the appearance of the model in a Section-type view.

Available for Source-type Sections and Elevations, 3D Documents and Interior Elevations.

Use the following controls:

CUT ELEMENTS

UNCUT ELEMENTS

FADE DISTANT ELEMENTS

ELEMENT CONTOURS AT SECTION/ELEVATION BOUNDARY

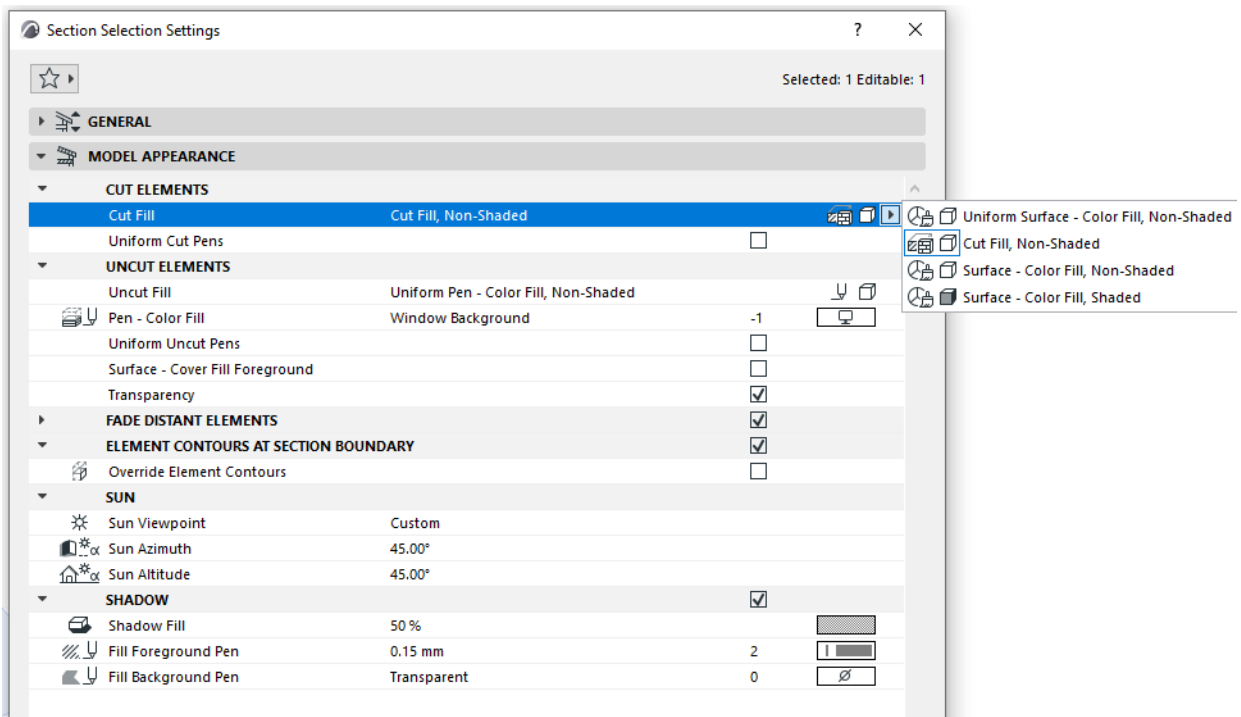
SUN

SHADOW

See also:

Note on Transparent Fills in Plotter Output

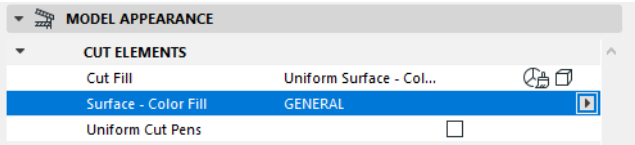
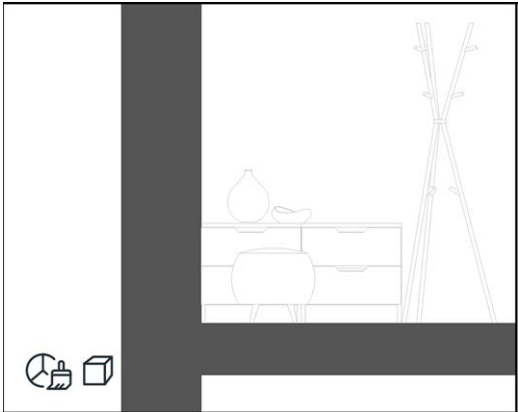
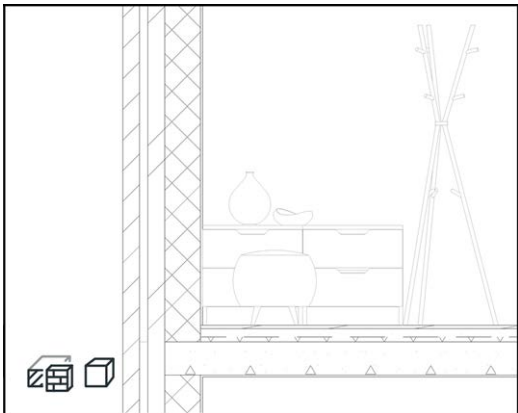
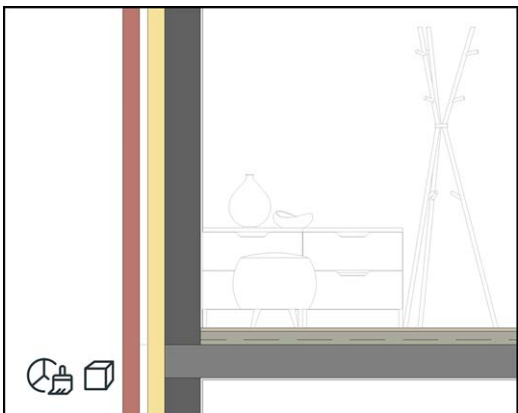
Display of Elements in Interior Elevation

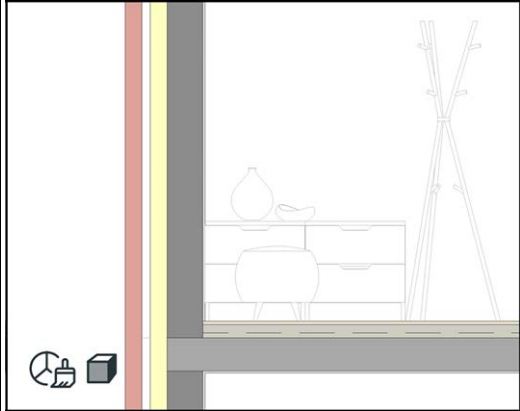


CUT ELEMENTS

Choose how to display cut elements in this Section.

Note: For Elevations and Interior Elevations: You can hide cut elements altogether by unchecking the Cut Elements box.

CUT FILL OF ELEMENTS IN SECTION	
<p>Uniform Surface - Color Fill, Non-Shaded</p> <p>Use a single surface to display all cut elements. Use the Surface - Color Fill control to choose the color.</p> 	
<p>Cut Fill, Non-Shaded</p> <p>Cut fill of Building Material</p>	
<p>Surface - Color Fill, Non-Shaded</p> <p>Surface of Building Material</p>	

CUT FILL OF ELEMENTS IN SECTION	
<p>Surface - Color Fill, Shaded</p> <p>Surface of Building Material</p>	


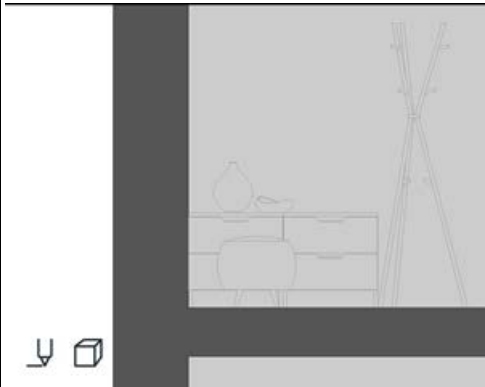
Uniform Cut Pens

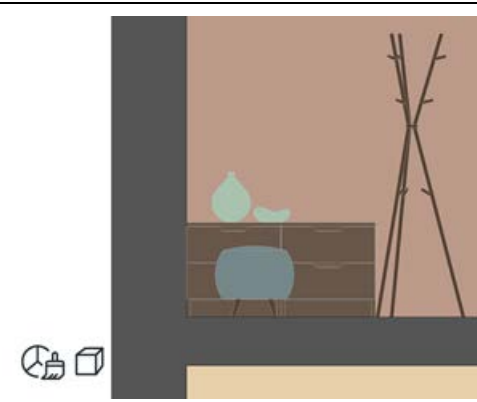
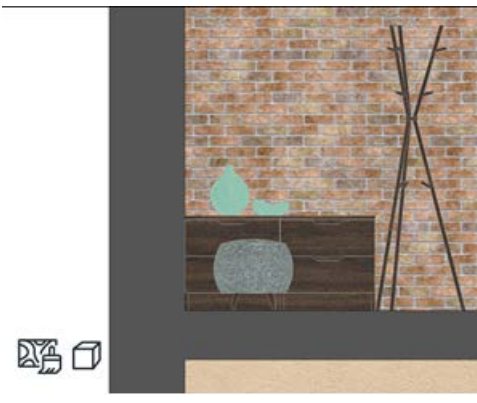
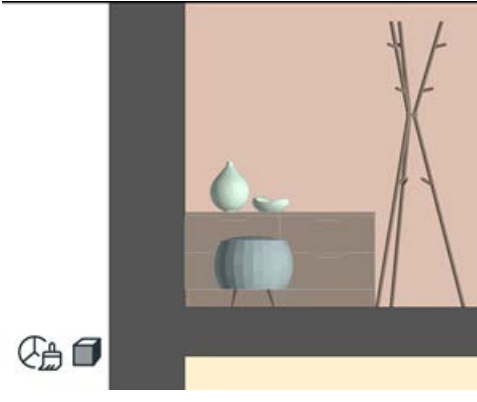

Use the same pens for all cut elements (cut line pen and fill foreground/background pens, as applicable).

UNCUT ELEMENTS

Choose fill attributes for uncut elements.

Note: Applying fills may increase rebuild times for large models. If this is a problem, choose “Nothing”.

UNCUT FILL OF ELEMENTS IN SECTION	
<p>Nothing</p> <p>No color applied to uncut surfaces.</p>	
<p>Uniform Pen - Color Fill, Non-Shaded</p> <p>Fill all uncut surfaces with a single color. Use the Pen-Color Fill control to choose the color.</p> <div style="border: 1px solid #ccc; padding: 5px; margin-bottom: 10px;"> <p>MODEL APPEARANCE</p> <ul style="list-style-type: none"> ▶ CUT ELEMENTS ▶ UN CUT ELEMENTS <ul style="list-style-type: none"> Uncut Fill Uniform Pen - Color Fi... ⌵ ⌲ <li style="background-color: #e0e0e0;">Pen - Color Fill Window Background -1 ⌵ ⌲ Uniform Uncut Pens <input type="checkbox"/> Surface - Cover Fill Foregro... <input type="checkbox"/> Transparency <input checked="" type="checkbox"/> </div>	

UNCUT FILL OF ELEMENTS IN SECTION	
<p>Surface - Color Fill, Non-Shaded</p> <p>Surface of Building Material (no shading effect)</p>	
<p>Surface - Texture Fill, Non-Shaded</p> <p>Surface plus texture of Building Material (no shading effect)</p>	
<p>Surface - Color Fill, Shaded</p> <p>Surface of Building Material (with shading effect)</p>	
<p>Surface - Texture Fill, Shaded</p> <p>Surface plus texture of Building Material (with shading effect)</p>	

Uniform Uncut Pens

All uncut lines use the same pen. Choose the desired pen from the Uncut Line Pen picker, below.

Surface - Cover Fill Foreground

Also activate the Cover Fill foreground of the surface. (Can be defined for each Surface attribute.)

Transparency

Display see-through surfaces (e.g. glass) as transparent.

FADE DISTANT ELEMENTS

Check this box to divide the Section into “close” and “distant” areas.

You can then differentiate the distant area by applying a separate fill to it.

To define the limit where “close” ends and “distant” begins, go to the Floor Plan and drag the “distant” marker line.

The “distant” marker line is an on-screen-only element.

- Show or hide it at View > On-Screen View Options > Marker Range
- Change its line type/color in Options > Work Environment > On-Screen Options, at Source Marker Color.

ELEMENT CONTOURS AT SECTION/ELEVATION BOUNDARY

Check this box to apply a contour to any displayed element which extends past the Section’s horizontal limit.

This contour marks the element's boundary, as it is shown in the section, even though the element does not really end there.

Choose a Line Type and Pen for the contour.

SUN

Sun Viewpoint:

Choose one of two sun positions relative to the Section/Elevation Line:

- **As in 3D Window:** Use this option to inherit the sun position set up in the 3D Projection or 3D Perspective Settings dialog box.

[See also 3D Window.](#)

- **Custom:** Use the **Sun Azimuth/Sun Altitude** controls below to define a custom sun position for this Section/Elevation.

Note: These custom values will apply to this Section/Elevation only and are calculated differently from the values set in the 3D Projection/Perspective Settings dialog box.

Sun Azimuth

The Azimuth value is an angle value in degrees. The line that is at a right angle to the Section/Elevation line represents zero degrees. As I increase the azimuth by increasing this angle value, the sun moves toward the right - meaning the shadows start to lengthen to the left of the shadow-casting surfaces.

If you do not want the visible side of the building to be overshadowed, you should use an Azimuth value between -90 and +90 degrees.

SHADOW

Check this box to activate Shadows (or Shadow effect).

Shadows have no contours, but you can customize their appearance.

Choose a Fill and Fill Pens for the Shadow. These can be different for Uncut and Distant elements.

Note: If your fill is set to any of the “Shaded” option, you cannot customize the Shadow Fill or its pens.

Note on Transparent Fills in Plotter Output

Plotters cannot handle transparent Fills. In Texture-type fills, shadows and shading are based on transparent Fills - these will appear as solid Fills in plotter output.

In fills based on Surface Color, shadows and shading are created using color mixing, and are not affected by this limitation - they will look good on plotter output, too.

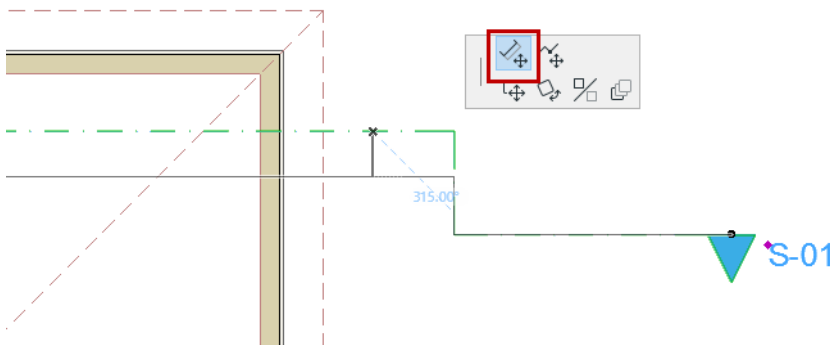
Editing Section/Elevation/IE Lines on Floor Plan

You can stretch, offset and break all types of Section/Elevation lines on the Floor Plan:

- the main Section line, and any break lines
- the Fade Distant line, if any
- the limit line (Depth break), if the Section/Elevation's horizontal range is limited

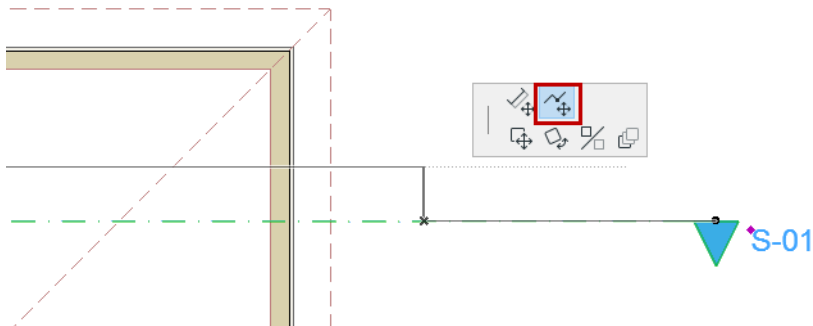
Offset Section Line Segment

1. Select Section line.
2. Click anywhere on the line segment to be offset.
3. From the pet palette, choose **Offset Edge**.
4. Drag and click to offset.



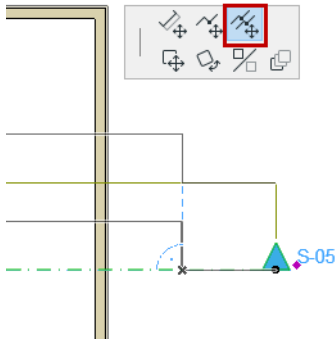
Break and Offset Line

1. Select the Section line.
2. Click at the point where you want to insert the break.
3. From the pet palette, choose **Insert Break and Offset**.
4. Drag and click to offset the broken segment.



Break All Section Lines

If the Section has a displayed Limit Line and/or Fade Distant line, use the pet palette command: **Insert Break on All Lines** to break and offset all lines simultaneously.



Stretch Line

1. Select the Section line.
2. Click an endpoint of any segment.
3. From the pet palette, choose **Stretch**.



Note: In Interior Elevations, the endpoints of the IE line cannot be stretched, because its length depends on the Room Boundary.

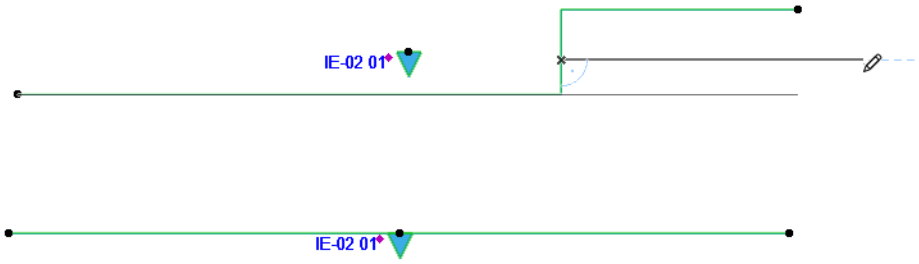
See [Display and Edit Interior Elevation Marker Lines and Marker Range](#).

Remove Break

To remove a break in the line, select the segment and do one of the following:

- Use the Offset pet palette command to realign it with another line of the same type
- Select the break edge, and drag it outside of the Section/Elevation horizontal limit





Related Topic:

[Edit Range or Limit Lines Graphically](#)

[Watch the Video](#)

Place a Linked Section Marker

Note: This process is identical for Elevations, Details and Worksheets.

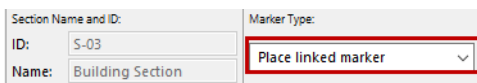
A linked marker is for reference purposes only. It has no model source and does not generate a new viewpoint.

To place a linked Section marker, use the Section tool in any of the following windows: Floor Plan; Section; Elevation; Interior Elevation; 3D Document; Worksheet; Detail. Then link the marker to an existing viewpoint, view or drawing whose information is displayed in the marker.

Define Reference for Linked Marker

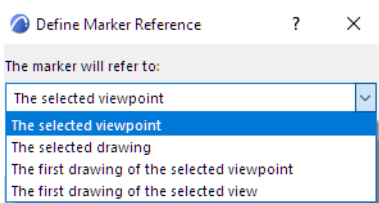
Follow these steps to define reference information for linked Section, Elevation, Detail and Worksheet markers.

1. Activate the Section tool.
2. In the Marker Type control of the Info Box or Section Default Settings, select “Place Linked Marker”.



3. In the appearing dialog box, define the Marker Reference.

For a new linked marker, or to redefine an already placed selected marker, you have these choices:



- **the selected viewpoint** (i.e., the viewpoint you select from the directory list after choosing “Browse”)
- **the selected drawing** (i.e., the drawing you select from the directory list after choosing “Browse”)
- **the first placed drawing of the selected viewpoint** (i.e., the first drawing created from the *viewpoint* you select from the directory list after choosing “Browse.”). “First” means the topmost eligible drawing in the Navigator Layout Book.
- **the first placed drawing of the selected view** (i.e., the first drawing created from the *view* you select from the directory list after choosing “Browse.”). “First” means the topmost eligible drawing in the Navigator Layout Book.

Choosing any of these reference options will bring up a directory dialog box showing the relevant view of the Navigator (Project Map, View Map, or Layout Book). Choose the desired viewpoint, view or drawing. The chosen item’s information will be displayed in the linked marker.

Thus, a linked marker can refer to *any* viewpoint, view or drawing in the project.

Note: The option to refer to a view is available in case the desired Drawing is located in a different project’s Layout Book (and is therefore not listed in the “Define Marker Reference” dialog box). Since you cannot refer to the actual drawing, you can refer to the View (in the current project) from which the Drawing was created (in the other project). The marker will display the drawing data of the first drawing created out of this view as autotext (e.g. #Drawing Name, #DrgID). Once the Layout Book that contains

the reference drawing is opened alongside the project containing the marker, the marker will fill in the correct drawing data.

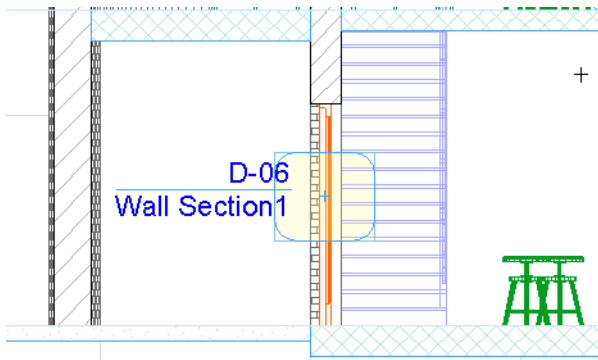
See [Place view from external Archicad project \(menu command\)](#).

After the marker is placed, you can still go back and change its marker information at any time.

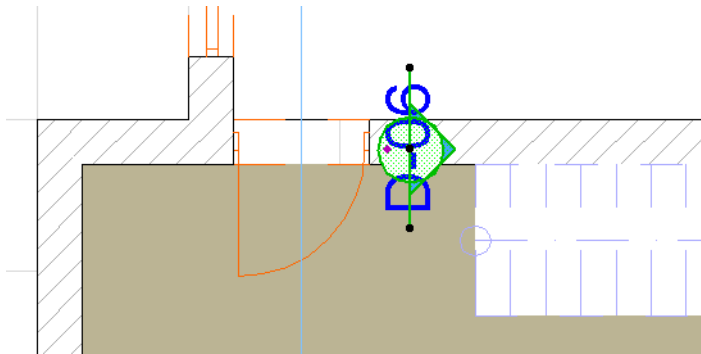
Linked Section Example

For example, you might create a Wall Section that functions like a Detail:

First, place a source Detail marker in the Section window;



then place a linked Wall Section marker on the Floor Plan which is linked to this Detail.



Place Unlinked Marker

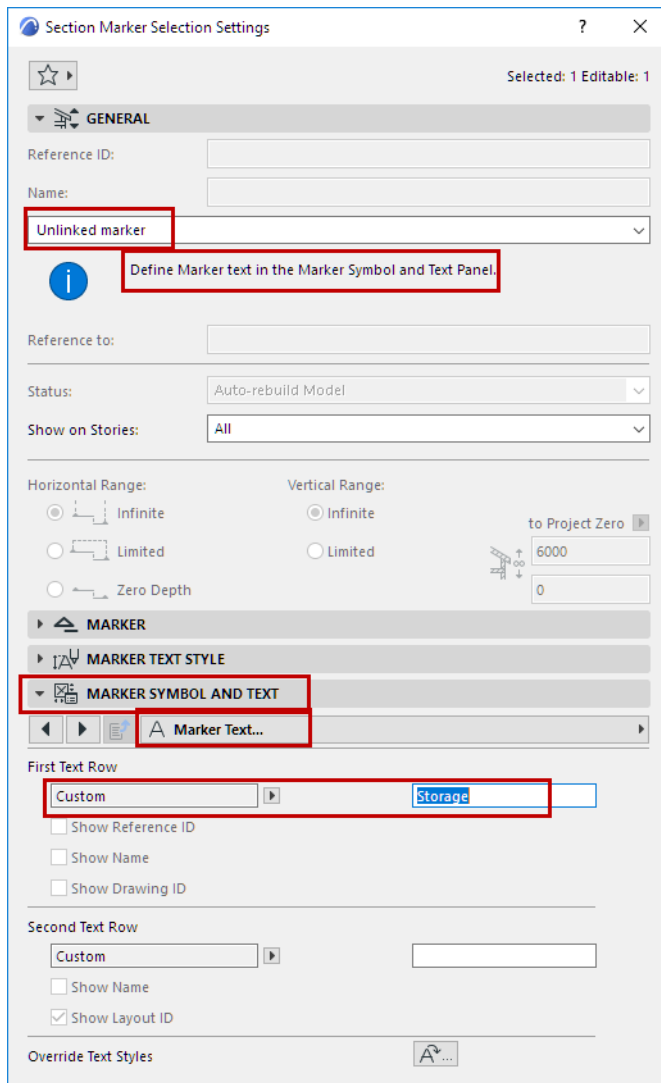
An **unlinked marker** can be useful in your published documentation, for example, if you want to refer (with a page number) to the location of an external drawing that is not integrated into the Archicad project.

Follow these steps to place an unlinked marker with the Section, Elevation, Detail or Worksheet tool.

1. In Default Settings or the Info Box, choose “Placed Unlinked Marker.”

Such a marker displays no linked information.

2. Define any custom text for the marker using the Custom text fields in the parameter list in the Marker Symbol and Text Panel of the Settings dialog box.



3. Draw a line or boundary (depending on the tool) and place a marker in any of the following windows: Floor Plan; Section; Elevation; Interior Elevation; Worksheet; Detail; 3D Document.

Create an Independent Section Viewpoint

Note: This process is identical for Independent Elevation Viewpoints.

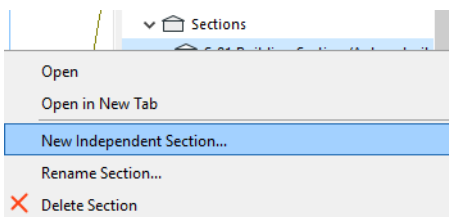
“Independent” sections, with or without a marker, can be created in some windows. These viewpoints have no model source; they are listed as “Independent” viewpoints in the Navigator.

In an independent section, you can use 2D tools to create or add new content. Use copy-paste to transfer selected items from a Trace Reference into the drawing window in 2D form for further editing.

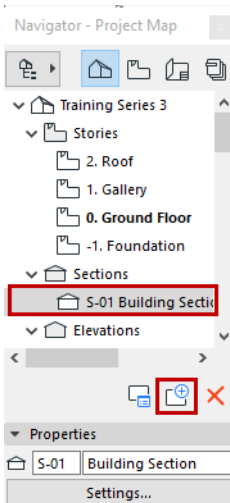
To create an independent Section without a marker, use the **“Create New Independent Section”** command. This creates an empty viewpoint without a marker, but listed in the Navigator.

This command is accessible:

- from the context menu of a Section item in the Navigator Project Map. Right-click on this folder and choose the **“New Independent Section”** command from the context menu



- from the **Document > Documenting Tools** submenu
- by clicking the New Viewpoint icon at the bottom of the Navigator Palette Project Map, if a Section is selected in the Project Map.

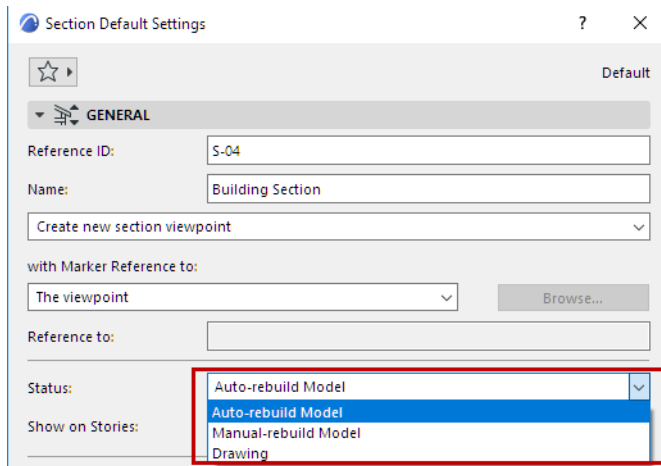


If you wish this independent Section viewpoint to be **linked to a marker**, place a linked Section marker into the project and link it (“Marker Reference to”) to the existing independent Section.

Assign Section Rebuild Status

Note: The process for assigning a status is identical for Sections, Elevations and Interior Elevations. 3D Document status can be either Auto-rebuild or Manual Rebuild, but not Drawing.

When creating each new Section, you assign it a status in the General panel of **Section Settings**. A Section can have one of three statuses: **Auto-rebuild Model**, **Manual-rebuild Model**, and **Drawing**.



Each Section's status defines the link between the Section and the model on the Floor Plan, and its rebuild method.

- In the two **Model** statuses (Auto-rebuild and Manual-rebuild), the window consists of construction elements; any changes made in a Model Section window can be updated in the **Floor Plan** window, as well as in the **3D** window and in other **Section** windows, and vice versa.
- In a Section of **Drawing** status, construction elements are exploded into 2D fills, arcs and lines. Changes made in this kind of window are not updated in other windows. You can, however, update the drawing to reflect recent changes made to the model.

Construction elements are editable in both Autorebuild and Manual-rebuild section windows. You cannot create new model elements of most types here, but you can duplicate, view and modify them. You can, however, create new Morphs and Curtain Walls in these windows.

Note: Clearing a construction element from a Section Window of model status (either Autorebuild or Manual-rebuild) will also clear it from both the Floor Plan and the 3D model.

Regardless of a section's status, you can add 2D graphics, annotation and dimensioning.

For more information, see [Updating Sections](#).

Updating Sections

Note: The updating process is identical for Sections, Elevations, Interior Elevations and 3D Documents. Each Section's status defines the link (and rebuild method) between the Section and the model on the Floor Plan.

For further information, see [Assign Section Rebuild Status](#).

Updating an Autorebuild Model Section

An **Autorebuild Model** Section is linked to the Floor Plan model and can be edited interactively. If the Floor Plan has changed, a Section in Autorebuild Model status will be **automatically rebuilt** every time it is opened.

Note: If background processing is on (see the Background Updating controls at **Options > Work Environment > Model Rebuild Options**), then the rebuild will occur in the background, even without opening the window.

Changes to existing construction elements in the Section window (including associative dimensions) will automatically be updated in other windows in the background. The update process works on construction elements only; 2D elements added to either the Floor Plan or Section window will not be reflected in the other window.

If you are in Autorebuild status and you are editing elements inside the Section window, the rebuild function is continuous and immediate within this window. However, in large plans, continuous auto-rebuild can cause slowdowns. If you prefer that your Section window be rebuilt only when it is opened or brought to the front of the screen, go to the **Options > Work Environment > Model Rebuild Options** dialog box and uncheck the "Update Auto-rebuild Model viewpoints continuously" checkbox.

While an update is underway, switching to another window or tab will cancel the update, so you can tab through successive views without waiting for the updates to finish.

If you nevertheless need to Rebuild manually, the **View > Refresh > Rebuild from Model** command is available. (The plain **View > Refresh > Rebuild** command, used to refresh 2D drawing elements, is not available for Autorebuild Model windows.)

Note: Some complex changes to the plan (e.g., modifying a Library Part) do not appear automatically in Model Section Windows. For a Manual-rebuild section, use the **View > Refresh > Rebuild** command; for an Autorebuild section, use **View > Refresh > Rebuild from Model** to see these changes reflected properly.

Updating a Manual-Rebuild Section

A Section having **Manual-Rebuild Model** status is not rebuilt automatically. It can be rebuilt from the model only by using the **View > Refresh > Rebuild from Model** command in the menu.

While a rebuild is underway, you receive feedback on the progress. Click the progress window to cancel the rebuild.

Updating a Drawing Section

Not applicable for 3D Documents, which are either Autorebuild or Manual-Rebuild type documents.

- To rebuild a **Drawing-status** Section, use **View > Refresh > Rebuild**. These commands refresh the view as a 2D drawing, fixing any potential temporary display errors.
- For a complete refreshing of a **Drawing-status** Section window, choose **View > Refresh > Rebuild from Model**. This will refresh the contents of these windows to fully reflect the current state of the Virtual

Building model. In the process, all 2D elements originating from the model will be removed and an up-to-date view will be generated. This means that any manual editing of these 2D elements will be lost. Manually added 2D elements will remain in place.

Summary of Rebuild Commands

The **View > Refresh > Rebuild** command is available in all construction windows. It rebuilds the model in the currently active window.

Additional Rebuild commands in the **View > Refresh** menu are available if the active window is a Section/Elevation/IE, 3D Document, or a Detail or a Worksheet.

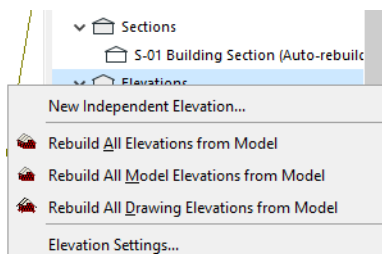
Rebuild from Model: Use this command to rebuild the active window; this will refresh the window information to reflect modifications in the Floor Plan.

Rebuild from Source View: This command is available only in the Detail and Worksheet windows.

The following commands might not be visible in the **View > Refresh** menu by default. You can customize your menu (**Options > Work Environment > Menus**) to include these commands if needed.

[See Customizing Menus.](#)

Also, these commands are available from the context menu of the folders in the Navigator Project Map and View Map.



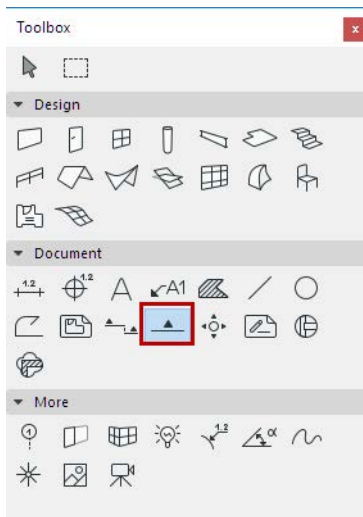
Rebuild All from Model: Use this command to rebuild all Section/Elevation/IE windows in your project.

Rebuild All Model Sections from Model: Use this command to rebuild all the Section windows of Model status (either **Autorebuild Model** or **Manual-Rebuild Model**).

Rebuild All Drawing Sections from Model: Use this command to rebuild all Drawing-status Section windows.

Elevations

Archicad has a separate tool for creating Elevation viewpoints and linked or unlinked Elevation markers.



In contrast to Sections:

- Elevations generally do not “slice through” the structure, but rather create a cross-section view of the structure from a distant point.
- There is no “zero depth” option for the Elevation’s horizontal range.
- Elevation markers are conventionally different from Section markers; consequently, the Marker options in Elevation Settings vary from their counterparts in Section Settings.
- Unlike the Section Line, the Elevation line is an on-screen-only Marker item, and is not shown on the Layout.

See [Display of Marker Range Lines](#).

In all other respects, the Elevation Tool works the same way as the Section Tool.

- To create an Elevation viewpoint, you must place a source-type Elevation marker on the Floor Plan.
- The resulting Elevation viewpoint has a Status (Model or Drawing) which determines its update process.
- The Elevation Settings Model Appearance Panel defines the display of the Elevation viewpoint, while the Marker and Marker Head panels determine the content and display of the Elevation marker.
- A linked-type Elevation marker, containing reference information only (without creating a viewpoint), can be placed in the Floor Plan, Section, Elevation, Interior Elevation, 3D Document, Detail or Worksheet Window.
- Unlinked Elevation markers can also be placed.

For more information, see the following topics:

[Create a Section Viewpoint](#)

[Horizontal/Vertical Range of Section Viewpoint](#)

[Assign Section Rebuild Status](#)

[Model Appearance \(Section-Type Views\)](#)

[Define Marker Reference for Source Marker](#)

[Place a Linked Section Marker](#)

Place Unlinked Marker

Create an Independent Section Viewpoint

Updating Sections

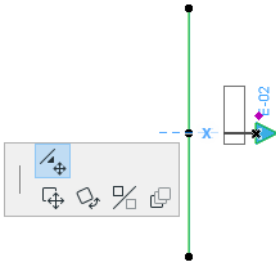
Floor Plan Display of Section Markers

Editing Section/Elevation/IE Lines on Floor Plan

Section and Elevation Settings

Moving an Elevation Marker

This option is available only for Elevation and Interior Elevation markers.



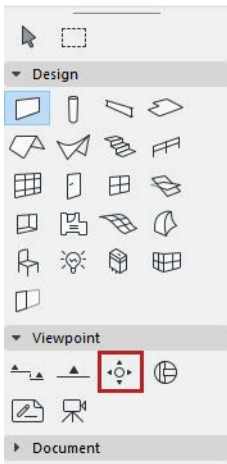
Select the Elevation line; the marker will be selected and will display a node.

Click on this node and choose the **Move Elevation Marker** command from the pet palette, then drag the marker to the desired position.

The marker will retain this position even if the elevation is dragged or rotated to a new position.

Interior Elevations (IE)

This dedicated Archicad tool is optimized for creating interior elevations of interior spaces.



Related Topics:

[About Interior Elevations](#)

[Create Interior Elevation Group](#)

[Create Single Interior Elevation Viewpoint](#)

[Interior Elevation Horizontal Range](#)

[Interior Elevation Vertical Range](#)

[Display and Edit Interior Elevation Marker Lines and Marker Range](#)

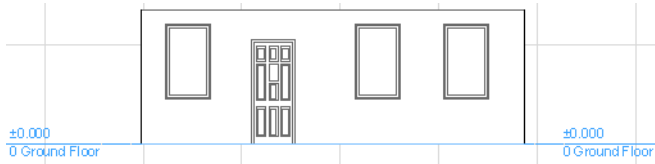
[Interior Elevation IDs and Names](#)

[Delete/Restore IE Viewpoint](#)

[Display of Elements in Interior Elevation](#)

[Model Appearance \(Section-Type Views\)](#)

About Interior Elevations



Interior Elevations (IEs) work similarly to Sections and regular Elevations: choose an input method; define the view and its limits; and place a marker with custom-defined marker reference information.

Each Interior Elevation is a separate viewpoint in the Navigator Project Map.

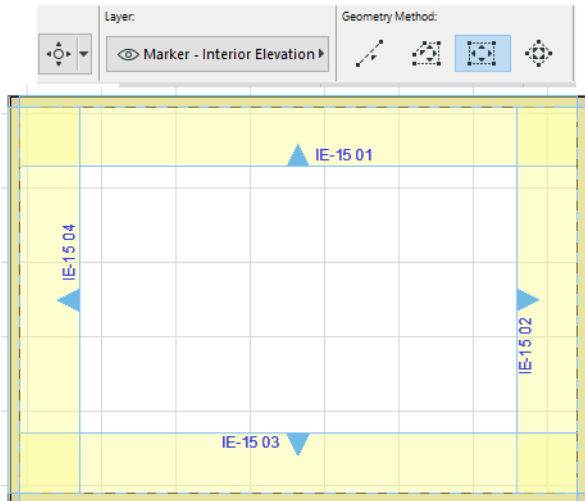
Unlike the Section and Elevation tools, the Interior Elevation tool is used to create new viewpoints only; you cannot place a linked, reference-only IE marker.

Many other aspects of Interior Elevation viewpoints - status, display, marker reference, and updating - work the same as in Section viewpoints.

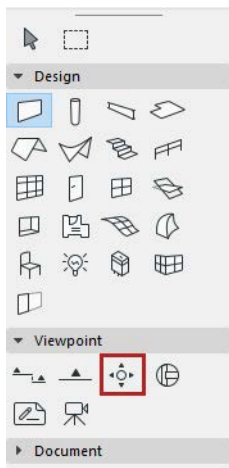
Create Interior Elevation Group

Interior Elevations are typically created in groups (e.g. four IE's created by placing a four-segment polyline around a single room).

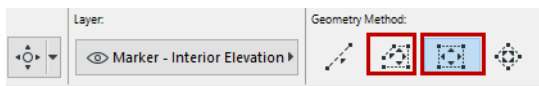
An IE created using one of the polyline or rectangle input methods will have a separate IE viewpoint for each segment of the line; each view is created perpendicular to the segment.



1. Activate the Interior Elevation tool.

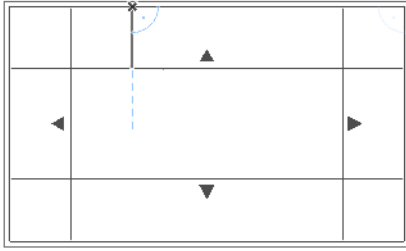


2. Use the rectangular or polyline input method to draw the Room Boundary, or space-click on a Zone.

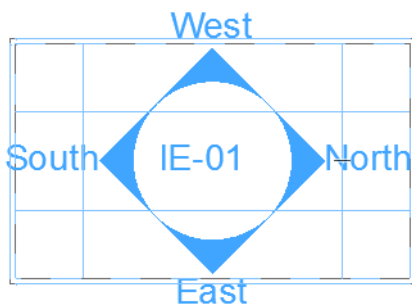


The resulting rectangle/line/polyline represents the Room Boundary of the Interior Elevation group.

3. Move the cursor to the desired standpoint. The arrows on the feedback indicate the view direction of the resulting Interior Elevation viewpoint.



4. Click to place the IE marker(s).



Interior elevations use a dedicated IE marker, a GDL object with editable parameters.

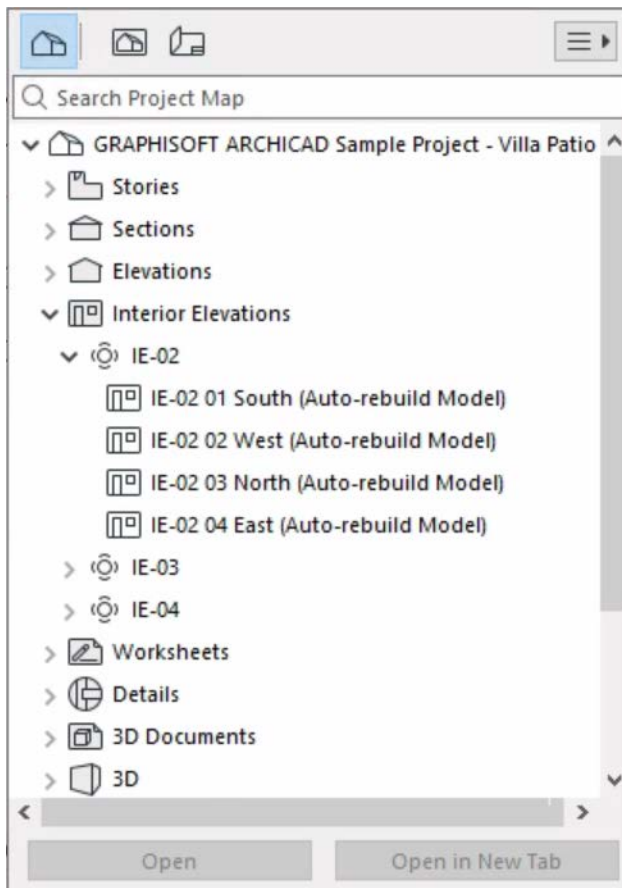
IE marker(s) are placed at the middle of each segment of the IE line or at the center of the room.

To define the Marker style and Position, use the controls in the Marker Panel of Interior Elevation Settings.

For any or all of the viewpoints, you can opt not to show a marker, by choosing the No Marker option in this panel.

[See Interior Elevation Marker Panel.](#)

5. The new group of viewpoints is created and listed in the Project Map. Each new IE viewpoint is assigned to an IE Group, even if there is only one viewpoint in the Group. The IE Group is automatically listed in its own subset in the Project Map.



You cannot move individual IE viewpoints from one Group to another.

IE's are handled as a group for purposes of numbering, display, Show on Stories, Layers, Trace functions, and placement on a layout.

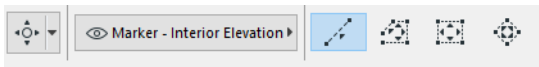
Typically, you will use the Master Layout's "Auto Arrange Setup" options - in Master Layout Settings - to fine-tune the appearance of IE views on the Layout.

To select an Interior Elevation group, select any of its Room Boundary lines.

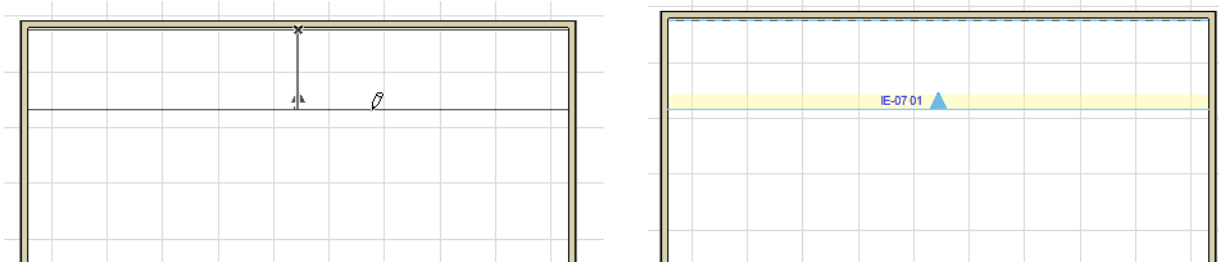
However, the settings of any single segment are accessible individually.

Create Single Interior Elevation Viewpoint

1. Use the single line input method to draw the Room Boundary.



2. Move the cursor to the desired standpoint. The arrow on the feedback indicates the view direction of the resulting Interior Elevation viewpoint.
3. Click to place the IE marker.



To define the **Marker style and position**, use the controls in the Marker Panel of Interior Elevation Settings.

See [Interior Elevation Marker Panel](#).

4. A new IE viewpoint is created and listed in the Project Map. It is assigned its own IE Group, even if there is only one viewpoint in the Group.

Many other aspects of Interior Elevation viewpoints - status, display, marker reference, and updating - work the same as in Section viewpoints.

Related Topics

[Assign Section Rebuild Status](#)

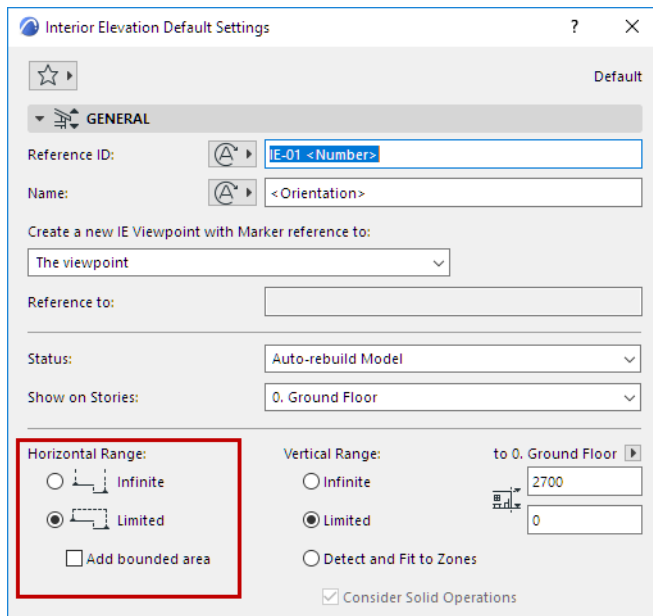
[Model Appearance \(Section-Type Views\)](#)

[Define Marker Reference for Source Marker](#)

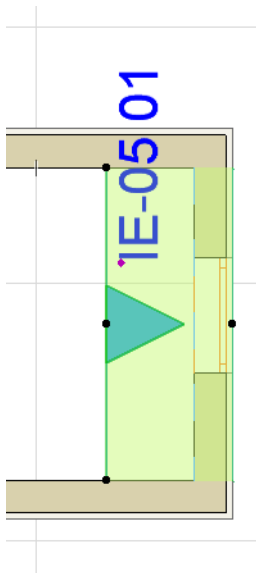
[Updating Sections](#)

Interior Elevation Horizontal Range

The Horizontal Range can be either Infinite or Limited.



The default choice is **Limited**. In this case, the IE view will extend horizontally until the Room Boundary line. If the Room Boundary is a Wall, then the view automatically extends to the depth of the Wall; you can see this by the Marker Range highlight, which overlaps the Wall area.



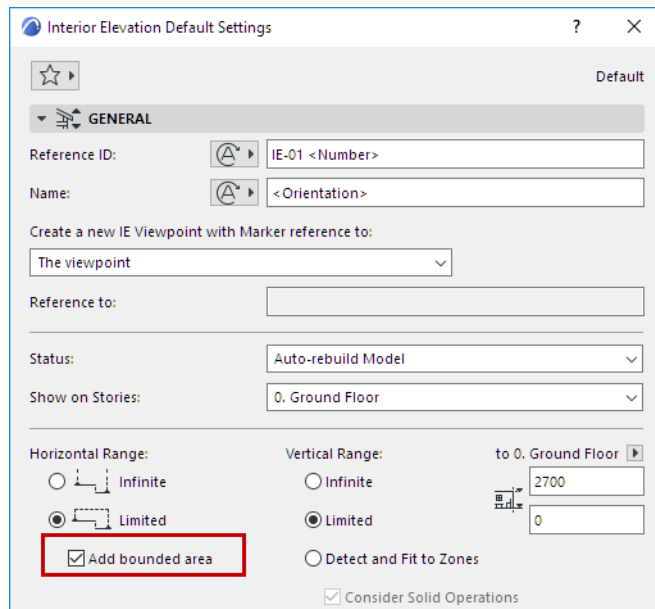
To adjust the horizontal range, select the Room Boundary line and stretch or shrink it using the commands of the pet palette. (Stretch/shrink modifications to this segment will affect the neighboring line segments accordingly, as when editing any polyline.)

An **Infinite** Horizontal Range means that the Interior Elevation viewpoint will extend horizontally to include all visible parts of the model.

Add Bounded Areas (Interior Elevation Default Settings)

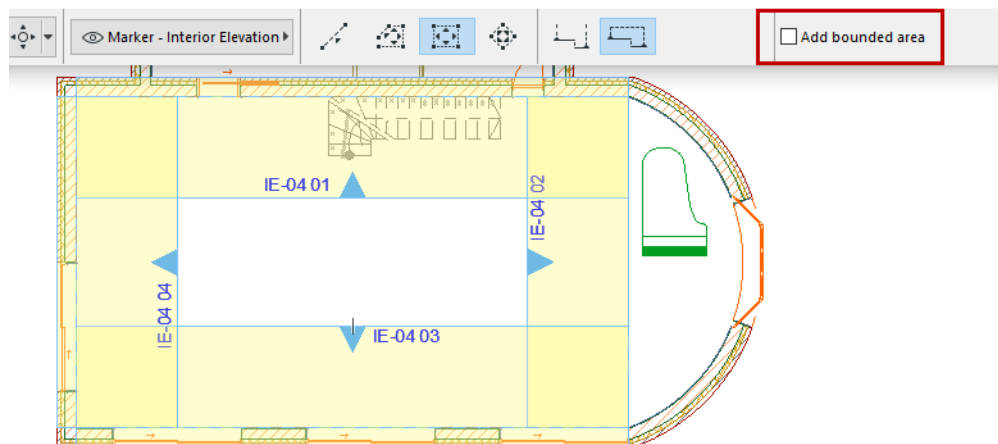
To include connected enclosed areas in the Interior Elevation viewpoint, choose **Add bounded areas**. This checkbox is available

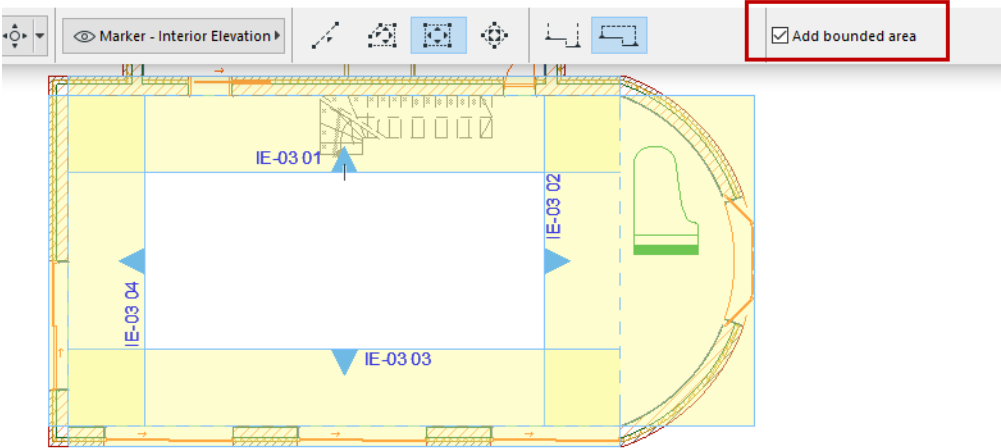
- in Interior Elevation Default Settings (you must set it before you place the IE; you cannot set this control for an already placed IE)
- if the Limited option is checked for Horizontal Range



This way, if the room contains an odd-shaped alcove, the IE viewpoint will include structures that fall beyond the Room Boundary Line, but still fall inside the room.

The following two illustrations show the result of the checkbox setting:





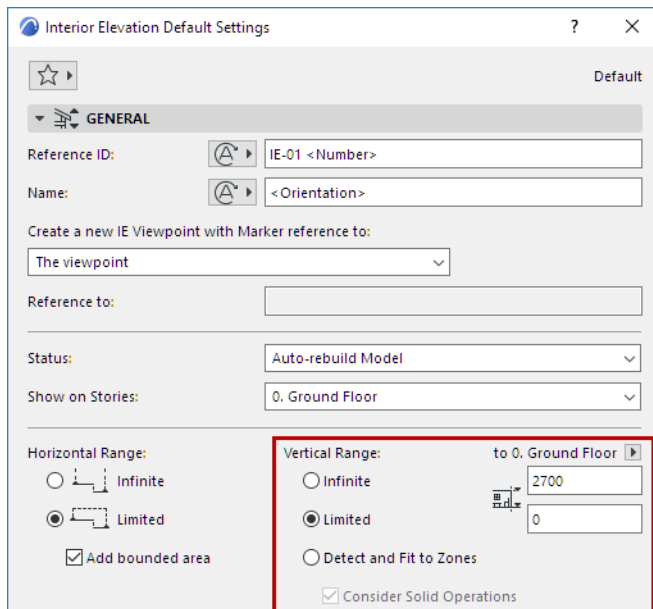
Interior Elevation Vertical Range

Define the Interior Elevation's vertical range either manually (in Settings), or graphically, in the Interior Elevation window.

Define IE Vertical Range in Settings

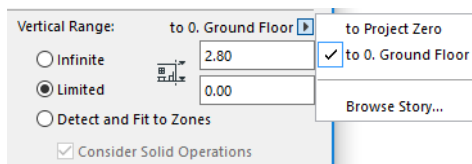
An IE viewpoint's **vertical range** is set manually by the user in the General Panel of Interior Elevation Settings, or the Info Box.

See [Interior Elevation \(IE\) Tool Settings](#).



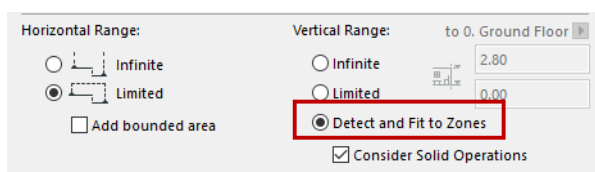
Each IE view (i.e. each segment of the IE Room Boundary polyline) can have a separate, custom vertical range value, if desired: choose the **Limited** option as the Vertical Range and enter the desired elevation limits.

Use the pop-up to define whether this vertical range should be measured from Project Zero, or from the Interior Elevation's Home Story (the story onto which it was placed).



Detect and Fit to Zones

If you want the Interior Elevation geometry to correspond to a zone, choose the **Detect and Fit to Zones** Vertical Range option from the General Panel of Interior Elevation Settings.



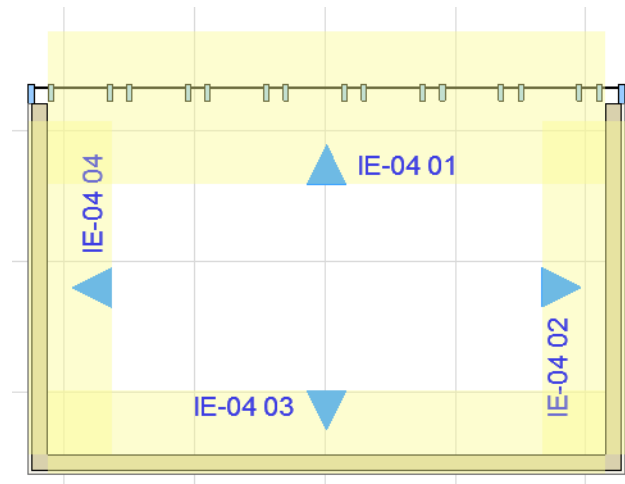
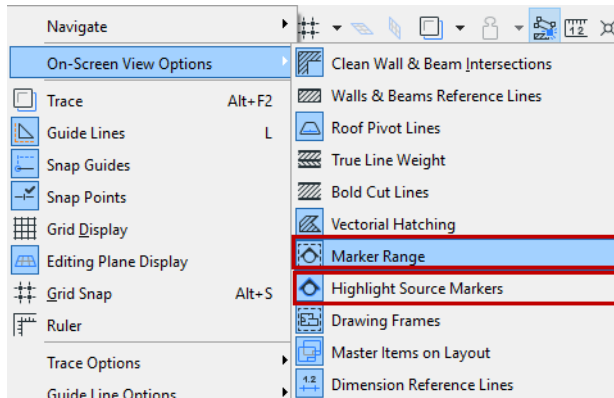
If a zone is detected, its height will be used as the vertical range of the selected Interior Elevation viewpoint.

Zones edited by Solid Element Operations are also calculated accurately, provided that you also check the **Consider Solid Operations** box. For example, a single zone trimmed to a slanted roof will have several different zone heights. These are accurately reflected in each viewpoint of the Interior Elevation group.

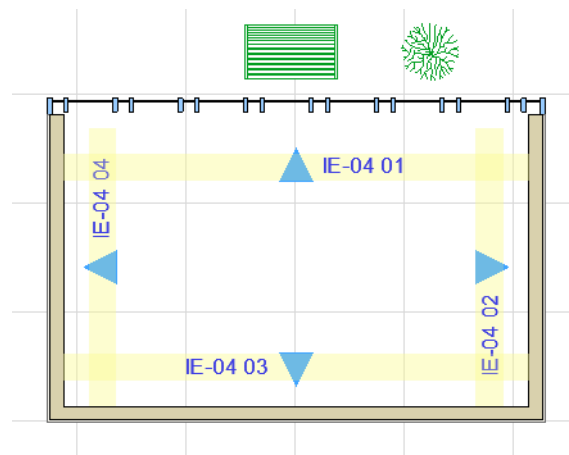
Display and Edit Interior Elevation Marker Lines and Marker Range

You can vary the display of Interior Elevation lines/range using the following two options at **View > On-Screen View Options**:

- Marker Range: displays the interior elevation limit line and Room Boundary line
- Highlight Source Markers: displays a highlight that indicates the range of the viewpoint. The color of this fill can be set in **Options > Work Environment > On-Screen Options**.



Marker Range On (Limited IE Viewpoint)



Marker Range On (Infinite IE Viewpoint)

Edit Room Boundary

1. Select the IE or the IE Group:

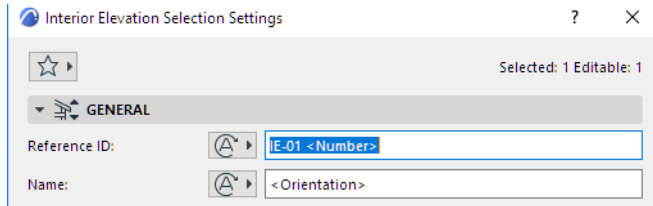
Note: If the IE group uses a single, common marker, select the marker to select the entire IE group.

2. Use pet palette commands to edit the line.

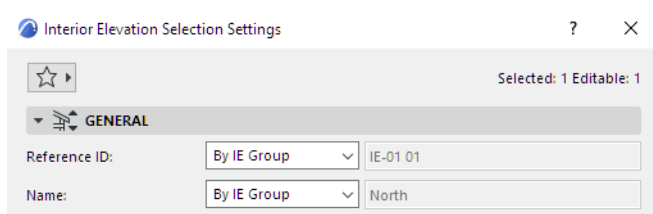
Note: Inserting a new node will create a new IE viewpoint.

Interior Elevation IDs and Names

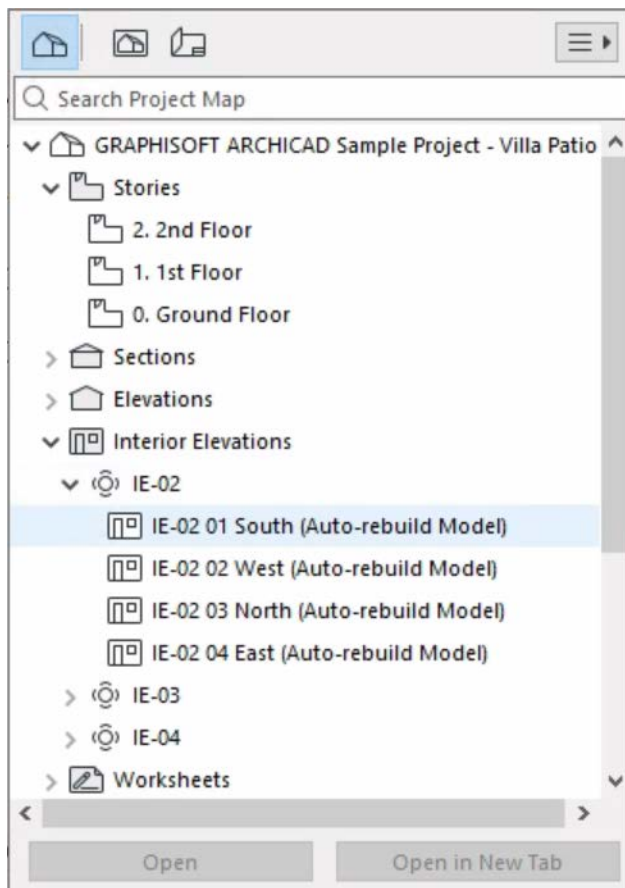
To assign IDs and names to IE Groups and their viewpoints, use the General Panel of Interior Elevation Settings. The options in the General Panel vary depending on whether you are defining Settings for an IE Group:



or for an individual IE viewpoint:



Each new IE viewpoint is automatically assigned to an IE Group (even if there is only one viewpoint in the group).



In each individual viewpoint's IE settings, the Reference ID and Name fields are set to "By IE Group" by default.

This means that each IE viewpoint will have the same ID and Name as its parent IE Group.

Archicad's default IE Settings further define the ID and Name of each IE viewpoint to include Autotexts.

Use Autotext to Create Meaningful IDs/Names for Interior Elevation Group

You can define a logical naming convention for your IE Groups by including an Autotext in the parent IE Group's ID and Name.

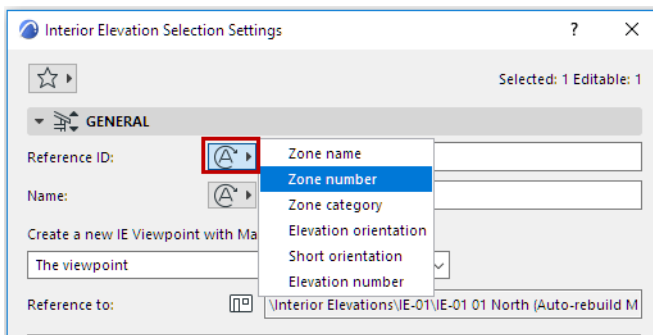
This way, each individual Interior Elevation contained in the group will display a logical Autotext value that can vary for each elevation.

By default, Archicad's IE Group ID adds an Autotext of <Number> to the ID of each viewpoint in the IE group.

Also, the default IE Group Name adds an Autotext of <Orientation> to each viewpoint Name. Consequently, each viewpoint name consists of its specific orientation, which automatically assigned using Autotext (North, South, East, West...).

For an IE Group, use the General Panel of IE Settings to define any other Autotext to include in the IE viewpoints' name and/or ID. For example, you can set the Interior Elevation Group's ID to be the Zone Number:

1. Click the Autotext button next to the ID field in General Settings and choose Zone Number from the list.



2. In the Navigator, note that the ID of the Group is now equivalent to the number of the zone contained in the Interior Elevation.

This ID is inherited by all of the viewpoints in the Group.

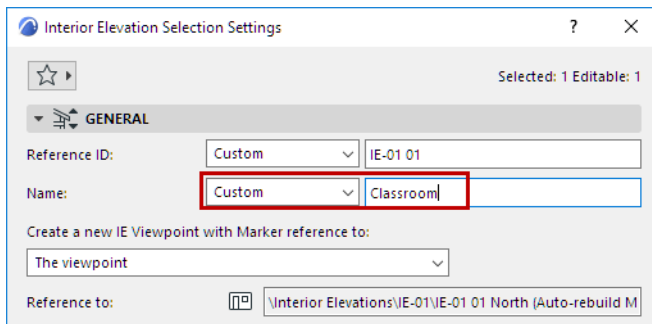
(Naturally, assigning a Zone Number autotext works only if the Interior Elevation includes a zone.)

Customize IDs and Names

You can give a customized ID and/or Name to any IE Group or individual viewpoint:

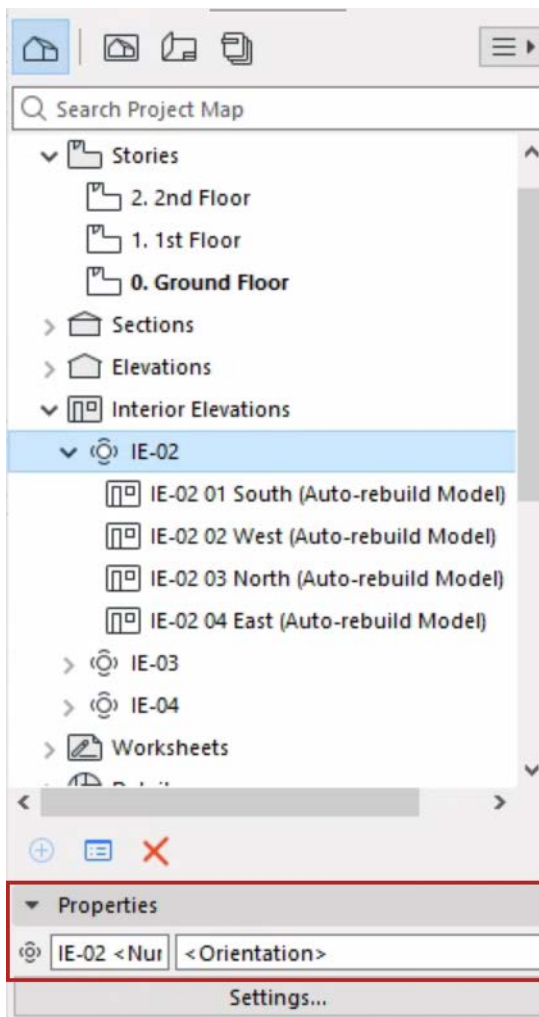
- Select the IE limit line.
- Open Interior Elevation Settings.

If it is an individual IE Viewpoint, make sure the pop-up is set to Custom.



- Enter the desired ID and/or Name.

You can also use the Navigator Palette to change IDs and Names for an IE Group.

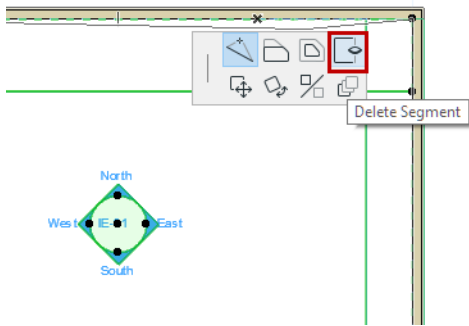


Delete/Restore IE Viewpoint

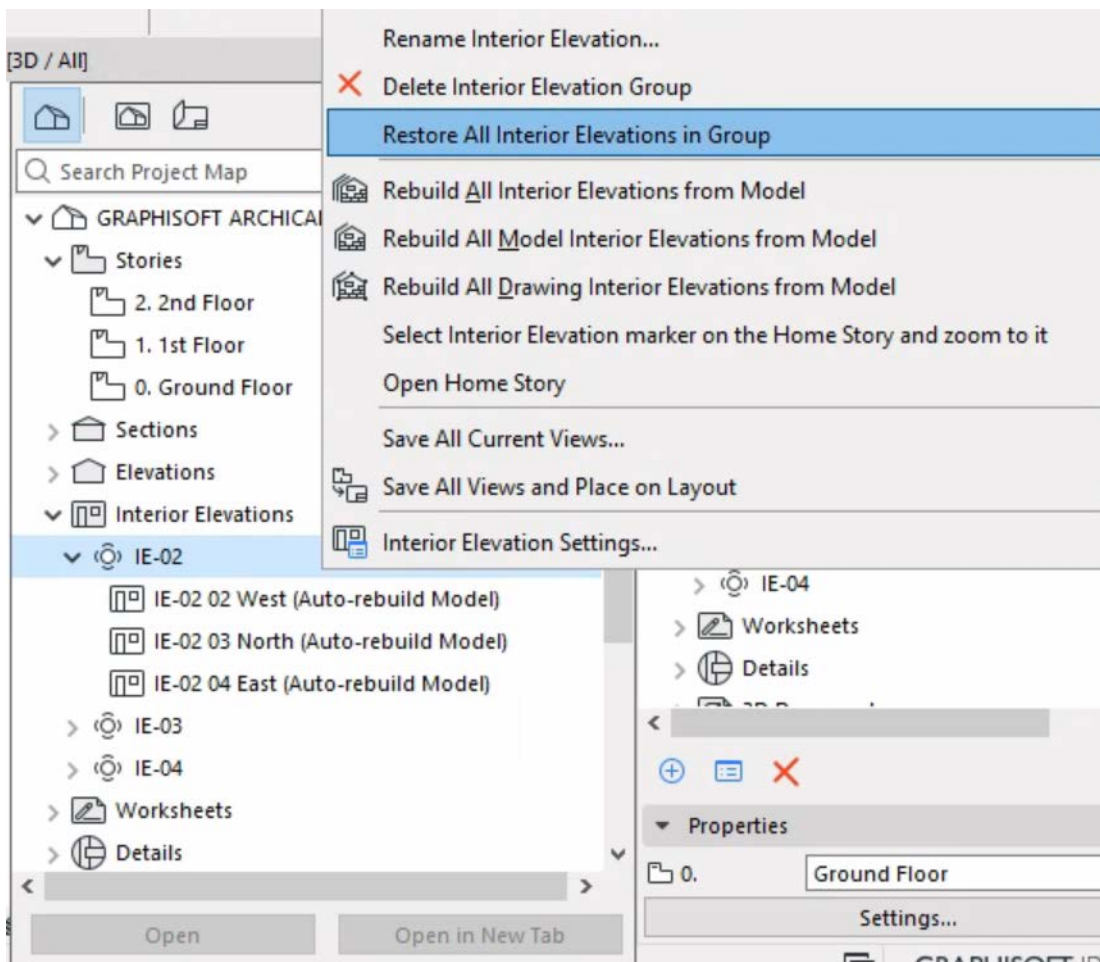
If you have selected an individual IE viewpoint, you can delete that single viewpoint with the context menu's **Delete** command.

Alternatively, if you have selected the entire IE group, use the **Delete Segment** command from the pet palette. The segment that you clicked to select the IE group will be deleted.

Note: To select an Interior Elevation group, select any of its Room Boundary lines.



You can restore this viewpoint even after deleting it: In the Navigator or in the project, select the IE group to which it belongs, then use the **Restore all Interior Elevations in Group** command from the context menu.



Note: This command regenerates the IE viewpoint, but any 2D edits you made in this viewpoint before deleting it have been lost.

Display of Elements in Interior Elevation

Use the Model Appearance controls in Interior Elevation Settings to define the appearance of cut and uncut elements in the view.

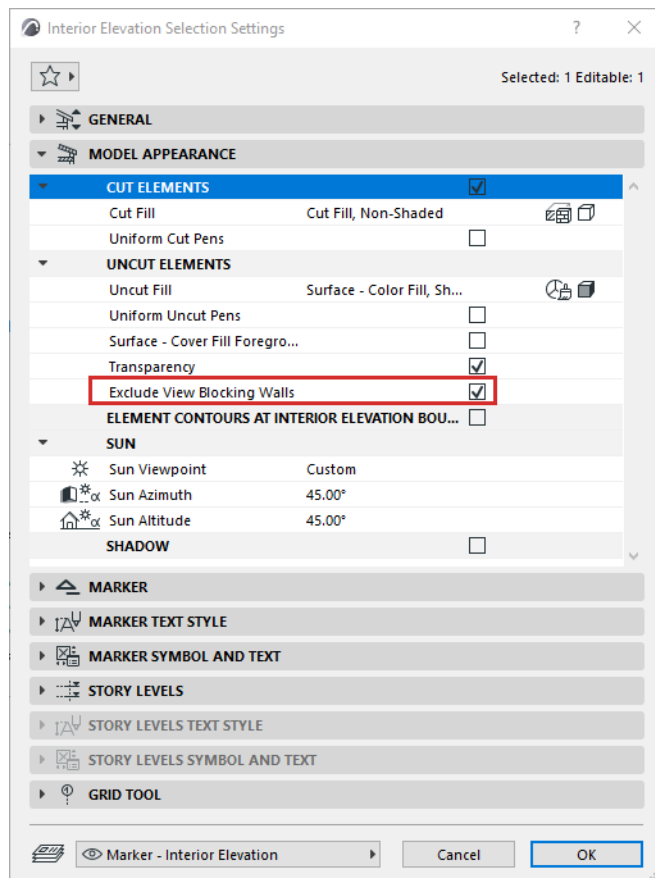
These options are largely identical to those in the Model Appearance panel of Section Settings.

Interior Elevations do not have a “Fade Distant Elements” display option.

[See Model Appearance \(Section-Type Views\).](#)

Exclude View Blocking Walls

Check this if you do not want intervening walls to be displayed in your Interior Elevation viewpoint.



3D Document

The 3D Document allows you to use either the Floor Plan or the 3D view of the model as the basis for creating a document, to which you can add dimensions, labels and additional 2D drawing elements.

Topics in this section:

[About the 3D Document](#)

[Create a 3D Document](#)

[Redefine the 3D Document](#)

[Model Appearance \(Section-Type Views\)](#)

About the 3D Document

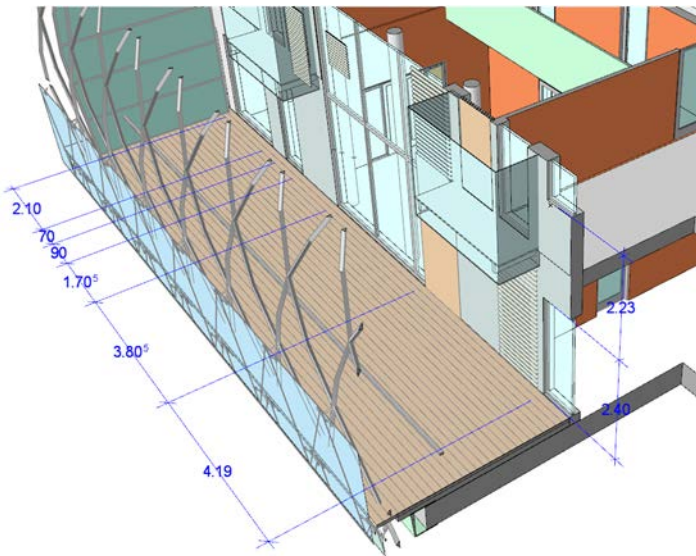
3D Document: Floor Plan Projection

Create this type of 3D Document based on a projection of the Floor Plan. Define the projection by setting a 2D cut plane height and defining projection direction either upwards (toward the ceiling) or downwards (to the floor). This is useful, for example, to display the load-bearing structures on a slab plan, or a reflected ceiling plan. Use it also to create shaded Floor Plan views for presentation/interior design.

3D Document: 3D Projection

This 3D Document has its source in the 3D window. Specific parameters and options affecting 3D window display (such as projection and filtering/selection of elements) will affect 3D Document display.

To add dimensions, just use Archicad's dimensioning tools, plus certain dimensioning functions that are available only for 3D Documents.



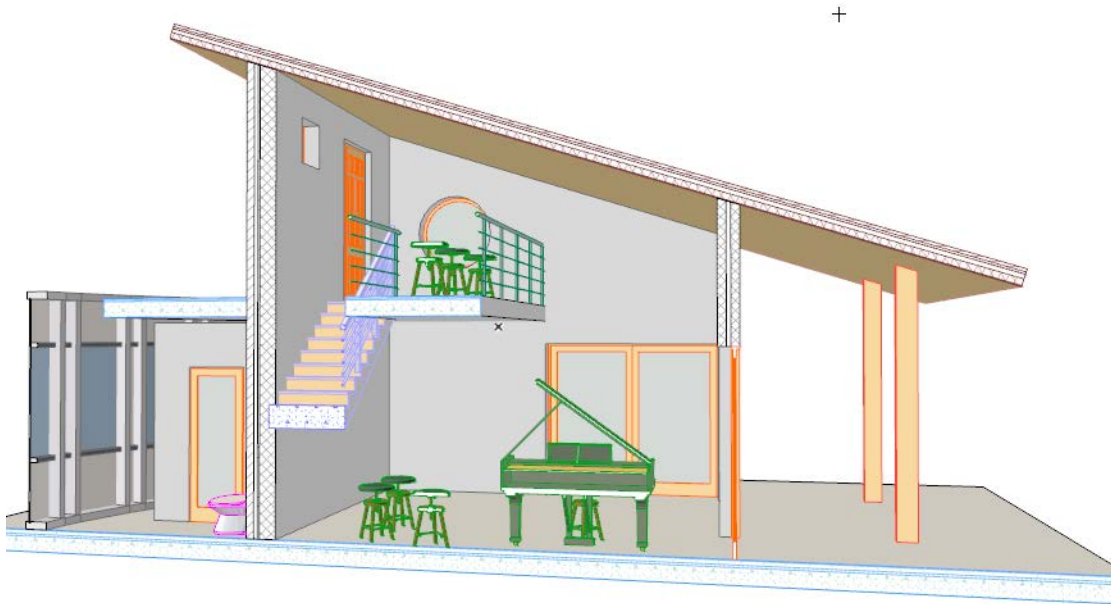
The 3D Document Viewpoint

The 3D Document is a viewpoint which appears in the Navigator Project Map.

You can also access 3D Documents through the **Go** command of the Mini-Navigator Toolbar, or from the **Window > Navigate > 3D Documents** menu, or from the Window menu.

In many respects, the 3D Document is analogous to the Section viewpoint: it is an integral part of the Archicad model, and its model elements are rebuilt automatically or manually, depending on its status.

In the 3D Document, you can select model elements and access their settings dialog boxes to make changes in the model, but you cannot edit them graphically or create new model elements.



Related Topics:

[Create a 3D Document](#)

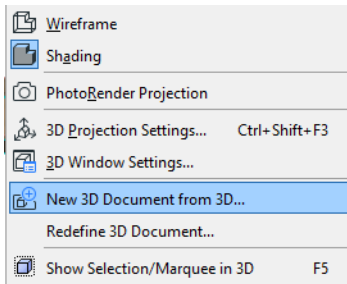
[3D Document Settings](#)

[Linear Dimensions in the 3D Document Window](#)

Create a 3D Document

From the 3D window:

1. In the 3D window, set up the desired 3D view of the model:
2. Create a new 3D Document by doing one of the following:
 - Use **New 3D Document from 3D...** from the context menu of the Navigator's 3D Documents folder.
 - Use **New 3D Document from 3D** command from the context menu of the 3D window



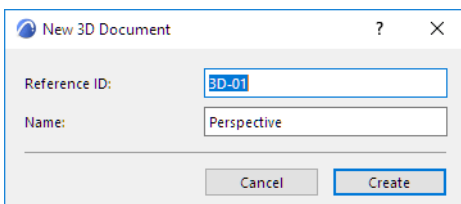
- Use a menu command (**Document > Documenting Tools > New 3D Document**).

From the Floor Plan:

1. On any story of the Floor Plan, set up the model view you want to see in your 3D Document.
2. If needed, limit the view using Selection or Marquee. (View settings for the 3D window, such as Filter and Cut Elements in 3D, have no effect on 3D Documents created from the Floor Plan window.)
3. Use the **New 3D Document from Floor Plan...** from one of the following locations
 - Floor Plan context menu
 - context menu of the Navigator's 3D Document folder
 - context menu of any Story from the Project Map.
 - a menu command (**Document > Documenting Tools > New 3D Document**).

New 3D Document Dialog Box

The New 3D Document dialog box appears.



Enter a Reference ID and Name for the new 3D Document; click **Create**.

Note: Optionally, the name of the source view can be set as the default 3D Document Name.

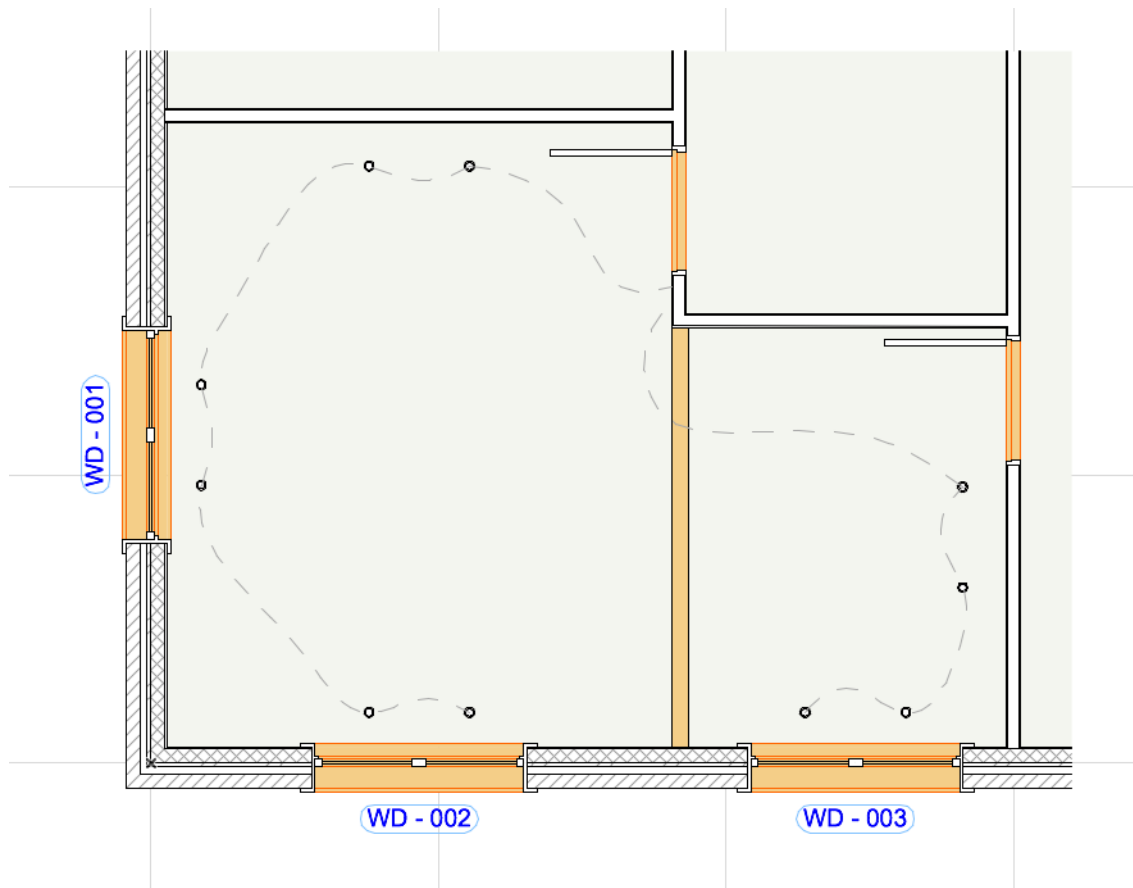
To set naming defaults, see [3D Document General Panel](#).

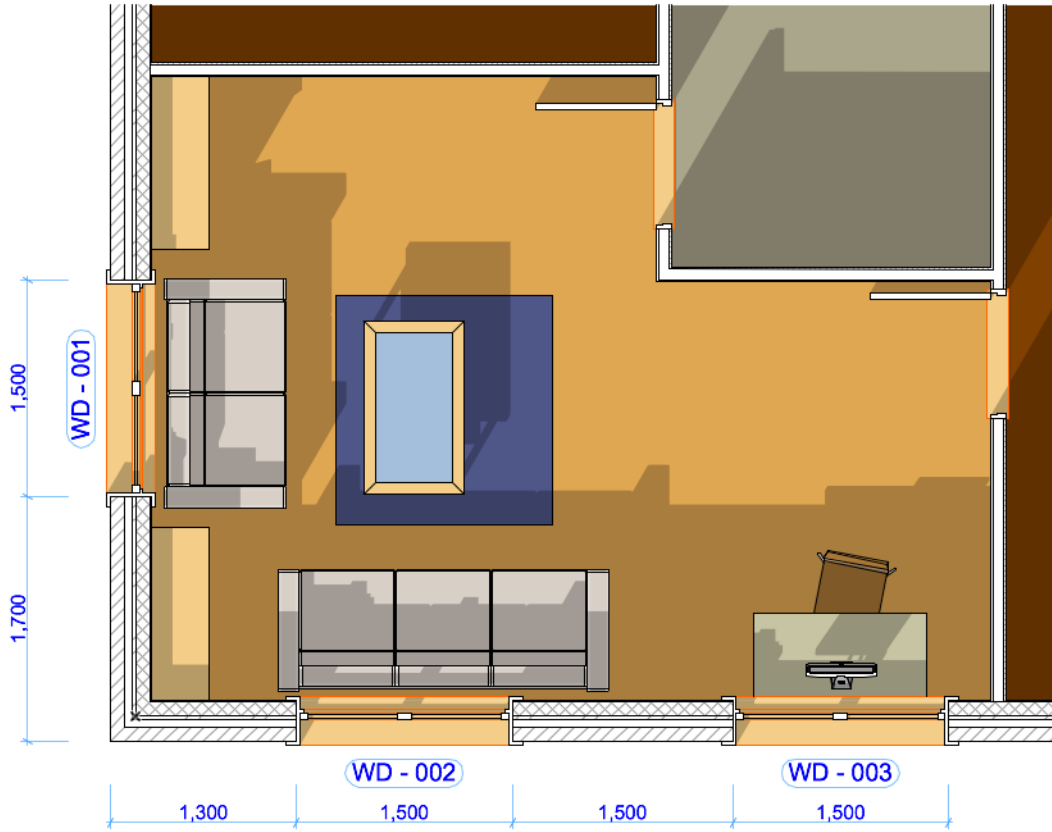
The 3D Document is created and opened in its own window. It is listed in the 3D Document folder in the Project Map of the Navigator.

To adjust the content, settings and display of the 3D Document, you can adjust its source view and/or its display settings, and then redefine it.

See the following for more information:

**Redefine the 3D Document
Model Appearance (Section-Type Views)**





Redefine the 3D Document

There are three ways to redefine a 3D Document after you have created it. Each method is described below:

Redefine 3D Document: Open and Edit Source Window

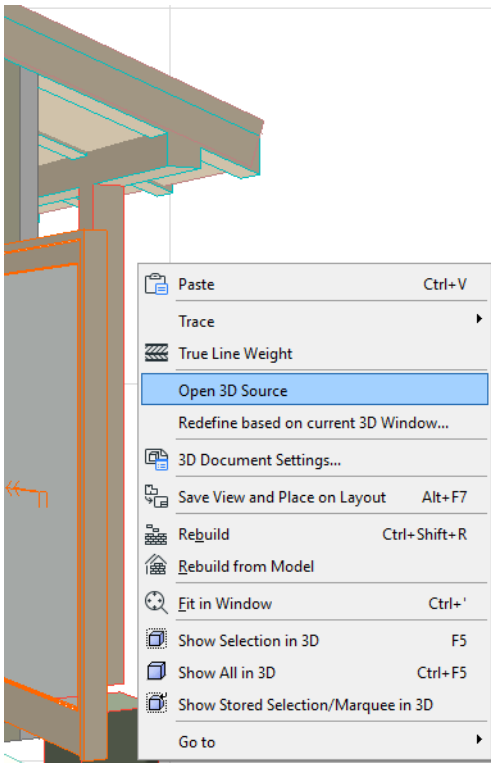
Redefine 3D Document Based on Current (3D or Floor Plan)

Redefine Source Using 3D Document Settings

Redefine 3D Document: Open and Edit Source Window

If the 3D Document window is in front:

From the context menu, use **Open 3D Source** (for 3D projections) or **Open Source Story** (for Floor Plan Projections).



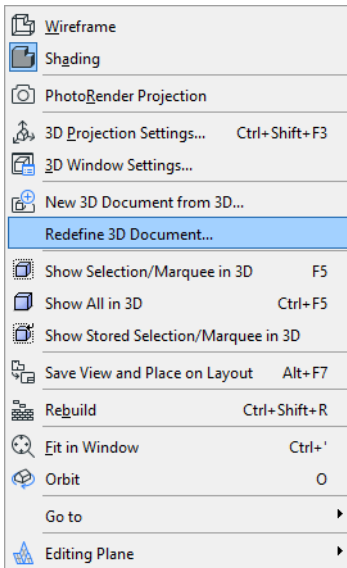
Or right-click the 3D Document viewpoint or view from the Navigator, then click the **Open Source View** command:

Make your changes in the source window:

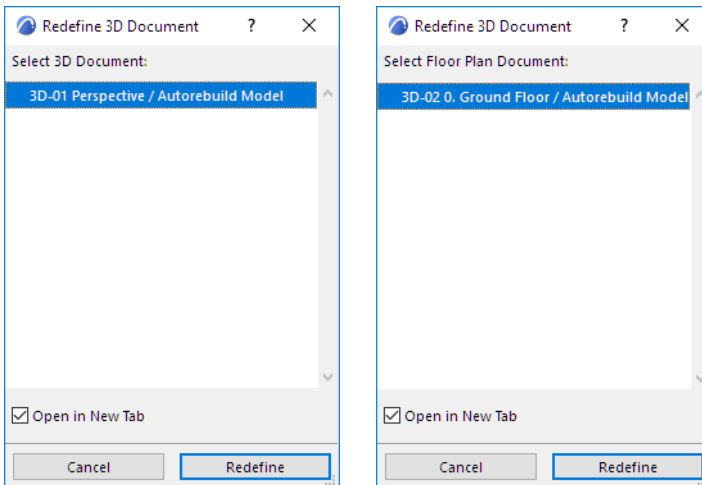
- Edit elements, layer settings, orientation or projection
- Change element visibility using Selection/Marquee
- If the source is a 3D Window, you can also change the Filter Elements settings (**View > Elements in 3D View > Filter and Cut Elements in 3D**) and adjust Cutting Planes

Then use the **Redefine** command by doing one of the following:

- Context menu of the 3D Document in the Navigator Project Map
- Context menu of the 3D window or Floor Plan



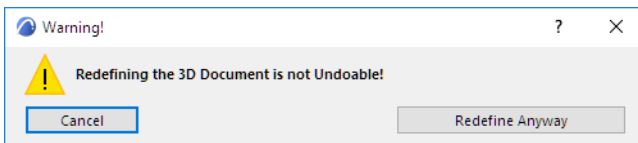
This brings up the **Redefine 3D Document** dialog box.



Choose the 3D Document which you want to redefine based on the current 3D window or current Floor Plan window.

Click **Redefine**.

A warning reminds you that this Redefine operation is not undoable.



Click **Redefine Anyway** to proceed.

Redefine 3D Document Based on Current (3D or Floor Plan)

Access this command by doing one of the following:

- Context menu of the 3D Document window
- Context menu of the 3D Document in the Navigator Project Map

- In 3D Document Settings, check the box to **Redefine the projection, cutting planes, and visible elements** of the 3D Document based on the current 3D window (available for 3D Documents based on a 3D projection.)

Redefine based on Current 3D Projection:

3D Document is redefined based on the following settings of the current 3D window:

- The current 3D projection, including zoom
- Filter and Cut Elements to Show in 3D (**View > Elements in 3D View**)
- Selection/Marquee limitations
- Cutting Planes

Note: The current window is not necessarily the same as the original source of your 3D Document. “Current” is the one currently open, or the one that was last open.

Redefine based on current Floor Plan Projection:

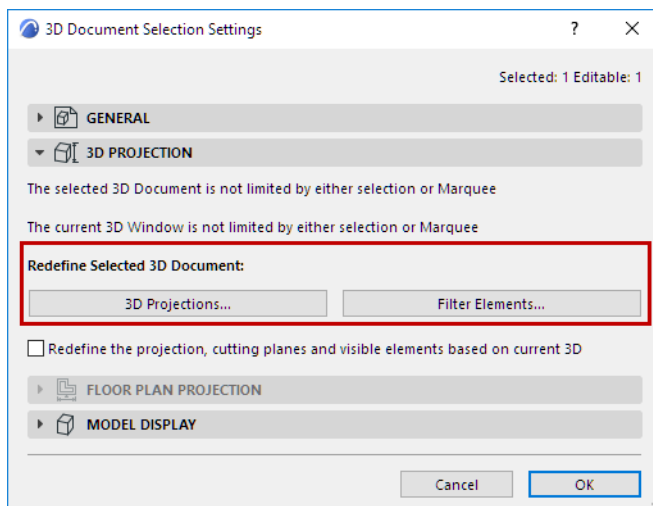
3D Document is redefined based on the following settings of the current Floor Plan:

- Selection/Marquee limitations
- Orientation
- Zoom
- Story

Redefine Source Using 3D Document Settings

In 3D Document Selection Settings: access dialog boxes to modify the 3D Document and its 3D window *source*.

Changes in your 3D Document will take effect when you click OK to leave 3D Document Settings. The current 3D window remains unchanged.



Redefine 3D Projections: Click to access the dialog box and readjust the 3D projection of the *source* 3D window.

[See also 3D Projection Settings.](#)

Redefine Filter Elements: Click to access the **Filter and Cut Elements in 3D** dialog box. Make any adjustments you need to the element filter of the *source* 3D window.

[See also Filter and Cut Elements in 3D Dialog Box.](#)

Details

The Detail tool is used to generate a Detail drawing from the model. Although the contents of the Detail viewpoint consist of 2D items, a Detail can be updated from the model.

Topics in this section:

[About Details](#)

[Create a Detail Drawing Viewpoint with Model Source](#)

[Contents of the Detail Viewpoint](#)

[Place a Linked Detail Marker](#)

[Place Unlinked Marker](#)

[Create an Independent Detail Viewpoint](#)

[Display of Detail/Worksheet Boundary](#)

[Update Detail/Worksheet Marker Boundary](#)

[Updating the Detail Window](#)

About Details

The Detail tool is used to generate a Detail drawing **viewpoint** from the model by placing a Detail marker in any of the following windows: Floor Plan; Section; Elevation; Interior Elevation; 3D Document; Worksheet; Detail window. It is also used to place **linked** Detail markers, as a reference to any viewpoint, view or drawing. The Detail Marker (a GDL object) is displayed according to the settings you made in the Detail Settings dialog box. You can also create an independent Detail drawing, with or without a marker.

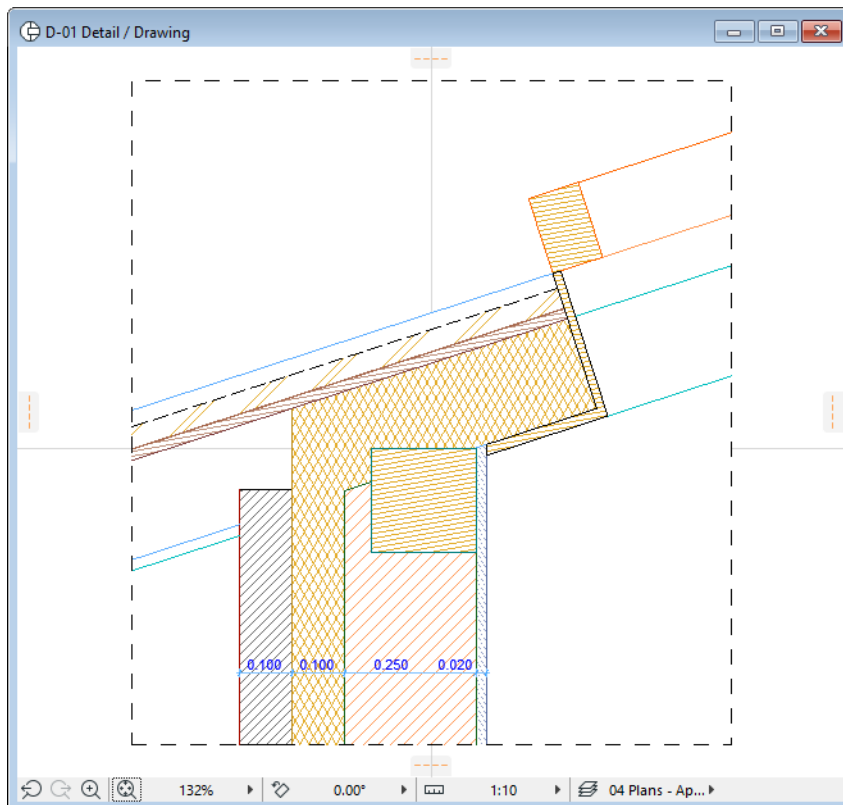
Detail viewpoints are displayed in a dedicated Detail Drawing Window and listed in Navigator Project Map.

Detail viewpoints generated from existing elements in the project are 2D-only copies of these elements. The viewpoints can be updated to reflect changes in the plan. You can add additional 2D elements, text, labels and objects to a Detail Drawing.

You may wish to fine-tune the Detail Drawing before final output. You will find, however, that the “exploded” 2D elements generated from the model often contain superfluous elements (extra line segments, overlapping or superfluous fills) that make such editing difficult. To make editing easier, first use the **Linework** and **Fill Consolidation** functions on selected items in the window.

For more information, see [Consolidate Lines and Fills in Drawing Windows](#).

To open the Detail Drawing in its own window, double-click the detail view name in the Navigator.



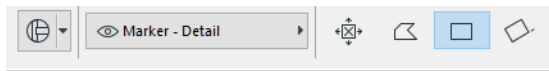
Create a Detail Drawing Viewpoint with Model Source

To create a Detail with its source in the model, place an area-based Detail marker in any of the following windows: Floor Plan, Section/Elevation/Interior Elevation, 3D Document, Worksheet or Detail.

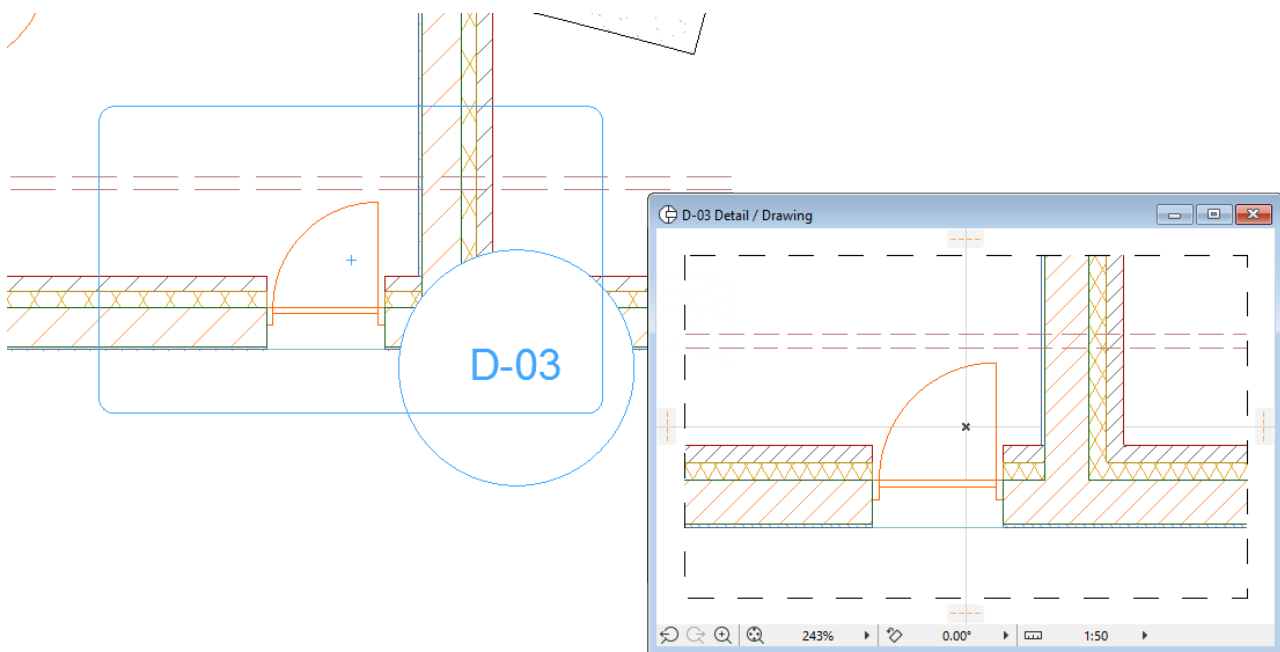
1. Activate the Detail tool.
2. In the Info Box or Detail Default Settings, make sure that "Create new detail viewpoint" is selected.



3. Choose an area-based geometry method from the Info Box: Polygonal, Rectangular, or Rotated Rectangular. (The first, "No Boundary" geometry method, creates a marker linked to an independent viewpoint having no model source.)



4. Draw the Detail boundary and click to complete.
5. After drawing the detail boundary, click with the hammer cursor to place the Detail Marker. The X represents the Detail's origin.



Note: Source-type Detail markers are distinguished on screen by an optional semi-transparent fill. (Use **View > On-screen View Options > Highlight Source Markers** to enable or disable this distinguishing fill for all source markers. The color of this highlight can be set in **Options > Work Environment > On-Screen Options**.)

6. A new Detail Drawing viewpoint is created and listed in the Navigator Project Map.

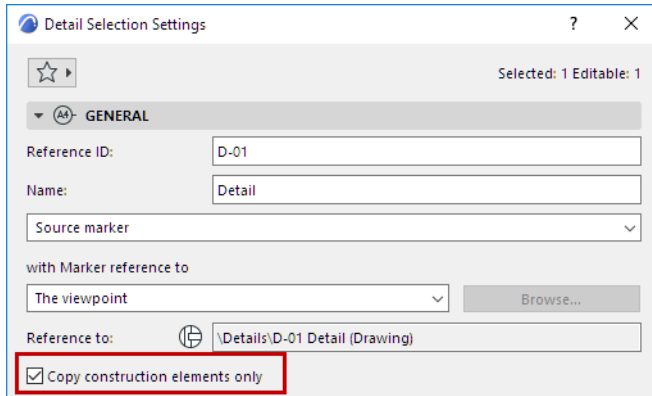
To define the **reference information** displayed in a source-type Detail marker:

[See *Define Marker Reference for Source Marker*](#).

Other information about the new Detail viewpoint - its Markers and Reference information - can be defined in Detail Settings.

Contents of the Detail Viewpoint

The source-based detail drawing consists of only 2D lines/fills, including the 2D exploded representations of all construction elements inside the detail boundary. By default, only construction elements are transferred; annotations and dimensions are *not* transferred to the Detail viewpoint. (You can change this setting using a checkbox in the General Panel of Detail Settings.)



For information on what happens to annotation elements cut by the Detail boundary, see [Annotations, Dimensions and Markers Cut by the Boundary](#).

Only 2D tools are available in the Detail window. All types of 2D information can be added to the Detail Drawing: Lines, Fills, Hotspots, Text, Figures, Dimensions and 2D Symbols of Objects.

The Detail Drawing will include a line representing the **Detail Boundary** you drew on the plan.

See [Display of Detail/Worksheet Boundary](#).

Section Details (Detail viewpoints created from a Section) reveal the elevation of the original construction, which means that Elevation Dimensioning placed on Detail viewpoints shows correct elevation values, unless you displace drawing elements vertically. (The same is true for Elevation Dimensions in Details generated from Elevations and Interior Elevations.)

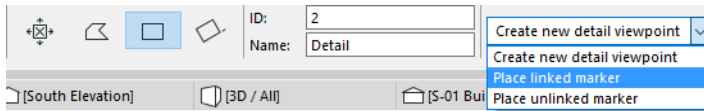
Related Topics:

[Worksheet vs. Detail](#)

Place a Linked Detail Marker

To place a linked detail marker without creating a viewpoint:

1. Activate the Detail tool.
2. In the Info Box or Detail Default Settings, make sure that “Place Linked Marker” is selected.



3. Define the Marker Reference using the appearing dialog box.

See [Define Reference for Linked Marker](#).

4. Choose any of the Detail geometry methods to place the marker.



- If you use the No Boundary geometry method: click once to place the marker
- If you use any of the area-based methods: draw a Detail boundary, then click with the hammer cursor to place the marker.

Such a marker has no model source and does not generate a new viewpoint. To define the **reference information** displayed in a linked Detail marker:

See [Define Reference for Linked Marker](#).

Create an Independent Detail Viewpoint

An Independent Detail drawing is one whose content you create yourself, starting with an empty detail viewpoint. You can then use 2D drawing tools or paste an object or existing drawing.

An Independent Detail drawing is not generated from existing project elements and so is unlinked to any modifications that take place in your plan.

You may wish to create an Independent Detail for your documentation which depicts something that does not otherwise exist in the model - for example, detail of a fence.

These independent Detail Drawings always appear in the Navigator tree structure. They can be marked or unmarked: Independent *marked* details have a corresponding detail marker on the plan. Independent *unmarked* details do not have a detail marker on the plan, but they exist as viewpoints which you can open from the Navigator.

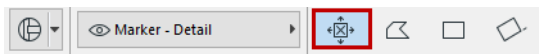
Create an Independent Detail viewpoint with a marker

1. In Detail Settings or the Info Box, choose “Create New Detail Viewpoint.”
2. Define the Marker Reference:
 - the viewpoint, or
 - the first placed drawing of the viewpoint

The link information of the chosen viewpoint/drawing will appear in the “Reference to” text field.

After the marker is placed and the new Detail is created, you can still go back and change its marker information at any time.

3. Choose the single-click geometry method (the first icon among the Info Box geometry methods) and click to place a detail marker on the plan.



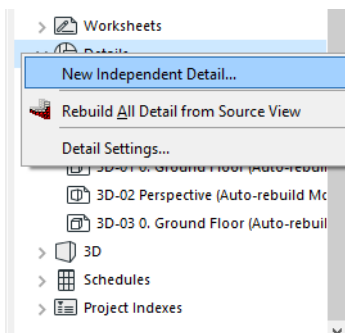
4. A new, empty Detail viewpoint is created (“independent marked detail”) and listed in the navigator. This viewpoint has no model source.

To open the independent Detail viewpoint, double-click its name in the Navigator or access it with the “Open Detail Drawing” in the marker’s context menu. Use the available 2D drawing tools, place objects, text, labels, or paste an existing drawing into the detail drawing window.

Create an Independent Detail viewpoint without a marker

Use the New Independent Detail command from one of the following locations:

- Context menu of the Detail folder in the Navigator Project Map



- New Viewpoint icon at the bottom of the Navigator Palette Project Map, if a Detail or the Detail folder is selected
- **Document > Documenting Tools > Create Independent Detail** command.

Independent unmarked details are listed in the Navigator, though they have no associated marker.

Later, if needed, you can attach a Independent Detail viewpoint to a marker: place a linked marker and point the marker to your Detail viewpoint.

Display of Detail/Worksheet Boundary

When you create a new Detail or Worksheet viewpoint using a polygon input method, the resulting drawing will show the Boundary as a dashed line. If you do not want this boundary included in the Drawing window, disable the relevant checkbox in **Options > Work Environment > More Options**.

▼ MORE OPTIONS

- Create Detail/Worksheet Boundary in Detail/Worksheet Window
- Save Navigator Preview Data with Project files
- Assign New Element ID to Each New Element
- Enable Archicad Sounds
- Show New Message Alert in Teamwork
- Show Tracked Changes Alert in Teamwork
- Use large alert cues
- Do not launch new instance of Archicad when placing a view onto a Layout
- When placing model views onto Layouts, adjust each drawing frame to show only zoomed area of model view

When opening another View or Layout of a similar type:

- Open views in a new tab
- Open layouts in a new tab

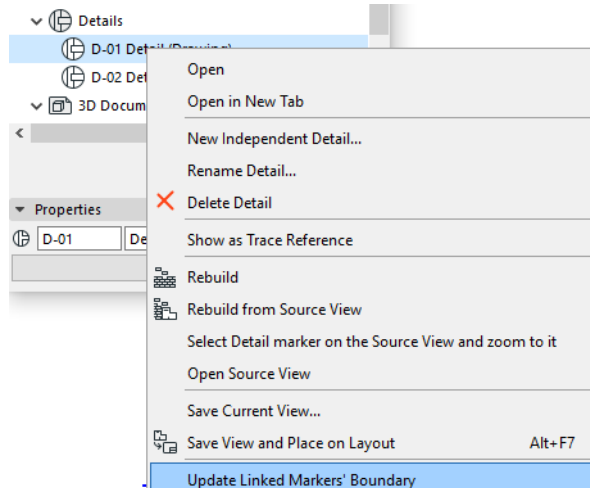
Note: This option will not affect Floor Plan or 3D windows.

- Show Preview in Tab Bar
- Launch new instance of Archicad when opening a project from the recent list or by dragging onto a running Archicad

▶ EXPERIMENTAL FEATURES

Update Detail/Worksheet Marker Boundary

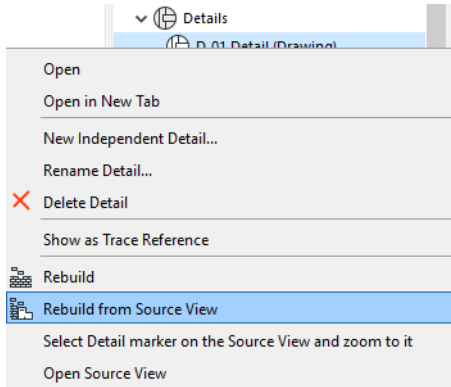
If you redraw the boundary of a source detail or worksheet marker, any markers linked to this marker's viewpoint will not be updated until you issue the **Update Linked Markers' Boundaries** command from the Detail/Worksheet viewpoint item's Navigator context menu.



This command affects the shape of marker boundaries only, not any associated model content.

Updating the Detail Window

If the source view of a Detail viewpoint has been modified, update its contents with the **Rebuild from Source View** command (from context menu of the detail drawing listed in the Navigator, or from the **View > Refresh** sub-menu).



For more information, see [Summary of Rebuild Commands](#).

Worksheets

The Worksheet window provides a dedicated environment for 2D model-based drawings, such as partial floor plans and partial sections, and for drawings created entirely in 2D.

Topics in this section:

[About Worksheets](#)

[Worksheet vs. Detail](#)

[Create a Worksheet Drawing with Model Source](#)

[Contents of the Worksheet Window](#)

[Editing in the Worksheet Window](#)

[Create an Independent Worksheet Viewpoint](#)

[Place a Linked Worksheet Marker](#)

[Place Unlinked Marker](#)

About Worksheets

The Worksheet window provides a dedicated environment for 2D model-based drawings, such as partial floor plans and partial sections, and for drawings created entirely in 2D. Worksheets can be used as a dedicated environment for finishing drawings using 2D tools, and for working with consultants limited to 2D methods.

You might want use a Worksheet to add a diagram or organizational chart, which is totally independent of your model, yet useful for your documentation.

As in the Detail window, elements on the Worksheet consist of 2D-only duplicates of any construction elements in “exploded” form (e.g. 2D lines and fills). Only 2D tools are available in the Worksheet window.

You may find that the “exploded” 2D elements generated from the model often contain superfluous elements (extra line segments, overlapping or superfluous fills) that make such editing difficult. To make editing easier, first use the **Linework** and **Fill Consolidation** functions on selected items in the window.

For more information, see [Consolidate Lines and Fills in Drawing Windows](#).

The user can link any type of marker to the Worksheet.

You can place a Section marker onto a Worksheet to create a Drawing-type section. You can also place a Detail marker onto a Worksheet to create a Drawing-type Detail. Such Drawing Sections and Drawing Details cannot be refreshed based on the content of the Worksheet window.

The Worksheet tool is active in any window. It has marker and linking options similar to those of the other Archicad marker tools (Section, Detail).

To open a Worksheet window, double-click its name in the Navigator. (You can also access Worksheets through the **Go** command of the Mini-Navigator Toolbar, or from the **Window > Navigate > Worksheets** menu, or from the Window menu.) Use the available 2D drawing tools, place objects, text, labels, or paste an existing drawing into the drawing window.

Worksheet vs. Detail

While creating a Worksheet based on the source model is similar to creating a model-based Detail, the Worksheet is optimized for creating partial floor plans and partial sections.

Unlike the Detail tool, the Worksheet tool has a single-click “capture” geometry method which reproduces the entire contents of the current window as 2D elements in the Worksheet viewpoint.

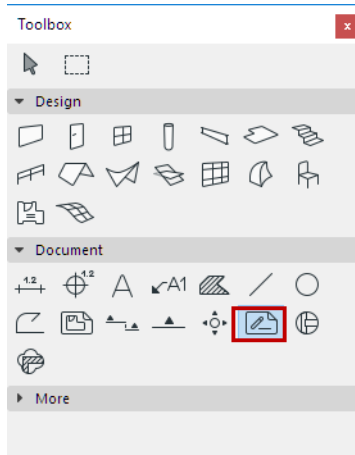
Consequently, when transferring all or part of a source model view to a Worksheet window:

- Annotation elements and dimensions will, by default, appear in the resulting Worksheet, in addition to construction elements exploded into 2D components. (In contrast, a Detail viewpoint created from the model will - by default - contain the 2D-only copies of construction elements only, without the source model's annotation/dimension elements.)
- The resulting Worksheet will have the same scale as the source. (In contrast, a Detail is created, by default, at half the scale of the original.)

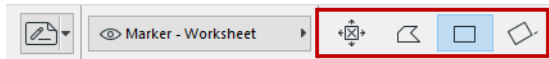
Create a Worksheet Drawing with Model Source

To create a Worksheet with its source in the model, place a Worksheet marker in any of the following windows: Floor Plan, Section/Elevation/Interior Elevation, Worksheet or Detail.

1. Activate the Worksheet tool.



2. In the Info Box or Default Settings, make sure that "Create new Worksheet viewpoint" is selected.
3. Choose any of the geometry methods from the Info Box.



- **Capture Screen input method:** Use the single-click method: the first input method in the Worksheet info box. Choose this icon, then click anywhere in the active project window. You will create a Worksheet based on the entire contents of the window.
 - **Area-based input method:** Use one of the polygonal or rectangular/rotated rectangle geometry methods to define the Worksheet area.
4. Draw the Worksheet boundary and click to complete.
 5. After drawing the Worksheet boundary, click with the hammer cursor to place the Worksheet Marker.

Note: Source-type Worksheet markers are distinguished on screen by an optional semi-transparent fill. (Use **View > On-screen View Options > Highlight Source Markers** to enable or disable this distinguishing fill for all source markers. The color of this highlight can be set in **Options > Work Environment > On-Screen Options**.)
 6. A new Worksheet viewpoint is created and listed in the Navigator Project Map.

To define the **reference information** displayed in a source-type Worksheet marker.

See [Define Marker Reference for Source Marker](#).

Other information about the new Worksheet viewpoint - its Markers and Reference information - are defined in Worksheet Settings.

See [Worksheet and Detail Tool Settings](#).

Contents of the Worksheet Window

A project-based Worksheet only contains drawing primitives, that is, lines and fills. These originate from the construction elements on the Floor Plan or Section/Elevation/IE, or 3D Document.

Section/ Elevation and Detail/Worksheet markers within the Worksheet boundary are “live” markers: they continue to function as markers.

When transferring all or part of a model view to a Worksheet window, annotation elements and dimensions will, by default, also appear in the resulting Worksheet.

To copy construction elements only, without annotations/dimensions, check the “Copy construction elements only” box in Worksheet Settings.

The resulting Worksheet will have the same scale as the source, if any. (If the source has no scale, the Worksheet will be shown at a default scale value.)

Annotations, Dimensions and Markers Cut by the Boundary

If the Worksheet boundary intersects annotation elements, dimensions or marker-type elements (Section/Elevation markers; Detail/Worksheet markers), then these will be transferred to the Worksheet window if they meet the relevant criteria, as follows.

Note: The following also apply to Detail drawings. (However, by default, Detail drawings do not include annotations or dimensions at all, unless you uncheck “Copy Construction elements only” in Detail Settings.)

Figures and Drawings: if at least one reference point is inside the worksheet boundary, the entire Figure or Drawing will be included in the Worksheet.

Labels: Arrowhead must be inside the boundary.

Dimensions: The relevant reference point(s) or the dimension’s endpoints must be inside the boundary.

- Dimensions with two endpoints must have both endpoints inside.
- Dimension chains will be transferred by segment: if both endpoints of any segment are within the boundary, that segment is included in the Worksheet.

If any part of the Wall falls within the boundary, then all Doors, Windows, and Door/Window markers in that Wall are also included in the Worksheet.

Detail markers: If a detail viewpoint’s origin (the “X”) or endpoint of a marker falls within the boundary, the entire marker is included in the Worksheet.

Section and Elevation Markers: If any part of a Section or Elevation line is included in the boundary, the Section/Elevation marker (a GDL object) will be included in the Worksheet, although the line’s geometry may be different (the line may be shorter) if the boundary includes only part of the Section/Elevation line.

Note: Interior Elevation markers, unlike Section/Elevation markers, are cut by the boundary and appear in the Worksheet as exploded line/fill components.

Related Topics:

[Display of Detail/Worksheet Boundary](#)

[Update Detail/Worksheet Marker Boundary](#)

Editing in the Worksheet Window

Only 2D tools are available when the Worksheet window is active.

All types of 2D information can be added to the Worksheet: Lines, Fills, Hotspots, Text, Figures, Drawings, Dimensions and 2D Symbols of Objects.

You can add content to your Worksheet window in several ways:

- Select any element(s) from any window and paste them into the Worksheet window. All elements, including construction elements, will be pasted in 2D form.
- To transfer elements from a Trace Reference, first switch the Reference with the active window, then copy-paste from there into the Worksheet (in exploded form)
- Draw all new 2D elements into the Worksheet window.

Create an Independent Worksheet Viewpoint

An independent worksheet is one which you create yourself, starting with an empty worksheet viewpoint. You can then use 2D drawing tools or paste an object or existing drawing.

An independent worksheet is not generated from existing project elements and so is unlinked to any modifications that take place in your plan.

Independent Worksheets without a marker are also listed in the Navigator. However, you can subsequently link a marker to such a Worksheet as needed.

Create an Independent Worksheet without a marker

Use the **New Independent Worksheet** command from one of the following locations:

- Context menu of the Worksheet folder in the Navigator Project Map
- New Viewpoint icon at the bottom of the Navigator Palette Project Map, if a Worksheet or the Worksheet folder is selected
- **Document > Documenting Tools > Create Independent Worksheet** command.

Place a Linked Worksheet Marker

To place a linked Worksheet marker without creating a viewpoint:

1. Activate the Worksheet tool.
2. In the Info Box or Default Settings, make sure that "Place Linked Marker" is selected.
3. Choose any of the Worksheet geometry methods to place the marker.

Such a marker has no model source and does not generate a new viewpoint.

To define the **reference information** displayed in a linked Worksheet marker:

See [Define Reference for Linked Marker](#).

Managing Markers in Archicad

Several Archicad tools that create a view use markers. The behavior and management of these markers is analogous for all tools with markers:

- Section
- Elevation
- Interior Elevation
- Worksheet
- Detail

Topics in this section:

[About Markers](#)

[Display of Marker Range Lines](#)

[Source Marker Highlight](#)

[Changing Marker Type](#)

[Transfer Marker Parameters](#)

[Copying a Marker](#)

[Find Linked Markers](#)

[Check Markers Palette](#)

[Deleting a Viewpoint/View/Drawing with a Marker](#)

[Deleting a Source Marker](#)

About Markers

Markers can be placed using any of the following tools:

[Sections](#)

[Elevations](#)

[Interior Elevations \(IE\)](#)

[Worksheets](#)

[Details](#)

With any of the Marker tools, you can opt to place a **source marker**. A source marker creates a new viewpoint (e.g. a Section, Elevation, Worksheet, Detail) involving the model as their source.

Any of the Markers - except Interior Elevation - can optionally be defined as a **linked marker**. Such markers have no model source and do not generate a new viewpoint. Instead, they are placed in the project, then linked by the user to an existing viewpoint, view or drawing. This information is displayed in the marker, which can be used to navigate to the referred item.

An **unlinked marker** has no model source and displays no linked information. You cannot use it to navigate among markers, but you can define any custom text for it.

Topics in this section:

[Display of Marker Range Lines](#)

[Source Marker Highlight](#)

[Changing Marker Type](#)

[Transfer Marker Parameters](#)

[Copying a Marker](#)

[Find Linked Markers](#)

[Check Markers Palette](#)

[Deleting a Viewpoint/View/Drawing with a Marker](#)

[Deleting a Source Marker](#)

Source Marker Highlight

Source markers are distinguished on screen by an optional semi-transparent fill. This option makes it easy to tell at a glance which of your markers on screen are source markers.

To toggle this highlight on or off for all source markers, use the **View > On-Screen View Options > Highlight Source Markers** command.

To change the color of the source marker highlight, use the color chooser in **Options > Work Environment > On-Screen Options**.

Related Topics:

[Display of Detail/Worksheet Boundary](#)

[Update Detail/Worksheet Marker Boundary](#)

Changing Marker Type

You can change the type of an already placed marker, so long as all the Marker references point to items that originate with a Marker tool.

To change a linked marker to a source marker:

1. Select the marker.
2. Open the marker's Settings dialog box and choose the Source Marker option

The linked marker becomes the source marker, while the original source marker becomes a linked marker.

This function is available for:

- Linked markers which are linked to a viewpoint/view/drawing that is located within the current project. The linked marker must have been placed **using the same tool** as the item to which it was linked (e.g. a Section marker linked to a Section viewpoint).

Note: A viewpoint's cut/model attributes are defined by its source marker. If a linked marker becomes the source of a section, that marker's cut/model attributes are now applied to the viewpoint.

Note: If you change a linked detail marker into a source detail marker, the viewpoint's defining polygon will change to reflect the new source detail marker's polygon.

- Linked markers linked to an **independent viewpoint** of the same type.
The independent viewpoint is deleted and replaced by a source viewpoint. (If the original linked marker had no boundary, the source will take on a default boundary value.)

Markers linked to an external drawing cannot be changed to a source marker.

To change a source marker to a linked marker:

1. Select the marker.
2. Open the marker's Settings dialog box and choose the Linked Marker option

The original source marker becomes a linked marker containing the same reference information as before.

Transfer Marker Parameters

You can use the **Pick Up Parameters** and **Inject Parameters** commands between markers.

If you Pick Up Parameters **from a source marker** and inject them into another marker, the second marker will take on all of the first marker's parameters except its link information, and except for its geometry (e.g. the range of a section marker.)

If you Pick Up Parameters **from a linked marker** and inject them into another marker, the second marker will become identical to the first: a second linked marker with the same parameters, and containing the identical reference information, as the first.

See also [Parameter Transfer](#).

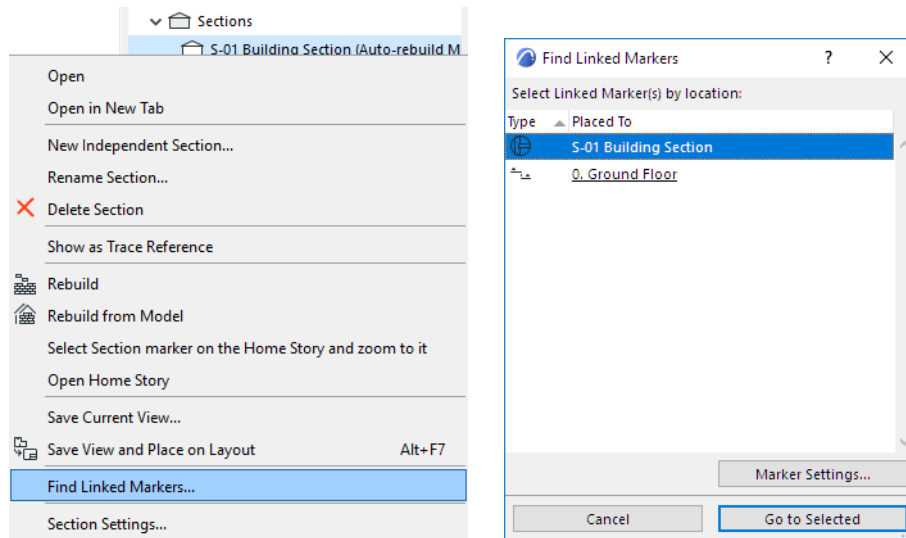
Copying a Marker

Any interaction (Copy/Paste, Drag/Rotate/Mirror Copy/Copies) that results in creating a new copy of a marker will have the following results:

- Copying a source marker (Section/Elevation/IE or Worksheet) results in a second source marker. This creates a new viewpoint having the same parameters as the viewpoint associated with the original marker. The new viewpoint's name, however, will be different. Moreover, the Marker Link information is *not* copied.
- Copying a source Detail marker will create a new linked Detail marker, which is linked to the original detail viewpoint.

Find Linked Markers

To find all the markers linked to a particular viewpoint/view/drawing, select the item from the Navigator. Use the **Find Linked Markers** command from the item's context menu.



The Find Linked markers dialog box lists all the linked markers:

- By marker type (shown as an icon)
- Placed to: ID and name of the viewpoint in which the marker is placed. (If it is a source marker, the viewpoint name is underlined.)

Clicking either column header will sort the list accordingly.

Click **Marker Settings** to open the Tool Settings dialog box of the selected marker.

Click **Go to Selected** to zoom to the selected marker. (The button is disabled if multiple markers are selected.)

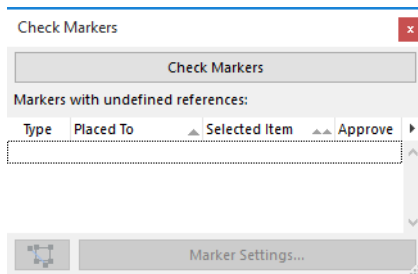
Check Markers Palette

The Check Markers function lists which marker-referred items are missing from the project.

The Check Markers Palette is available:

- from **Window > Palettes > Check Markers**
- by clicking the **Check Markers** command on the Layouts and Drawings toolbar

To list undefined marker references, click the **Check Markers** button at the top of the palette.



The palette lists all the markers in the project which the user has automatically or manually linked a viewpoint, drawing or view, but whose referred viewpoint/drawing/view *has since been deleted* from this project. (In this case, the marker will display Autotext in place of the missing data.)

Markers are listed, and can be sorted, according to the following:

- By marker type (shown as an icon)
- **Placed to:** ID and name of the viewpoint in which the marker is placed.
- **Selected item:** Name and ID of the marker's referenced item (i.e. the viewpoint/view/drawing one you chose when defining the marker reference). If this referenced item has been deleted from the project, this column will show "N/A".
- **Approve:** Editable checkbox for each marker.

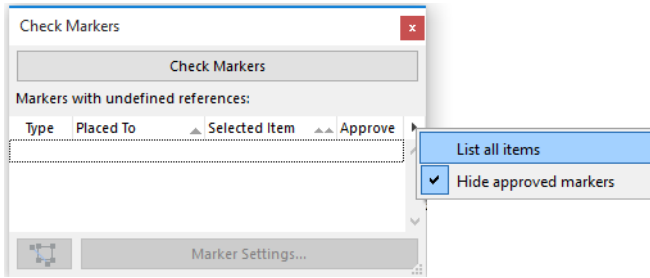
You can sort items by any of these properties by clicking on its column header. You can then perform a secondary sorting function: clicking a second column header will further sort within the hierarchy obtained by the first sorting.

An icon at the bottom of the palette allows the user to zoom to the selected marker. (The zoom button is disabled if multiple markers are selected.)

For each problematic marker listed (or multiple selected markers), the user can click **Marker Settings** to re-link the marker(s) or otherwise modify them.

The user may decide that the marker is OK the way it is (even though it continues to display undefined Autotext). In this case, the **Approve** checkbox can be used as a sorting criterion; check the box for these markers. (For example, if you know that the undefined Autotext refers to an external project that has not yet been loaded.)

If you don't want the palette to re-list these approved markers as problems the next time you click the Check Markers button, use the **Hide Approved Markers** option from the pop-up at top right of the dialog box.



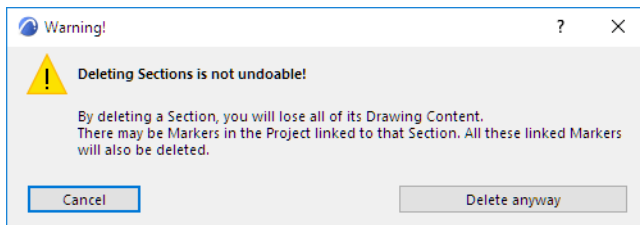
Otherwise, use the **List all items** option.

Deleting a Viewpoint/View/Drawing with a Marker

If you delete a viewpoint, the program will first check whether its source/linked markers are located on a locked layer or in a different Teammate's workspace, or whether the marker elements themselves have been locked. These markers include:

If none of these markers is locked or reserved, then the program displays a Warning dialog box informing you that proceeding with the Delete procedure will delete all of the following:

- The viewpoint (even if its source marker displays the data of a different viewpoint)
- Any views created out of the viewpoint
- All its source/linked markers



If you delete a view, any markers linked to that view will remain in place. However, they will no longer display any linked information, just an undefined Autotext. These markers will be listed as problematic the next time you run the Check Markers function.

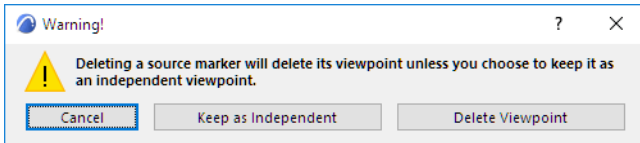
Markers linked to the deleted drawing with "first drawing from viewpoint" will henceforth be linked to the next drawing created out of the same viewpoint.

If you delete a drawing, any markers linked to that drawing will remain in place. However, they will no longer display any linked information, just an undefined Autotext.

Deleting a Source Marker

If you delete a source marker, a dialog box appears informing you that deleting a source marker will also delete the viewpoint and any markers linked to it.

However, the same dialog box gives you a different option: delete the source marker, but keep the viewpoint as an independent viewpoint with no model source.



Deleting a linked marker has no effect on any other element.

Interactive Schedule

Use the **Interactive Schedule** function to generate schedules. The Interactive Schedule not only displays quantities and other parameters - you can actually edit it. Any changes you make in these fields of the Schedule are automatically and instantly reflected in the Floor Plan and in other views, when activated. Conversely, all changes made on these elements in the Floor Plan or another editable view are updated in the Schedule when you return to it.

Schedules are viewpoints, which you can save as views and output as drawings on a Layout, or save in various other file formats.

See [Save from Schedule Window \(Interactive Schedule\)](#).

A formatted schedule can be placed into any 2D window (typically a Worksheet), by using Copy/Paste. The contents of the schedule, after pasting, will consist of lines and text, which you can freely edit, but will no longer be associative to the model.

Topics in this section:

[Types of Interactive Schedules](#)

[Units in Interactive Schedules](#)

[Open and View a Schedule](#)

[Show Schedule Data for Selected Floor Plan Items](#)

[Editing and Updating Schedule Items](#)

[Select Schedule Items on Floor Plan/3D](#)

[Define Schedule Content Using Scheme Settings](#)

[Surface Schedule](#)

[Format Options \(Interactive Schedule\)](#)

[Merge Schedule Items](#)

[Schedule Headers](#)

[Previews and Pictures in the Interactive Schedule](#)

[Restructure Schedule to Fit Layout](#)

[Split Schedule into Multiple Layouts](#)

Related Topics:

[Element Listing Parameters](#)

[Component Data in Archicad](#)

[Import/Export Property Data via Spreadsheet](#)

For Teamwork Users:

See [Reservation in Interactive Schedules \(Teamwork\)](#).

Types of Interactive Schedules

There are three categories of Interactive Schedules:

- **Element Lists** show characteristics of basic construction elements

An illustrated list of the parameters, divided by element type, is available here:

[See Element Listing Parameters.](#)

- **Component Lists** are optimized to list information about *each* component of composite, profile or basic elements: walls, slabs, roofs, morphs, meshes, and/or multi-component profile elements).

[See Component Data in Archicad.](#)

- **Surface Lists** list elements by their exposed surfaces.

[See Surface Schedule.](#)

Door Schedule			
Element ID	DOO - 001	DOO - 002	DOO - 003
2D Symbol			
3D Back View			
ID by Classification	Door - 001	Door - 002	Door - 003
Opening Name	Door 27	Door with 2 Sideligh...	Double Door with Si...
Quantity	1	1	1
From Zone	<Undefined>	<Undefined>	<Undefined>
To Zone	<Undefined>	<Undefined>	<Undefined>
W x H Size	0,900x2,100	2,100x2,400	2,100x2,100

Units in Interactive Schedules

An **editable value** (such as wall height) is shown on the Interactive Schedule in the units defined on the Dimensions page of Project Preferences.

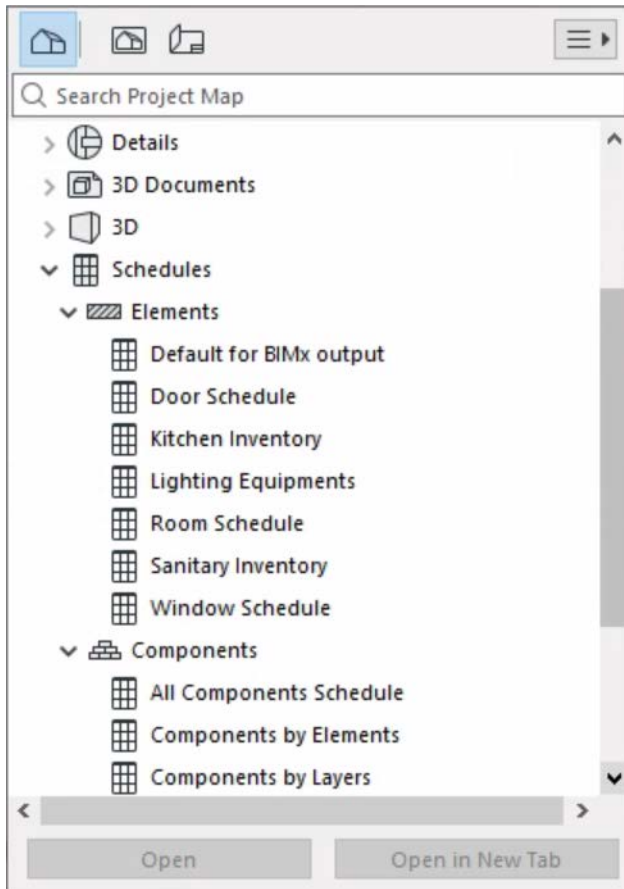
[See *Dimensions Preferences*](#).

A **calculated value** (such as wall volume) is listed in the Interactive Schedule using the units defined on the Calculation Units page of Project Preferences.

[See *Calculation Units Preferences*](#).

Open and View a Schedule

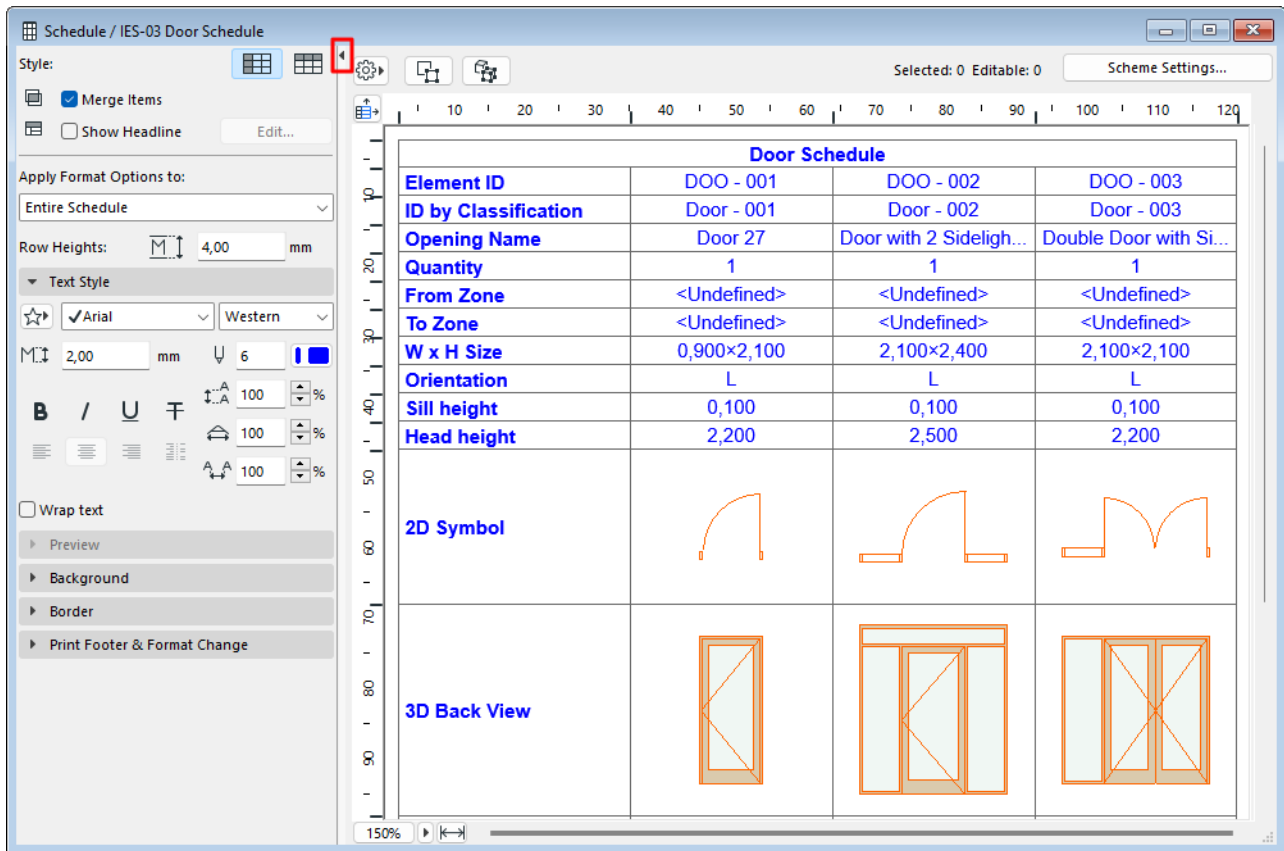
Archicad comes with predefined Schedules, such as Door List; Wall List; Window List. To open any of these schedules to list the elements in the current project, double-click the Schedule name in the Navigator.



You can also access Schedules through

- the Mini-Navigator Toolbar
- **Document > Schedules**
- **Window > Navigate > Schedules**

The Schedule window opens. Formatting options are in the left column. To display the schedule only, without the formatting panel, click the black arrow on the dividing line.



Navigating in the Interactive Schedule Window

To navigate among fields in the Interactive Schedule window, use familiar keyboard shortcuts:

Move right or left: Enter, Shift+Enter

Move down or up: Tab, Shift+Tab

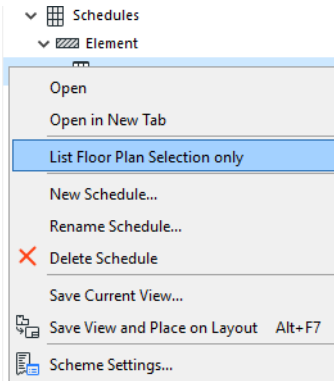
Show Schedule Data for Selected Floor Plan Items

To view a Schedule that shows data only for selected Floor Plan items:

1. Select the desired elements (on the Floor Plan only). Use Find and Select if needed.

See also [Find and Select Elements](#).

2. Select the desired Schedule in the Navigator Project Map or View Map.
3. Right-click the schedule name and choose “List Floor Plan Selection Only” to view schedule data for the selected items only. (The schedule lists only the selected elements which also meet the schedule Criteria.)



This is a quick way to filter elements for editing as a group, by taking advantage of the Interactive Schedule capabilities.

Note: A view saved from such a schedule does not store the selection information.

Editing and Updating Schedule Items

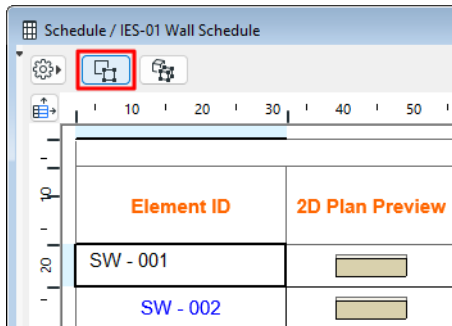
To work on an item of the Interactive Element Schedule, click its column (or row). You will see that some of the cells of this column become editable, while others cannot be modified.

Door Schedule			
Element ID	DOO - 001	DOO - 002	DOO - 003
2D Symbol			
3D Back View			
ID by Classification	Door - 001	Door - 002	Door - 003
Opening Name	Door 27	Door with 2 Sideligh...	Double Door with Si...
Quantity	1	1	1
From Zone	<Undefined>	<Undefined>	<Undefined>
To Zone	<Undefined>	<Undefined>	<Undefined>
W x H Size	0,900x2,100	2,100x2,400	2,100x2,100
Orientation	L	L	L
Sill height	0,100	100	0,100
Head height	2,200	2500	2,200
Fire Resistance Rating [minutes]	<Undefined>	<Undefined>	<Undefined>
Thermal Transmittance [W/m²K]	<Undefined>		<Undefined>
Sound Transmission Class	<Undefined>		<Undefined>
Fire Exit	<Undefined>	<input type="checkbox"/>	<Undefined>

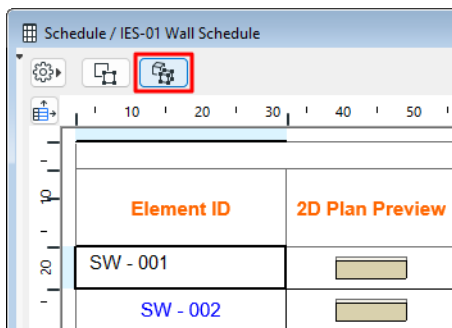
Any changes you make in these fields of the Schedule are automatically updated in the Floor Plan and in other views, when activated. Conversely, all changes made on these elements in the Floor Plan or another editable view are updated in the Schedule when you return to it.

Select Schedule Items on Floor Plan/3D

For any selected item in the Schedule Window, click the **Select on Floor Plan** icon to view it in the Floor Plan window.

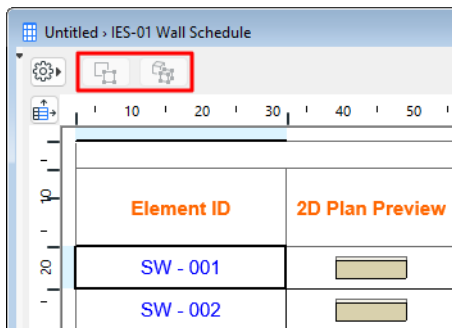


The icon to its right, **Select in 3D**, takes you to the 3D Window and zooms onto the selected item.



If nothing in the Schedule is selected, these commands will select *all* scheduled elements in the Floor Plan/3D window.

Note: You cannot edit or select scheduled elements that are on a locked or hidden layer.



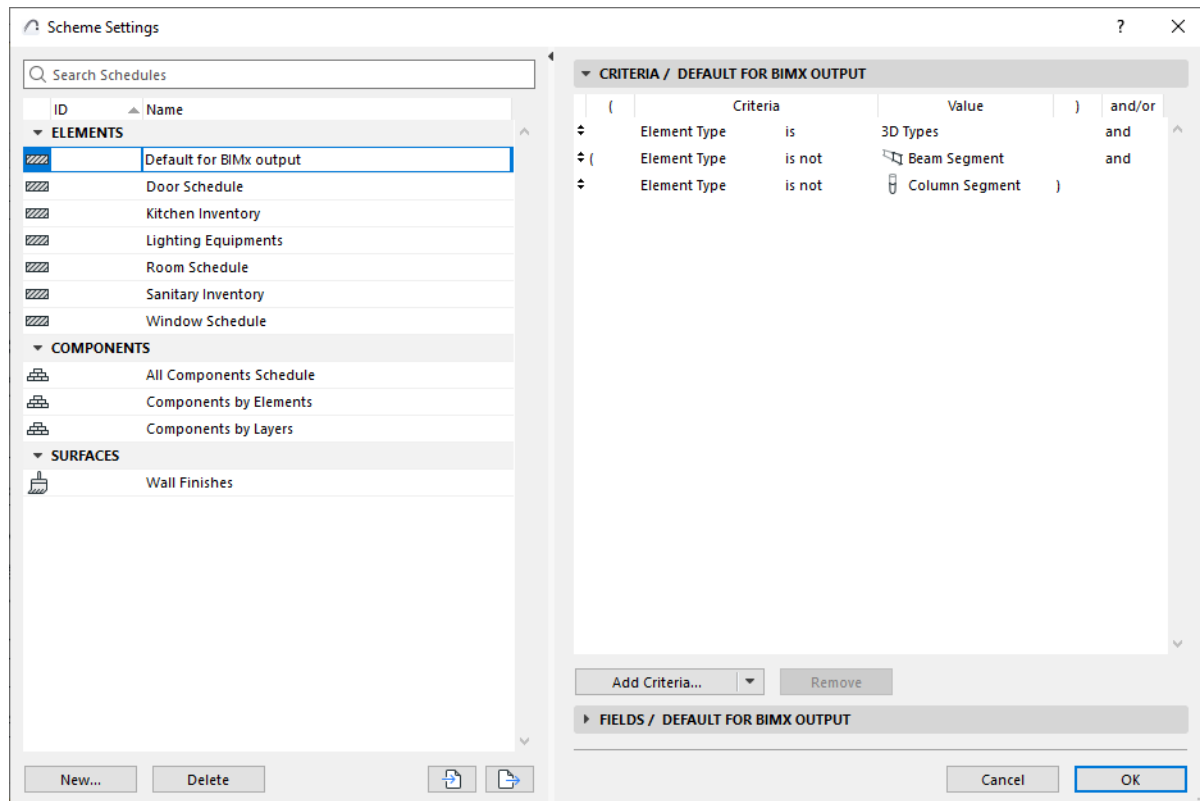
See [Scheduled Elements on Locked or Hidden Layer](#).

Define Schedule Content Using Scheme Settings

The contents of your Interactive Schedule are defined in the **Scheme Settings** dialog box.

To access this dialog box, do one of the following:

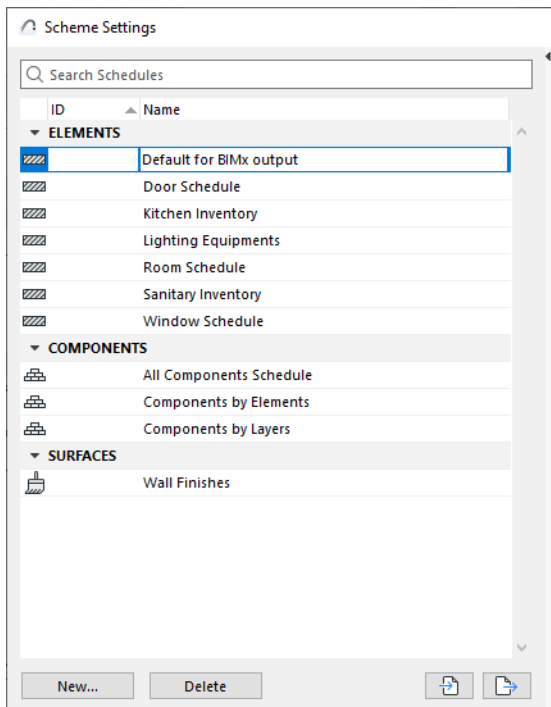
- Click **Scheme Settings** from the opened Schedule window



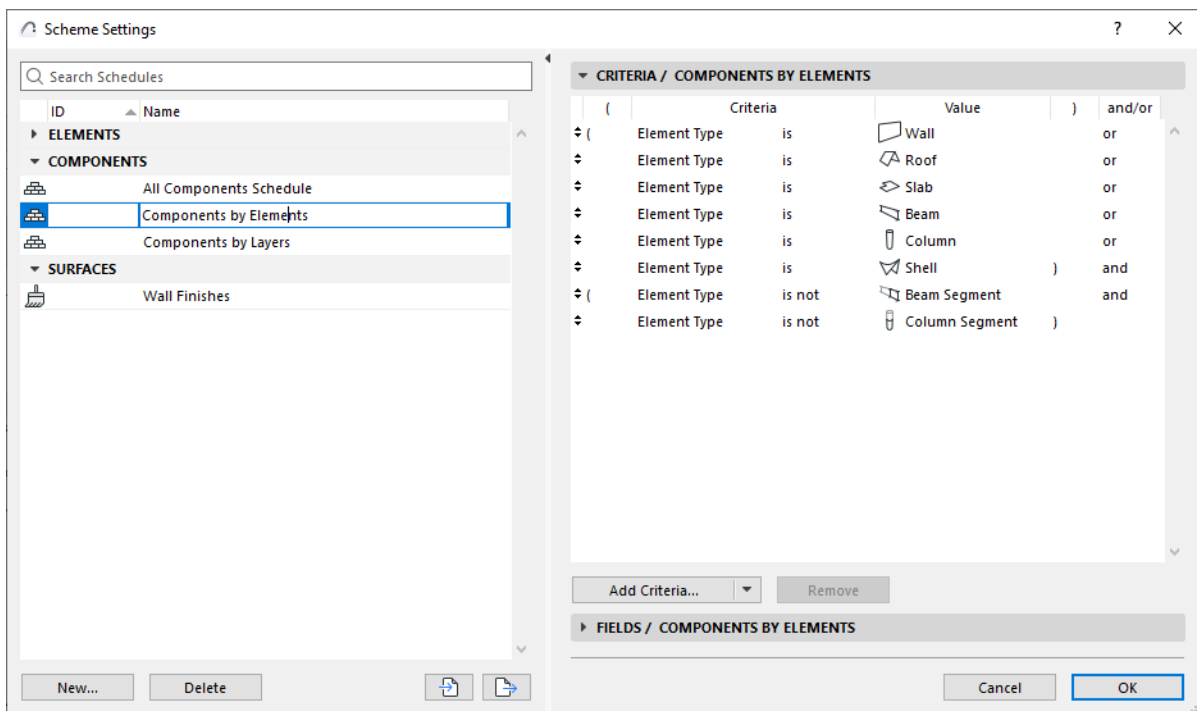
- Use the Schedule's context menu in the Navigator
- In the Navigator Palette, select a Schedule, then click the "Settings" button at the bottom of the Navigator.
- Use **Document > Schedules > Scheme Settings**

The left panel lists the project's schemes, grouped by type (e.g. Element Schemes, Component Schemes). Select the scheme you want to edit, or use the buttons at the bottom to create, duplicate, delete, import and export schemes.

For details, see [Schemes Panel](#).



The right side shows the Criteria and Fields of the selected Schedule.



Use the Criteria Panel of Scheme Settings to define the elements/components/surfaces to be listed in the schedule.

[See Criteria Panel.](#)

In the Fields Panel of Scheme Settings, choose which parameters of the chosen criteria should be displayed in your schedule.

[See Schemes Fields Panel.](#)

Surface Schedule

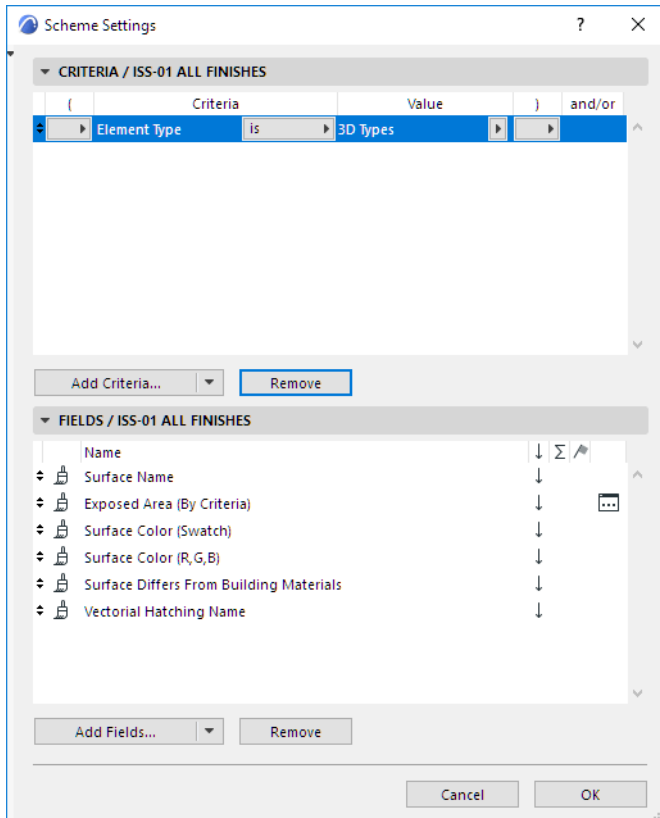
The Surface Schedule is the third type of interactive schedule, alongside Element Lists and Component Lists.

Note: Before using the Surface Schedule feature, make sure you are *not* working in “legacy” intersection mode. (Uncheck the “Use legacy intersection” box at **Options > Project Preferences > Legacy.**)

Surface Schedule Fields

In Scheme Settings, use the available fields as needed to adjust the content of your Surface lists.

For details on setting up a scheme, see [Scheme Settings Dialog Box \(Interactive Schedule\)](#).



Surface Differs from Building Materials

This field is displayed in the schedule as a checkbox (e.g. entitled “Painted”) to indicate it has been “painted over” with an override.

Note: This field is not evaluated for Openings, which do not have Building Materials.

Related Topics:

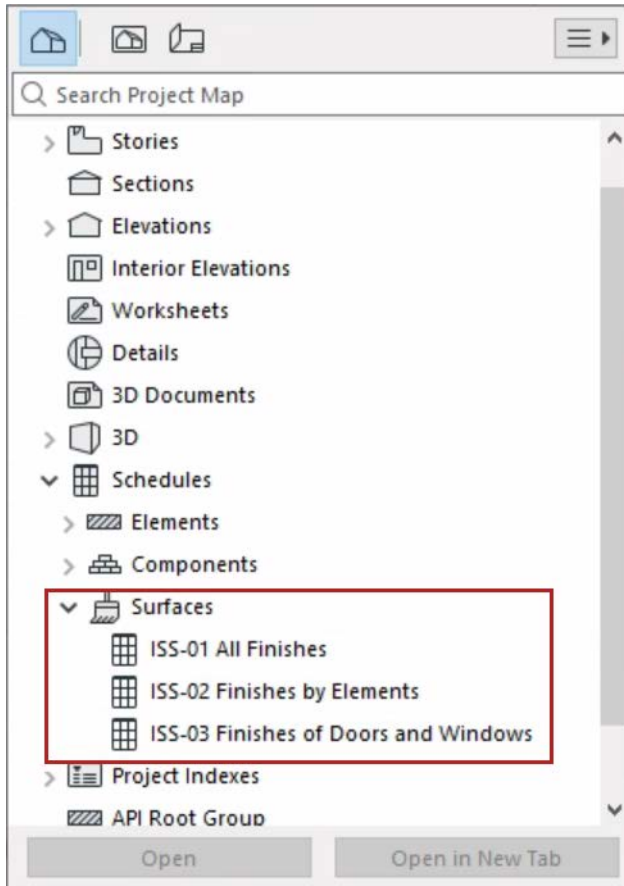
[Surface Schedule: Predefined schemes](#)

[Exposed Area parameter in Surface Schedules](#)

[Opening surface in Surface Schedule](#)

Surface Schedule: Predefined schemes



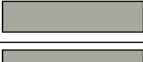







The predefined schemes for Surface Schedules are optimized to list finishes and their exposed areas in several formats.



In this template:

All Finishes

Lists every visible surface in the model with its total exposed area.

All Finishes					
Name	Exposed Area [m2]	Color Sample	Color [R,G,B]	Paint	Hatching
Brick - Red	98.50		225,102,57	<input type="checkbox"/>	Brick - Running Bond
Ceramic - Porcelain	38.05		247,253,255	<input checked="" type="checkbox"/>	Background
Concrete - 02	3.94		168,169,158	<input type="checkbox"/>	Solid + Dashed
Concrete - 04	594.87		168,169,158	<input type="checkbox"/>	Background
Concrete - 04	9.92		168,169,158	<input checked="" type="checkbox"/>	Background
Earth - Brown	10,570.31		148,104,75	<input type="checkbox"/>	Facing Tile - Large Scale
Floorboards - 03	25.16		237,217,184	<input checked="" type="checkbox"/>	Plank Floor
Foliage - Leaves Tree Small	3,680.39		11,79,26	<input checked="" type="checkbox"/>	Background
Generated from GDL	3,311.60		---	<input checked="" type="checkbox"/>	Generated from GDL
Glass - Blue	0.72		164,191,219	<input type="checkbox"/>	Background
Glass - Blue	203.65		164,191,219	<input checked="" type="checkbox"/>	Background

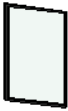

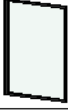



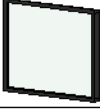





Finishes by Elements

Lists the visible surfaces used by each element type, with their respective exposed areas.

Finishes by Elements				
Type	Element ID	Finish	Exposed Area [m2]	Painted
Column	COL-E-001	Wood - Pine Grained Vertical	1.02	<input checked="" type="checkbox"/>
	COL-E-002	Wood - Pine Grained Vertical	1.11	<input checked="" type="checkbox"/>
	COL-E-003	Wood - Pine Grained Vertical	1.18	<input checked="" type="checkbox"/>
	COL-E-004	Wood - Pine Grained Vertical	1.21	<input checked="" type="checkbox"/>
	COL-E-005	Wood - Pine Grained Vertical	2.79	<input checked="" type="checkbox"/>
	COL-E-006	Wood - Pine Grained Vertical	2.69	<input checked="" type="checkbox"/>
	COL-E-007	Wood - Pine Grained Vertical	2.58	<input checked="" type="checkbox"/>
	COL-E-008	Wood - Pine Grained Vertical	2.27	<input checked="" type="checkbox"/>
	COL-E-009	Wood - Pine Grained Vertical	2.17	<input checked="" type="checkbox"/>
	COL-E-010	Wood - Pine Grained Vertical	2.09	<input checked="" type="checkbox"/>
	COL-E-011	Wood - Pine Grained Vertical	2.01	<input checked="" type="checkbox"/>
	COL-E-012	Wood - Pine Grained Vertical	1.92	<input checked="" type="checkbox"/>

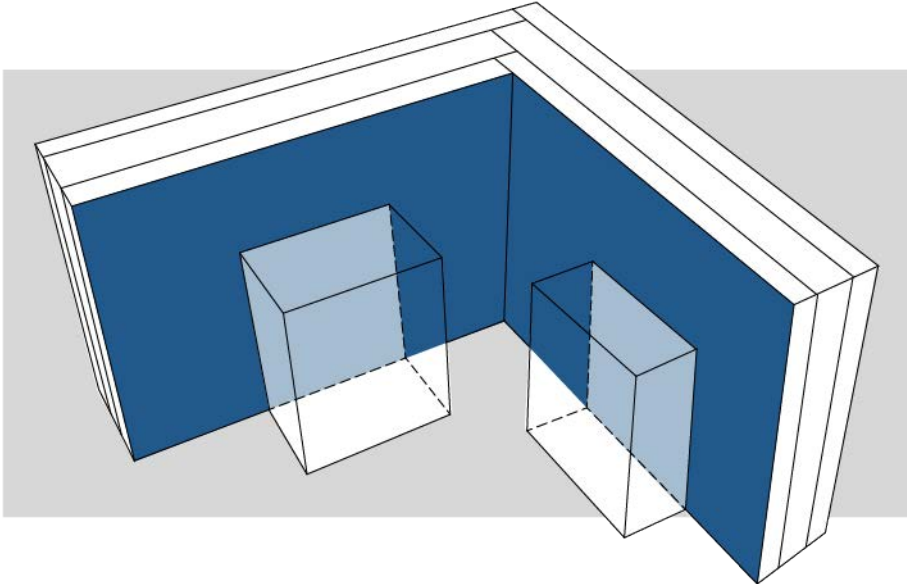
Finishes of Doors and Windows

Lists the surfaces (excluding glass) of Doors and Windows. That is, the schedule lists the surfaces used for frame, casing, and accessories such as shutters, board, and sill).

Finishes of Doors and Windows						
Type	Element ID	3D Front Axonometry	Surface Name	Surface Color [Sample]	Surface Color [R,G,B]	Casing Total Area [m2]
Corner-Window						
	WDW-001		Paint - Anthracite		46,46,46	7.44
	WDW-001		Stucco - White Rough		255,255,255	0.49
	WDW-002		Paint - Anthracite		46,46,46	1.83
	WDW-003		Paint - Anthracite		46,46,46	1.74
Door						
	DOR-001		Metal - Iron		75,78,84	0.00
	DOR-001		Metal - Stainless Steel		215,211,217	0.07

Exposed Area parameter in Surface Schedules

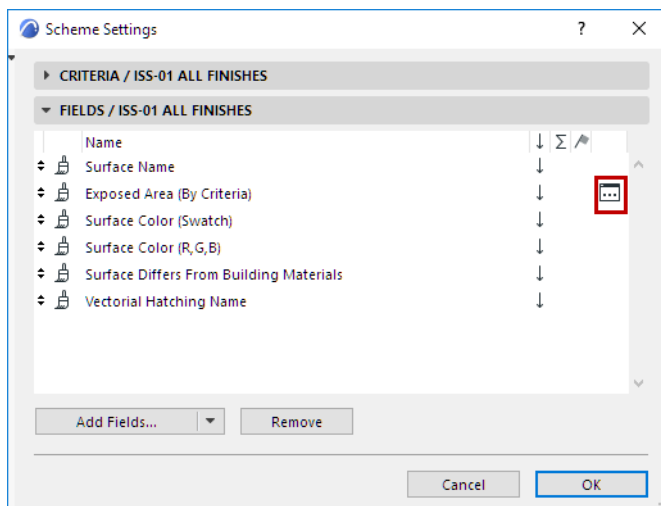
Exposed Area is the part of an element's surface that would be covered by a paint bomb. In the follow diagram, the dark blue area is considered as "Exposed", the light blue areas are considered as "Covered".



You can define exactly what counts as exposed, using the **Reduced Exposed Area** control in the Scheme Settings of your Surface List.

Reduce Exposed Area

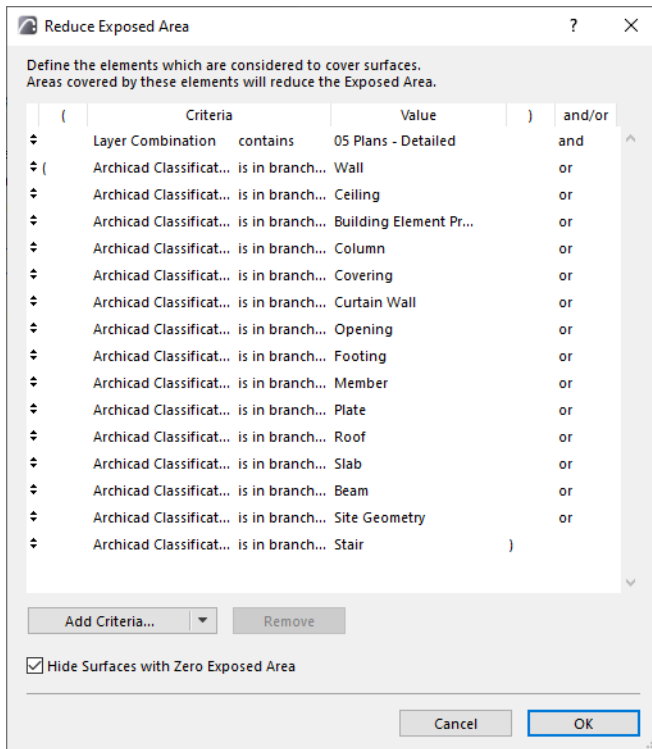
1. In Scheme Settings, Fields panel: find the **Exposed Area (By Criteria)** field.



2. Click the button to open the **Reduce Exposed Area** dialog box.
3. Define which elements are covering elements (that is, they should cover up other element surfaces).

For example: normal construction elements are covering elements: if they overlap a wall's surface, that wall's surface is covered, and the wall's exposed area is reduced accordingly.

However, some elements are usually not covering elements. In this example: objects, lamps and zones should *not* affect a wall's exposed area, so they are *not* listed here:

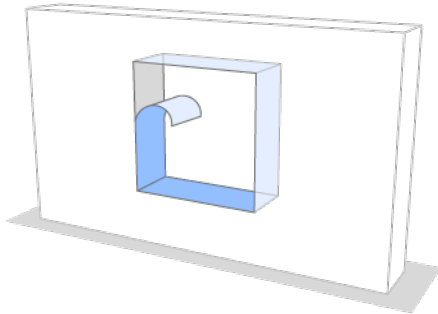


Hide Surfaces with Zero Exposed Area: This is checked by default. It means that unexposed areas, with zero visible area, are not shown in the schedule.

Opening surface in Surface Schedule

Suppose you have an Opening in a wall.

When you assign an override to the Opening surface, it is like a thin sheet of paint that has two sides to it (like a strip of wallpaper).



Thus, when listing surfaces, keep in mind that there are a total of three potential surfaces here:

- one wall surface
- plus the two sides of the opening surface override (“wallpaper”)

Depending on what you want to list, make sure the Surface schedule only counts the surfaces you need.

Surfaces that are blocked by a "covering" element will not be counted in the schedule.

To set elements as "covering", use the Reduce Exposed Area dialog of Surface Schedule Scheme Settings. (see [Exposed Area parameter in Surface Schedules](#)).

See examples below.

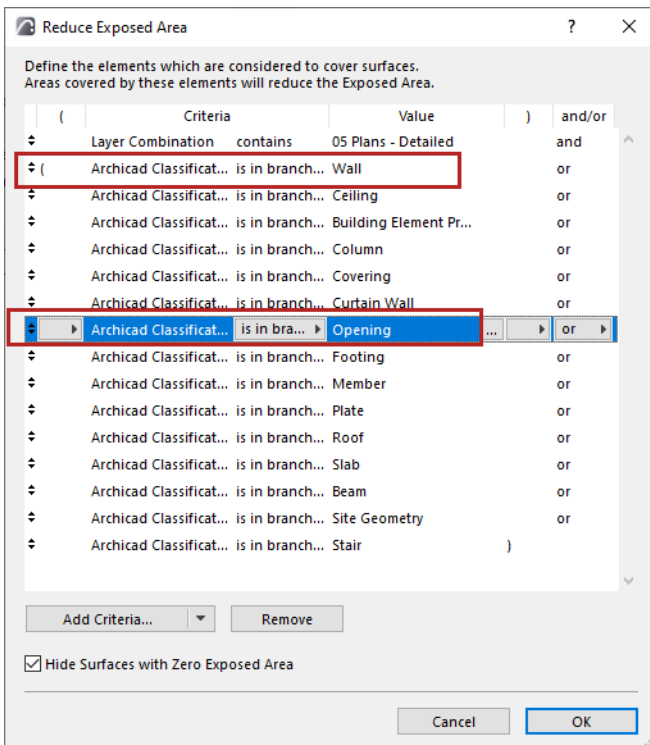
Example: List opening's painted surface

An Opening cuts through a wall, and you have painted the Opening edges with an override.

The painted Opening surface has two sides to it (like a strip of wallpaper).

You want the schedule to list the Opening surface override, but only its visible side. To get this result:

- **Add walls to the “Reduce Exposed Area” list** (to exclude the non-visible side of the "wallpaper")
- **Add openings to the “Reduce Exposed Area” list** (to exclude the original, unpainted face of the wall)



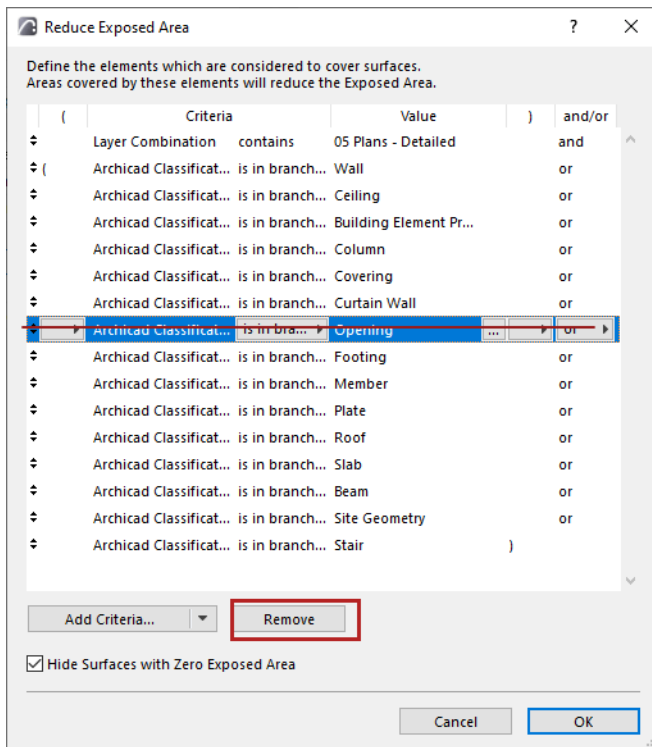
Example: List wall surface only; the opening is just a hole

The Opening is just a hole cut into the host structure and has NO surface override.

Note: If the Opening has no override, you cannot list its surface. But such an Opening is still capable of covering the host element.

You want to count the host element surface only, ignoring the Opening. Make sure the Openings are not set as covering the host element surfaces. To get this result:

- **Remove Openings from the “Reduce Exposed Area” list** (to reveal the wall’s opening face)



Remove Openings from "Reduce Exposed Area" list

Format Options (Interactive Schedule)

Use the formatting controls at the left side of the Schedule Window to format the current schedule.

The screenshot shows the 'Schedule / AC27 | GFA by Zone 02' window. On the left, a red box highlights the 'Format Options' panel, which includes:

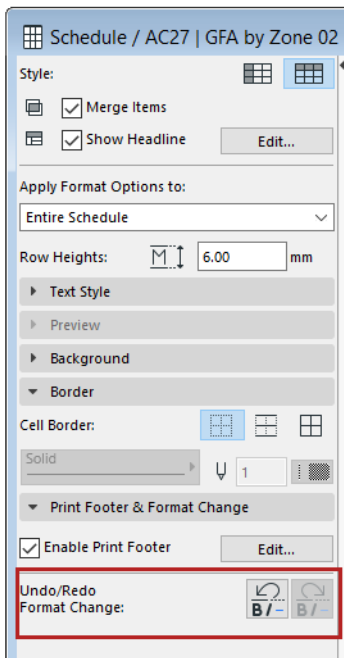
- Style:** Merge Items (checked), Show Headline (checked), Edit...
- Apply Format Options to:** Headline
- Row Heights:** 6,00 mm
- Text Style:** [Helvetica Neue], Western
- Text Formatting:** Bold (B), Italic (I), Underline (U), Strikethrough (ABC), Bullets (A), Numbering (1), Indent (0), Outdent (0), Increase (100%), Decrease (100%), Paragraph (100%)
- Wrap text:** (checked)
- Background:** Cells (checked), Stripes (unchecked)
- Apply colors to:** Display Only (unchecked), Display and Output (checked)
- Border:** (collapsed)
- Print Footer & Format Change:** (collapsed)

The main table displays the 'GROSS FLOOR AREA' with the following data:

Zone Name	Area Reducement	Reduced Area	Calculated Area
01. COMMON			
AMENITIES	24,00	26,04	634,20
LIFTS + STAIRS	24,00	71,40	1713,60
LOBBY	1,00	282,24	282,24
PLANT	50,00	8,40	2 433,00
			5 063,04 m ²
02. COMMERCIAL			
COMMERCIAL	10,00	1177,92	11 779,20
			11 779,20 m ²
03. RESIDENTIAL			
RESIDENTIAL	12,00	956,16	11 473,92
UNIT CORNER	2,00	151,09	302,18
UNIT MIDDLE	1,00	95,37	95,37
			11 871,47 m ²
04. RETAIL			
RETAIL 1	1,00	138,24	138,24
RETAIL 2	1,00	155,52	155,52
RETAIL 3	1,00	129,60	129,60
RETAIL 4	1,00	129,60	129,60
			552,96 m ²
			29 266,67 m ²

Important:

Use the Undo/Redo Format Change to revoke or redo format changes in the Schedule.



Topics in this Section

[Resize Rows/Columns](#)

[Hide Rows/Columns](#)

[Adjust Zoom/Fit to Current Window](#)

[Display by Rows or Columns](#)

[Merge Schedule Items](#)

[Apply Format Options to:](#)

[Show Headline](#)

[Text Style](#)

[Cell Background Colors](#)

[Cell Borders](#)

[Add Printable Footer](#)

[Use the Undo/Redo Format Change to revoke or redo format changes in the Schedule.](#)

[Scheduled Elements on Locked or Hidden Layer](#)

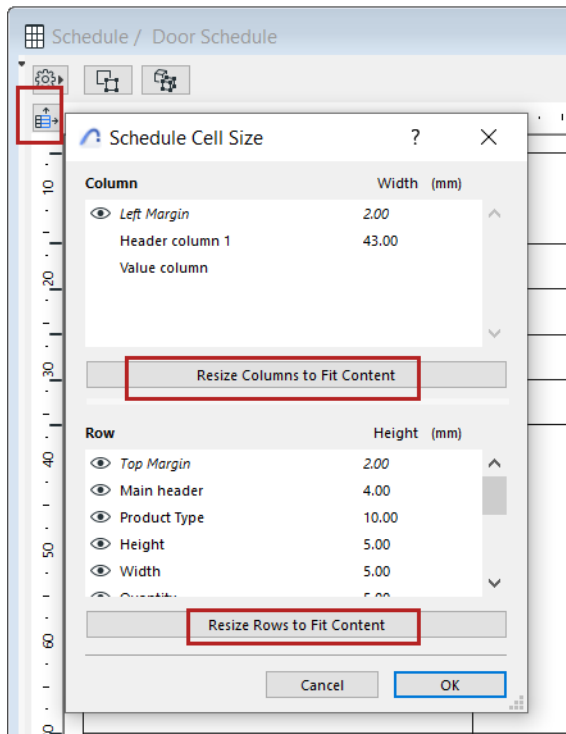
Related Topics

[Select Schedule Items on Floor Plan/3D](#)

Resize Rows/Columns

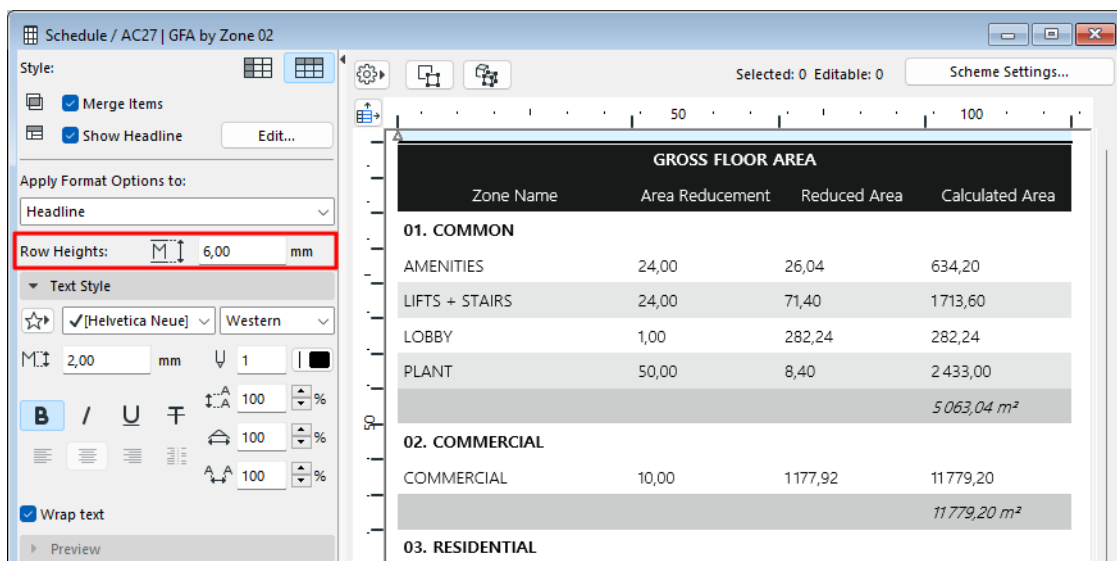
Do one of the following:

- To adjust **Schedule Cell Size**, click the icon at the top of the schedule and use the dialog box.



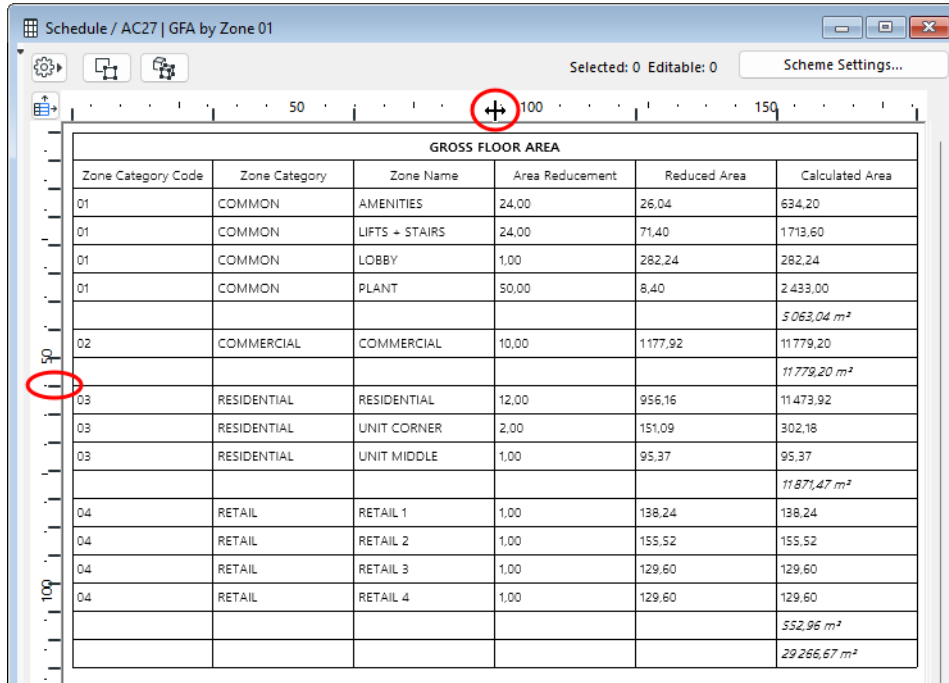
Use the **Resize** buttons to automatically adjust rows and columns to fit the content, or enter width/height values as needed (multi-select available for the listed items).

- Set the **Row Heights** field



- To adjust individual row or column heights, manually drag the cell boundaries on the top and left side of the schedule.

- Double-click the row or column edge to fit its contents.

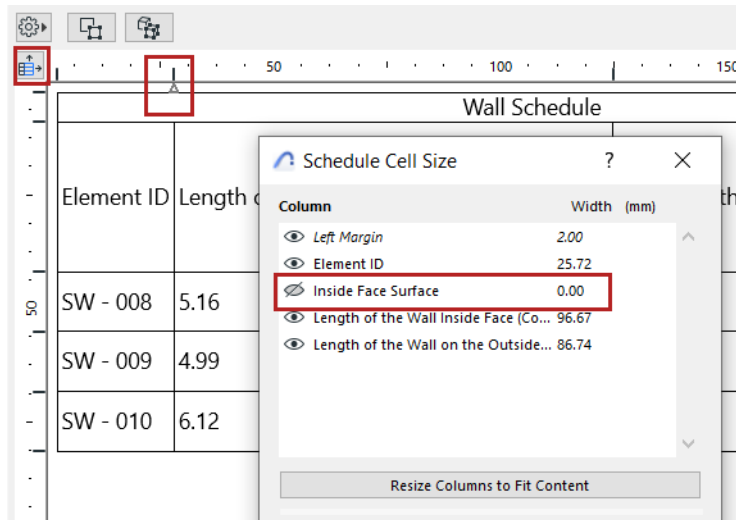


Double-Click: Fit Width to Cell Contents

Hide Rows/Columns

In Schedule Cell Size dialog, do one of the following:

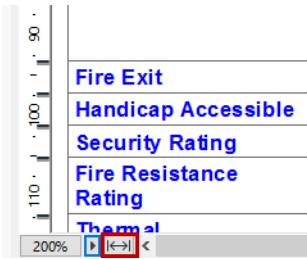
- Use the eye icon to show or hide the Row or Column
- Set Row height or Column width to zero



An arrow appears in the schedule to mark a hidden column or row. Double-click to unhide.

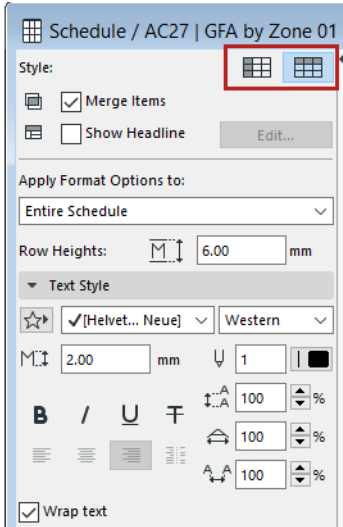
Adjust Zoom/Fit to Current Window

- Adjust the Zoom using the pop-up from the bottom scrollbar
- To fit the schedule to the current window size, click the Fit to Width icon



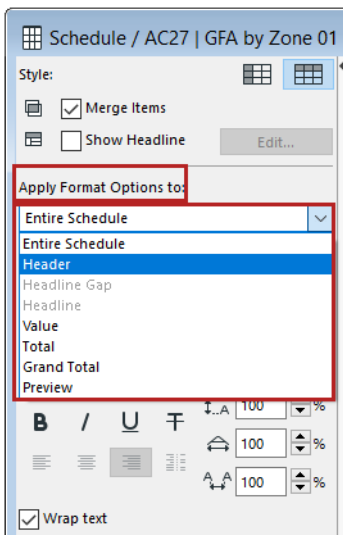
Display by Rows or Columns

From the two Style buttons, choose to display the Schedule fields by either rows or by columns.



Apply Format Options to:

Choose the items that you will format using the controls below.

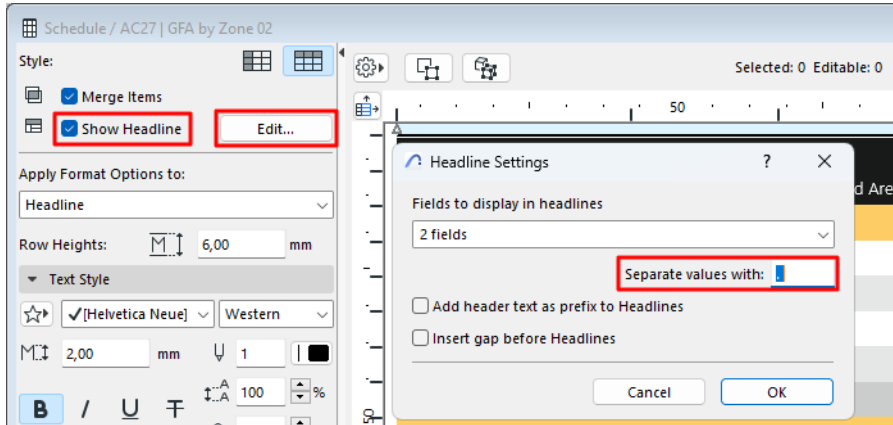


The Preview refers to the element Preview pictures, if any, in the Schedule. [\(See Format Preview Picture.\)](#)

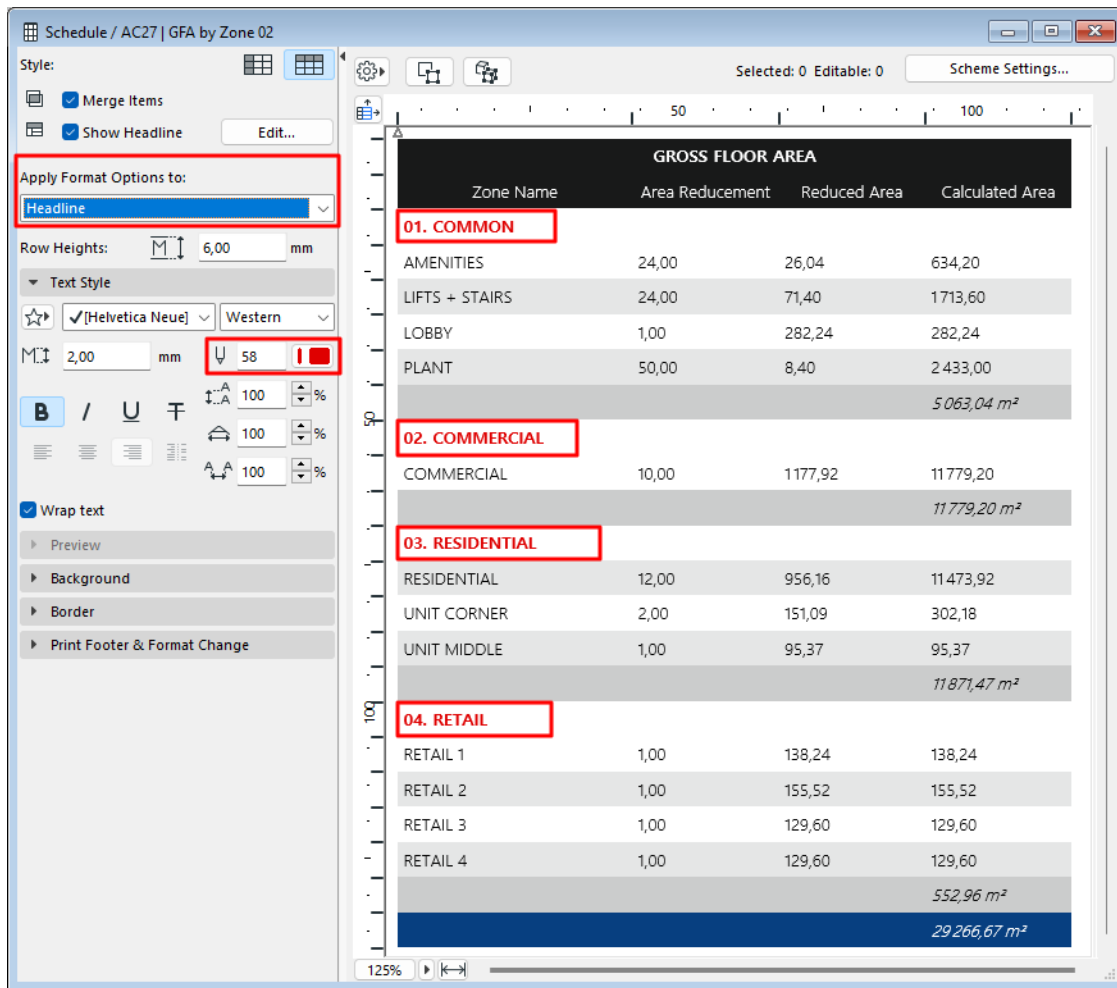
Show Headline

This option lets you display data from one or more schedule fields in a separate row, above (or to the left) of the rest of the element data.

Click **Edit** to open Headline Settings, to define the content and appearance of the headline data:



- **Fields to display:** Choose how many schedule fields to display in the headline.
- **Separator:** Choose any character to separate the headline fields (such as comma, period, vertical line). Add a space if needed.
- **Add header text as prefix to headlines:** Adds the Column/Row header name as a prefix to its value in the headline row.
- **Insert gap before headlines:** Inserts an empty row before each headline row.
- Customize the text **format** of Headline data, as needed: at Apply Format Options to, choose the Headline item.



See also [Schedule Headers](#), below.

Text Style

Set text format options for the item type selected at **Apply Format Options to**:

Note: Not available if you have selected “Preview”.

Text Favorite: Click the pop-up to apply Favorites saved with the Text Tool.

See also [Apply a Favorite](#).

Leading: This control modifies the leading value (vertical line spacing), as a percentage. Enter a number or use the up-and-down arrows to change the value in increments of 25 percent.

Wrap text: Check this option to automatically wrap text so that it fits into the column width.

Cell Background Colors

1. From “Apply Format Options,” choose the item (such as Headline or Grand Total) whose cell color you want to set. Or click in a cell.
2. Click **Cells**.
3. Choose a background color.

Screenshot of Archicad software showing a 'GROSS FLOOR AREA' table. The sidebar on the left contains formatting options. The 'Background' section is highlighted with a red box, showing 'Cells' checked and 'Stripes' unchecked. The table lists zones like COMMON, COMMERCIAL, RESIDENTIAL, and RETAIL with columns for Area Reducement, Reduced Area, and Calculated Area.

Zone Name	Area Reducement	Reduced Area	Calculated Area
01. COMMON			
AMENITIES	24,00	26,04	634,20
LIFTS + STAIRS	24,00	71,40	1713,60
LOBBY	1,00	282,24	282,24
PLANT	50,00	8,40	2 433,00
			5 063,04 m ²
02. COMMERCIAL			
COMMERCIAL	10,00	1177,92	11 779,20
			11 779,20 m ²
03. RESIDENTIAL			
RESIDENTIAL	12,00	956,16	11 473,92
UNIT CORNER	2,00	151,09	302,18
UNIT MIDDLE	1,00	95,37	95,37
			11 871,47 m ²
04. RETAIL			
RETAIL 1	1,00	138,24	138,24
RETAIL 2	1,00	155,52	155,52
RETAIL 3	1,00	129,60	129,60
RETAIL 4	1,00	129,60	129,60
			552,96 m ²
			29 266,67 m ²

To apply stripes to alternating rows of values:

1. From "Apply Format Options," choose Values. Or click in a row.
2. Click Stripes.
3. Choose a stripe color.

Cell Borders

Set cell border options for the item type selected at **Apply Format Options to**.

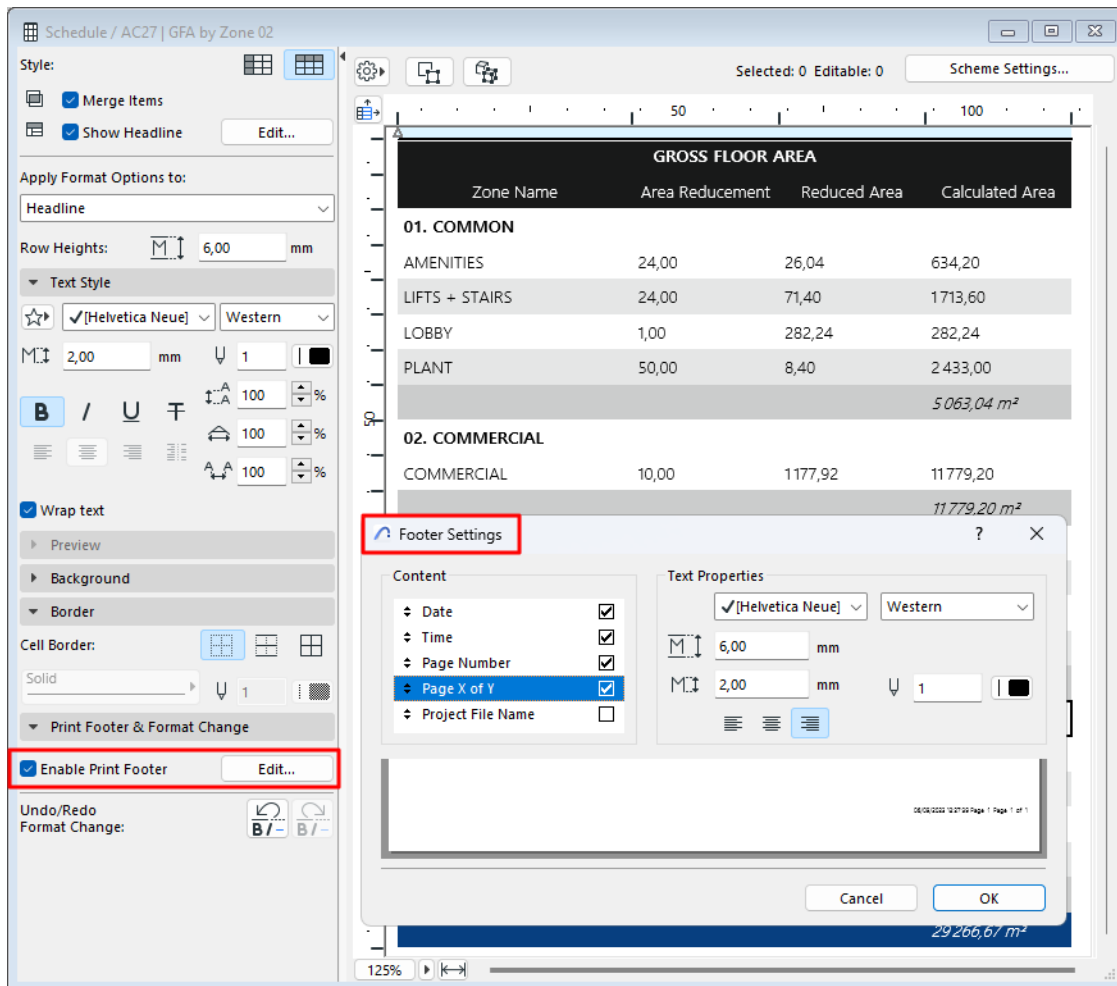
The screenshot shows the Archicad Schedule window for 'AC27 | GFA by Zone 02'. The main table displays the Gross Floor Area breakdown by zone. The 'Border' section in the left sidebar is highlighted with a red box, indicating that the 'Cell Border' is set to 'Solid'.

GROSS FLOOR AREA			
Zone Name	Area Reducement	Reduced Area	Calculated Area
01. COMMON			
AMENITIES	24,00	26,04	634,20
LIFTS + STAIRS	24,00	71,40	1713,60
LOBBY	1,00	282,24	282,24
PLANT	50,00	8,40	2 433,00
			5 063,04 m ²
02. COMMERCIAL			
COMMERCIAL	10,00	1177,92	11779,20
			11 779,20 m ²
03. RESIDENTIAL			
RESIDENTIAL	12,00	956,16	11 473,92
UNIT CORNER	2,00	151,09	302,18
UNIT MIDDLE	1,00	95,37	95,37
			11 871,47 m ²
04. RETAIL			
RETAIL 1	1,00	138,24	138,24
RETAIL 2	1,00	155,52	155,52
RETAIL 3	1,00	129,60	129,60
RETAIL 4	1,00	129,60	129,60
			552,96 m ²
			29 266,67 m ²

See [Previews and Pictures in the Interactive Schedule](#).

Add Printable Footer

1. Check **Enable Print Footer** to add a print-only footer to the Schedule.
2. Click **Edit** to open Footer Settings and define the Footer.



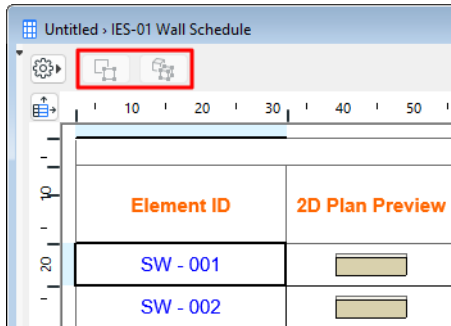
- **Content:** Check the items that should appear in the print footer. Drag them up and down to modify the order.
- Set text format options for the printable footer. Use the preview at the bottom as a guide.

3. Click **OK**.

Scheduled Elements on Locked or Hidden Layer

You cannot edit or select scheduled elements that are on a locked or hidden layer.

This information is displayed for selected elements in the Information panel:



See also [Select Schedule Items on Floor Plan/3D](#).

Merge Schedule Items

You may wish to merge certain records in your schedule. Apart from the default merge behavior, you can customize how each schedule field and their values should be merged.

Default Merge

Customize Display of Merged Fields

Merge uniform values - Display the uniform value

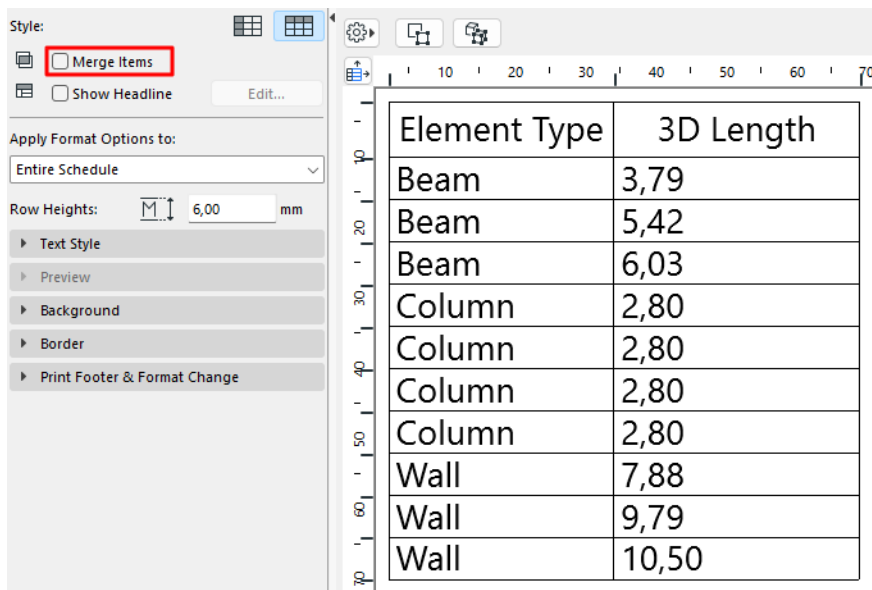
Merge uniform values - Display total sum

Merge all values (uniform or various) - Display total sum

Merge all values (uniform or various) - List each value

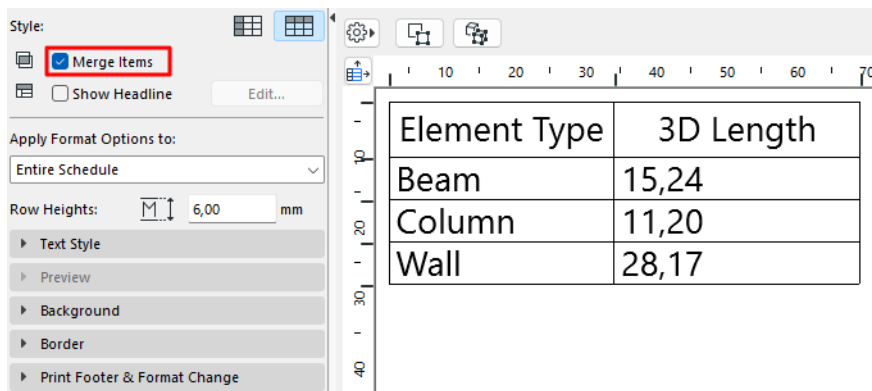
Default Merge

Suppose you have the following schedule. To merge schedule items (using a default logic), check the **Merge items** box in Schedule Settings.



Element Type	3D Length
Beam	3,79
Beam	5,42
Beam	6,03
Column	2,80
Column	2,80
Column	2,80
Column	2,80
Wall	7,88
Wall	9,79
Wall	10,50

The default result looks like this. Listed element types are combined in a single record, with all their length values (whether uniform or not) shown as a sum total.

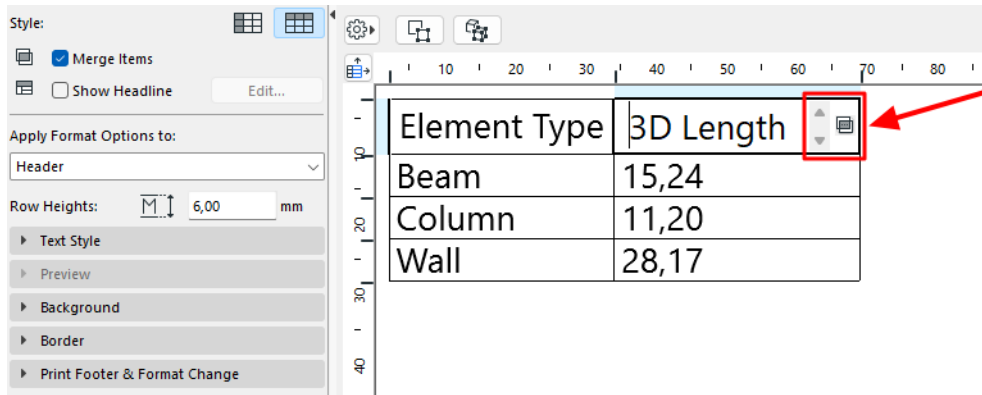


Element Type	3D Length
Beam	15,24
Column	11,20
Wall	28,17

See below for other ways to merge and display schedule items.

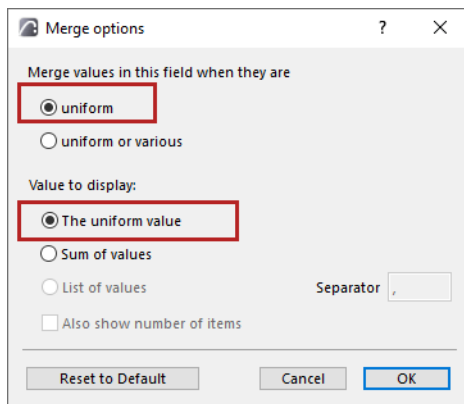
Customize Display of Merged Fields

1. Check **Merge Items**.
2. In the schedule, click the field whose values you want to merge (for example, “3D Length”), then click the “Merge Items” icon.



3. The Merge Options dialog opens. Define which values to merge and how to display them. Each scenario is shown below.

Merge uniform values - Display the uniform value



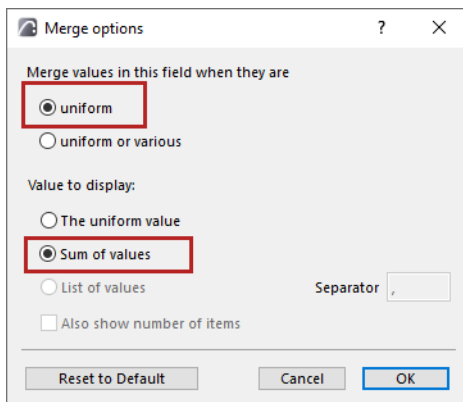
1. Choose the **uniform** option.
2. Choose **uniform value**:

Result: the Column items are merged into a single item. (Only the Columns have a uniform value; the other elements have various length values). The field displays the Column length value, which is the same for each Column.

Element Type	3D Length
Beam	3,79
Beam	5,42
Beam	6,03
Column	2,80
Wall	7,88
Wall	9,79
Wall	10,50

Merge uniform values - Display total sum

1. Choose the **uniform** option.
2. Choose **Sum of values**.



Result: the Column items are merged into a single item. The displayed value here is the sum of all the uniform Column lengths.

3D length	
Element Type	3D
Beam	2.37
Beam	2.52
Beam	3.10
Beam	3.34
Beam	4.45
Beam	8.98
Column	14.00
Wall	1.99
Wall	4.09
Wall	4.85
Wall	4.92
Wall	4.95
Wall	6.32
Wall	7.90
Wall	7.95
Wall	19.82

Merge all values (uniform or various) - Display total sum

Here, all lengths for each element type (even if the length values vary) should be displayed in a single field.

1. Choose the **uniform or various** option.
2. Choose **Sum of values**.

Merge options

Merge values in this field when they are

uniform

uniform or various

Value to display:

The uniform value

Sum of values

List of values Separator ,

Also show number of items

Reset to Default Cancel OK

Result: all the Beam, Column and Wall items are merged into one item each, and the displayed value is the sum of their lengths. (This is the default behavior for this example.)

3D length	
Element Type	3D
Beam	24.76
Column	14.00
Wall	62.79

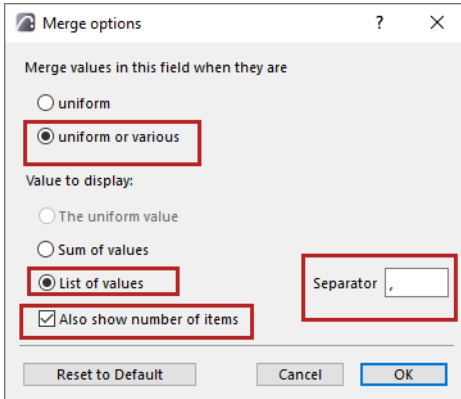
Merge all values (uniform or various) - List each value

All lengths for each element type (even if the length values vary) should be displayed in a single record. However, the field should list each varying length value separately.

1. Choose the **uniform or various** option.
2. Choose **List of values**:

Customize how to display the value list:

- Enter a separator character: e.g. comma, vertical line.
- Check the box to also display the number of each listed item



Result: all the Beam, Column and Wall items are merged into a single record, but with each length value shown in a list.

3D length	
Element Type	3D Length
Beam	2.37 (1),2.52 (1),3.10 (1),3.34 (1),4.45 (1),8.98 (1)
Column	2.80 (5)
Wall	1.99 (1),4.09 (1),4.85 (1),4.92 (1),4.95 (1),6.32 (1),7.90 (1),7.95 (1),9.91 (2)

Schedule Headers

The **Main Header** field is generated automatically at the top of the Schedule: its content is the name given to the Schedule in Scheme Settings.

Wall Schedule								
Full Element ID	Wall Type	Volume [m3]	Thickness [m]	Height [m]	Area [m2]	Perimeter [m]	Construction Type	Fire Resistance Rating
SW - 001	Exterior Wall	9.15	0.470	7.300	4.90	22.19	Structural frame	25 minutes
SW - 002	Exterior Wall	11.16	0.470	7.300	3.02	14.19	Structural frame	25 minutes
SW - 003	Exterior Wall	0.97	0.470	3.100	0.97	5.05	Structural frame	25 minutes
SW - 003	Exterior Wall	1.77	0.470	3.100	0.60	3.53	Structural frame	25 minutes
SW - 003	Exterior Wall	22.80	0.470	7.300	4.90	22.19	Structural frame	25 minutes
SW - 003	Reinforced Conc...	3.14	0.600	0.900	3.90	14.70	Structural frame	25 minutes

Column Headers are also generated automatically, at the top (or the left) of the columns (or rows) of data. Their content comes from the respective field names.

To change a header text, just click into the field and type.

Wall Schedule								
Full Element ID	Wall Type	Volume [m3]	Thickness [m]	Height [m]	Area [m2]	Perimeter [m]	Construction Type	Fire Resistance Rating
SW - 001	Exterior Wall	9.15	0.470	7.300	4.90	22.19	Structural frame	25 minutes
SW - 002	Exterior Wall	11.16	0.470	7.300	3.02	14.19	Structural frame	25 minutes
SW - 003	Exterior Wall	0.97	0.470	3.100	0.97	5.05	Structural frame	25 minutes
SW - 003	Exterior Wall	1.77	0.470	3.100	0.60	3.53	Structural frame	25 minutes
SW - 003	Exterior Wall	22.80	0.470	7.300	4.90	22.19	Structural frame	25 minutes
SW - 003	Reinforced Conc...	3.14	0.600	0.900	3.90	14.70	Structural frame	25 minutes

Topics in this Section:

[Format Headers](#)

[Show/Hide Headers](#)

[Insert Header Cell](#)

[Freeze Schedule Header](#)

Format Headers

To format header text, make sure that "Header" is selected from the Apply To pop-up, then choose text formatting options.

The screenshot shows the Archicad interface with a schedule table and a formatting panel. The table has the following data:

Element ID	2D Plan Preview	Wall Type	Height [m]	Thickness [m]	Perimeter [m]	Area [m2]	Net Volume [m3]
SW - 001		Generic Wall/Shell	3,000	0,30	20,18	2,94	5,41
SW - 002		Generic Wall/Shell	3,000	0,30	21,61	3,15	9,45
SW - 003		Generic Wall/Shell	3,000	0,30	16,36	2,36	7,09
							21,95 m³

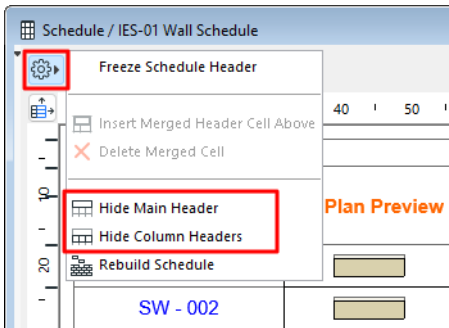
The formatting panel on the left shows the following settings:

- Apply Format Options to: Header
- Row Heights: 4,00 mm
- Text Style: Arial, Western
- Font Size: 2,00 mm
- Line Spacing: 23
- Wrap text: checked

To select multiple cells, use Shift-click.

Show/Hide Headers

From the **Settings** pop-up, click Show/Hide Main Header or Show/Hide Column Header.

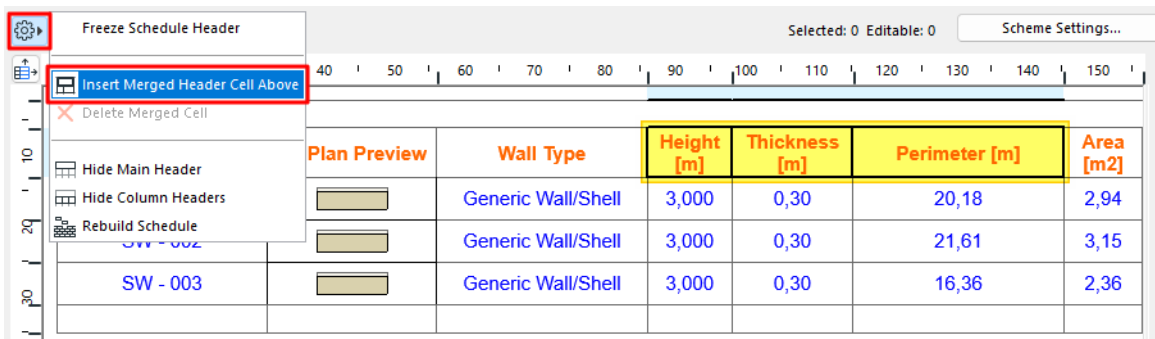


Insert Header Cell

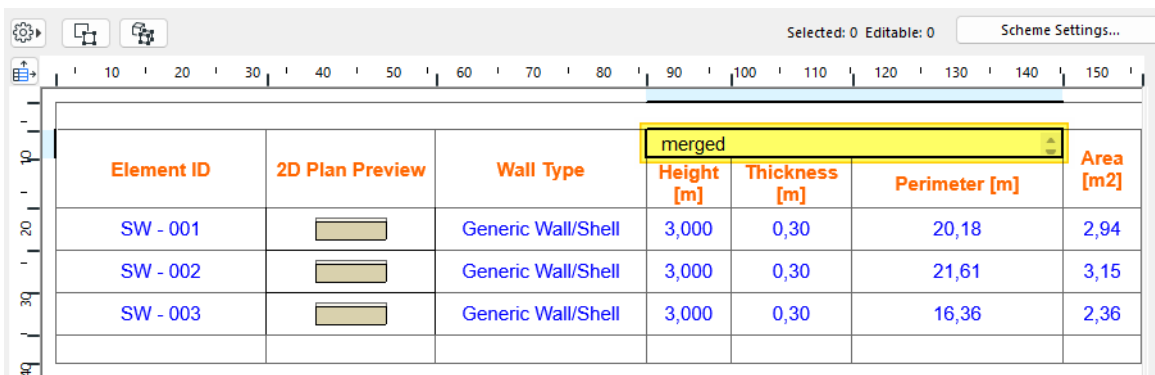
Use this command to create a new header cell above any set of selected column headers. For example, select three column headers that refer to Wall dimensions, and create a new header cell above these three columns, entitled “Dimensions”.

1. Select two or more column headers (use **Shift-click to select multiple cells**).
2. Choose **Insert Merged Header Cell Above** from Header Options.

Note: Not available unless you have selected at least two adjoining header cells!



3. A new cell (named “merged” by default) appears above the selected headers.



4. Type in any header text and format as usual. Any number of header cells can be created in a hierarchical order.

Wall Schedule						
Full Element ID	Wall Type	Volume [m3]	Thickness [m]	Dimensions		
				Height [m]	Area [m2]	Perimeter [m]
SW - 001	Exterior ...	9.17	0.470	7.300	4.90	22.19
SW - 002	Exterior ...	11.14	0.470	7.300	3.02	14.19
SW - 003	Exterior ...	22.77	0.470	7.300	4.90	22.19
SW - 003	Reinforce...	3.31	0.600	0.900	3.90	14.70
SW - 003	Stud Parti...	0.74	0.100	2.590	0.35	7.44
SW - 004	Exterior ...	10.66	0.470	7.300	3.02	14.19
SW - 004	Reinforce...	4.87	0.600	0.900	6.30	22.70

- **Delete Merged Cell:** This is only active if you have manually created a new merged header cell, and have selected that cell for deletion. Not available in any other cell.

Freeze Schedule Header

This command ensures that the main header and/or row/column headers rows remain visible on top when you scroll down the schedule.

Previews and Pictures in the Interactive Schedule

Interactive Schedules can include 2D or 3D previews pictures:

Element ID	2D Symbol	3D Axonometry	GDL Preview Picture	Quantity
DOO - 001				1
DOO - 002				1
DOO - 003				1

Topics in this Section:

[Annotate Preview](#)

[Automatic Dimensioning of Schedule Previews \(Doors/Windows\)](#)





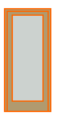
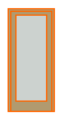


[Format Preview Picture](#)

Annotate Preview

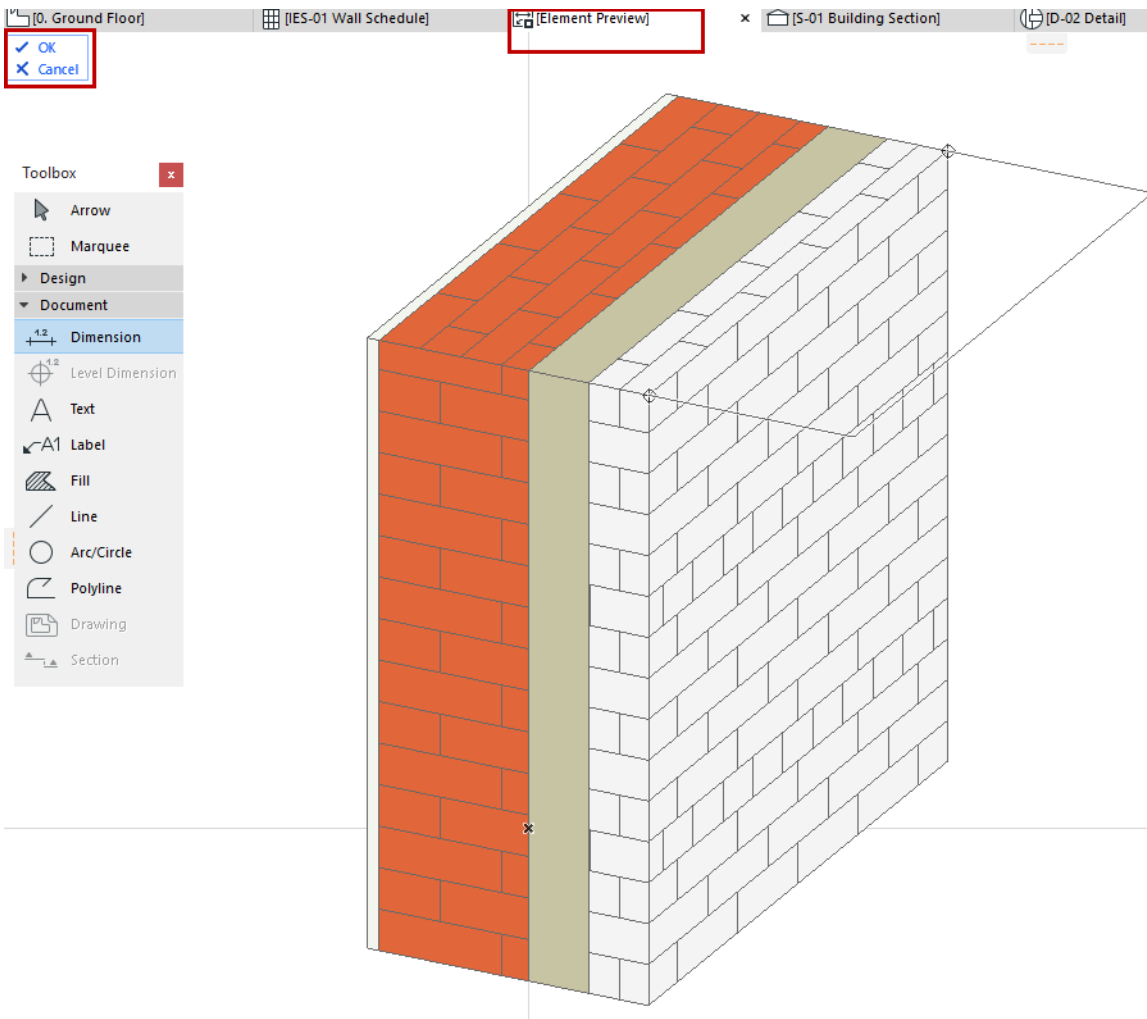
You can add associative dimensions to these previews in the Schedule.

1. Double-click the cell you wish to annotate, or

Select the cell, then click on **Annotate**.

Door List			
D1 12	D1 12	D1 1 Sidelight 12	D1 Double Acting 12
1	1	2	1
0.900x2.100	0.900x2.100	1.200x2.100	0.900x2.100
L	R	R	L
0.000	0.000	0	0.000
2.100	2.100	2.100	2.100
Annotate			
			
			

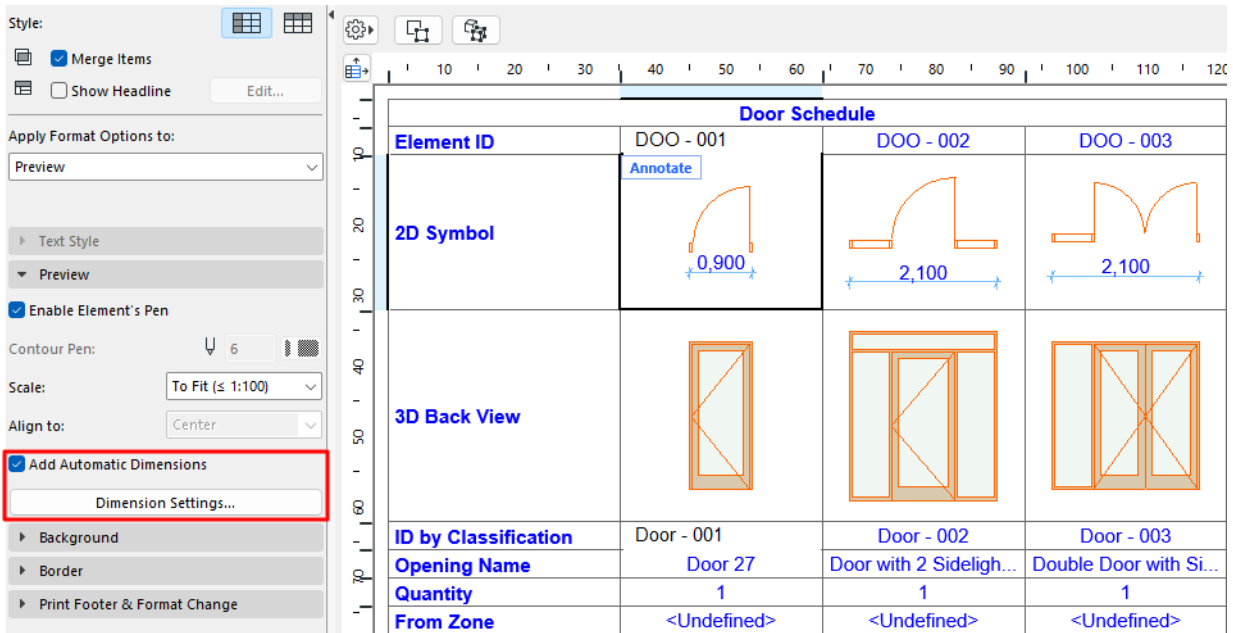
- The view opens in a separate tab/window entitled **Element Preview**.
- Use the available 2D tools to annotate it: dimensions, labels, text. You can also add annotations from available libraries, using the Object tool.



- Click **OK** from the menu at the top left to close this tab/window and apply the annotations to the selected view; or click **Cancel** to close without applying any changes.

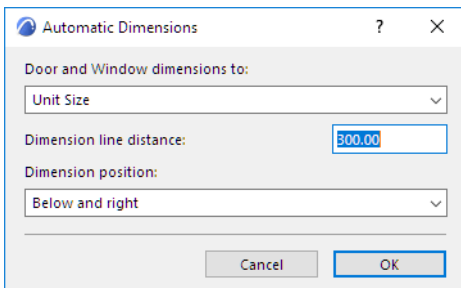
Automatic Dimensioning of Schedule Previews (Doors/Windows)

For Schedules showing Door/Window previews, the Schedule Options include the **Add automatic dimensions** option.



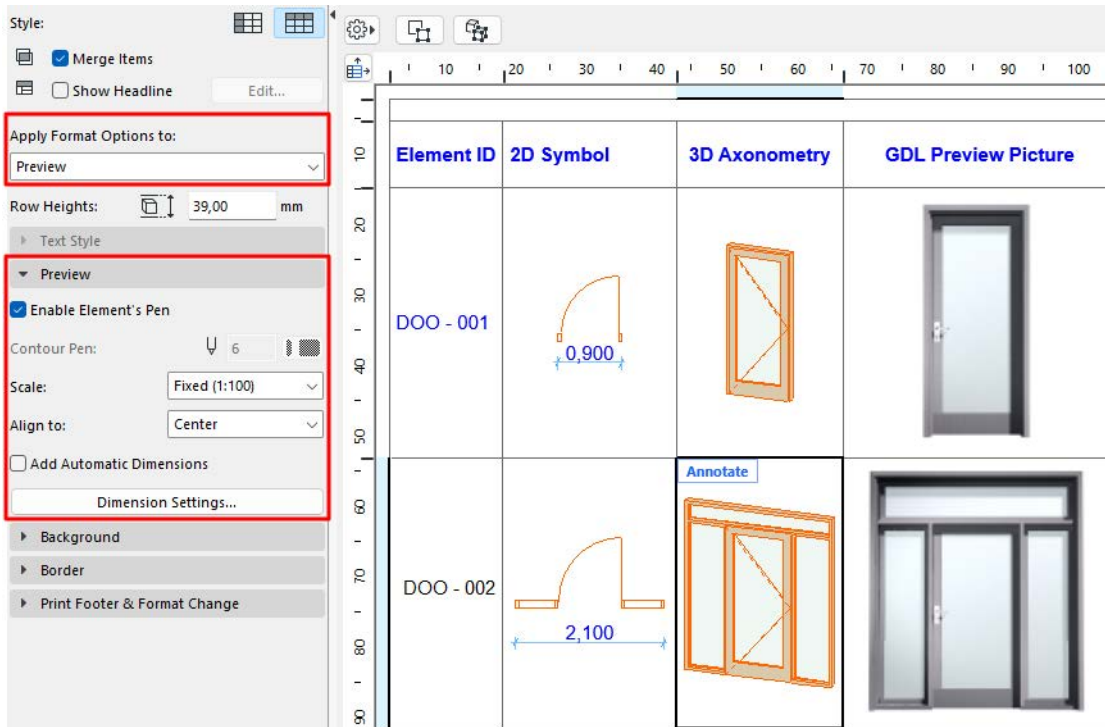
Check this box to add automatic dimensions to the selected picture.

Then click **Dimension Settings** to set dimensioning options for the picture, in the appearing **Automatic Dimensions** dialog box.



Format Preview Picture

This panel is available if **Apply Format Options to** is set to **Preview**, or if a preview cell is selected in the Schedule.



Enable Element's Pen: Check to use the pencolors defined for the element for its preview picture in the schedule.

- If it is unchecked, choose a pencolor in the **Contour Pen** field below.

Scale: Set the optimal scale for the schedule's Preview pictures. **To fit** ensures that the picture will fit in the cell, by scaling the preview down if needed (will not scale up).

Align to: Align the preview pictures in this schedule within their cells: Center, Top or Bottom of the cell, or (for 2D previews only) to correspond to their relative elevations from the chosen reference level (Story Zero, Project Zero, or other defined reference level.)

Note: Not available when Scale is "To Fit".

Add Automatic Dimensions: Add dimension information Previews in this schedule. Dimensions are associative; if the element is modified in the project, the dimensions will be updated.

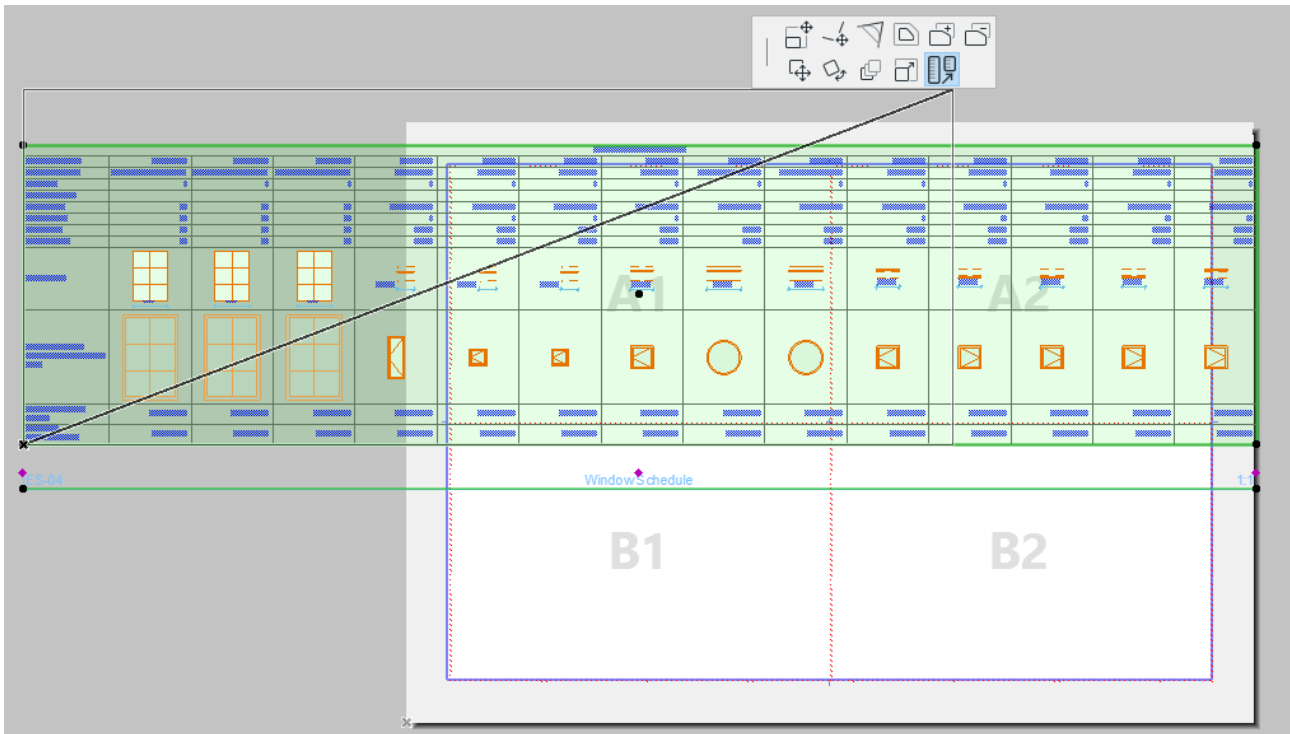
Note: You can add dimensions manually to any Preview picture by double-clicking its cell, or clicking the on-screen "Annotate" command.

Restructure Schedule to Fit Layout

Note: This function is also available for Project Indexes placed on Layouts.

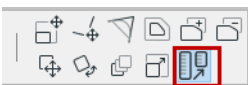
Use the **Restructure Table** command to automatically split up large Drawings (e.g. based on Schedules) which exceed the Layout size, and to lay out the parts next to each other.

The following image shows a Window Schedule that does not fit onto an A4 sized sheet.



Follow these steps:

1. Make sure that the Schedule (on which the Drawing is based) is using the “Records by Columns” format ([see Display by Rows or Columns.](#))
2. Make sure that the Drawing uses the “Fit Frame to Drawing” option ([see Drawing Frame.](#))
3. On the Layout: select the Drawing.
4. From the Pet Palette, choose the **Restructure Table** command, then manually resize the Drawing frame.



The result: the Schedule is split in two portions which are placed under each other.



Note: If the selected Drawing is set to **Split Drawing into Multiple Layouts**, then the **Restructure Table** command will place part of the Schedule onto an additional Layout.

See [Split Schedule into Multiple Layouts](#).

Split Schedule into Multiple Layouts

Note: This function is also available for Project Indexes placed on Layouts.

If a Schedule-based Drawing does not fit onto a single Layout - as can easily happen with a long Schedule - use this feature to quickly and easily split up a long Schedule and place it onto multiple Layouts, so that all data on the Schedule can be published.

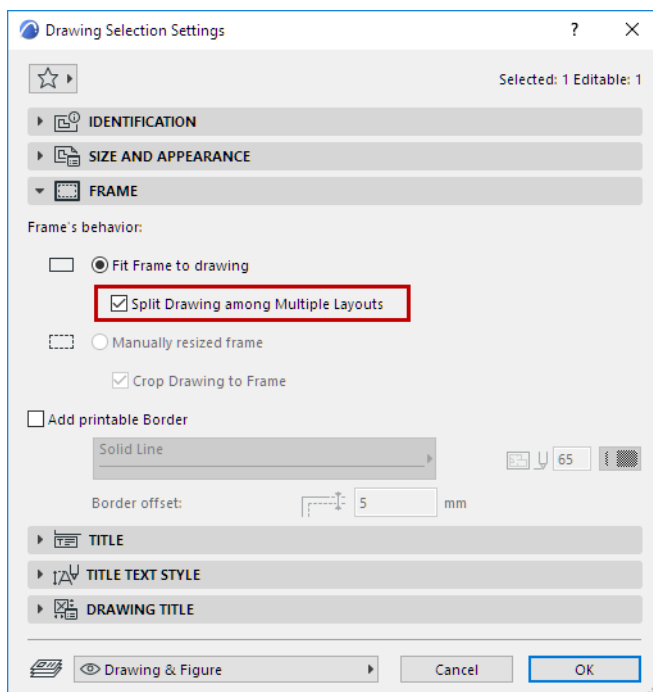
This feature is available only for Drawings created from Element Schedules and Project Indexes.

1. For the selected Drawing on the Layout: Use the **Restructure Schedule** pet palette command to graphically resize the Drawing's enclosing rectangle, to enable it to fit on the Layout.

[See Restructure Schedule to Fit Layout.](#)

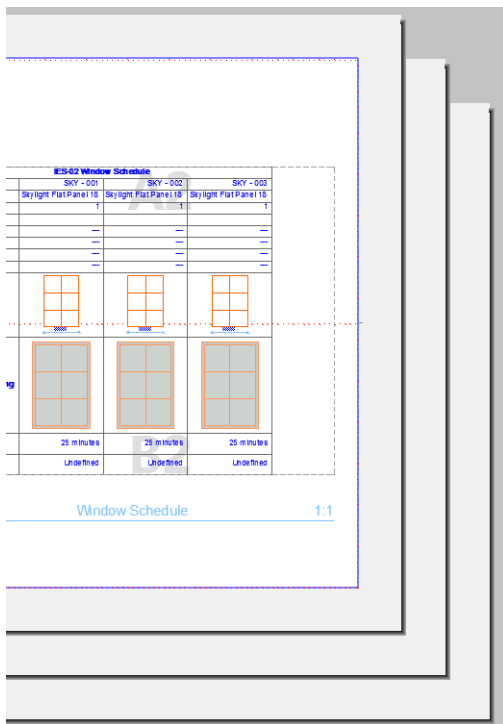
2. In Drawing Selection Settings: go to the Frame Panel and check **Split Drawing among Multiple Layouts**.

Note: This checkbox is available whenever you place a Drawing based on an Element Schedule onto a Layout. For Drawings with any other source, the checkbox is gray.

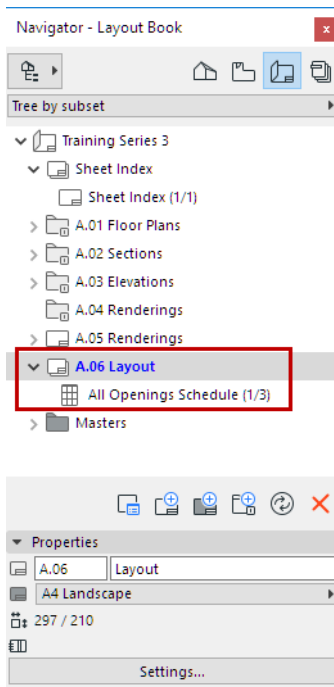


3. Now Archicad will create as many Layouts as needed to display the entire contents of the Schedule.

In this example, the Window List has been placed on the Layout. The Split Drawing feature has placed the total Window List onto three Layouts. In the Layout Window, Layout is drawn to indicate that it consists of multiple pages:



In the Navigator Layout Book, only one Layout is shown, but it has a “multi-page” icon, different from the single Layout icon, and the total number of Layouts used for this Drawing is displayed after the Drawing Name.



If a Layout Window displays a multi-Layout Drawing, the Window’s title bar shows the range of Layouts which display the Drawing.

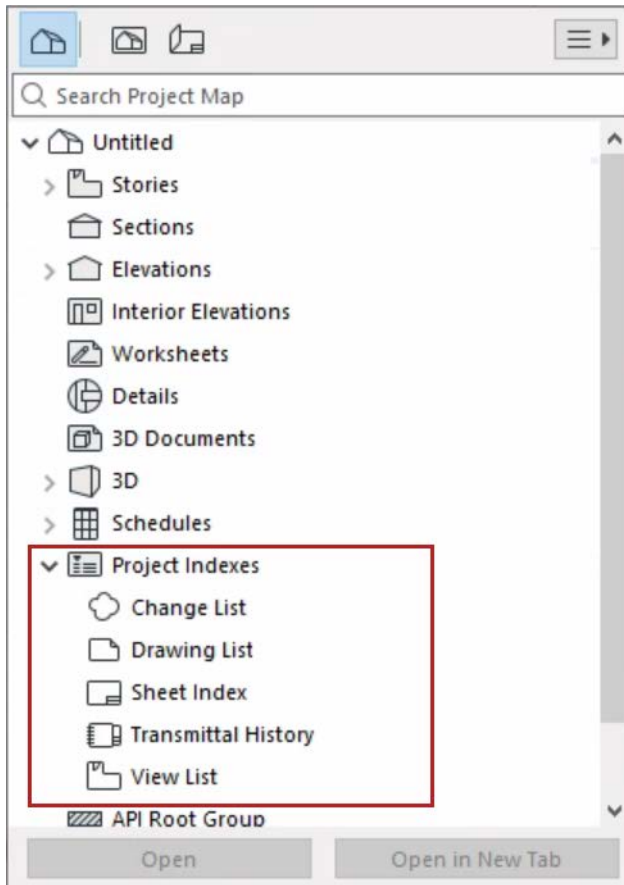
In the Layout Book, you can navigate between Layout pages as usual, using the navigation arrows at the bottom of the Layout Window.

Project Indexes

Project Indexes are table of contents-type items listed in the Project Map of the Navigator.

Use Project Indexes to make lists of revisions, issues, views, layout sheets and drawings, based on customized criteria.

To open an index, go to the Navigator palette and double-click its name.



Project index content is based on **Project Index Schemes**.

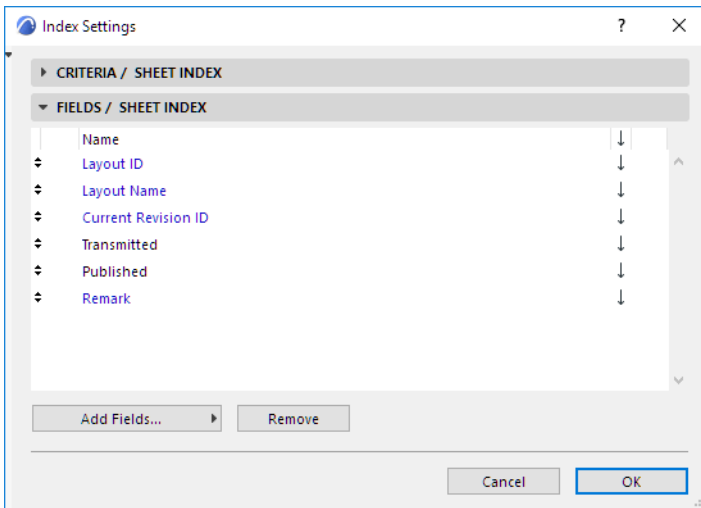
Like Interactive Schedules, Project Indexes can be saved as views, and you can edit project data directly from the Project Index window.

Subset ID and Name	Layout	Drawing		Last Updated
		ID	Name	
	Sheet Index		Sheet Index	2/14/2019 9:40 AM
	A.05 Renderings		Back - CineRender	6/6/2015 10:01 PM
	A.05 Renderings		Back - Internal	6/6/2015 9:58 PM
	A.05 Renderings		Entrance - Sketch	6/6/2015 9:58 PM
	A.05 Renderings		Entrance - White	6/6/2015 9:58 PM
	A.05 Layout		All Openings Schedule	2/14/2019 9:42 AM
	A.07 Layout		All Openings Schedule	2/14/2019 9:42 AM
	A.08 Layout		All Openings Schedule	2/14/2019 9:42 AM
A.01 Floor Plans				
	A.01.1 Ground Floor	0.	Ground Floor	4/25/2016 9:30 AM
	A.01.2 1. Story	1.	Gallery	4/25/2016 9:30 AM
	A.01.3 2. Story	2.	Roof	4/25/2016 9:30 AM
A.02 Sections				
	A.02.1 S-01 Building Section	S-01	Building Section	7/19/2017 11:26 AM
	A.02.2 S-02 Building Section	S-02	Building Section	7/19/2017 11:26 AM
A.03 Elevations				
	A.03.1 E-01 North Elevation		North Elevation	2/13/2019 4:43 PM
	A.03.2 E-02 East Elevation		East Elevation	7/19/2017 11:26 AM
	A.03.3 E-03 South Elevation		South Elevation	4/25/2016 9:30 AM
	A.03.4 E-04 West Elevation		West Elevation	4/25/2016 9:26 AM

Index of Published Items

Suppose you are publishing a number of Layouts in a Publisher Set, and you want to include a Sheet Index which indicates which items have been published. (The same is available for published views.)

One way to do this is to add the field “Published” to the Index.



This places a field called “Published” on the Sheet Index. Its checkbox will be activated for those items which you have published, but only in the published copy of the Index (not while you work in Archicad).

For example, if you publish Elevation layouts, as well as this Sheet Index, then the published Sheet Index Layout will look like this:

Sheet Index					
Layout ID	Layout Name	Revision	Issued	Published	
	Sheet Index	01	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
A.01.1	Ground Floor	02	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
A.01.2	1. Story	01	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
A.01.3	2. Story	01	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
A.02.1	S-01 Building Section	02	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
A.02.2	S-02 Building Section	01	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
A.03.1	E-01 North Elevation	01	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
A.03.2	E-02 East Elevation	01	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
A.03.3	E-03 South Elevation	01	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
A.03.4	E-04 West Elevation	01	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
A.05	Renderings	01	<input checked="" type="checkbox"/>	<input type="checkbox"/>	

Related Topics:[Index Settings Dialog Box](#)[Change List and Transmittal History](#)

Model View Options

The display and output of Model View elements can be set globally in **Document > Model View > Model View Options**.

Model View Options affect the on-screen and output display of construction elements and certain GDL objects throughout the project.

Model View Options can be applied to each view. Saved Model View Options Combinations are part of View Settings.

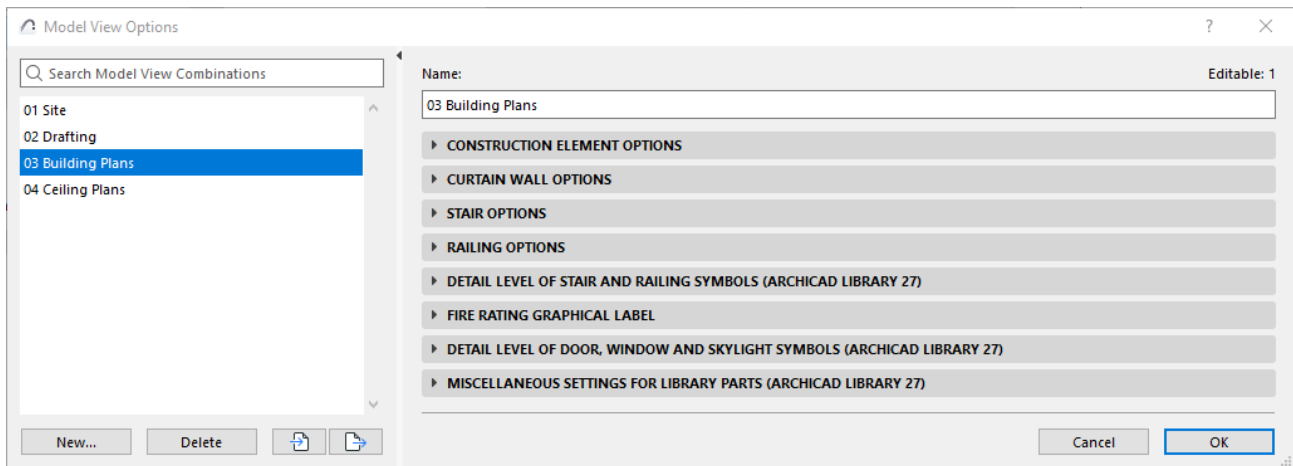
Topics in this section:

[Define Model View Options](#)

[Model View Options Combinations](#)

Define Model View Options

Use the following panels at **Document > Model View > Model View Options**:



Construction Element Options

Stair Options

Railing Options

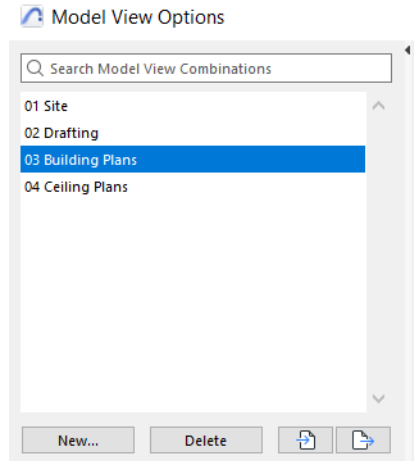
Detail Level of Stair and Railing Symbols

Detail Level of Door, Window and Skylight Symbols

Miscellaneous Settings for Library Parts

Model View Options Combinations

The list of pre-set Model View Options Combinations is available at **Document > Model View > Model View Options**.



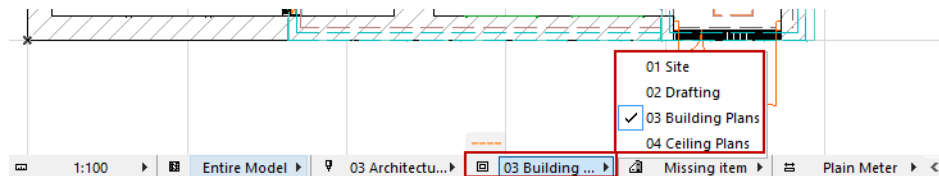
Model View Option Combinations can be applied to individual views depending on their purpose (for example, for use as Drafting plans or Site plans).

The active Model View Options Combination is displayed in the Quick Options Bar of the Archicad model window.

Apply a Model View Options Combination to the Project

To apply a Model View Options Combination to a currently active view, do one of the following:

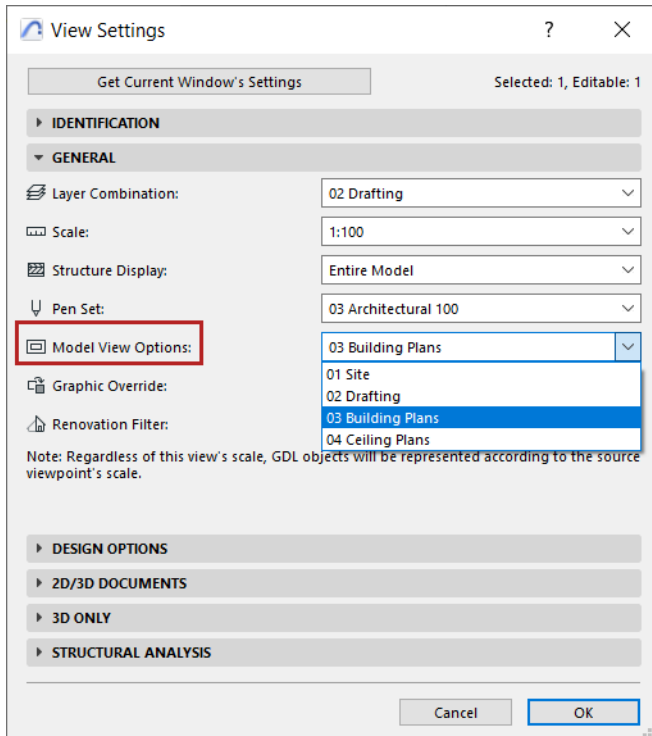
- Use the MVO Combinations pop-up from the Quick Options Bar at the bottom of the window



- Use the **Document > Model View** menu
- Use the **View Settings** dialog box
- Open **Document > Model View > Model View Options** and choose one of the defined Model View Options Combinations from the top panel.
- Use View Settings and Storing Options (at the bottom of the Project Map in the Organizer)
- Use the pop-up list in the Properties section of the Navigator Palette (where available)

Apply a Model View Options Combination to a Saved View

When saving a view from any model window, the view's current Model View Options Combination is also saved. To apply a different Model View Options Combination to any single view, use View Settings.



Use Model View Options Combination in Another Project

You can export and save your stored combination as an .xml file, so you can import and use it in another project.

In the Model View Options Combinations panel, click **Export** to export the selected Model View Options Combination as an .xml file to the desired location.

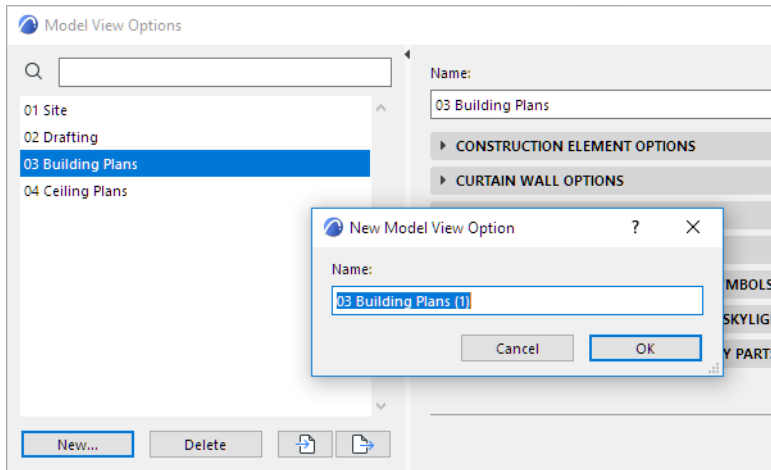
To use it any other Archicad project, open the project and click **Import** from the same panel to browse for accessible .xml files to import as Model View Options Combinations.

Create and Delete Model View Options Combinations

On the left side of the Model View Options dialog box, use the following functions:

New

Creates a duplicate of the currently selected Model View Option Combination, with a new name.



Delete

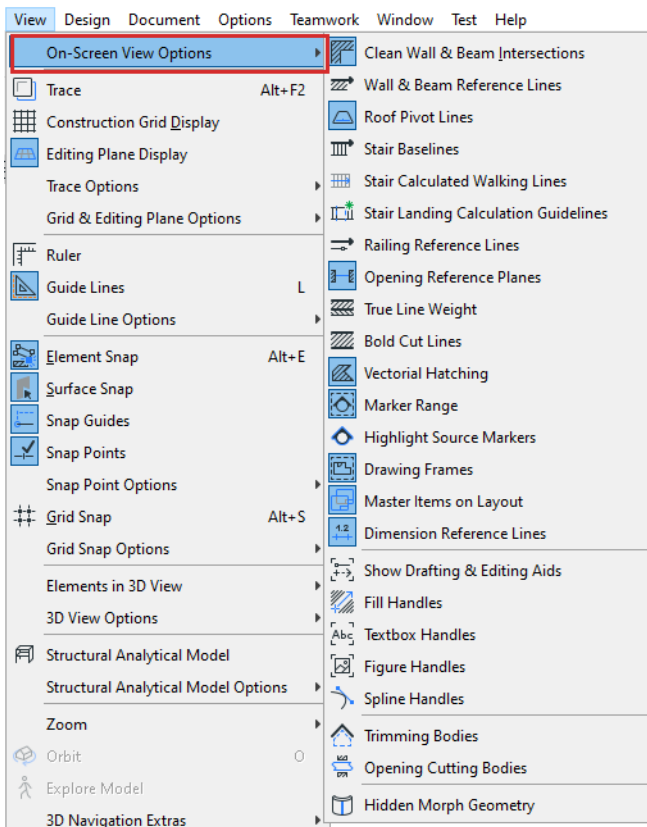
Click this button to delete the selected Model View Options Combination. **This operation is not undoable.**

On-Screen View Options

The **View > On-Screen View Options** command produces a list of items (generally aids in editing) that can be shown or hidden on screen, depending on your preferences. Other items in this list let you toggle between line weight options and vectorial vs. bitmap display of fills.

Most of these options affect element display on the Floor Plan only.

Click any item to toggle its status.



This status has no effect on whether the item will be displayed on your output; it affects only on-screen display.

For a quick way to access these commands, activate the **Window > Toolbars > On-Screen View Options** toolbar.

On-Screen View Options Commands

Use the **View > On-Screen View Options** command to access these options (for a quick way to access these commands, activate the **Window > Toolbars > On-Screen View Options** toolbar).

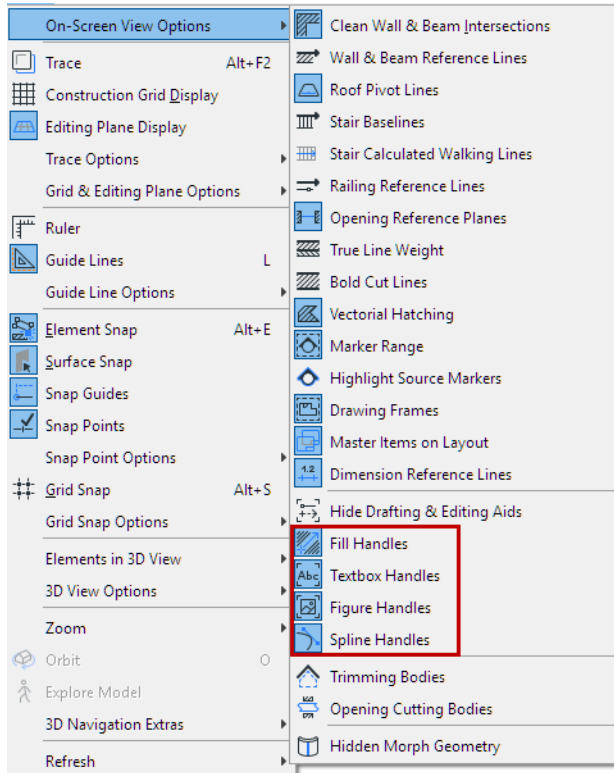
- **Clean Wall & Beam Intersections:** Toggles the clean wall and beam intersection mode on and off.
- **Walls & Beams Reference Lines:** Toggles between displaying walls and beams with their reference lines and their contour lines.
- **Roof Pivot Lines**
- **Stair Baselines**
- **Stair Calculated Walking Lines**

- **Stair Landing Calculation Guidelines:** Shows distribution of extra landing length in case of Automatic Landing constraint
- **Railing Reference Lines**
- **Opening Reference Planes:** This toggle ensures that an Opening will have an on-screen display on the Floor Plan, even if it is located entirely outside of its associated host element.
- **True Weight:** Use this option to toggle the True Weight display of Lines on or off. If it is off, lines will be displayed in Hairline, and the following option (Bold Cut Lines) is not applicable.
Note: The Hairline option in On-Screen View Options affects on-screen display only. For hairline output, use the Hairline checkbox in the Print or Plot dialog box.
- **Bold Cut Lines:** Use this option to specify whether Construction Elements are displayed with bold cut lines.
- **True Fill Appearance:** Use this option to turn Fill pattern display on or off. If it is off, then fills will be displayed in their bitmap patterns. This control affects on-screen display only for all Fills (regardless of fill type).
- **Marker Range:** Toggle the on-screen display of Marker Range items - e.g. the input line or polygon placed with any of the marker-type tools and the Change tool.

[See Display of Marker Range Lines.](#)

- **Highlight Source Markers:** Toggle the on-screen display of source marker highlights. Source markers are those which create a viewpoint when you place them. The color of source marker highlights is set in **Options > Work Environment > On-Screen Options**.
- **Drawing Frames:** Use this option to show/hide Drawing Frames on the layout.
- **Master Items on Layout:** Use this option to specify whether Master Items are displayed on the layout. If you want a uniform for all Master items, set this in **Options > Work Environment > On-Screen Options**.
- **Dimension Reference Lines:** Show/Hide a Dimension Line (on-screen only) while using the **Dimension Text Only** setting.
- **Show/Hide Drafting & Editing Aids:** Use this option to show or hide all available drafting aids. Drafting aids include Fill Handles, Figure Handles, Text Boxes, and Bézier Spline handles.

Use the next four toggles to show or hide each of these drafting aids individually.



Note: When the Profile Editor window is active, the Fill Handles toggle changes to **Component Area Base Line/Circle**.

See [Components Panel of the Profile Editor](#).

- **Trimming Bodies:** Use this option to turn the display of Trimming Bodies on or off. Trimming Bodies can be shown in the 3D Window only, for Shells and Roofs.
- **Opening Cutting Bodies:** Turn the display (in both 2D and 3D windows) of Openings' Cutting Bodies on or off.
- **Hidden Morph Geometry:** Use this option to show hidden Morph geometry, and thus be able to select all Morph edges, including "soft" ones.

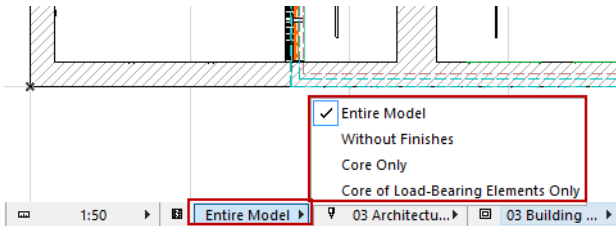
Partial Structure Display

The Partial Structure Display function allows you to display construction elements in various ways depending how you define the components of elements.

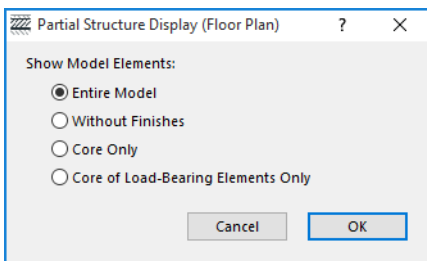
For example, when collaborating with other designers or with engineers, you may prefer to ignore non-load-bearing walls, and to display/output only the load-bearing components of complex elements.

To switch the preferred Partial Structure Display Option, do one of the following:

- Use the Partial Structure Display pop-up from the Quick Options Bar at the bottom of the window



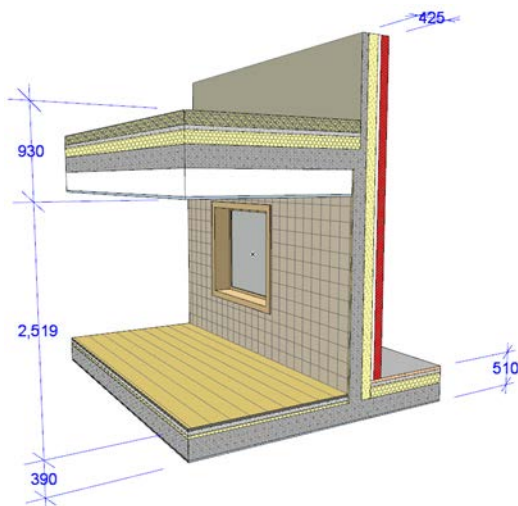
- Use **Document > Partial Structure Display** to set these preferences; they will be applied to the current window and can be saved separately for each view.



Use the first three options to display or hide the components of composite or complex elements, and column veneers, depending upon how you have defined those components.

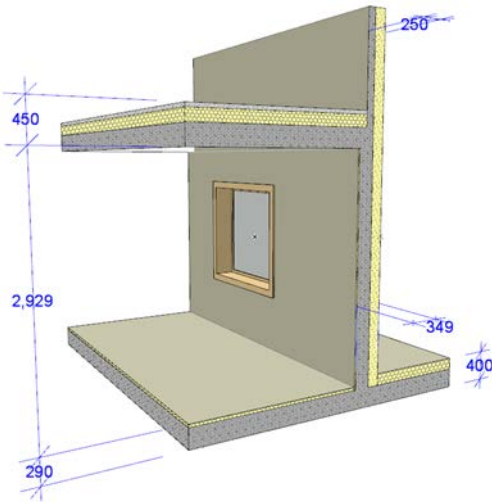
[See *How to Define Components as “Core” or “Finish”*.](#)

1. **Entire Model:** this is the default setting. All parts of the model are displayed.

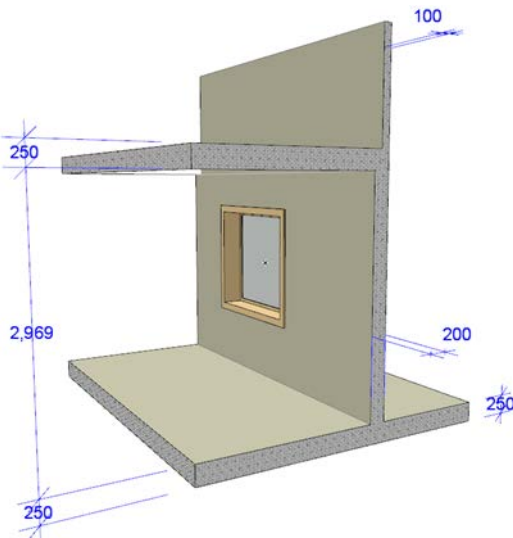


2. **Without Finishes:** Model is displayed *without*:
 - skins/components that are defined as “Finish” and

- column veneers that are defined as “Finish”



- 3. Core Only:** Displays only the structures defined as “Core” (applicable to complex or composite structures, and columns having a veneer)



See also [Dimension Skins of Composite/Profile Element Automatically](#).

- 4. Core of Load-Bearing Elements Only:** The fourth option considers all construction elements (not just composite/complex). This option will hide any elements you have classified as “non-load-bearing” or “undefined”.

The “Core of Load-Bearing Elements Only” option has three effects:

- Elements classified as “Non-Load-Bearing” or “Undefined” will be hidden; all elements classified as “Load-Bearing” are displayed
- Load-bearing composite and complex structures are shown with their Core only
- Column veneers defined as “Core” in Column Settings will be shown in addition to the Column cores

Topics in this section:

[Applying Partial Structure Settings to Model Windows](#)

[Save Partial Structure Settings by View](#)

How to Define Components as “Core” or “Finish”

Visualizing the Entire Model

Partial Structure Display Settings: Effect on Elements

Partial Structure Display Settings in Windows Based on a Source Marker

Applying Partial Structure Settings to Model Windows

Unlike Model View Options, which affect 2D windows only, Partial Structure Display affects the display of elements in 2D and 3D windows alike.

Partial Structure Display settings can be assigned in the following window types:

- **Floor Plan** (a single Partial Structure setting is in effect for every Floor Plan window)
- **3D Window** (one setting for the 3D window)
- **Section/Elevation/Interior Elevation/3D Document:** Each instance of these window types can have its own Partial Structure Display setting (i.e. you can have three Section windows with three different Partial Structure settings).

Exception: You cannot set a Partial Structure Display setting for a Drawing-type Section or Elevation, or for any Independent viewpoint.

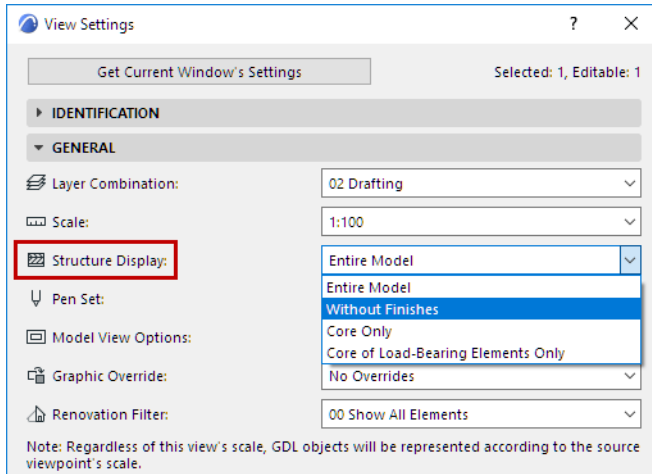
See [Partial Structure Display Settings in Windows Based on a Source Marker](#).

- **Worksheet and Detail:** If the source viewpoint is in the Floor Plan or is an Auto-update viewpoint, then you can set Partial Structure Display settings for Worksheets and Details (separate ones for each window, if needed.)

For a quick way to assign Partial Structure Display to the current window, use the Partial Structure pop-up in the Quick Options Palette.

Save Partial Structure Settings by View

You can save different Partial Structure Display settings on a view-by-view basis; thus, your settings will affect the content of published views and layouts.



This means you can save different Partial Display Settings for each view: for example, you can save two different views of the same Section, each having a different Partial Structure Display.

Exception: You cannot set a Partial Structure Display setting for a Drawing-type Section or Elevation, or for any Independent viewpoint.

See [Partial Structure Display Settings in Windows Based on a Source Marker](#).

How to Define Components as “Core” or “Finish”

Composite Elements

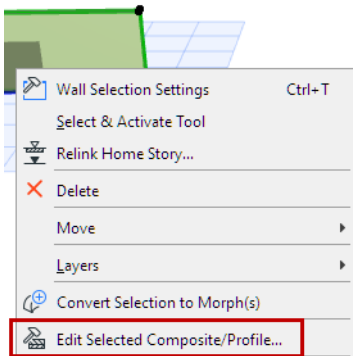
For composite elements (Wall, Slab, Roof, Shell): skins can be defined as either **Core**, **Finish**, or **Other** in the Composite Structures Dialog Box.

This is relevant if you wish to display the model elements depending on their structure type: this is the Partial Structure Display function.

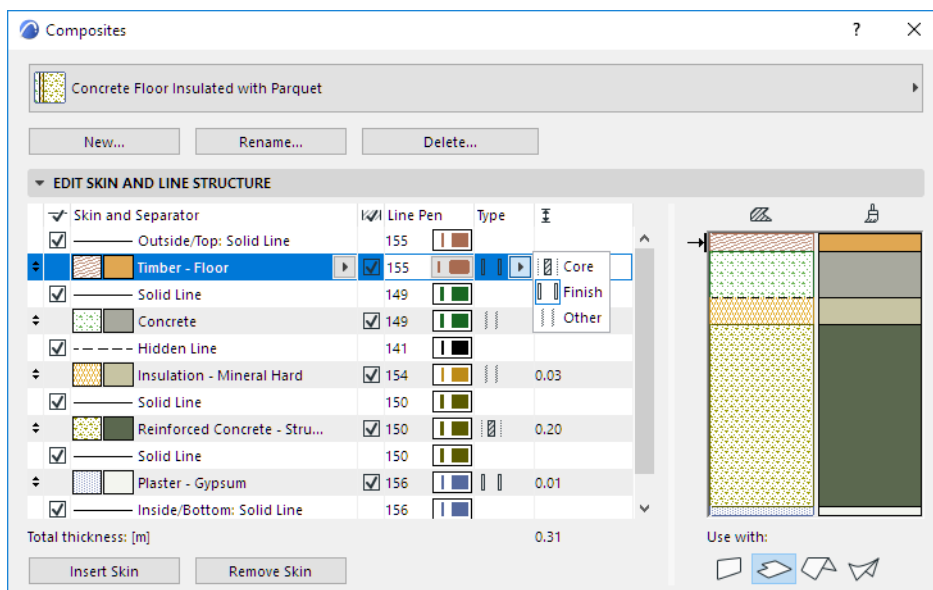
For information, see [Partial Structure Display](#).

1. Open the Composite Structures dialog box from **Options > Element Attributes > Composites**.

You can also edit a selected composite by accessing its settings from its context menu:



2. Click on a skin in the list to select it.
3. In the Type column, click the pop-up and choose Core, Finish, or Other.



Notes:

- You can define multiple skins as Core, but these skins must be adjacent to each other.
- You can define multiple adjacent skins as Finish, but they must include one or both of the outermost skins.

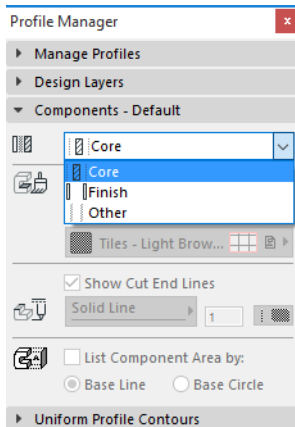
Column Veneers

Each veneer can be defined as Core, Finish or Other, in Column Tool Settings.

[See *Core and Veneer*.](#)

Profile (Complex) Elements

For Profile elements (Wall, Beam, Column): Components can be defined as **Core**, **Finish**, or **Other** in **Design > Profile Manager**.



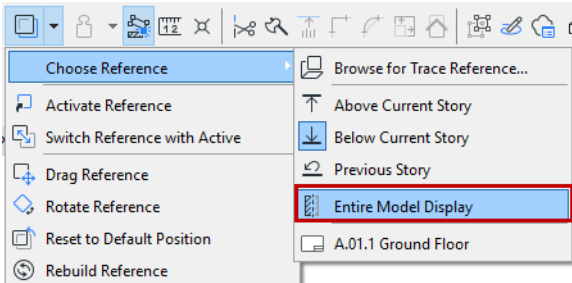
[See also *Components Panel of the Profile Manager*.](#)

Visualizing the Entire Model

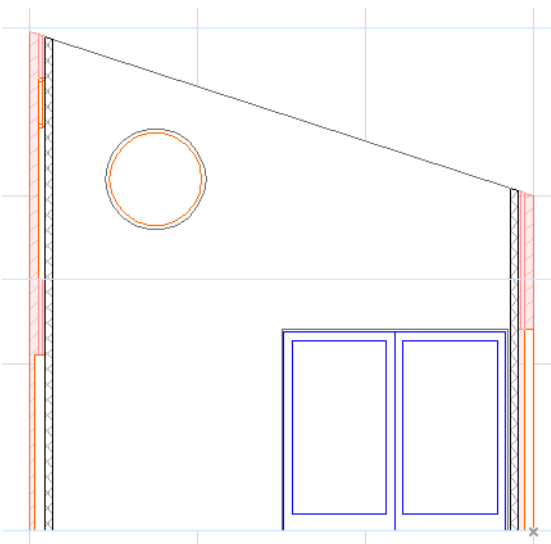
Even if you are using a Partial Display option, Archicad gives you several types of feedback to help you identify the entire original structure if needed.

You can always choose the **Entire Model Display** option for your **Trace Reference**:

1. Activate the **Trace** function.
2. From the Trace drop-down menu, use **Choose Reference**, and select **Entire Model Display as the Trace Reference**.



In this Section, the composite walls are displayed with Core-Only settings, but the “Entire Model Display” reference (in red) indicates the full wall structure.



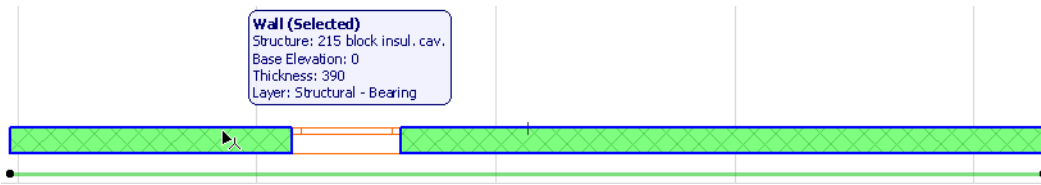
When selecting a partially displayed element, **hotspots** will appear at their original location. The selection highlight, however, is shown only on the displayed parts of the elements.



Preselection feedback will also help you identify the entire as well as the partially displayed parts of the elements. The pre-selection highlight is shown only on the displayed components, but the cursor recognizes the reference line even if it is not on a displayed part of the element - as shown by the bold Mercedes cursor in this image.



The **Info Tag** displays the Thickness of the original, entire element, even if it is currently only in Partial Display:



Partial Structure Display Settings: Effect on Elements

This page contains element-specific information on the effect of Partial Structure Display.

Columns In Partial Structure Display

Zones and Partial Structure Display

Floor Plan Cover Fills in Partial Structure Display

Dimensions in Partial Structure Display

Intersections of Partially Displayed Elements

Columns In Partial Structure Display

Each veneer can be defined as Core, Finish or Other, in Column Tool Settings.

[See *Core and Veneer*.](#)

For Columns “wrapped” by walls: No wrapping occurs if you are using Core only display.

Zones and Partial Structure Display

Regardless of which Partial Display option is in effect, zones are not affected: zones that extend to an “Inner Edge” are calculated to include the “finish” component of a surrounding structure, even if the finishes are hidden according to the current Partial Structure Display Setting.

Floor Plan Cover Fills in Partial Structure Display

The Partial Structure Display setting has no effect on cover fills.

If you have assigned a Cover Fill to any composite or complex element (this might be a Roof, Slab, or Mesh), then the Cover Fill will appear regardless of the Partial Display option that is in effect.

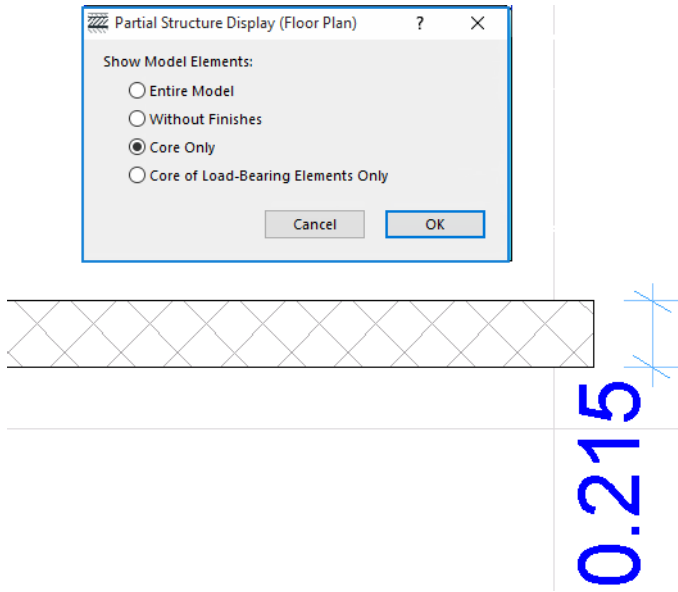
Note: Cover Fills can use Graphic Overrides.

[See *Graphic Override Rules*.](#)

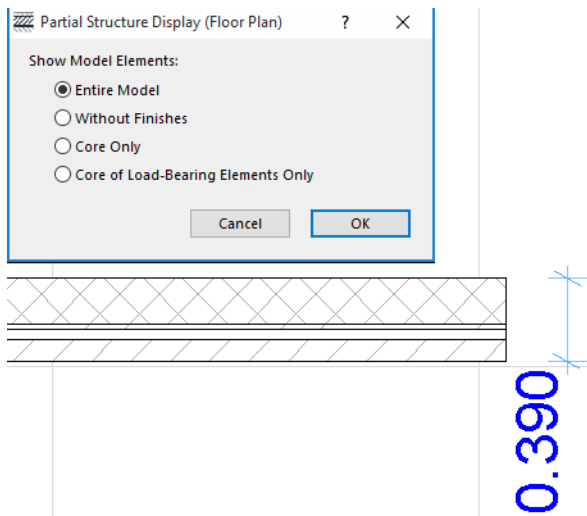
Dimensions in Partial Structure Display

Each Partial Display mode has its own set of dimensions: you can have a set of dimensions in your Core Only view, and other sets of dimensions in your Entire Model and Without Finishes views. **If you switch between Partial Structure Display settings, the dimensions associated with the earlier Display setting will not be visible.** To see the Dimensions again, switch back to the Partial Structure Display setting in which the Dimensions were placed.

This wall is displayed with the Core Only display setting:



If you now switch the display setting to Entire Model, the dimension shown in Core Only disappears, and the other dimension - the one previously created with the Entire Model display setting - appears.



You can copy-paste dimensioned elements between two views whose Partial Structure Display settings are dissimilar. However, if you try to paste dimensioned elements that do NOT appear in the given Partial Structure display, those dimensions will be lost after you paste.

Intersections of Partially Displayed Elements

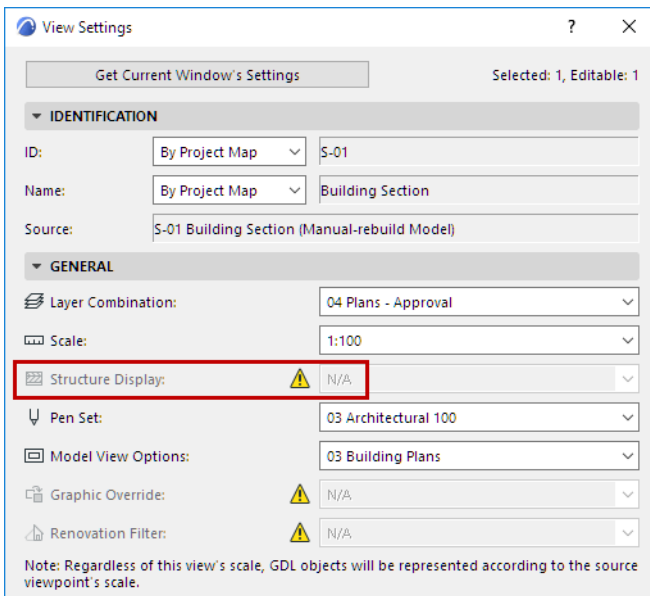
Intersections will behave just as they do when the entire model is displayed: No further skin intersections will be cleaned up as a result of changing to partial display.

Partial Structure Display Settings in Windows Based on a Source Marker

Often, you will be working in a model view (Section, Elevation, Interior Elevation, Detail or Worksheet) in which the elements have their source in a different model window.

However, if a your source viewpoint is a Manual-Rebuild or Drawing-type view/viewpoint, you would run into problems if you tried to change the derived view’s Partial Structure Display Settings, because the source viewpoint might not be in updated condition. Therefore, Archicad does not let you change the Partial Structure Display settings for views derived from Manual-Rebuild or Drawing-type sources.

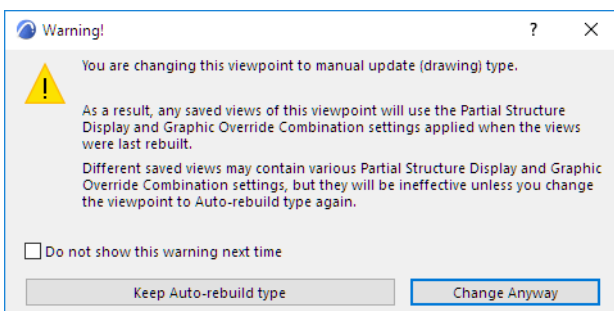
The following view settings belong to a view whose source is a Manual-Rebuild model; the Partial Structure Display settings cannot be changed.



The solution: go back to the source view (or make a copy of it) and change its update type to Auto-update.

In the opposite case: Suppose you have an Auto-update viewpoint with several views derived from it. Then you change the viewpoint to “Manual Update” or “Drawing type.” This is possible, but be aware that the views deriving from this source - even if you’ve already created these views - will, when rebuilt, change their Partial Structure Display settings to match those of the source viewpoint.

If you do change the Partial Structure Display settings of a source viewpoint which is of Manual Rebuild type (even though its derived views are based on an Auto-Rebuild source), Archicad warns you of the consequences for the derived views: the views’ Partial Structure Display and Graphic Override settings might change:



You can go ahead and continue Rebuild, or you can cancel the process. Again, the solution is to go back to the source view (or make a copy of it with the Partial Display Structure/Graphic Override settings you need) and change its update type to Auto-update.

Renovation

Use the Renovation feature set to create views that provide visual feedback on the status of each element at a particular stage of a renovation project.

To achieve this, each element is assigned one of three **Renovation Statuses**:

- Existing;
- To be Demolished;
- or New.

Note: By default, all elements have a Renovation Status of “Existing.” New elements that you add to the project are also set to “Existing.”

You can then assign view settings, called **Renovation Filters**, to display the project at various stages: such as “Demolition Plan” or “New Construction.” Renovation Filters show/hide elements and/or provide graphical highlighting (**Graphic Override**) depending on their status, to convey information on their renovation status.

Archicad comes with predefined Override Rules for each of the three Renovation Statuses. If you have different style preferences, use the **Graphic Override Rules** dialog box to redefine them.

Since the Renovation Filter is saved with the view, it also affects the appearance of the Drawings in your documentation.

[Watch the Video](#)

Related Topics:

[Assign Renovation Status to Elements](#)

[Renovation Filters](#)

[Graphic Overrides By Renovation Status](#)

[Best Practices for Creating Renovation Views](#)

[Workflows for Tricky Renovation Situations](#)

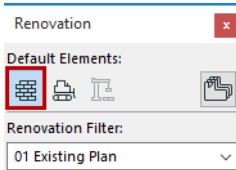
[Reset Renovation Status](#)

Assign Renovation Status to Elements

See also [Reset Renovation Status](#).

Use Renovation Palette to Set Default

1. Open the Renovation Palette from **Window > Palettes > Renovation**.
2. With nothing selected, choose a Status to assign to all newly placed elements:



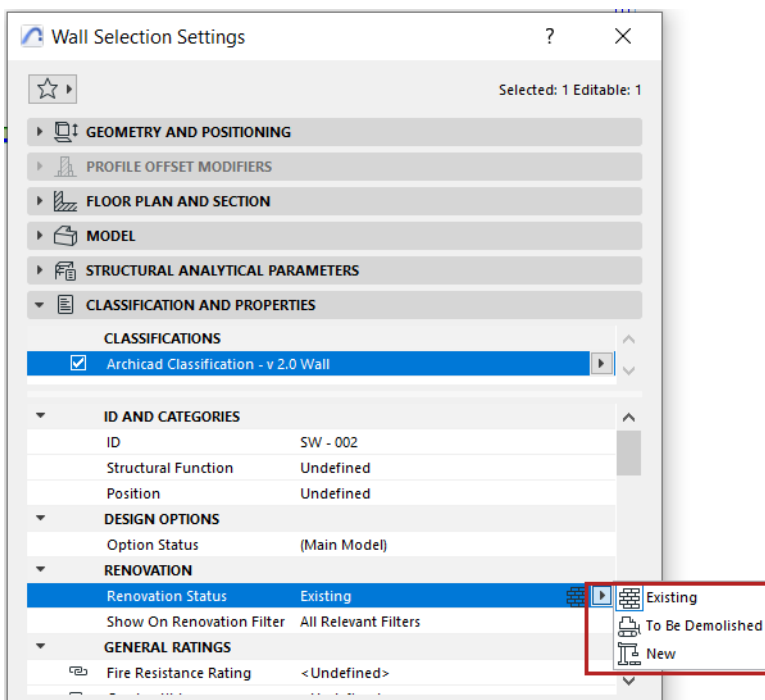
Use Renovation Palette to Change Renovation Status

1. **Select** one or more elements to which you want to assign a different renovation status.
2. From the **Renovation Palette**, click the icon representing the status you need.
 - **Existing** Element: to be retained
 - **New** Element: to be built during the current renovation process
 - **To Be Demolished** Element: to be demolished during the current renovation process

For example, in an existing project, select all the elements to be demolished, then click the “To be Demolished” icon. This way, you change the status of all those elements to “To Be Demolished” with a single click.

Use Element Settings to Change Renovation Status

Change the renovation status of *selected* elements (if they are all of the same type) in the Classification and Properties panel of the element Settings dialog box:



Note: If you have several element types selected, use the Renovation Palette or the Edit Selection Set palette to change their Renovation Status simultaneously.

The Renovation controls in the Settings dialog box are active only for selected elements.

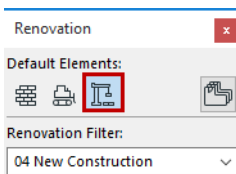
(If nothing is selected, the Renovation controls in the tool Default Settings are gray; use the Renovation Palette to set a default Renovation Status for all elements.)

Choose Default Renovation Status for Newly Placed Elements

By default, all elements have a Renovation Status of “Existing.” New elements that you add to the project are also “Existing.”

However, you can reset this default for all subsequently placed elements, using the Renovation Palette.

1. With nothing selected, click the icon of the Renovation Status you wish to be the default - for example, “New.”



2. Now, all elements you place - regardless of which tool you use - will automatically be assigned the “New” status.

It may be useful to set a default (e.g. New) before you begin to place additional elements in an existing project. This will save you the step of having to define those elements’ status as “new” retroactively.

Renovation Filters

Use Renovation Filters to display your project to reflect different stages of renovation

The given Renovation Filter (such as Demolition) defines how or whether elements are displayed in the project, depending on their Renovation Status.

- **Show** the elements with their default display, or
- **Hide** the elements, or
- **Show** the elements **with an Override**.

Since the Renovation Filter is saved with the view, it also affects the appearance of the Drawings in your documentation.

Exception: You cannot set a Renovation Filter setting for views that are based on Drawing-type or Manual-Rebuild Section or Elevation markers.

Archicad comes with several predefined **Renovation Filters**, but you can change the options at **Document > Renovation > Renovation Filter Options**.

Topics in this section:

[Apply Renovation Filter](#)

[Show an Element in a Single Renovation Filter Only](#)

[Renovation Filter Options](#)

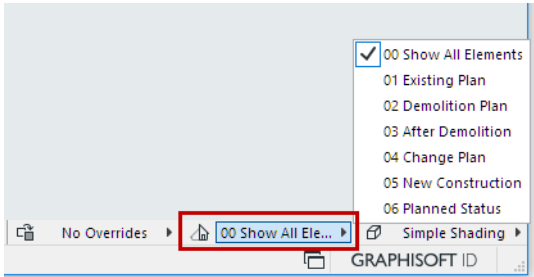
[Examples of Renovation Filters](#)

Apply Renovation Filter

To apply a different Renovation Filter to the current tab/window, do one of the following:

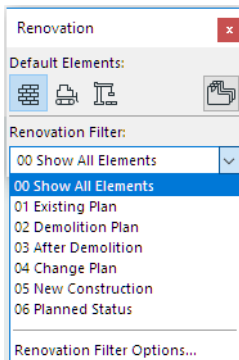
Use Quick Options

- Use the Renovation Filter pop-up from the Quick Options Bar at the bottom of the window



Use Renovation Palette:

From the Renovation Palette, click the desired filter from the Renovation Filter pop-up:



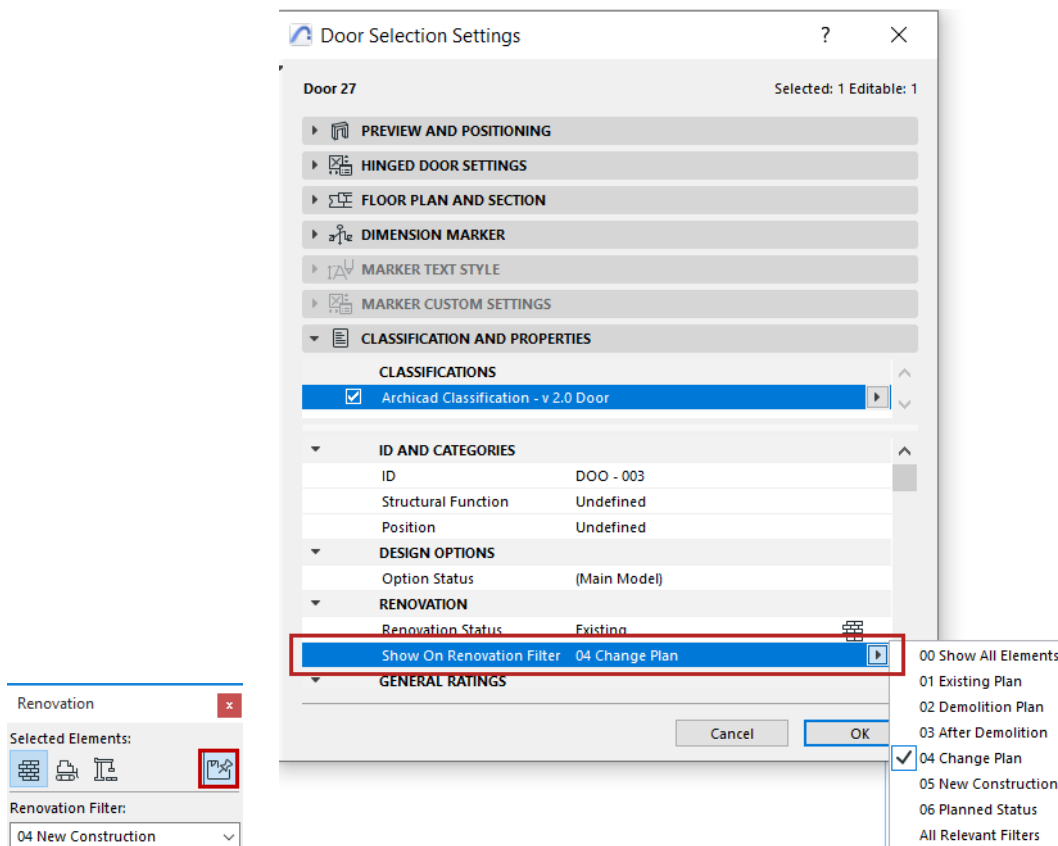
Show an Element in a Single Renovation Filter Only

During the course of your Renovation project, you may wish *certain specific elements* to be visible only if a particular Renovation Filter is in effect.

To achieve this, you must define the selected element(s) as specific to the current Renovation Filter. The filter-specific element(s) will be shown *only* in that single Renovation Filter, and *will not exist* in any other Renovation Filter.

Note: Of course, the filter-specific element will be shown in that Renovation Filter only if its Renovation Status would ordinarily be shown in that Renovation Filter.

You can define a selected element as “filter-specific” either on the Renovation Palette, or in its Selection Settings (Classification and Properties Panel).



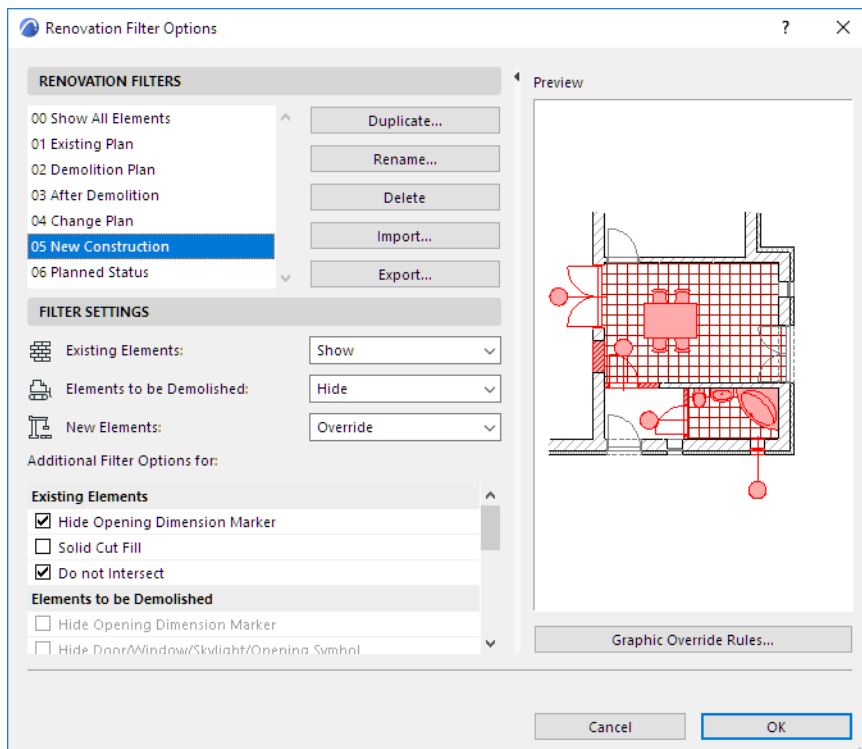
Renovation Filter Options

Use this dialog box (**Document > Renovation > Renovation Filter Options**) to define which elements (by Renovation Status) should be shown, hidden, or overridden in a Renovation Filter.

You can change the settings of existing filters, or create new Renovation Filters.

Renovation Filter Options are stored with the project.

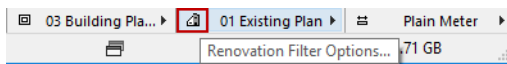
See also [Examples of Renovation Filters](#).



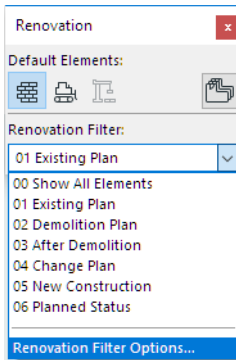
Open Renovation Filter Options Dialog Box

Do one of the following:

- Click the Reno Filter Options icon from the Quick Options Tab

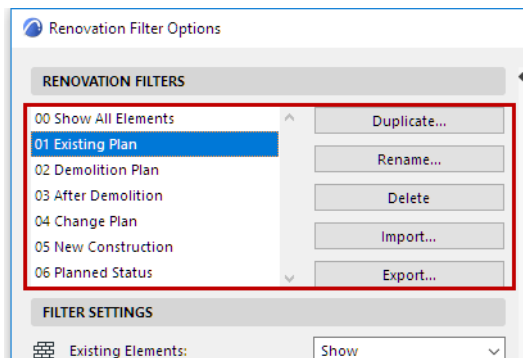


- Go to Document > Renovation > Renovation Filter Options
- Click Renovation Filter Options from the pop-up in the Renovation Palette.



Choose a Filter to Edit

Use the pop-up at the top of Renovation Filter Options to choose the Filter you want to view or edit.

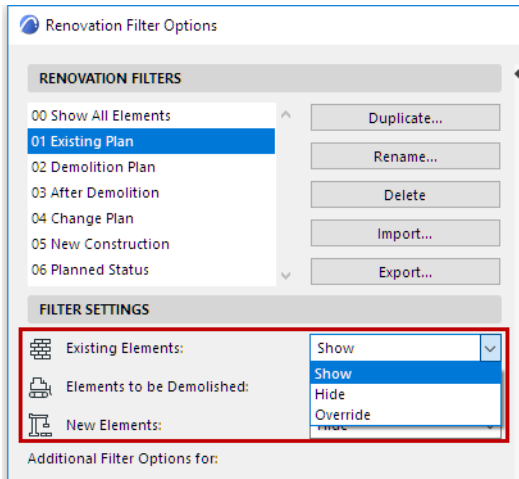


- **Duplicate:** Create a new Renovation Filter by duplicating the selected filter under a new name and editing its properties.
- **Rename:** Rename the currently selected Renovation Filter.
- **Delete:** Remove the selected Renovation Filter from the project.
- **Import:** Add a saved Renovation Filter (.xml file) to your project.
- **Export:** Save the selected Renovation Filter (.xml file) to an external location. It can then be imported to another Archicad project.

Define Show/Hide/Override By Renovation Status

For the selected Renovation Filter, define which elements, depending on their Renovation Status, should be shown, hidden, or overridden whenever this Renovation Filter is in effect.

For each Renovation Status, choose either **Show**, **Hide**, or **Override**.



- **Show:** Elements of this Status will be shown using the fills/colors/lines as set in their Settings dialog box.
- **Hide:** Elements of this Status will not be shown at all.
- **Override:** Elements of this Status will be shown, using the Override Style defined in the Graphic Override Rules dialog box.

Note: To open this dialog box, click the button at the bottom of Renovation Filter Options, or use **Document > Graphic Overrides > Graphic Override Rules**.

For more information, see [Graphic Overrides By Renovation Status](#).

Add Additional Overrides to the Renovation Filter

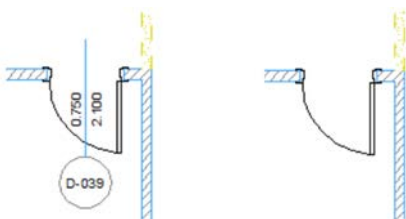
Additional Override options are available for “Existing” elements, for “To be Demolished” elements, and for Annotation Elements.

These extra options help you meet local standards or make your Renovation views easier to interpret.

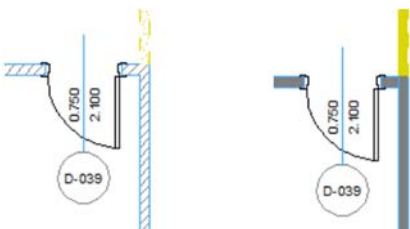
Check the box of any of the Additional Overrides you wish to add to your Renovation Filter definitions:

For “Existing” Elements:

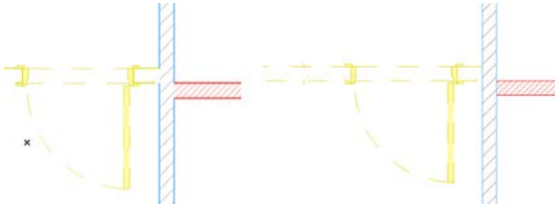
- **Hide Opening Dimension Marker** (if you don’t need to see dimensions of openings already in place)



- **Solid Cut Fill** (if you don’t need to see the structure of walls already in place)

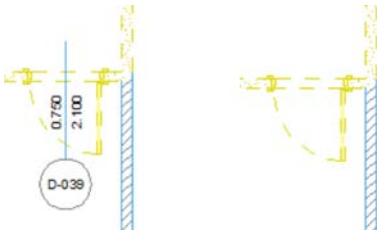


- **Do not Intersect** (use this to ensure that intersections among New or Demolished walls do not interfere with existing wall intersections)

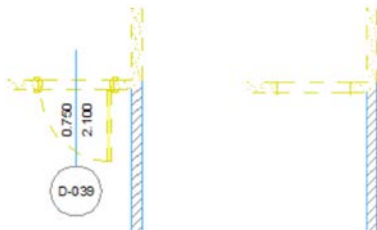


For “To Be Demolished” Elements:

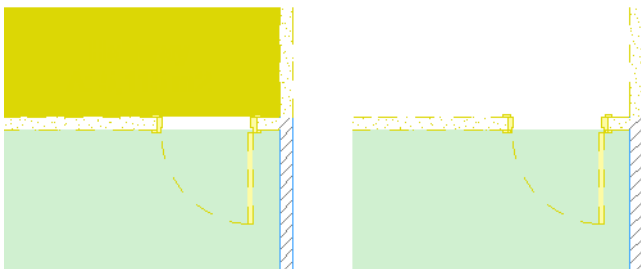
- **Hide Opening Dimension Marker** (if you don’t need to see dimensions of openings that will soon no longer exist)



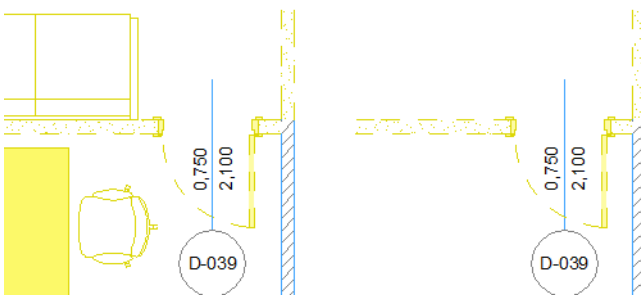
- **Hide Door/Window/Skylight/Opening Symbol** (if you don’t need to see openings that will soon no longer exist, especially if they are part of walls to be demolished anyway)



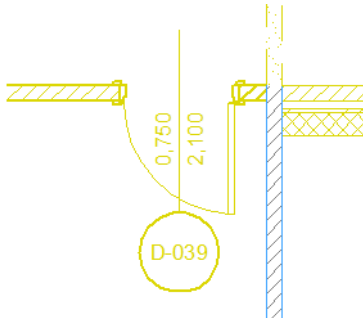
- **Hide Zone** (if you don’t want to see information for zones that will be eliminated)



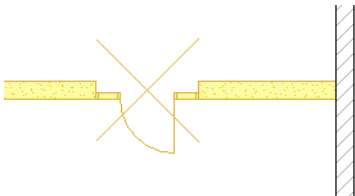
- **Hide Objects** (since most furnishings will not really be demolished; and they don’t need to be visible on the Demolition Plan)



- **Solid Cut Fill** (if you don't need to see the structure of walls that will be demolished)
- **Transparent Fill** (to indicate the place of a demolished element with just lines, no fill)
- **Do not Intersect** (to ensure that intersections among New or Demolished walls do not interfere with existing wall intersections)



- **Mark Openings with “X” Symbol:** Place an X on Doors and Windows to be demolished.

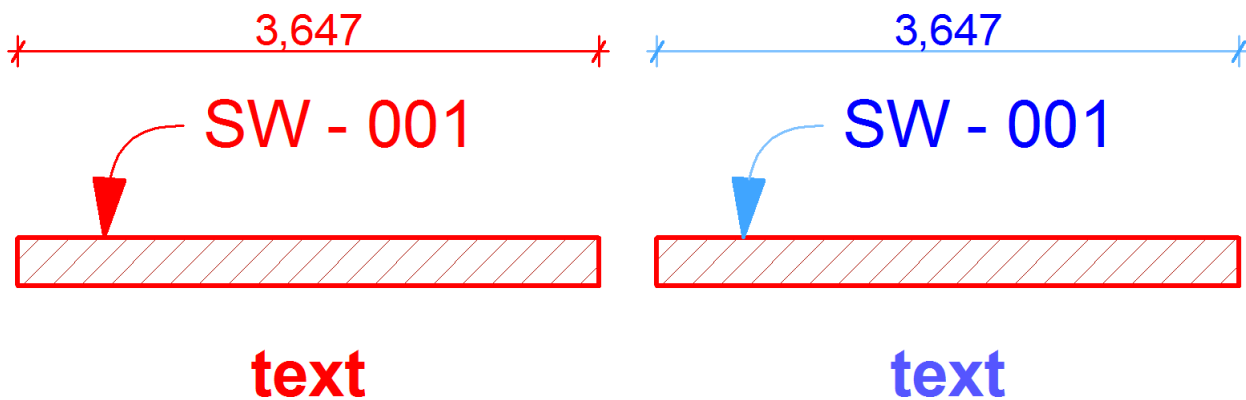


For Annotation Elements:

- Do not Override Dimensions/Texts/Labels
- Do not Override Drafting Fills/Lines

Use these checkboxes to exclude Annotation Elements - Dimensions, Texts and Labels, and/or Drafting Fills and Lines - from the filter settings.

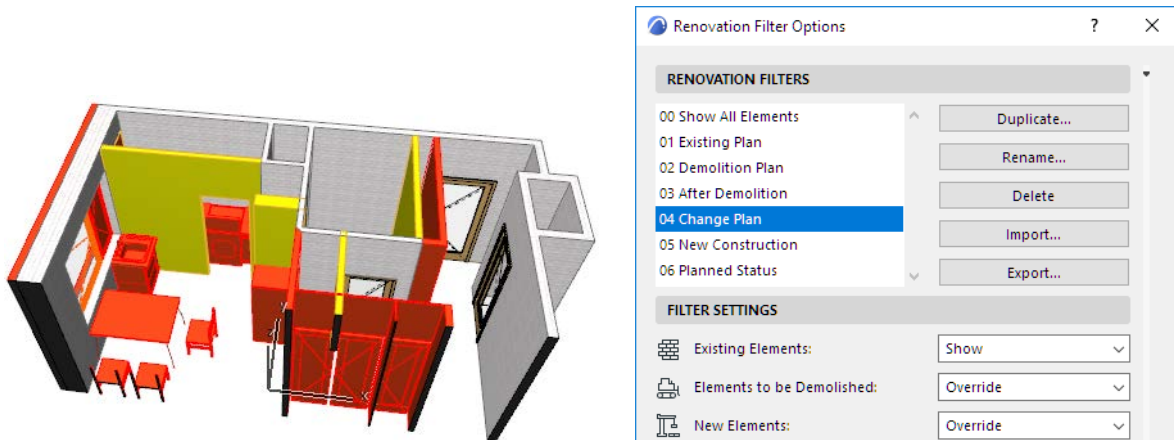
For example, this way you can make sure your labels and texts continue to be shown normally (as in the image on the right), even on a Demolition Plan:



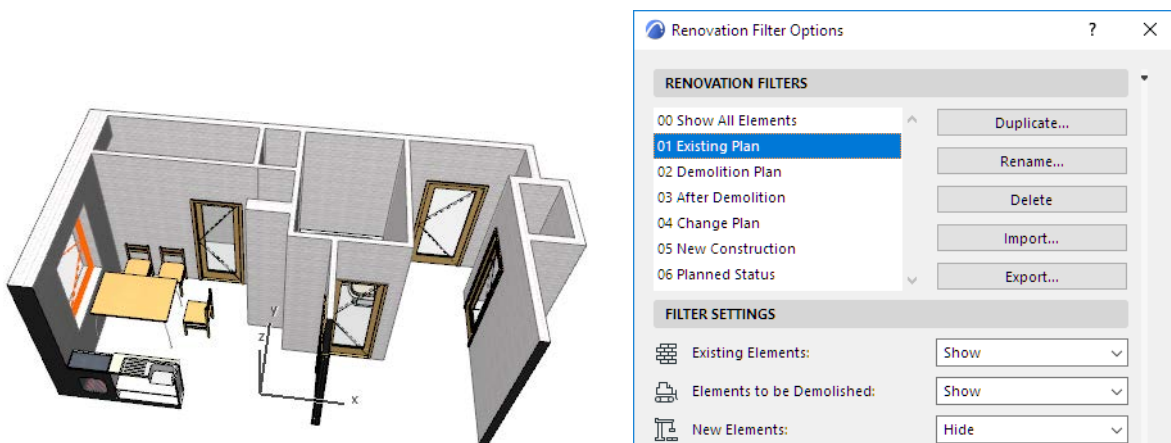
Examples of Renovation Filters

The following are examples of how different Renovation Filters can affect the display of your project.

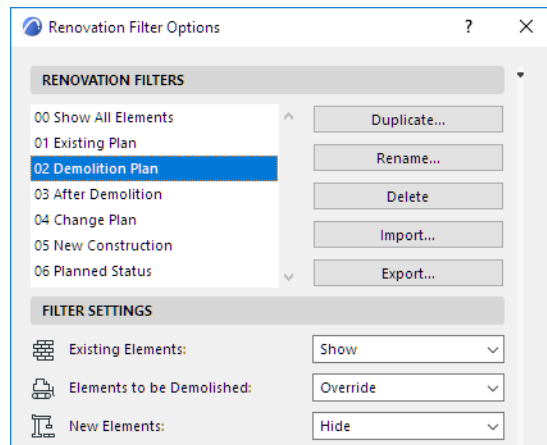
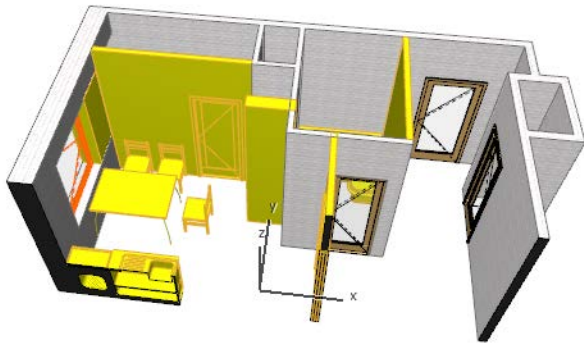
1. **Change Plan:** Here, existing elements are shown in their default display (“Show”), while new elements and elements to be demolished are highlighted with their respective Graphic Overrides (here, red and yellow).



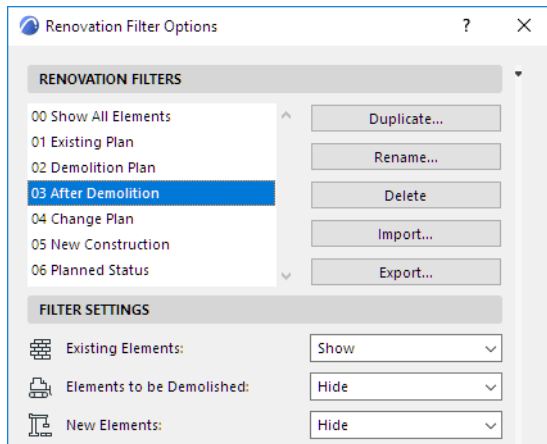
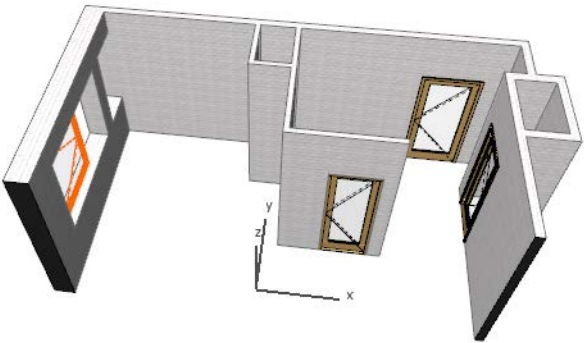
2. **Existing Plan:** In this filter, “New” elements are hidden. Other elements (with a status of either “Existing” or “To Be Demolished”) are all shown with their own settings. In this Existing Plan filter, you get no visual information on demolition plans.



3. **Demolition Plan:** Here, elements with a status of “To Be Demolished” are shown with Overrides, so you see at a glance what is to be demolished, and what is to remain. (“New” elements are not seen at all - the New Elements setting is “Hide”).



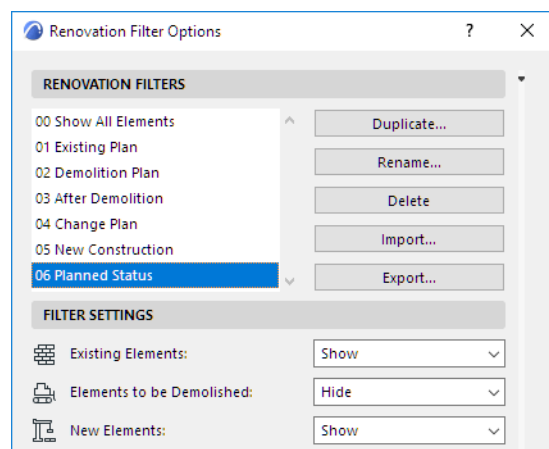
4. **After Demolition:** This is just like the Demolition Plan filter, except that the “To Be Demolished” elements are gone. Only Existing elements are shown.



One way to use this filter is to help determine the total quantities of new and demolished construction materials in the project.

See [Listing Material Quantities for Renovation Projects](#).

5. **Planned Status:** This filter is used to show the project’s final status. New elements and Existing elements are all shown with their default display (“Show”). Demolished elements are not shown at all (“Hide”).



Related topics:

[Graphic Overrides By Renovation Status](#)

[Openings By Renovation Status: Override Examples](#)

Graphic Overrides By Renovation Status

Archicad comes with predefined Override Styles for each of the three Renovation Statuses.

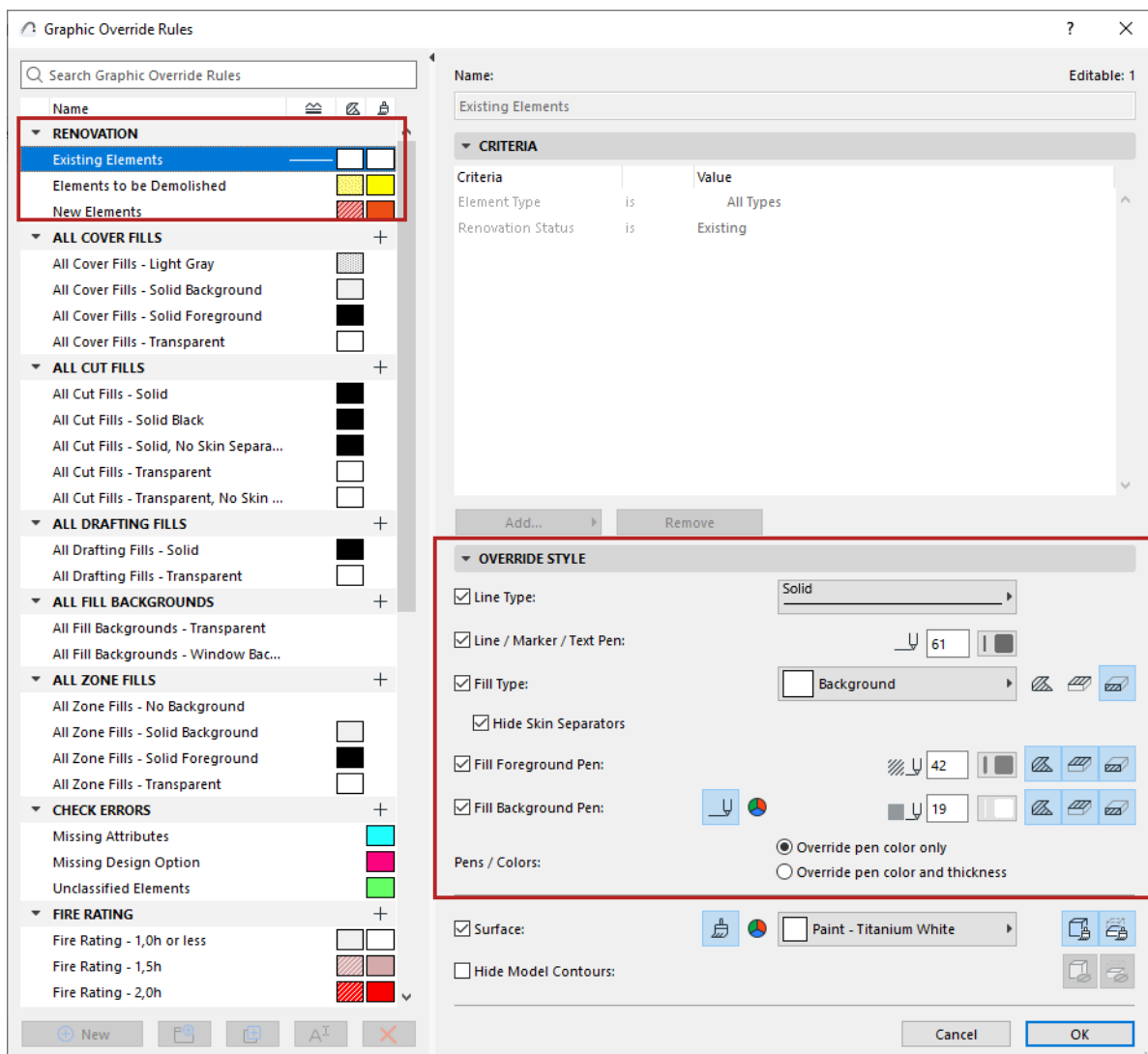
To change these default styles, use the **Graphic Override Rules** dialog box (**Document > Graphic Overrides > Graphic Override Rules**).

See [Graphic Override Rules](#).

Note: The Graphic Override Rules dialog box is also available from the Renovation Filter Options dialog box (click the “Graphic Override Rules” button).

The first three Rules (at the top left of the dialog box) correspond to the three Renovation Statuses: New, Existing, and To be Demolished.

Select the Rule for which you want to redefine the override style.



For Renovation-based rules, the “Criteria” panel is not editable. (Renovation Status is assigned to model elements individually, rather than by criteria.)

Use the **Override Style** panel to adjust the style settings for each Renovation Rule:

- Line type

- Line/Marker/Text pen
- Fill type
- Fill foreground pen
- Fill background pen
- Surface

For details, see [Define Override Style](#).

See also [Exceptions: When Overrides Don't Apply](#).

In addition to the override settings listed here, you may wish to include one or more additional override settings in your Renovation Filter. These can be chosen in the **Renovation Filter Options** dialog box.

See [Add Additional Overrides to the Renovation Filter](#).

[Watch the Video](#)

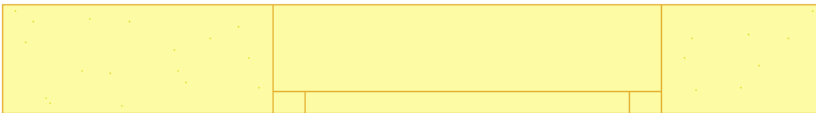
Openings By Renovation Status: Override Examples

Examples of default display of walls with windows shown with Override:

NEW wall in red (-> window is also red for NEW, no demolition)



TO BE DEMOLISHED wall in yellow (-> window is also yellow for TO BE DEMOLISHED)



Window MOVED from right to left

To communicate a modified element with the Renovation feature - such as a window that is moved - you must model the window twice: in its original location and in its new location.

Select the original window and set its Renovation Status to To be Demolished. Note that the window frame is yellow (it will be demolished), but the wall beneath it is red (it will be rebuilt).

Select the window in its new location and set its Renovation Status to New. Note that the window frame is red (it will be built as new), but the wall beneath it is yellow (it will be demolished to make room for the window).

Archicad automatically displays any overlap (between the window's old locations) in white.



- On the right: the yellow window frame must be demolished. The red part - formerly the window opening - is new wall.
- On the left: the yellow wall part must be demolished to make room for the opening. The red frame is the new frame.
- In the middle: the white part was part of the opening before, and remains part of the opening after.

Best Practices for Creating Renovation Views

Save Filter Options with Project Template

Renovation Filter Options are stored with the project. Save the project as a template file, then use that template when creating new projects to ensure that your preferred renovation options are available by default.

Export Configured Filter Options to Another Computer

From the Renovation Filter Options dialog box, export your preferred settings as an XML file. You can then import this file to be able to use your Renovation Filter Options on another computer.

Building Modeling

1. While you are working in Existing Plan view, keep the Renovation Status setting on “Existing”. This way, all new elements are classified as Existing.
2. Next, switch the Renovation Filter to “New Construction” or “To be Demolished”. Select the elements to be demolished, then use the Palette icons to switch their Renovation Status from Existing to “To Be Demolished.”
3. Finally, before you start modeling the new elements, make sure you are in the “New Construction” Renovation Filter. Switch the default Renovation Status to “New”. All new elements you create now get a Renovation Status of “New.”

Workflows for Tricky Renovation Situations

Archicad's Renovation feature depends on assigning the proper Renovation Style to individual elements, and then displaying them in the different filters corresponding to the different stages of the renovation process. In most cases, once you have assigned element styles and defined filter options, you can see the graphical feedback just by clicking from view to view - from Existing Plan to New Construction to Planned Status. The display of elements changes automatically.

However, some renovation processes involve element components - not just discrete elements - and so their style cannot be conveyed just by switching from one view to another. There are ways to get around this constraint: by recreating the needed elements separately in each view, and assigning their correct renovation status.

You can use the Renovation palette's "Show on current Renovation Filter Only" function to restrict the display of any element to a certain Filter - so that the element is visible only in a particular view filter.

See [Show an Element in a Single Renovation Filter Only](#).

You can set up views for Interactive Schedules that help you determine the total quantities of new building components to be acquired, and of demolished building materials to be carted away.

See the examples below:

[Adding Insulation to Walls: Renovation Workflow](#)

[Adding New Door Panel to Existing Door Frame: Renovation Workflow](#)

[Zone Stamps for Renovated Rooms: Renovation Workflow](#)

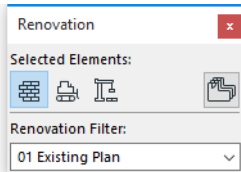
[Listing Material Quantities for Renovation Projects](#)

Adding Insulation to Walls: Renovation Workflow

If you want to add a layer of insulation to an existing wall, you simply add an insulation skin to the composite wall. However, you cannot assign “New” status to just a single component of a wall - only to the wall itself. To distinguish (for Renovation Filter purposes) the new insulation added to the original wall, the following workflow is recommended:

Existing Plan View

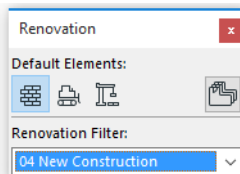
1. In the Renovation Palette, assign the original wall (Wall 1) a status of “Existing”.



New Construction View

In this view, you want to show both the Existing and the New components of the wall.

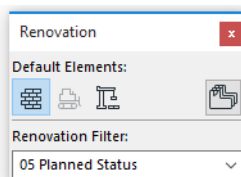
2. Alongside Wall 1, place another wall (Wall 2) alongside it to represent insulation. Define Wall 2 as “New”.



This way, only the new part of the Wall - the insulation part - is displayed as “New” in this view.

Planned Status View

3. Here, both Wall 1 and Wall 2 are shown with their own original attributes.



Note: Door and Window openings must be added to each separate Wall layer that is modeled in this workflow.

Adding New Door Panel to Existing Door Frame: Renovation Workflow

If you are adding a new door panel to an existing frame, the following workflow can be used for your Renovation filters:

Existing Plan View

In the Renovation Palette, assign the original frame (Door 1) a status of “Existing”, but set it to be shown on the current filter only (i.e. Existing Plan).

New Construction View

In this view, you want to show both the Existing frame and the New door panel to be added to it. In the place of the original door frame (which was displayed only in Existing Plan), place an identical door frame (Door 2) with a status of “Existing”.

To represent the new door panel, draw an opening line and assign this line a “New” status.

Both elements (Door 2 - “Existing” and the opening line - “New”) should be Filter-specific for the current filter. This way, only the new part of the door is displayed as “New” in this view.

Planned Status View

Here, place the final Door (Door 3), complete with frame and panel, with a status of “Existing”, and filter-specific for current (Planned Status) filter.

Zone Stamps for Renovated Rooms: Renovation Workflow

This workflow is used to reflect the renovation process in Zone Stamps.

Suppose you have a zone that represents a closet, and it is to be transformed into a bathroom.

In Existing Plan View (the starting point) and in Planned Status view (the ending point), place Zone Stamps accordingly:

- The first Zone Stamp, in Existing Plan, has a status of “Existing” and is filter-specific. (It shows data for the closet.)
- The final Zone Stamp, in Planned Status view, has a status of “New” and is filter-specific. (It shows data for the bathroom.)

There is no way to use a single Zone Stamp to express two renovation styles (demolish the closet, build a new bathroom), so in the interim New Construction view, you should use two text boxes instead of a Zone Stamp:

- The first text box shows the closet’s data and has a status of “Demolish.”
- The second text box shows the bathroom’s data and has a status of “New.”

This way, in this view only, the first text box, for the item to be demolished, will be displayed in yellow (the override style for “Demolish.”). Below it, the second text box, for the item to be built, will be displayed in red (the override style for “New”).

Listing Material Quantities for Renovation Projects

This workflow explains how to set up views for Interactive Schedules that help you determine the total quantities of demolished building materials to be carted away, and of new building components to be acquired.

Note: The Renovation Filters cited below refer to filters whose settings are described at: [Examples of Renovation Filters](#).

List Quantities of Demolished Materials

1. Use the Interactive Schedule to set up two separate Component Lists. Both Component Lists should use identical Scheme Settings, listing the parameters you need for the relevant construction elements (e.g. Walls, Slabs, Columns, Roofs).

Note: Component Lists are recommended here (rather than Element lists), because they allow you to disregard non-material components such as air space in the calculation. Component Lists also lets you categorize demolished components (e.g. recyclable, hazardous waste, etc.)

2. Save these Component Lists as views.
3. In one view, apply the **Existing Plan** Renovation Filter.
4. In the other view, apply the **After Demolition** filter.
5. Compare the two lists to determine the differences in material quantities.

For Doors/Windows and other non-construction elements:

1. Use the Interactive Schedule to set up two separate Element Lists using identical Scheme Settings, listing the parameters you need for Doors and Windows. (You cannot make Component lists for Doors and Windows.)
2. In one view, apply the **Existing Plan** Renovation Filter.
3. In the other view, apply the **After Demolition** filter.
4. Compare the two lists to determine the differences in quantities.

List Quantities of New Materials:

This workflow is almost the same as the one detailed above, except that you will use different Renovation Filters in the views to be compared:

- In one view, apply the **After Demolition** Renovation Filter.
- In the other view, apply the **Planned Status** filter.

Compare the two lists to determine the differences in material quantities.

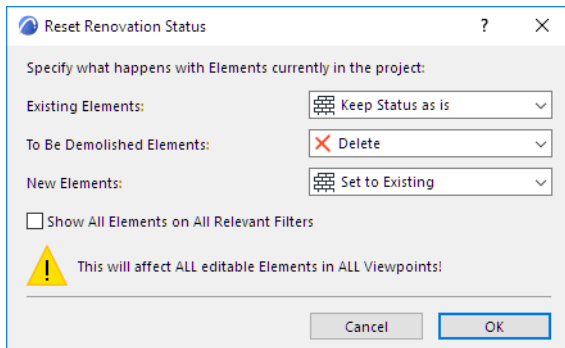
Reset Renovation Status

When a renovation project is finished, you will need to update all elements' renovation status: for example, those that were “To be Demolished” can now be deleted; those that were “New” are now “Existing.”

The reset will be applied to all elements in all views, including your annotations.

To do this, use **Document > Renovation > Reset Renovation Status**.

In the appearing dialog box, use the pop-ups to redefine element status for each of the three Renovation Statuses (or keep them as is).



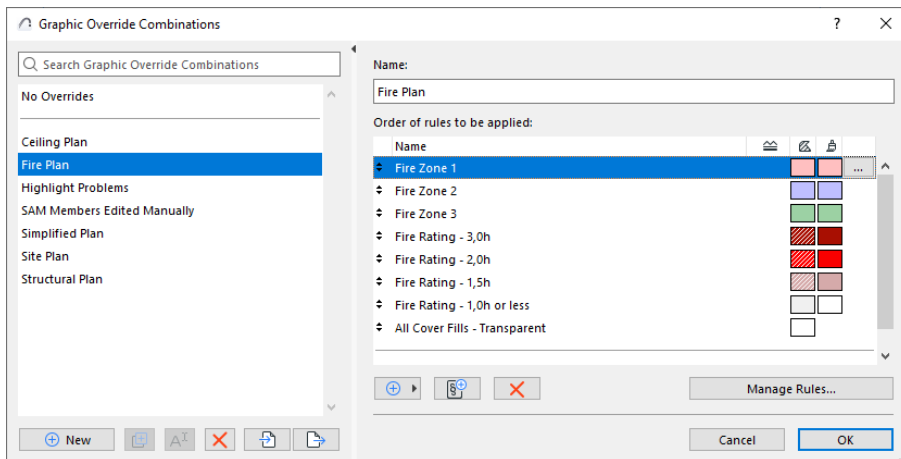
Note: To ensure that all of your elements have their status redefined, make sure you check the “Show All Elements on All Relevant Filters” box. Elements that remain pinned to a particular filter (“Show on specific Renovation Filter only”) cannot be reset to a status that is not visible on that specific filter.

Graphic Overrides

Use Graphic Overrides to apply a predefined appearance (color, fill) to model elements by view, to convey information at a glance.

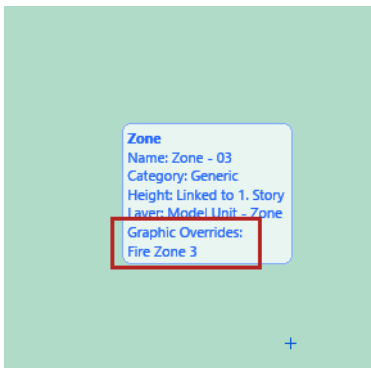
For example:

- Display elements in different colors based on their Renovation Status
- Display all plaster walls in blue
- Highlight a group of zones which belong together (e.g. unheated areas)
- Highlight all elements having a specific acoustic rating or fire rating



Zones by Fire Rating

The element's Info Tag lists the Graphic Override Rules in effect for this element:



Apply Graphic Override Rules

There are two ways to assign a Graphic Override:

- **By Graphic Override Combination:** A Graphic Override Combination is a collection of Graphic Override Rules, applied in a particular order to the elements that match the rule's criteria.
- **By Renovation Filter:** The Renovation Filter defines which elements (by Renovation Status) should be displayed with an override

Graphic Override Combination and Renovation Filter are saved with each view's settings. For any view, the rules of the Graphic Override Combination are applied first. The rules applied by the Renovation Filter are the lowest priority.

[Watch the Video](#)

Related Topics:

[Graphic Override Rules](#)

[Graphic Override Combinations](#)

[Exceptions: When Overrides Don't Apply](#)

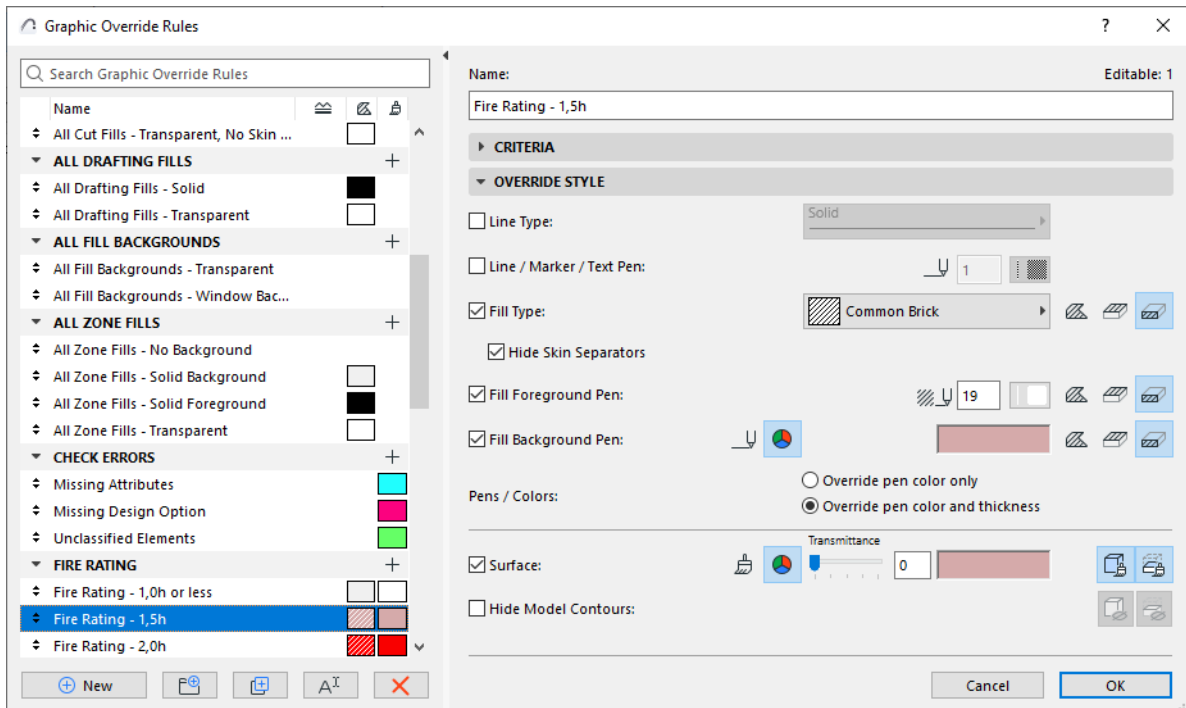
[Surface Criteria in Graphic Override Rules and Collision Detection](#)

Graphic Override Rules

Use this dialog to create and edit rules.

To open it, do one of the following:

- Document > Graphic Overrides > Graphic Override Rules
- in Graphic Override Combinations dialog box: click **Manage Rules**
- in Renovation Filter Options dialog box: click **Graphic Override Rules**



[Watch the Video](#)

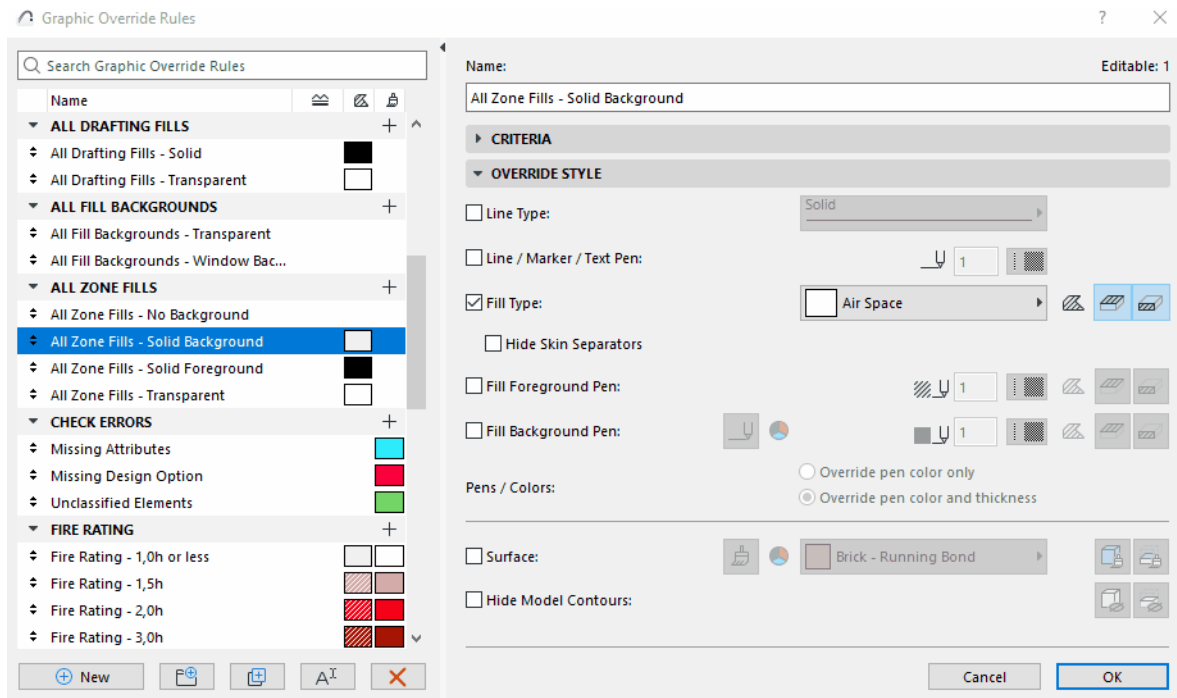
Rules are listed on the left side of the dialog box. Colorful icons indicate the Line Type, Fill Type, Foreground and Background Pens, and the Surface (if any) defined for each rule.

The first three rules correspond to the three Renovation Statuses: Existing, To be Demolished, and New.

Select any rule to see its **Criteria** and **Override Style** panels on the right.

Add New Rule

1. Click **New**
 - from the bottom of the Graphic Override Rules dialog box
 - from the context menu of a Rule or Rule Group

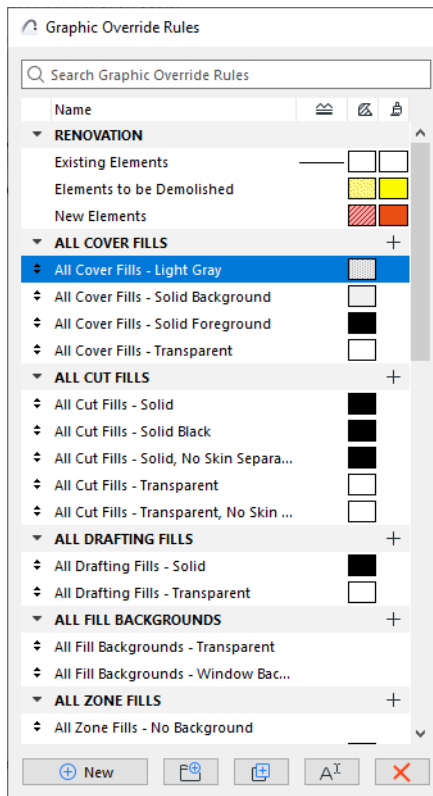


Note: The rules for the three Renovation Statuses cannot be duplicated.

2. Use the panels on the right side of the dialog to define rule's **Criteria** and **Override Style**.

Organize Rules in groups

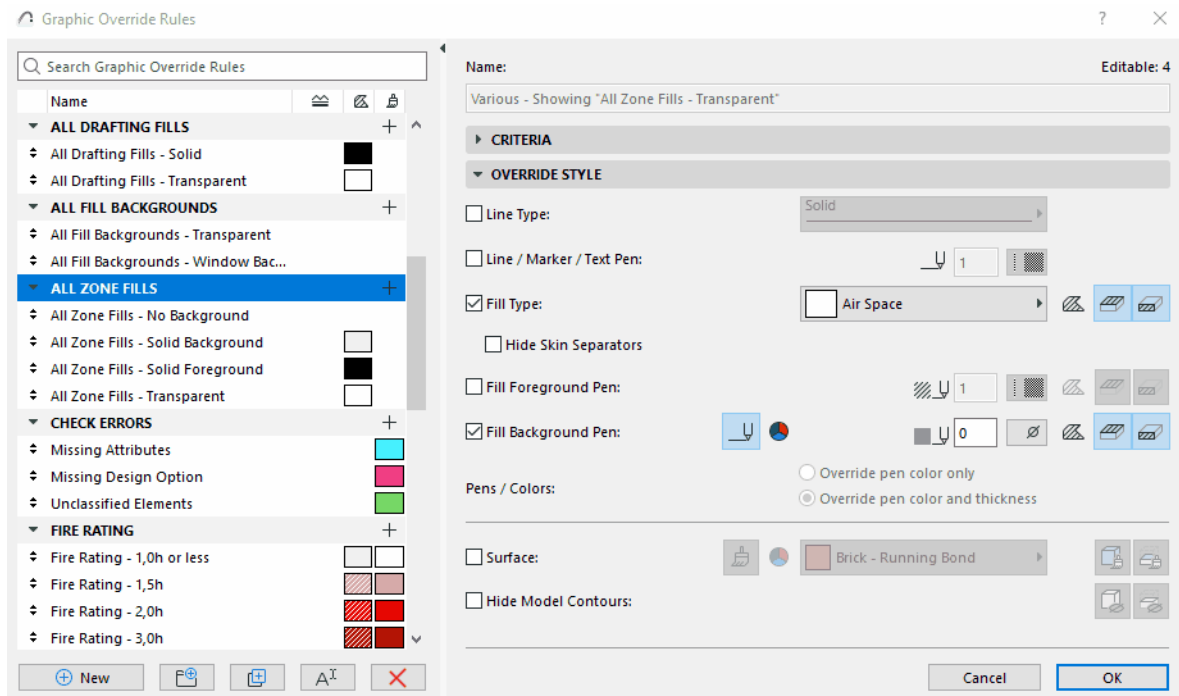
- Even lots of rules can be easily organized, sorted and viewed
- You can move rules from one group to the other by drag and drop.
- Context menus are there when you need them
- Search makes it easy to find any rule or group by their names.



Add new Rule Group

Click New Group

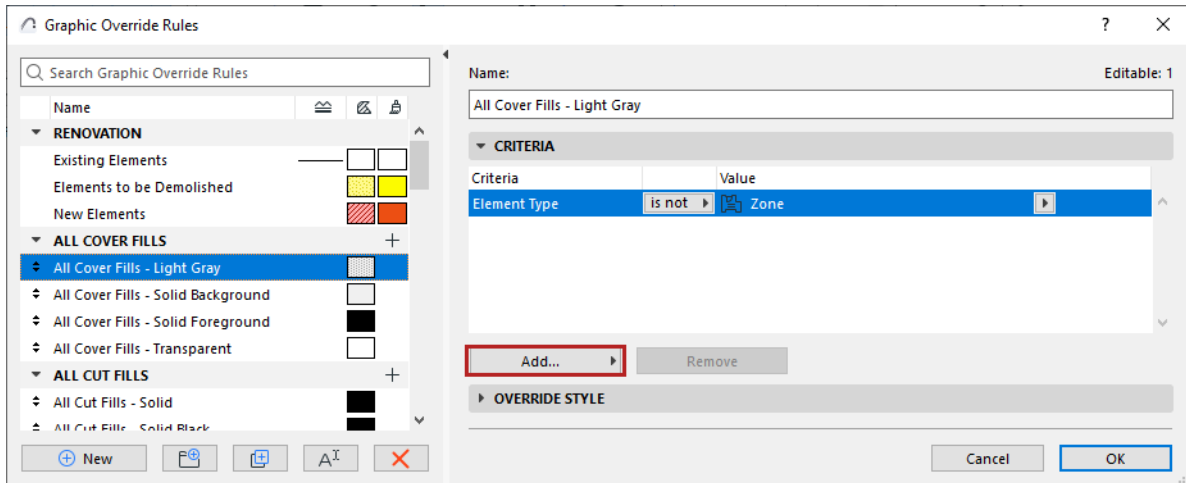
- from the bottom of the Graphic Override Rules dialog box
- from the context menu of a Rule Group



Define Rule Criteria

These criteria define which elements should get the override.

Note: Criteria for the three Renovation Statuses cannot be edited. (Renovation Status is assigned to model elements individually.)



1. In the **Criteria** list, the first line is always Element Type. (Sub-elements, such as Curtain Wall Panel or Morph face, are not available here.)
2. Select this first line, then click the black arrow at the end of the row to choose the element type(s) to which this rule should apply.
3. To define additional criteria, click the **Add...** button to bring up a list of Parameters & Properties.
4. Select one or more parameters (use the search field at the top to find it more quickly), then click **Add** (or press Enter) to add them to the list.
5. Define additional criteria. The list of available criteria depends on the element(s) defined in the row(s) above.

Note: This interface is similar to that of the Find and Select dialog box, but IFC Properties are not available here.

See [Define Additional Criteria: Parameters & Properties](#).

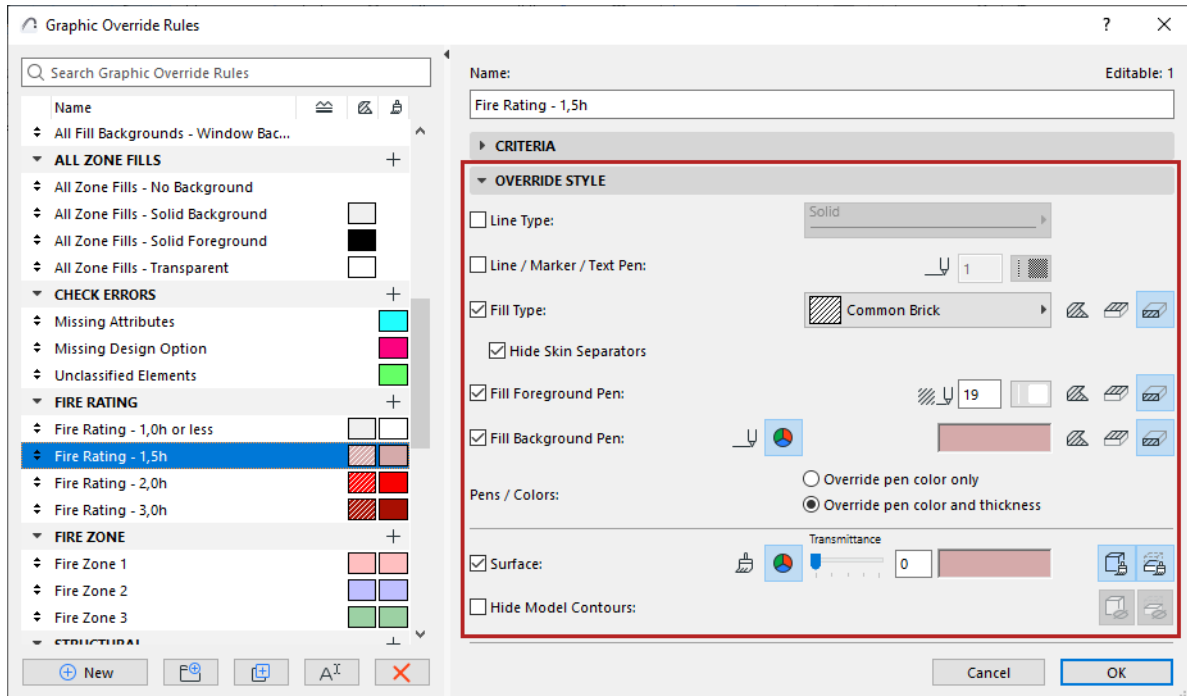
Note: Defining a “Layer” or “Layer Combination” as a criterion means that the override affects elements that are *visible* in the given Layer Combination.

[Watch the Video](#)

[Watch the Video](#)

Define Override Style

Override Style defines what the override should look like.



Check the boxes of the attributes you want use in this Graphic Override.

- **For Fill-related controls (Fill Type, Fill Pen, Fill Background Pen):**

- Click one or more icons to choose which category (Drafting Fill, Cover Fill, Cut Fill) should be affected by the override.
- Use either the pen number or color picker control to choose a color.

Notes:

- Cover Fills include Fill elements in the “Surface (Cover Fill)” category.
- Cut Fills include Fill elements in the “Building Material (Cut Fill)” category.

- **For Surface:** Click one or both icons to apply the override to Cut and/or Uncut Surfaces.

Note: In addition to the override settings listed here, you can include additional override settings in your Renovation Filter. Set these in the **Renovation Filter Options** dialog box.

[Watch the Video](#)

Related Topics

[Assign Renovation Status to Elements](#)

[Graphic Override Combinations](#)

[Renovation Filter Options](#)

[Apply Renovation Filter](#)

[Surface Criteria in Graphic Override Rules and Collision Detection](#)

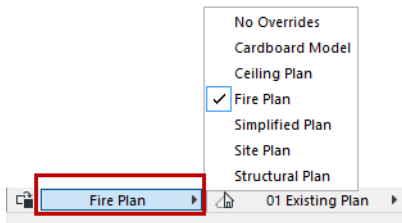
Graphic Override Combinations

A Graphic Override Combination is a collection of Graphic Override Rules, applied to the elements of the model view.

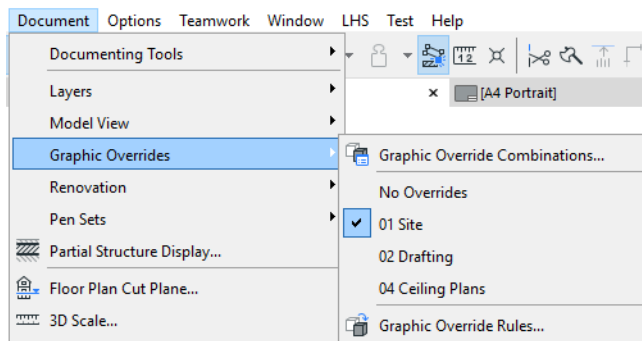
Apply Graphic Override Combination

To apply a Combination to the active view, select it from one of the following:

- **Quick Options pop-up** (at the bottom of the window)



- **Graphic Overrides Combination** dialog
- **Document > Graphic Overrides** menu

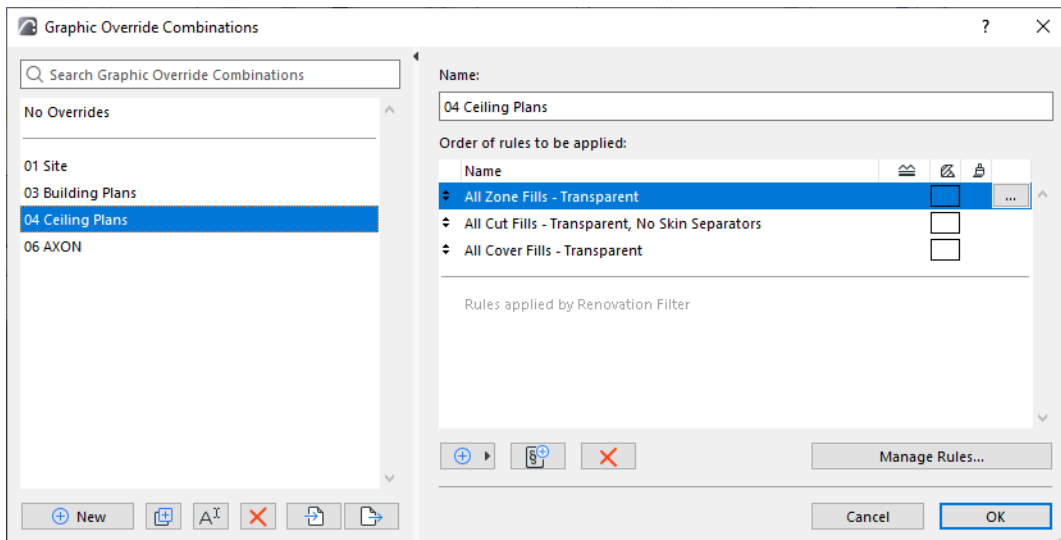


Graphic Override Combinations Dialog Box

Use this dialog to create named combinations of Graphic Override Rules.

To open it, do one of the following:

- Document > Graphic Overrides > Graphic Override Combinations
- Open from Quick Options



[Watch the Video](#)

The left side lists the project's defined Graphic Override Combinations.

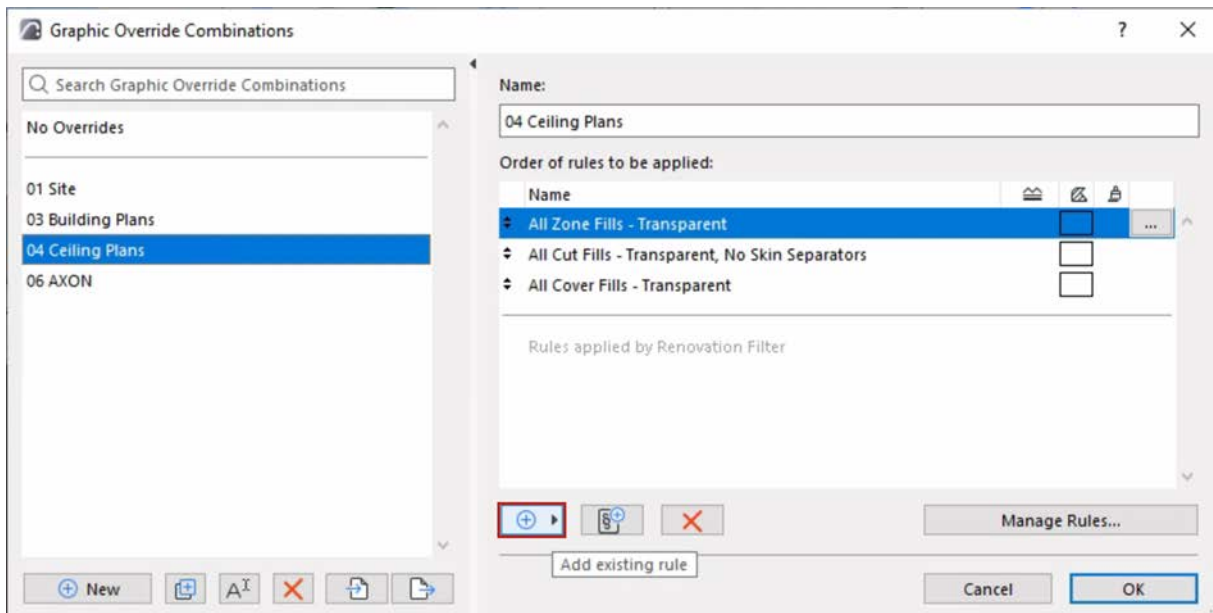
The top Combination, **No Overrides**, is predefined and not editable.

Click **New...** to create a new or duplicate Combination.

Add Existing Rule

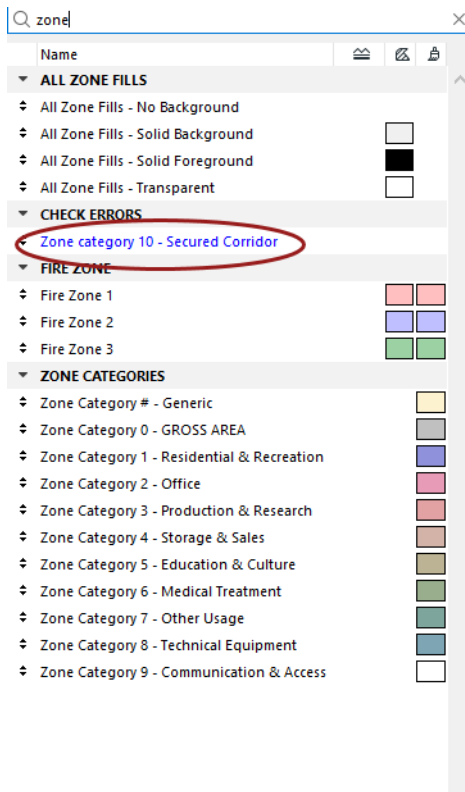
Choose from the list of defined Graphic Override Rules to add them to the current Combination:

1. Click **Add existing rule**



2. From the appearing list, search to find and select the rules you need.

If the name of the rule is in blue, this means it is a new you just created (in the Rules dialog, while the Graphic Overrides Combinations dialog is open), or just imported as part of a Combination - so it is easier to find.



3. To add selected rules to current Combination, do one of the following:

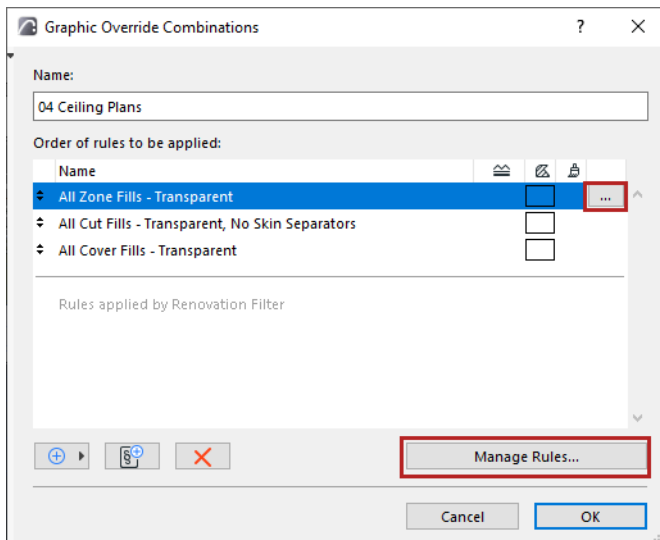
- click Enter
- click **Add** at the bottom of the list
- on Mac: double-click

Manage Rules

To manage and edit rules, you need the Graphic Override Rules dialog.

To open this from the Combination dialog, do one of the following:

- Double-click on a rule
- Select the Rule, then click **Edit Rule** at the end of the row
- Click **Manage Rules** at the bottom of the dialog box

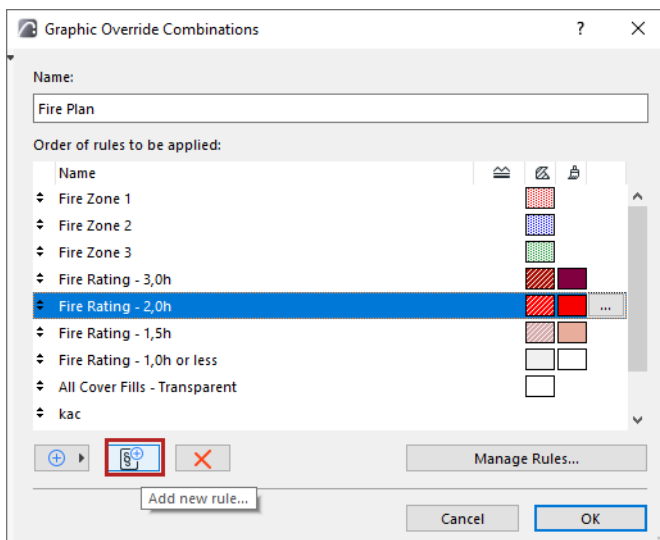


See [Graphic Override Rules](#).

Add New Rule

This lets you add a new or duplicate rule to the current Combination.

1. Click the **Add new rule** button



2. From the appearing dialog, choose New or Duplicate, and give the rule a name.
3. The new rule is added to the current Combination. Set up its criteria now or later.

For details, see [Graphic Override Rules](#).

Order of rules

Rules in a Graphic Override Combination are applied in the order that they are listed.

If two rules both apply, but can't be applied simultaneously, then the rule that is higher up applies.

- For example: all plaster walls are blue, and all elements with a Fire Rating of 30 min are red. A wall that fits both of these rules will get the override from the rule that is higher up in the list.

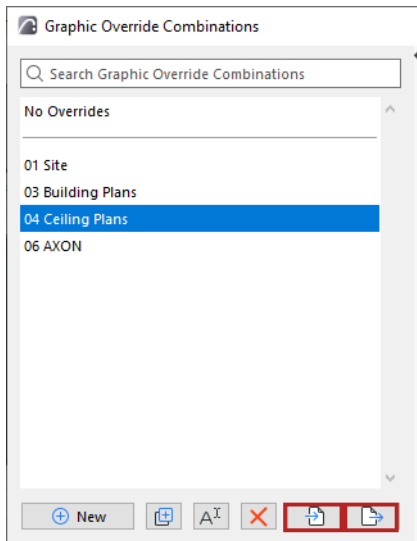
To change the order, drag a rule up and down using the arrows the left edge of its row in the list.

Rules applied by Renovation Filter are at the bottom of the list (lowest priority), and this position cannot be changed.

Export/Import Graphic Override Combination

Use Graphic Override Combinations in other Archicad projects.

1. Select one or more Graphic Override Combinations
2. Use the buttons at the bottom of the Combinations dialog to Export or Import selected Combinations in XML format



Merge Rules of Imported Combination

At import: the incoming Combination and all of the rules it contains are merged with those of the current project.

If an imported rule has the same name as a rule already in the project:

- If the two rules' contents are exactly the same, no duplicate is created
- If the two rules' contents differ, then the imported rule is merged to the project, and its name gets a new suffix.

Exceptions: When Overrides Don't Apply

Line Type override does not apply to:

- Dimension Lines
- Column Symbol lines
- X Symbol used for Openings
- Frames of Drawings, Text boxes and Labels
- Fill handles, Spline handles

Fill Type override does not apply to:

- Opaque fill of Texts and Labels
- Figures, Drawings, PDFs

Fill Background Pen override does not apply to:

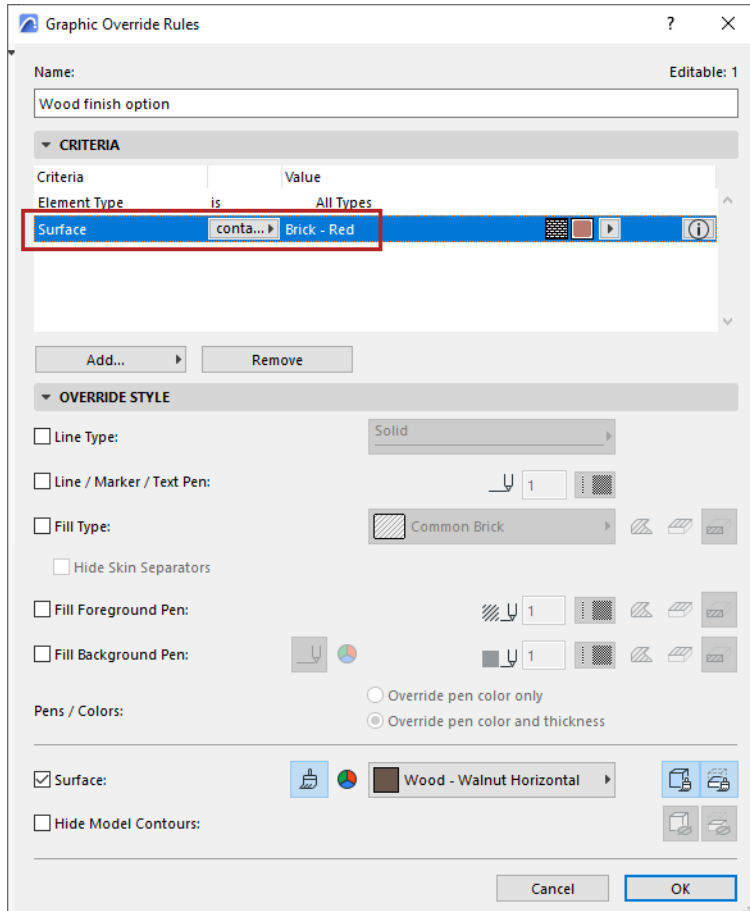
- Transparent (0) background pens

Related topic

[Surface Criteria in Graphic Override Rules and Collision Detection](#)

Surface Criteria in Graphic Override Rules and Collision Detection

Please be aware that the Graphic Override and Collision Detection functions recognize surfaces in a limited manner.



Surface criteria in these interfaces (Surface, Surface Name) might or might not apply, depending on the type of element and how the surface was assigned.

Please see the following sections to see how it works:

Construction elements: Criteria apply if surface was defined in element settings



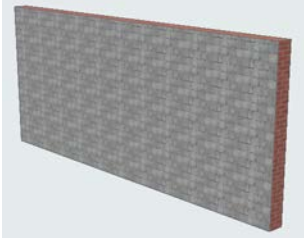

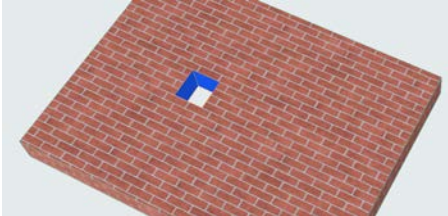

Library parts: Criteria apply only if surface comes from Model Panel override

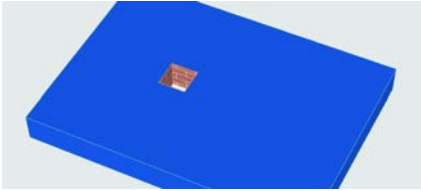
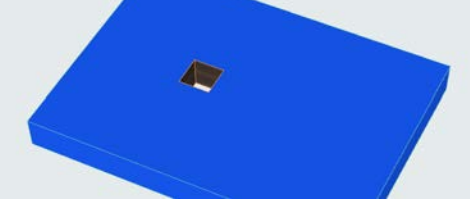
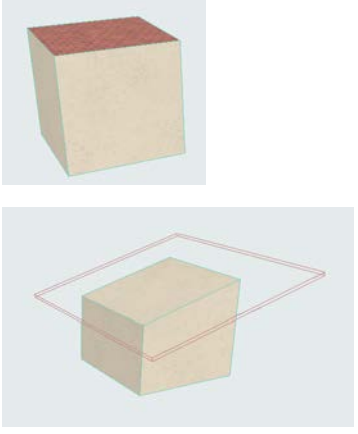
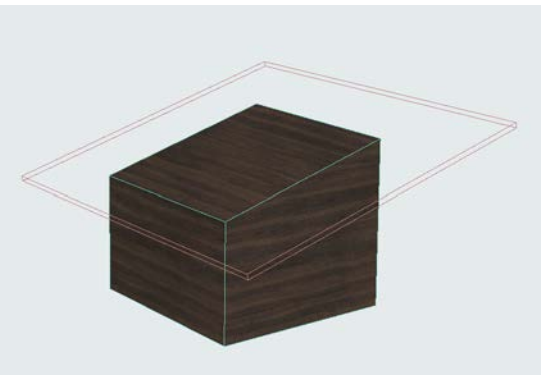
Surfaces resulting from 3D model edits: Criteria do NOT apply

Construction elements: Criteria apply if surface was defined in element settings

For construction elements, if a surface is defined in element settings, then the rule applies.

For example, a graphic override applies a Walnut finish to all elements containing a red brick surface.


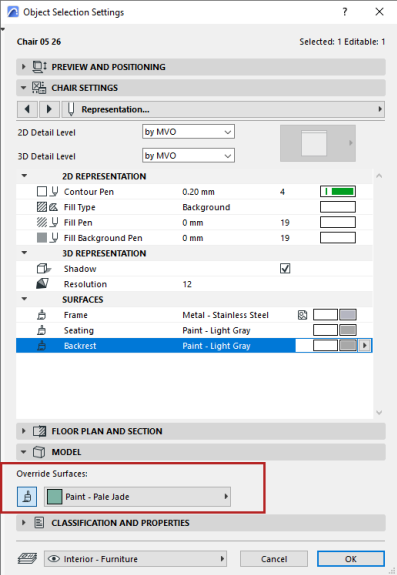
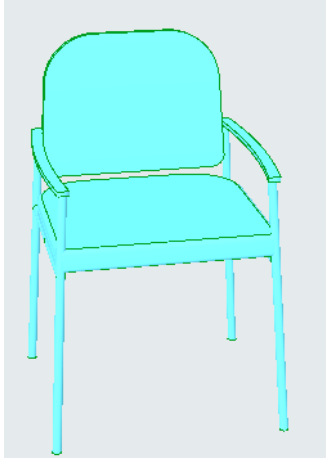
Original	Override Brick surface with Walnut
	
<p>Column segment with red brick building material</p>	<p>“Walnut” override applied to entire column, based on red brick surface assigned to segment in element settings (via Building Material)</p>
	
<p>Wall with brick edge surface</p>	<p>“Walnut” override applied to entire element, based on brick surface assigned in element settings (via Edge override in Model panel)</p>
	
<p>Red brick slab with blue Opening element</p>	<p>“Walnut” override applied to slab element (but not the Opening), based on brick surface assigned in Slab settings</p>

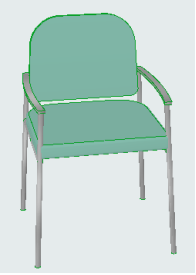
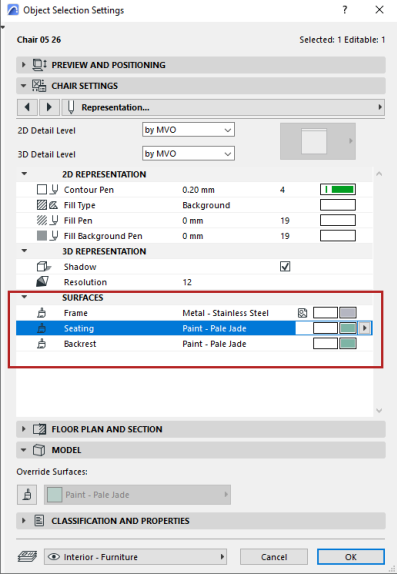
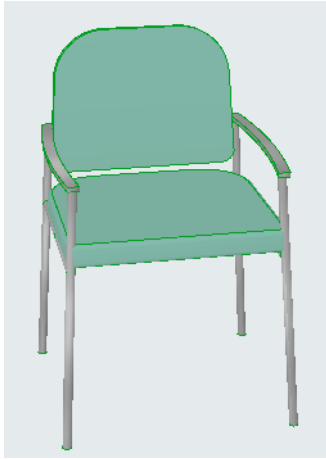
Original	Override Brick surface with Walnut
	
<p><i>Blue slab, plus Opening element with brick surface</i></p>	<p><i>“Walnut” override applied to Opening surface (but not the slab), based on brick surface assigned in Opening settings (via Model panel)</i></p>
	
<p><i>Slab with brick top surface (via Model panel override)</i> <i>Top surface cut off by roof</i></p>	<p><i>“Walnut” override applied to entire element, based on brick surface in element settings (even though the brick surface is not visible in the model)</i></p>

Library parts: Criteria apply only if surface comes from Model Panel override

For library parts, objects, GDL objects:

- Graphic Overrides and Collision Detection apply if the original surface was applied in the **Model Panel of Object Settings** (this is an Archicad parameter available on every Archicad output that is based on property data).
- In contrast: If the original surface was defined only in the Custom Settings panel of the library part (this is the panel that sets the GDL script, which is NOT an Archicad parameter), then Graphic Overrides and Collision Detection **will not apply** the surface criteria.

Original	Override green surfaces with blue
 	
<p><i>Chair's green surface assigned in Model panel</i></p>	<p><i>Surface-based Graphic Override applied</i></p>

Original	Override green surfaces with blue
 	
<p><i>Chair's green seat surface assigned in Custom Settings Panel (GDL script)</i></p>	<p><i>Surface-based Graphic Override does not apply</i></p>

Surfaces resulting from 3D model edits: Criteria do NOT apply

Construction elements: if surface is "inherited" due to model editing, the resulting surfaces are NOT considered for Graphic Overrides. For example, an element's surface might derive from

- A solid operation (target element inherits the operator surface)
- Connection with other elements
- Elements cut into them (like openings, doors or windows)

The resulting surfaces will NOT be considered, and the element will not get a Graphic Override or participate in Collision Detection.

Element Listing Parameters

The following sections list the general and tool-specific parameters available for the fields of Element Schedules.

- [General Listing Parameters](#)
- [Beam Listing Parameters](#)
- [Beam Segment Listing Parameters](#)
- [Column Listing Parameters](#)
- [Column Segment Listing Parameters](#)
- [Curtain Wall Listing Parameters](#)
- [Curtain Wall Accessory Listing Parameters](#)
- [Curtain Wall Frame Listing Parameters](#)
- [Curtain Wall Junction Listing Parameters](#)
- [Curtain Wall Panel Listing Parameters](#)
- [Window/Door Listing Parameters](#)
- [Fill Listing Parameters](#)
- [Lamp Listing Parameters](#)
- [MEP Listing Parameters](#)
- [Mesh Listing Parameters](#)
- [Morph Listing Parameters](#)
- [Object/Lamp Listing Parameters](#)
- [Opening Listing Parameters](#)
- [Railing Listing Parameters](#)
- [Railing Segment Listing Parameters](#)
- [Railing Members Listing Parameters](#)
- [Shell Listing Parameters](#)
- [Skylight Listing Parameters](#)
- [Roof Listing Parameters](#)
- [Shell Listing Parameters](#)
- [Skylight Listing Parameters](#)
- [Slab Listing Parameters](#)
- [Stair Listing Parameters](#)
- [Stair Finish Listing Parameters](#)
- [Stair Structure Listing Parameters](#)
- [Wall Listing Parameters](#)
- [Zone Listing Parameters](#)
- [Component Listing Parameters in the Interactive Schedule](#)

Related Topics

- [Interactive Schedule](#)
- [Component Data in Archicad](#)
- [Surface Schedule](#)

General Listing Parameters

Parameter	Short Description	Long Description
2D Cross Section Preview		
2D Plan Preview		
3D Axonometry		
3D Back View	Shaded 3D Symbol of the GDL element (Object, Lamp, Skylight, CW Panel, CW Frame).	3D Symbol of the Frame. Back = inside of the Curtain Wall
3D Front View	Shaded 3D Symbol of the GDL element (Object, Lamp, Skylight, CW Panel, CW Frame).	3D Symbol of the Frame (from the front, that is, the outside of the Curtain Wall). Front = outside of the Curtain Wall
3D Holes Perimeter	Total Perimeter of all the holes in the Shell or Roof. Does not take SEO's into account.	Relevant for Roof or Shell.
3D Left Side View	Shaded 3D Symbol of the Object/Lamp or CW Junction/Accessory.	Cross-section of the Accessory viewed from its left side.
3D Length	Maximum 3D length of construction element, measured along its reference axis.	Available for - Wall - Beam/Beam Segment See also Beam-specific parameters: Length Left and Length Right - Column/Column Segment - Railing/Railing Segment - Stair Structure
3D Perimeter	Total perimeter of the 3D shape of the element, including the perimeter of any holes.	Relevant for Roof, Shell, CW Panel
3D Right Side View	Shaded 3D Symbol of the Object/Lamp or CW Junction/Accessory.	Cross-section of the Accessory viewed from its right side
Absolute Top Link Story	Name of the element's top-linked story.	Relevant for Wall/Column/Zone/Stair For example: 1. Story
Archicad IFC ID	The Globalid IFC attribute automatically assigned by Archicad to each element	

Parameter	Short Description	Long Description
Area	For most elements: area includes the full extent of the element surface geometry (net of any holes) viewed from the top, as projected down to the floor plan.	For a Column (or column segment): Area of the column's bottom polygon only, as projected to the floor plan. For a Wall: Area of the Wall's bottom polygon only, as projected to the floor plan.
Building Material	Name of the element's Building Material	For non-composite, non-complex profile elements
Building Material / Composite / Profile / Fill	Name of the element's Building Material, or its composite structure, or its profile, or its fill	
Building Materials (All)	Lists all of the element's Building Materials in a single Schedule cell	
Colliding Zones	The name of the Zone with which the element collides in 3D.	Elements must intersect the Zone to be considered as "Colliding". Bordering the Zone is not considered collision. For Columns: If its Relation to Zones option is set to "Zone Boundary" or "No Effect on Zones", it is not considered collision, even if it is located in the Zone.
Complex Profile	For a complex profile element: Name of the profile	
Composite Structure	For a composite element: Name of the composite attribute	
Cover Fill	Name of the Cover Fill assigned to the element (if any).	Cover Fills are available for Zone, Mesh, Slab, Roof, Column, Beam, Morph, Opening.
Cross Section Area at Bottom/Start	Area of element cross section at its bottom (Column) or start (Beam).	Relevant for Beams and Columns and their segments For Columns: Includes the area of the Veneer, if any. For tapered Beam: Define "start" and "end" in Beam Settings (Segment panel). For tapered Column: Define "top" and "bottom" in Column Settings (Segment panel).
Cross Section Area at Top/End	Area of element cross section at its top (Column) or end (Beam).	Relevant for Beams and Columns and their segments For Columns: Includes the area of the Veneer, if any. For tapered Beam: Define "start" and "end" are defined in Beam Settings (Segment panel). For tapered Column: Define "top" and "bottom" in Column Settings (Segment panel).

Parameter	Short Description	Long Description
Cross Section Height at Bottom/Start (cut)	Height of element cross section at its bottom (Column) or start (Beam). Cross section is based on cut geometry.	Relevant for Beams and Columns and their segments. For tapered Beam: Define "start" and "end" in Beam Settings (Segment panel). For tapered Column: Define "top" and "bottom" in Column Settings (Segment panel)
Cross Section Height at Bottom/Start (perpendicular)	Height of element cross section at its bottom (Column) or start (Beam). Cross section is measured perpendicular to element axis.	Relevant for Beams and Columns and their segments. For tapered Beam: Define "start" and "end" in Beam Settings (Segment panel). For tapered Column: Define "top" and "bottom" in Column Settings (Segment panel).
Cross Section Height at Top/End (cut)	Height of element cross section at its top (Column) or end (Beam). Cross section is based on cut geometry.	Relevant for Beams and Columns and their segments. For tapered Beam: Define "start" and "end" in Beam Settings (Segment panel). For tapered Column: Define "top" and "bottom" in Column Settings (Segment panel).
Cross Section Height at Top/End (perpendicular)	Height of element cross section at its top (Column) or end (Beam). Cross section is measured perpendicular to element axis.	Relevant for Beams and Columns and their segments. For tapered Beam: Define "start" and "end" in Beam Settings (Segment panel). For tapered Column: Define "top" and "bottom" in Column Settings (Segment panel).
Cross Section Width at Bottom/Start (cut)	Width of element cross section at its bottom (Column) or start (Beam). Cross section is based on cut geometry.	Relevant for Beams and Columns and their segments. For tapered Beam: Define "start" and "end" in Beam Settings (Segment panel). For tapered Column: Define "top" and "bottom" in Column Settings (Segment panel).
Cross Section Width at Bottom/Start (perpendicular)	Width of element cross section at its bottom (Column) or start (Beam). Cross section is measured perpendicular to element axis.	Relevant for Beams and Columns and their segments. For tapered Beam: Define "start" and "end" in Beam Settings (Segment panel). For tapered Column: Define "top" and "bottom" in Column Settings (Segment panel).
Cross Section Width at Top/End (cut)	Area of element cross section at its top (Column) or end (Beam). Cross section is based on cut geometry.	Relevant for Beams and Columns and their segments. For tapered Beam: Define "start" and "end" in Beam Settings (Segment panel). For tapered Column: Define "top" and "bottom" in Column Settings (Segment panel).
Cross Section Width at Top/End (perpendicular)	Area of element cross section at its top (Column) or end (Beam). Cross section is measured perpendicular to element axis.	Relevant for Beams and Columns and their segments. For tapered Beam: Define "start" and "end" in Beam Settings (Segment panel). For tapered Column: Define "top" and "bottom" in Column Settings (Segment panel).

Parameter	Short Description	Long Description
Element ID	Lists Element ID.	For each element: editable ID field is located on the Classification and Properties panel of construction tools, Fill and Zone tools.
Element Type	Type of AC element	
Elevation to 1st/2nd Reference Level	Element's elevation relative to the project's 1st and/or 2nd Elevation Level.	Elevation levels are defined at Options > Project Preferences > Reference Levels. Element's elevation value (measured from its reference line/point) is displayed in its Info box or Settings.
Elevation to Linked/Home Story	Element's elevation relative to its linked story (Door/Window) or Home Story (other elements).	Relevant for Doors/Windows. Value is the door/window elevation from its linked (anchor) story to its sill height.
Elevation to Project Zero	Element's absolute elevation, measured from zero.	Project Zero is always at zero for the purposes of measuring actual element elevation. To set other reference levels, use Options > Project Preferences > Reference Levels. Element's elevation value (measured from its reference line/point) is displayed in its Info box or Settings.
Elevation to Sea Level	Element's elevation relative to Sea Level (Altitude as set in Location Settings).	Define Sea Level value (Altitude) at Options > Project Preferences > Location Settings. Element's elevation value (measured from its reference line/point) is displayed in its Info box or Settings.
External IFC ID	The Globalid IFC attribute assigned to each element upon IFC import (either Merge or Hotlink IFC)	
Fill Type	Name of the Fill attribute assigned to the element	To list the type of the Fill pattern (e.g. Solid, Vectorial, Symbol, Image) instead: Use the Fill Type parameter in the "General" parameter group.
Floor Plan Holes Perimeter	Total Perimeter of all the holes in the element.	Relevant for Slab, Mesh, Zone
Floor Plan Perimeter	Perimeter of the element, including hole perimeters. Measured according to the element's 2D polygon	Relevant for Wall, Slab, Mesh, Morph For Wall: Perimeter can vary depending on its Floor Plan Projection settings (e.g. Cut Only vs. Projected with Overhead) Wall intersections have no effect on the perimeter value.
From Zone	Name of the Zone from which the element opens.	Relevant for Door, Window, Skylight, CW Panel CW Panel: Zone (if any) from which the Panel faces outward. (The Zone that abuts the panel's inside surface.)
From Zone Number	Number of the Zone from which the element opens.	Relevant for Door, Window, Skylight, CW Panel CW Panel: Number of the Zone (if any) from which the Panel faces outward. (The Zone that abuts the panel's inside surface.)
GDL Preview Picture	Preview picture, of GDL Object type elements only.	

Parameter	Short Description	Long Description
Height	Height of the element.	D-W: Height of the window/door, as defined in the "Height" field of the opening's Settings dialog box or Info Box. Curtain Wall: defined in Curtain Wall System Settings or the Info Box Curtain Wall Panel: Height of the panel, not including the panel's extrusion into the surrounding frames. Zone: Height of the Zone body measured from the Zone's bottom elevation. Defined in the Name and Positioning Panel of Zone Settings. Skylight: Height parameter in the Preview and Positioning Panel (or the Skylight Settings Panel)
Home Offset	Offset of element from its Home Story. For Door/Window: The sill height, measured from the host Wall's home story.	
Home Story Name	Name of element's Home Story	As defined in Element settings. Openings: Home Story depends on the opening's vertical position in the host element. Skylight: Home Story is the same as the host Roof/Shell.
Home Story Number	Number of element's Home Story	As defined in Element settings. Door/Window: Home Story depends on the opening's vertical position in the host element. Skylight: Home Story is the same as the host Roof/Shell.
Hotlink and Element ID	If element is part of a Hotlink: Lists the Hotlink's Master ID as well as the Element's ID	
Hotlink Master ID	If element is part of a Hotlink: Lists the Hotlink's Master ID only	
ID of Connected Openings	IDs of opening(s) connected to this element.	
IFC Assignment	Name of IFC Assignment (if any) to which element belongs.	IFC Assignment takes place in IFC Project Manager. Assignments include e.g. IFC Group, IFC Zone, IFC System.
IFC Type	Name of element's IFC Type	Classification of Archicad elements for IFC is displayed in Element Settings (Classification and Properties panel), and in IFC Project Manager. IFC Type depends on "Type Mapping for Export" setting of the current Preview Translator (IFC Translators dialog box)
IFC Type Product	Lists the element's IFC Type Product Type and IFC Type Product Name.	IFC Type Product is defined in the "Type Mapping for Export" setting of the current Preview Translator (IFC Translators dialog box), and is displayed in IFC Project Manager.


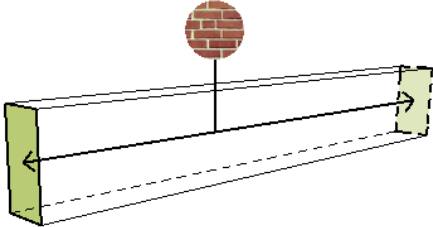
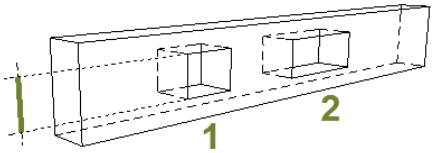
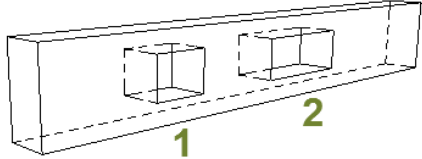
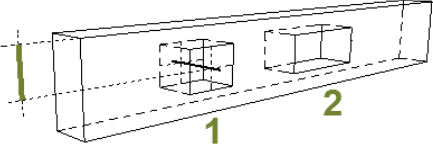
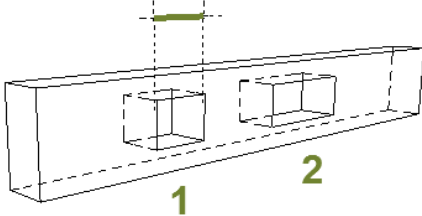
Parameter	Short Description	Long Description
Insulation Skin Thickness	Thickness of Wall, Roof, or Shell skins whose Building Material is defined as "Insulation"	Insulation Skin (Wall, Roof, Shell) is defined at Options > Project Preferences > Calculation Rules.
Issue Date (of last Change)		
Issue ID (of last Change)		
Issue Name (of last Change)		
Layer	Layer of element.	
Length	Length of element.	
Library Part Name	Name, of GDL Object type elements only.	
Linked Changes	If element or sub-element is linked to a Change: lists the number of the linked Change(s)	
Locked	Element's locked/unlocked status	This is a toggle (yes/no) value expressed as a checkbox in the Schedule field.
Number of Openings	Number of Opening elements.	Opening elements only (not Doors, Windows, or Skylights).
Number of Segments	(new)	
Parent ID	Element ID of the listed element's parent element	Relevant for sub-elements (e.g. Tread of Stair, or Panel of Curtain Wall, or Segment of a Beam/Column, or Door/Window/Skylight/Opening)
Position	Value of element's "Position" classification	Possible values: Interior, Exterior, Undefined Position is set in Classification and Properties panel of Element Settings. Used for IFC data exchange.
Property Object Name	Name of property object (if any) linked to this element.	Property Objects can be linked to elements using the Listing panel of Element Settings. (This panel is hidden by default. Consult the Calculation Guide for details.)
Quantity	For a particular listed element, displays the number of scheduled elements.	

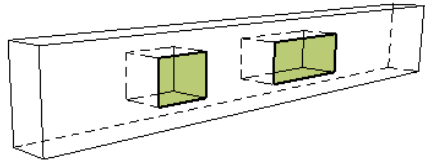
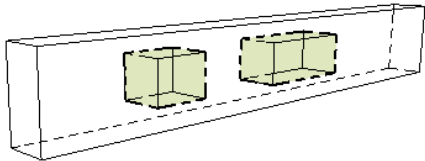
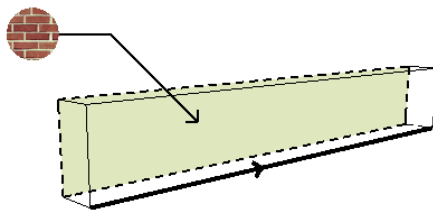
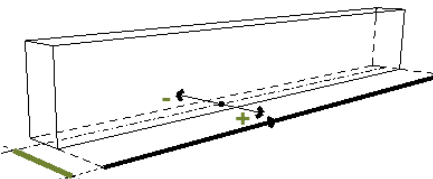
Parameter	Short Description	Long Description
Related Zone Name	<p>Name of the element's related Zone, if any.</p> <p>In general: the Zone which the element borders or intersects in 2D.</p> <p>Related Zone definitions vary by element. See Description.</p> <p>An element can have only one Related Zone. (If multiple Zones apply, the first created Zone is listed.)</p> <p>(To identify the Zone with which an element intersects in 3D, use the "Colliding Zones" parameter.)</p>	<p>Wall: The Zone which it borders, or the Zone which the Wall's 2D polygon intersects</p> <p>Door/Window: Related Zone of the host Wall.</p> <p>Curtain Wall: The Zone which it (or at least one of its segments) borders</p> <p>Curtain Wall Frame/Panel: The Zone which its CW segment borders, and which the Frame/Panel touches.</p> <p>Curtain Wall Junction: All of the connecting subelements (at the Junction's gridpoint) border the Zone; or two of the connecting Frames (at the Junction's gridpoint) border the Zone</p> <p>Curtain Wall Accessory: The Zone bordered by the Accessory's parent Frame.</p> <p>Shell/Roof: The Zone to which it connects via SEO, or by which it is cropped</p> <p>Object, Light, Morph, Stair, Railing: The Zone that has the same Home Story as the element. For an Object: the centerpoint of its 3D bounding box is within the Zone polygon.</p> <p>Beam: The Zone which the Beam's Reference line intersects in 2D. The Beam must be visible on this related Zone's home story.</p> <p>Skylight: The Zone to which its parent Roof or Shell connects via SEO, or by which it is cropped.</p> <p>Column: The Zone polygon intersected by any corner of the Column. The Column must be visible on this related Zone's home story.</p> <p>Tread, Riser, Stair Structure: The related Zone of its parent Stair Railing subelements: The related Zone of its parent Railing</p>
Related Zone Number	Number of the element's related Zone, if any. See explanation for Related Zone Name, above.	
Relative Top Link Story	Relative position of element's top-linked Story.	Relevant for Wall/Column/Zone/Stair For example: Home + 1
Renovation Status	Existing, To Be Demolished, or New	As set in element's Classification and Properties panel, or Renovation palette
Show On Renovation Filter	Renovation Filter on which this element is shown	As set in element's Classification and Properties panel, or Renovation palette
Slant Angle	Angle of slanted or inclined element.	For Wall: Same as Wall Outside Slant Angle. (See also Wall-specific Inside Slant Angle parameter). Beam: Angle of an inclined Beam CW Frame or Panel: Slant angle measured to the horizontal plane.
Structural Function	Load-bearing, or Non-load-bearing, or undefined	As set in element's Classification and Properties panel
Structure Type	Basic, Composite, or Complex Profile	

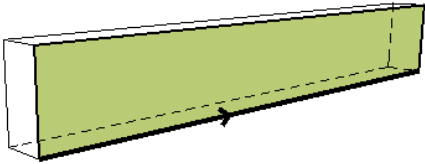
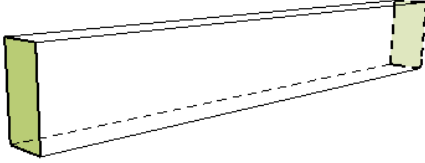
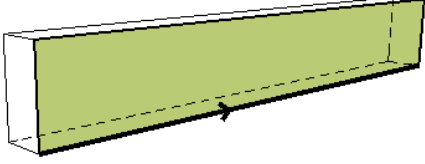
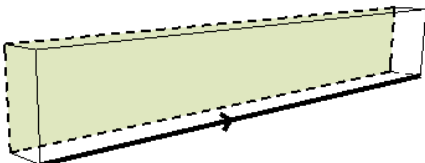
Parameter	Short Description	Long Description
Bottom Surface	Name of the Surface assigned to the bottom of the element.	
Bottom Surface Area (Conditional)	Surface area of the bottom of the element - this area may be conditioned on a defined Calculation Rule.	Condition: Subtracts the area of holes only if they exceed a given size, as defined in Options > Project Preferences > Calculation Rules Reduced by trims and SEO's.
Bottom Surface Area (Net)	Area of the Surface of the bottom of the element.	Reduced by any SEO's or holes.
Bottom Surface Area (Gross)	Area of the Surface of the bottom of the element.	
Edge Surface	Name of the Surface assigned to the element edge.	Lists the element's default edge surface (as defined in its Building Material). Not listed for composite elements. For Openings: name of its extrusion surface override, if any (applied in Opening Settings, Model panel) To list all Surfaces, including custom ones and those of composite elements, use a Surface-type list (e.g. All Finishes).
Edge Surface Area (Net)	Surface area of the element edge	Reduced by trims and SEO's
Edge Surface Area (Gross)	Surface area of the element edge	
Surface (All)	List of all surfaces used in the element, including any overrides	
Surface Area	Generally: the sum of all surfaces of an element.	Beam: The sum of all Beam surfaces (top, bottom, ends) Column: Surface area of the Column's cross-section Shell: Net surface area of the Reference Side
Top Surface	Name of the Surface assigned to the top of the element.	Defined in Element Settings (via Building Material, or Override in Model Panel)
Top Surface Area (Conditional)	Surface area of the top of the element - this area may be conditioned on a defined Calculation Rule.	Condition: Subtracts the area of holes only if they exceed a given size, as defined in Options > Project Preferences > Calculation Rules Reduced by trims and SEO's..
Top Surface Area (Net)	Area of the Surface of the Top of the element.	Reduced by any trims, SEO's or holes.
Top Surface Area (Gross)	Area of the Surface of the Top of the element.	
To Zone	Name of the Zone toward which the element opens.	Relevant for Door, Window, Skylight, CW Panel CW Panel: Name of the Zone (if any) that abuts the panel's outside surface.

Parameter	Short Description	Long Description
To Zone Number	Number of the Zone toward which the element opens.	Relevant for Door, Window, Skylight, CW Panel CW Panel: Name of the Zone (if any) that abuts the Panel's outside surface.
Top Link Story	Element's top-linked Story, expressed as both relative and absolute.	Relevant for Wall/Column/Zone/Stair For example: Home + 1 (2. Story)
Top Offset	Offset from element's Toplink Story	Applicable to elements with a top link: Wall, Column, Zone, Stair
Top/Bottom Elevation to Reference Level	Elevation of the element's topmost (or bottom) point in 3D space (regardless of its reference line or anchor point).	Use the parameter corresponding to the desired reference level for the measured elevation: First/Second Reference Level, Home Story, Project Zero or Sea Level. Define levels at Options > Project Preferences > Reference Levels.
Unique ID	Automatically generated ID that is conserved throughout the life of the project	Also known as the element's Archicad "Global unique identifier" (GUID).
Volume (Gross)	Volume of the element	Wall: Includes Volume of Openings, doors and windows. Column: Includes Volume of both Core and Veneer. Roof/Shell/Slab: Includes the volume of any holes. Reduced by trims. See also the Slab parameter: "Gross Volume of the Slab, Minus the Holes"
Volume (Net)	Net volume of the element. Reduced by volume of holes and Solid Element Operations	Column: (including the volume of the core and of the veneer, if any). Roof-Shell-Slab-Mesh: is reduced by the surface of any holes. Reduced by trims and SEO's. Wall/Beam/Morph: reduced by holes and SEO's Door-Window: Volume of the opening cut by the Door/Window. (Volume calculation based on hole that results from cutting the GDL openings out of the Wall.)
Volume (Conditional)	Volume of Construction element (Beam, Roof, Shell, Slab, or Wall). This volume may be conditioned on a defined Calculation Rule.	Condition: subtracts the volume of any openings that exceed a given size, as defined in Options > Project Preferences > Calculation Rules. Reduced by trims and SEO's.
Width	Element Width parameter, as defined by its geometry or in Element Settings.	Object: Width of the Object as defined in the "Dimension 1" field of the Preview and Positioning Panel of Object Settings. Column: Width of the column: Core Dimension 1 value, plus the width of any veneer.

Beam Listing Parameters

Parameter	Short Description	Long Description	
Beam Cross Section	Rectangular, Profiled or Rounded	Shape of the Beam's cross-section as defined in the Beam Settings dialog box.	
Beam Slant Angle	Angle of an inclined Beam.		
End Surface	Name of the Surface assigned to the two ends of the Beam.	Defined in Beam Settings (via Building Material, or Override in Model Panel)	
Extrusion Surface	Name of the Surface assigned to the Beam extrusion.	Use with round and profiled beams, which use a single surface on all sides of the extrusion. (Same value as the Left Side Surface parameter.)	
Hole Height	Height or diameter of the first-placed hole in the Beam.	Same value as "Hole Height or Diameter", on the Hole panel of Beam Tool Settings	
Hole Number	Total number of holes in the Beam.		
Hole Position from Reference Line	Distance of the centerpoint of the first-placed hole from the Beam Reference Line.	Same value as Position, on the Hole panel of Beam Tool Settings.	
Hole Width	Width of the first-placed hole in the Beam.	Same value as "Hole Height", on the Hole panel of Beam Tool Settings.	
Holes Edge Surface Area	Total edge area of all Beam holes.		


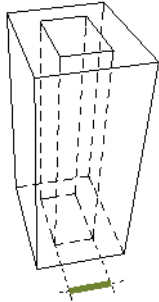
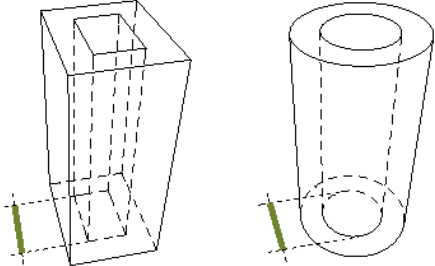
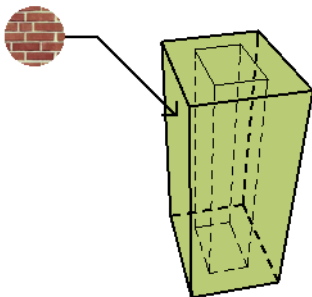
Parameter	Short Description	Long Description	
Holes Surface Area	Total area of the 2D Surface of all holes, as calculated on one Side of the Beam.		
Holes Volume	Total volume of all holes in the Beam.	Multiplies Beam Width by Holes Surface Area.	
Left Side Surface	Name of the Surface assigned to the left Side of the Beam.	Defined in Beam Settings (via Building Material, or Override in Model Panel). "Left" and "Right" are determined based on the direction of the Beam's reference axis. The axis direction is the direction in which you drag the cursor to draw the Beam.	
Length Left	Length of the Beam on its left Side.	"Left" and "Right" are determined based on the direction of the Beam's reference axis. The axis direction is the direction in which you drag the cursor to draw the Beam. To list the Beam's Reference Length, use the 3D Length parameter (General category).	
Length Right	Length of the Beam on its right Side.	"Left" and "Right" are determined based on the direction of the Beam's reference axis. The axis direction is the direction in which you drag the cursor to draw the Beam. To list the Beam's Reference Length, use the 3D Length parameter (General category).	
Offset	Beam offset distance from reference line.	By default, this value is zero. Entering an offset value offsets the Beam from its reference axis.	

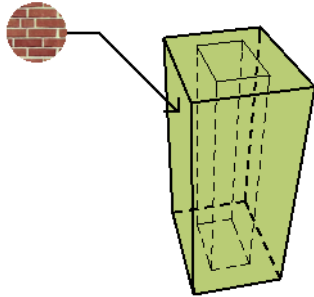
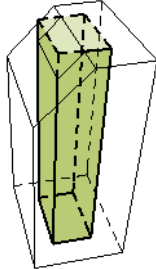
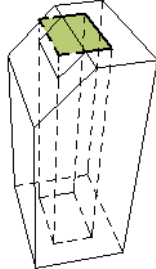
Parameter	Short Description	Long Description	
Right Side Surface	Name of the Surface assigned to the right Side of the Beam.	Defined in Beam Settings (via Building Material, or Override in Model Panel) "Left" and "Right" are determined based on the direction of the Beam's reference axis. The axis direction is the direction in which you drag the cursor to draw the Beam.	
Surface Area of Both Beam Ends	Area of the Surfaces of both ends of the Beam.	Area of the Surfaces of both ends of the Beam.	
Surface Area of the Beam Right Side (Net)	Area of the Surface of the right Side of the Beam.	Reduced by any SEO's or holes. "Left" and "Right" are determined based on the direction of the Beam's reference axis. The axis direction is the direction in which you drag the cursor to draw the Beam.	
Surface Area of the Beam Left Side (Net)	Area of the Surface of the left Side of the Beam.	Reduced by any SEO's or holes. "Left" and "Right" are determined based on the direction of the Beam's reference axis. The axis direction is the direction in which you drag the cursor to draw the Beam.	

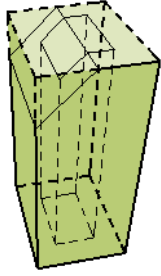
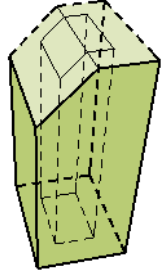
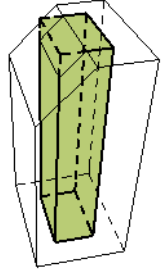
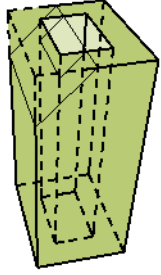
Beam Segment Listing Parameters

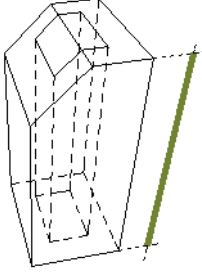
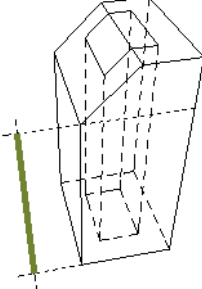
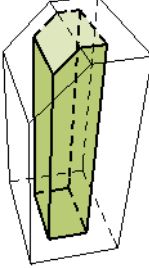
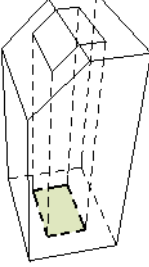
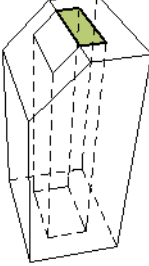
Parameter	Short Description	Long Description
Bottom Surface	Name of the Surface assigned to the bottom of the Beam Segment.	Defined in Beam Settings (via Building Material, or Override in Model Panel)
Bottom Surface Area	Area of the Surface of the bottom of the Beam Segment.	Reduced by any SEO's or holes.
Cross Section	Rectangular, Profiled or Rounded	Shape of the Beam Segment's cross-section as defined in the Beam Settings dialog box.
Ends Surface	Name of the Surface assigned to the two ends of the Beam Segment.	Defined in Beam Settings (via Building Material, or Override in Model Panel)
Ends Surface Area	Area of the Surfaces of both ends of the Beam Segment.	Area of the Surfaces of both ends of the Beam Segment.
Extrusion Surface	Name of the Surface assigned to the Beam Segment extrusion.	Use with round and profiled beams, which use a single surface on all sides of the extrusion. (Same value as the Left Side Surface parameter.)
Left Side Surface	Name of the Surface assigned to the left Side of the Beam Segment.	Defined in Beam Settings (via Building Material, or Override in Model Panel). "Left" and "Right" are determined based on the direction of the Beam's reference axis. The axis direction is the direction in which you drag the cursor to draw the Beam.
Left Side Surface Area	Area of the Surface of the left Side of the Beam Segment.	Reduced by any SEO's or holes. "Left" and "Right" are determined based on the direction of the Beam's reference axis. The axis direction is the direction in which you drag the cursor to draw the Beam.
Right Side Surface	Name of the Surface assigned to the right Side of the Beam Segment.	Defined in Beam Settings (via Building Material, or Override in Model Panel) "Left" and "Right" are determined based on the direction of the Beam's reference axis. The axis direction is the direction in which you drag the cursor to draw the Beam.
Right Side Surface Area	Area of the Surface of the right Side of the Beam Segment.	Reduced by any SEO's or holes. "Left" and "Right" are determined based on the direction of the Beam's reference axis. The axis direction is the direction in which you drag the cursor to draw the Beam.
Structure Type		
Top Surface	Name of the Surface assigned to the top of the Beam Segment.	Defined in Beam Settings (via Building Material, or Override in Model Panel)
Top Surface Area	Area of the Surface of the top of the Beam Segment.	Reduced by any SEO's or holes.

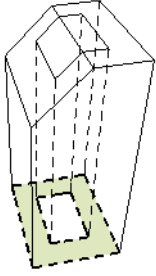
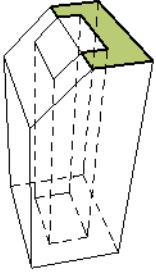
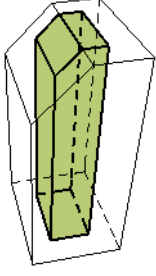
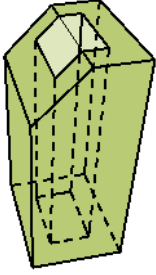
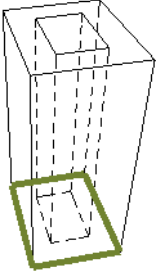
Column Listing Parameters

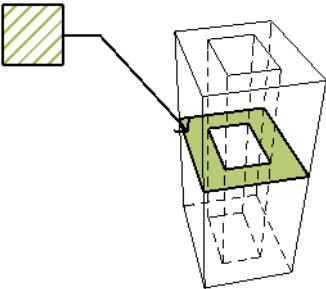
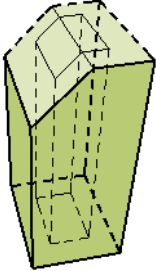
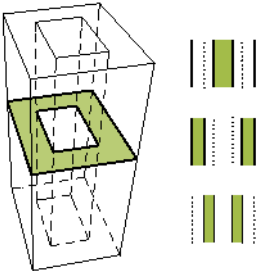
Parameter	Short Description	Long Description	
Column Cross Section	Rectangular, Circular or Profiled.	Defined in Column Settings	
Core Height / Diameter	<p>For a rectangular or profiled Column: One of the dimensions of the Column in cross-section. (The other dimension is "Core Width".)</p> <p>Does not include veneer. (To include veneer, use the General "Width" parameter.)</p>	For a circular Column: the diameter of the Column.	
Core Width	<p>For a rectangular or profiled Column: One of the two dimensions of the Column in cross-section. (The other dimension is "Core Height / Diameter".)</p> <p>Does not include Veneer. (To include veneer, use the General "Width" parameter.)</p>	For a circular Column: the diameter of the Column.	
End Surface	Surface assigned to Column Ends.	Defined in Column Settings (via Building Material, or Override in Model Panel)	

Parameter	Short Description	Long Description	
Extrusion Surface	Surface assigned to Column Extrusion.	Defined in Column Settings (via Building Material, or Override in Model Panel)	
Surface Area of the Column Core (Gross)	Gross Surface Area of the Column Core on all sides.	Solid Element Operations are ignored.	
Surface Area of the Column Core (Net)	Net Surface Area of the Column Core on all sides.	Reduced by Solid Element Operations.	
Surface Area of the Column Core (Gross, without Top/Bottom)	Gross Surface Area of the Column Core on all sides.	Surfaces of the Column top/bottom are ignored. Solid Element Operations are ignored.	
Surface Area of the Column Core Bottom (Gross)	Surface area of the cross-section of the Core at the bottom of the Column.	Solid Element Operations are ignored.	
Surface Area of the Column Core Top (Gross)	Surface area of the cross-section of the Core at the top of the Column.	Solid Element Operations are ignored.	

Parameter	Short Description	Long Description	
Surface Area of the Column Veneer (Gross, without Top/Bottom)	Gross Surface Area of the Column Veneer on all sides.	Surfaces of the Veneer top/bottom are ignored. Solid Element Operations are ignored.	
Surface Area of Column Veneer Bottom (Gross)	Surface area of the cross section of the Veneer at the bottom of the Column.	Solid Element Operations are ignored	
Surface Area of Column Veneer (Gross)	Surface area of the Column Veneer on all sides.	Solid Element Operations are ignored	
Surface Area of Column Veneer (Net)	Net Surface area of the Column Veneer on all sides	Surfaces of the Veneer top/bottom are ignored. Reduced by Solid Element Operations.	
Volume of the Column Core (Gross)	Gross Volume of the Column Core.	Solid Element Operations are ignored.	
Volume of the Column Veneer (Gross)	Gross Volume of the Column Veneer.	Solid Element Operations are ignored.	

Parameter	Short Description	Long Description	
Maximum Height of the Column	For a Column cut e.g. by a slanted Roof: the highest point of the Column.	For a straight or slanted Column: equivalent to Height (General field).	
Minimum Height of the Column	For a Column cut e.g. by a slanted Roof: the lowest point of the Column.		
Surface Area of the Column Core (Net, without Top/Bottom)	Net Surface area of the Column Core on all sides (but not the top or bottom).	Surfaces of the Column top/bottom are ignored. Reduced by Solid Element Operations.	
Surface Area of the Column Core Bottom (Net)	Surface area of the cross-section of the Core at the bottom of the Column.	Reduced by Solid Element Operations.	
Surface Area of Column Core Top (Net)	Surface area of the cross-section of the Core at the top of the Column.	Reduced by Solid Element Operations.	

Parameter	Short Description	Long Description	
Surface Area of Column Veneer Bottom (Net)	Surface area of the cross-section of the Veneer at the bottom of the Column.	Reduced by Solid Element Operations.	
Surface Area of Column Veneer Top (Net)	Surface area of the cross-section of the Veneer at the top of the Column.	Reduced by Solid Element Operations.	
Volume of the Column Core (Net)	Net Volume of the Column Core.	Reduced by Solid Element Operations.	
Volume of the Veneer (Net)	Net Volume of the Column Veneer on all sides.	Reduced by Solid Element Operations.	
Perimeter of Column Profile (Bottom)	Perimeter of the Column profile at the bottom of the Column.	Profile plane is perpendicular to the Column axis.	
Perimeter of Column Profile (Top)	Perimeter of the Column profile at the top of the Column.	Profile plane is perpendicular to the Column axis.	

Parameter	Short Description	Long Description	
Veneer Building Material	Name of Building Material assigned to Veneer	Defined in Column Settings	
Surface Area of Column Veneer (Net)	Net Surface area of the Column Veneer on all sides	Surfaces of the Veneer top/bottom are ignored. Reduced by Solid Element Operations.	
Veneer Type	Core, Finish or None	Defined in Column Settings This setting affects the display of the Column, depending on Partial Structure Display Settings.	

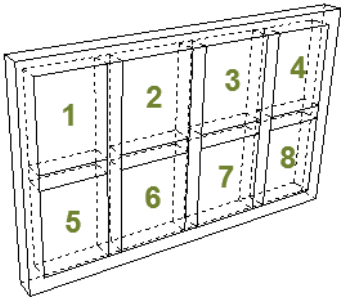
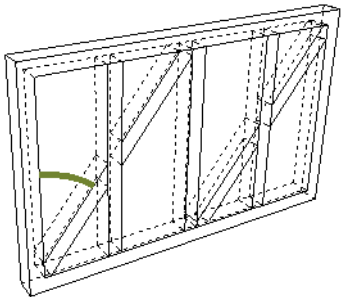
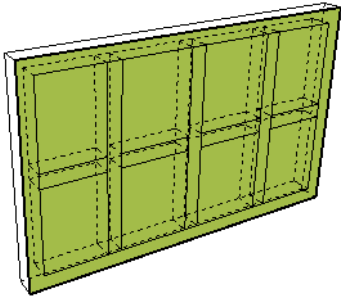
Column Segment Listing Parameters

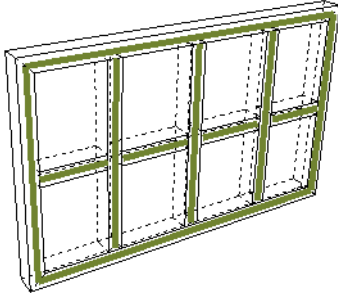
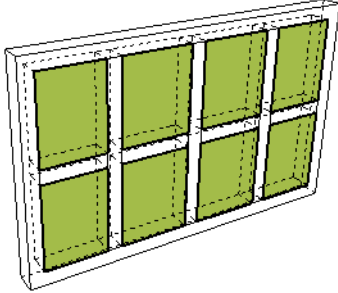
Parameter	Short Description	Long Description
Core Height / Diameter	For a rectangular or profiled Column Segment: One of the dimensions of the Segment in cross-section. (The other dimension is "Core Width".) Does not include veneer. (To include veneer, use the General " Width " parameter.)	For a circular Column Segment: the diameter of the Segment.
Core Width	For a rectangular or profiled Column Segment: One of the two dimensions of the Segment in cross-section. (The other dimension is "Core Height / Diameter".) Does not include Veneer. (To include veneer, use the General " Width " parameter.)	For a circular Column Segment: the diameter of the Segment.
Cross Section	Rectangular, Circular or Profiled.	Defined in Column Settings
Ends Surface	Surface assigned to Column Segment Ends.	Defined in Column Segment Settings (via Building Material, or Override in Model Panel)
Extrusion Surface	Surface assigned to Column Segment Extrusion.	Defined in Column Segment Settings (via Building Material, or Override in Model Panel)
Surface Area of Column Core (Gross)	Gross Surface Area of the Column Segment Core on all sides.	Surfaces of the Column Segment top/bottom are ignored. Solid Element Operations are ignored.
Surface Area of Column Core Bottom (Gross)	Surface area of the cross-section of the Core at the bottom of the Column Segment.	Solid Element Operations are ignored
Surface Area of Column Core Top (Gross)	Surface area of the cross-section of the Core at the top of the Column Segment.	Solid Element Operations are ignored.
Surface Area of the Column Veneer (Gross)	Gross Surface Area of the Column Segment Veneer on all sides.	Surfaces of the Veneer top/bottom are ignored. Solid Element Operations are ignored.

Parameter	Short Description	Long Description
Surface Area of the Column Veneer Bottom (Gross)	Surface area of the cross section of the Veneer at the bottom of the Column Segment.	Solid Element Operations are ignored
Surface Area of the Column Veneer Top (Gross)	Surface area of the cross section of the Veneer at the top of the Column Segment.	Solid Element Operations are ignored
Volume of the Column Core (Gross)	Gross Volume of the Column Segment Core.	Solid Element Operations are ignored.
Volume of the Column Veneer (Gross)	Gross Volume of the Column Segment Veneer.	Solid Element Operations are ignored.
Surface Area of the Column Core (Net)	Net Surface area of the Column Segment Core on all sides (but not the top or bottom).	Surfaces of the Column Segment top/bottom are ignored. Reduced by Solid Element Operations.
Surface Area of the Column Core Bottom (Net)	Surface area of the cross-section of the Core at the bottom of the Column Segment.	Reduced by Solid Element Operations.
Surface Area of the Column Core Top (Net)	Surface area of the cross-section of the Veneer at the top of the Column Segment.	Reduced by Solid Element Operations.
Surface Area of the Column Veneer Bottom (Net)	Surface area of the cross-section of the Veneer at the bottom of the Column Segment.	Reduced by Solid Element Operations.
Surface Area of the Column Veneer Top (Net)	Surface area of the cross-section of the Veneer at the top of the Column Segment.	Reduced by Solid Element Operations.
Volume of the Column Core (Net)	Net Volume of the Column Segment Core.	Reduced by Solid Element Operations.
Volume of the Column Veneer (Net)	Net Volume of the Column Segment Veneer on all sides.	Reduced by Solid Element Operations.
Veneer Building Material	Name of Building Material assigned to Veneer	Defined in Column Settings
Veneer Surface Area	Net Surface area of the Column Segment Veneer on all sides	Surfaces of the Veneer top/bottom are ignored. Reduced by Solid Element Operations.

Parameter	Short Description	Long Description
Veneer type	Core, Finish or None	Defined in Column Settings This setting affects the display of the Column Segment, depending on Partial Structure Display Settings.

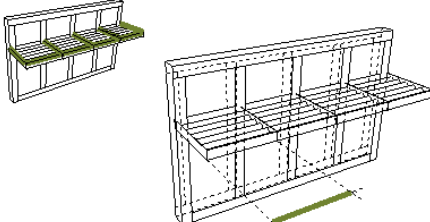
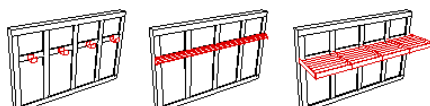
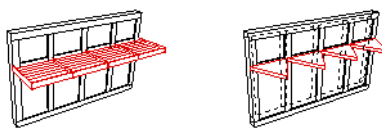
Curtain Wall Listing Parameters

Parameter	Short Description	Long Description	
Curtain Wall Slant Angle	Slant Angle of the Curtain Wall	Same value as the General " Slant Angle " parameter.	
Length	Length of element.		
Number of Panels	Number of panels in the Curtain Wall.		
Pattern Angle	Rotation angle of the row Grid Lines measured clockwise from the column Grid Lines. This angle is defined in the Curtain Wall Scheme Settings/ Grid Pattern panel, under Rows.		
Surface Area Including Boundary	Surface area of Curtain Wall, including the surface area of its Boundary Frames.	<p>If the Boundary Frames are placed "Inside Grid" (Curtain Wall Settings/Curtain Wall System/ Member Placement: Place Boundary Frames "Inside"), then this parameter is equal to the general Surface Area parameter for the Curtain Wall.</p> <p>If the Boundary Frames are placed "Outside Grid" or "Centered on Grid", then the value of this parameter will exceed the general Surface Area parameter, because the Boundary Frames hang out over the boundary line.</p> <p>If you use an Invisible type Frame for your Boundary Frames (Curtain Wall Settings/ Curtain Wall System/Frames: Boundary), then the value of this parameter will be less than that of the general Surface Area parameter.</p>	

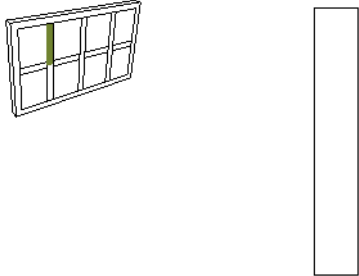
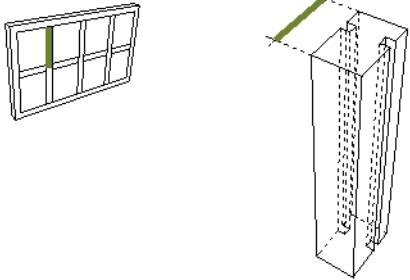
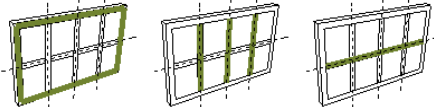
Parameter	Short Description	Long Description	
Total Length of Contour Frames			
Total Length of Custom Frames			
Total Length of Frames	Total of the lengths of all frames in the Curtain Wall. If the shape of the Frame is not uniform, then the value of the Frame length is calculated by taking the average of its shortest and longest lengths.		
Total Length of Frames Belonging to the Main Axis			
Total Length of Frames Belonging to the Secondary Axis			
Total Panel Surface Area	Total surface area of all the Curtain Wall panels. This value does not include the surface of the frames between the panels. Total Panel Surface Area is seen if you look at the Curtain Wall in Edit mode, with just the Panels visible.		

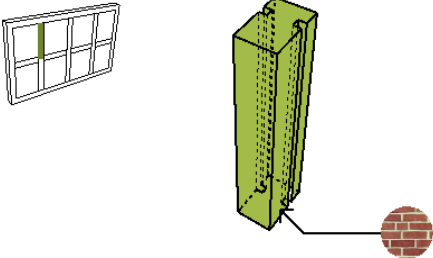
Parameter	Short Description	Long Description	
Total Surface Area of Panels Facing East			
Total Surface Area of Panels Facing North			
Total Surface Area of Panels Facing North-East			
Total Surface Area of Panels Facing North-West			
Total Surface Area of Panels Facing South			
Total Surface Area of Panels Facing South-East			
Total Surface Area of Panels Facing South-West			
Total Surface Area of Panels Facing West			

Curtain Wall Accessory Listing Parameters

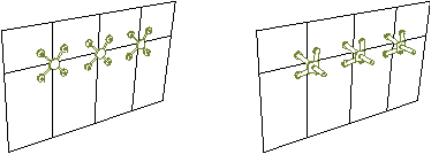
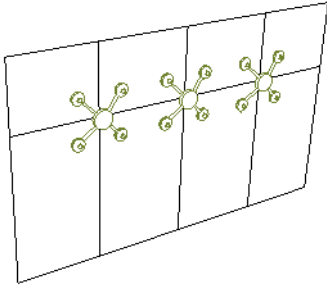
Parameter	Short Description	Long Description	
Length	Length parameter of the Accessory	Measured along the plane of the Curtain Wall	
Accessory Class	System or Custom	<p>Accessories from the "System" class use the settings at the Curtain Wall System Settings level.</p> <p>Accessories from the "Custom" class have been individually customized using Junction Settings, accessible through Curtain Wall Edit Mode.</p>	
Accessory Type	Name of the Accessory Type	Defined at Accessory Type in Curtain Wall Settings - Accessory page.	

Curtain Wall Frame Listing Parameters

Parameter	Short Description	Long Description	
Curtain Wall Frame Slant Angle	Frame's slant angle in the plane of the Curtain Wall	E.g. 0 or 90 degrees for orthogonally positioned frames, other angles for diagonal frames (regardless of whether the Curtain Wall itself is slanted.)	
Frame Class	Boundary, Mullion, Corner, Transom, or Custom	Assign Frame Class by clicking the frames in the Preview window of the Grid Pattern and Preview panel of Curtain Wall System Settings - Scheme Page. Frames from the "Custom" class have been individually customized using Frame Settings, accessible through Curtain Wall Edit Mode.	
Frame Depth	Depth of the Frame profile.	Defined in the Frame Type and Geometry panel of CW Frame Settings.	
Frame Length	Length of the Frame.		
Frame Type	Frame Type (e.g. Butt-glazed, Generic)	Defined in the Frame Type and Geometry panel of CW Frame Settings.	
Location	Frame's position as related to the Curtain Wall: Contour, Vertical, Horizontal, or Diagonal.		
Profile Height	Frame profile's height.	If the Frame Type is a profile element (e.g. Profiled Butt-Glazed Frame).	
Profile Width	Frame profile's width.	If the Frame Type is a profile element (e.g. Profiled Butt-Glazed Frame).	
Profile WxH	Frame profile's width and height dimensions.	If the Frame Type is a profile element (e.g. Profiled Butt-Glazed Frame).	

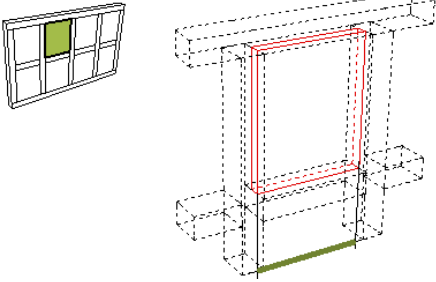
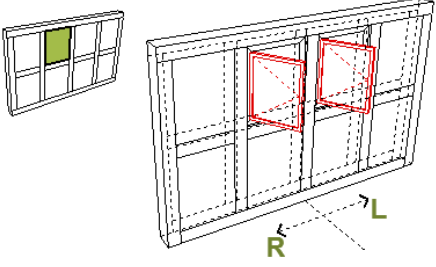
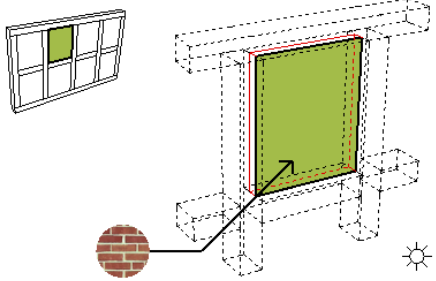
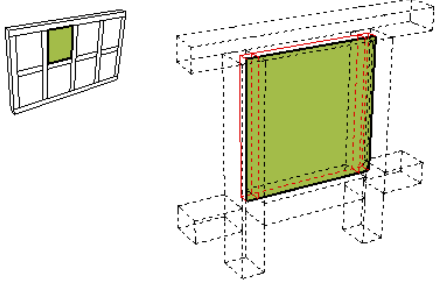
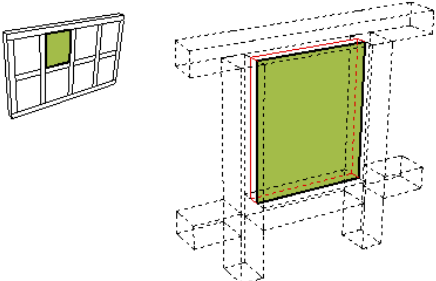
Parameter	Short Description	Long Description	
Surface	Surface assigned to the Frame.	From Frame's Building Material. Can be overridden in Curtain Wall Frame Settings.	 <p>The diagram consists of two parts. On the left is a 2D line drawing of a window frame with a central vertical mullion and two horizontal mullions, creating four panes. On the right is a 3D perspective view of a curtain wall frame. The frame is a tall, rectangular structure with a grid of vertical and horizontal lines representing the frame members. A red brick texture is shown at the bottom right, with a line pointing to the bottom edge of the frame, indicating that the brick texture is applied to the surface of the frame.</p>

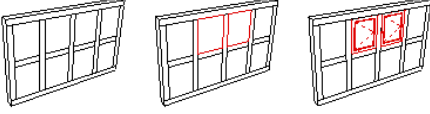

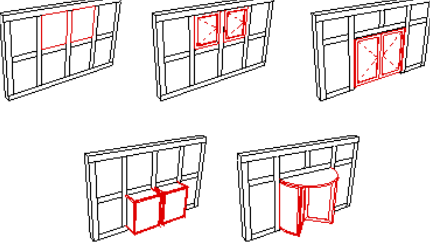
Curtain Wall Junction Listing Parameters

Parameter	Short Description	Long Description	
Junction Class	System or Custom.	<p>Junctions from the "System" class use the settings at the Curtain Wall System Settings level.</p> <p>Junctions from the "Custom" class have been individually customized using Junction Settings, accessible through Curtain Wall Edit Mode.</p>	
Junction Type	Name of the Junction Type	Defined at Junction Type in Curtain Wall Settings - Junctions page.	

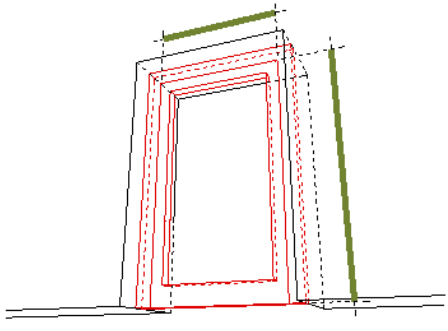
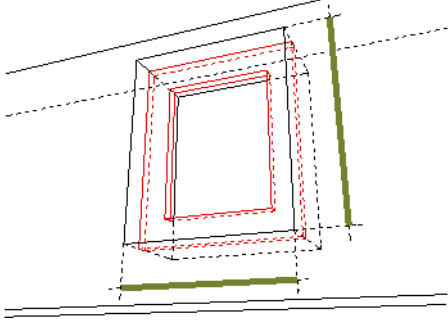
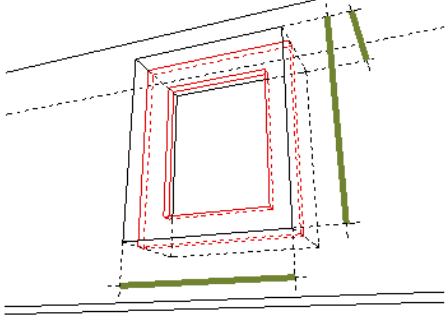
Curtain Wall Panel Listing Parameters

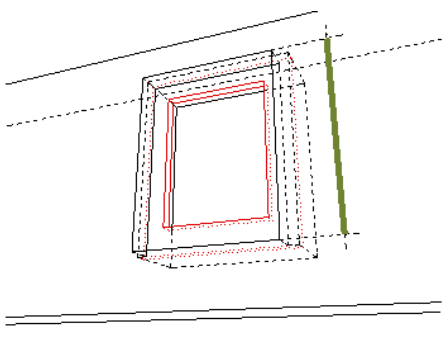
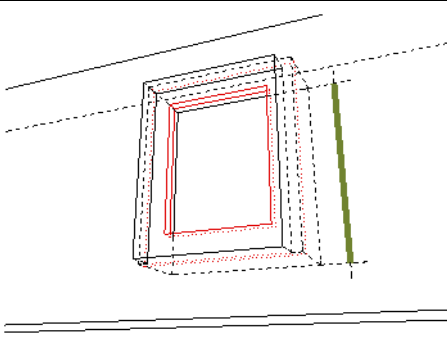
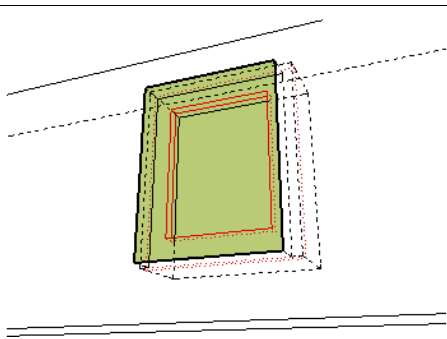
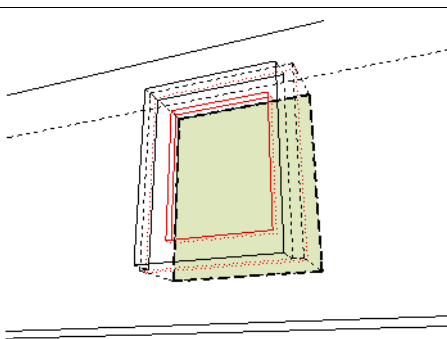
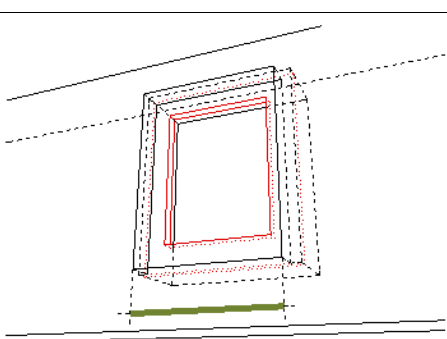
Parameter	Short Description	Long Description	
Abbreviation of Orientation	Facing direction of the panel (using abbreviation).		
Angle from North	Facing direction of the panel, in degrees measured from North.		
Curtain Wall Panel Slant Angle	Slant of the Curtain Wall Panel.	Equals the slant of the Curtain Wall	
Inside Surface	Panel's inside surface as defined in the Model Panel of CW Panel Settings.	For panels which are GDL Objects, the surface is defined in its GDL script.	
Nominal Height	Height of the panel.	Does not include the panel's extrusion into the surrounding frames	

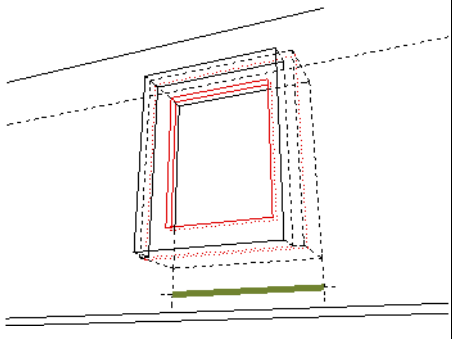
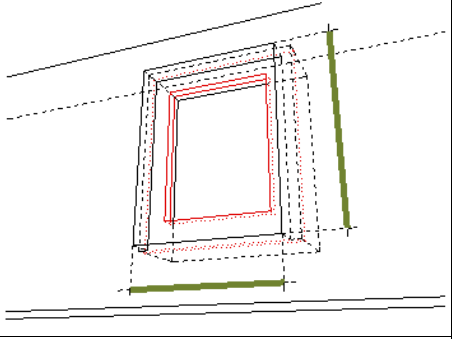
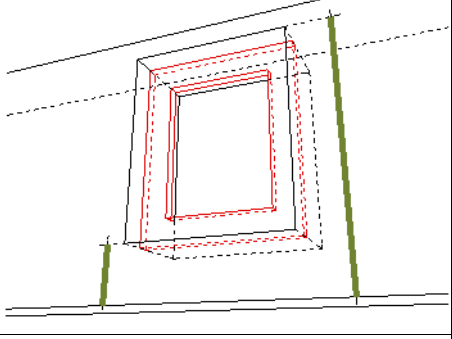
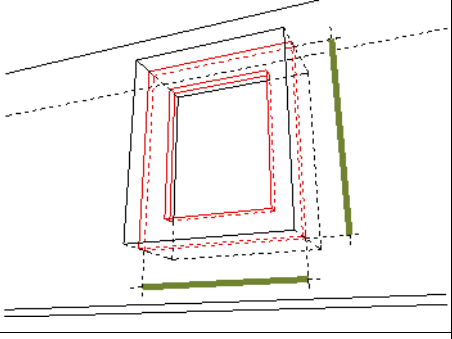
Parameter	Short Description	Long Description	
Nominal Width	Width of the panel.	Does not include the panel's extrusion into the surrounding frames For generic panels, this is the same as the regular Width parameter. For GDL panels, this value can be defined in the panel's parameters.	
Orientation	Lists the direction (Left or Right) of Panels which act as Doors/Windows.		
Outside Surface	Surface of the Panel outside surface as defined in the Model Panel of CW Panel Settings.	For panels which are GDL Objects, the surface is defined in its GDL script.	
Panel Area with Clamps	Gross surface area of the panel.	For rectangular-shaped panels. Includes the panel's extrusion into the surrounding frames (the "depth" value defined in the Frame Type and Geometry panel of CW Frame Settings.)	
Panel Area without Clamps	Surface area of the panel.	For rectangular-shaped panels. Does not include the panel's extrusion into the surrounding frames	

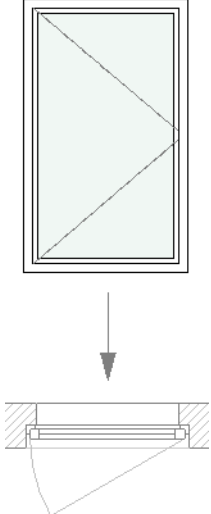
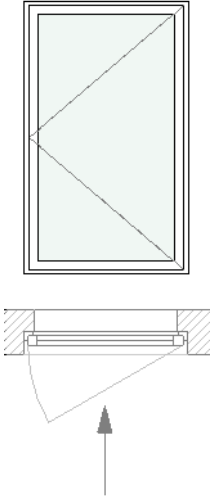
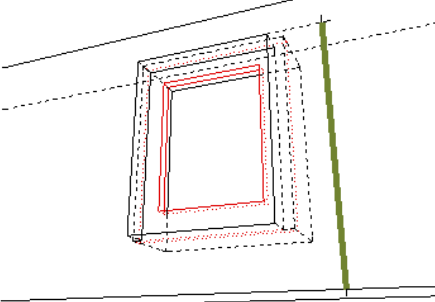
Parameter	Short Description	Long Description	
Panel Class	Main, Distinct, or Custom.	Assign Panel Class by clicking the panels in the Preview window of the Grid Pattern and Preview panel of Curtain Wall System Settings - Scheme Page. Panels from the "Custom" class have been individually customized using Panel Settings, accessible through Curtain Wall Edit Mode.	
Panel Function	Either Fixed, Window, or Door, depending on the Panel Type chosen.	Defined in the Panel Type and Geometry panel of CW Panel Settings.	
Panel Type	Name of the Panel Type.	Defined at Panel Type and Geometry in Curtain Wall Panel Settings.	

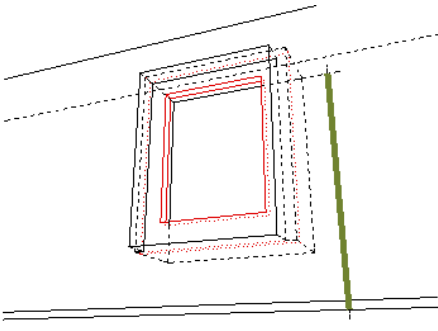
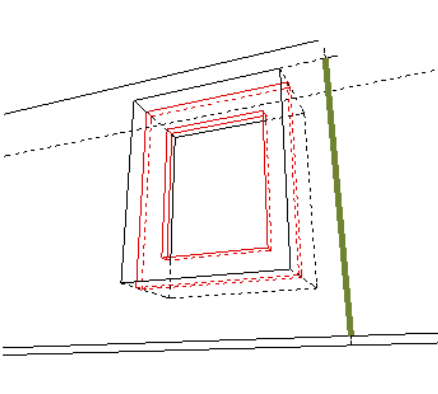
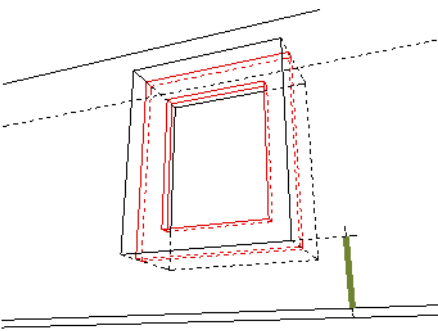
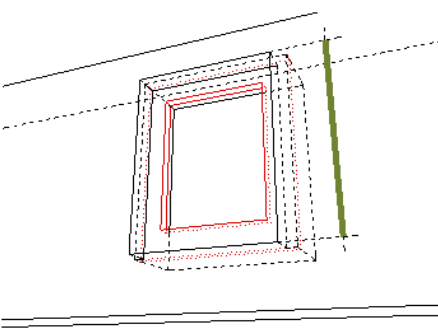
Window/Door Listing Parameters

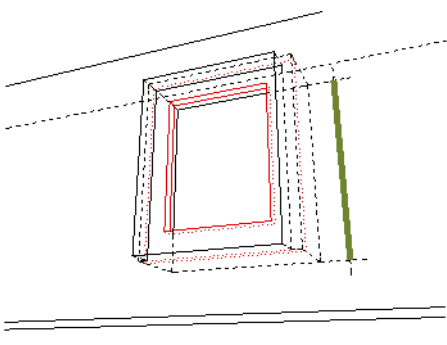
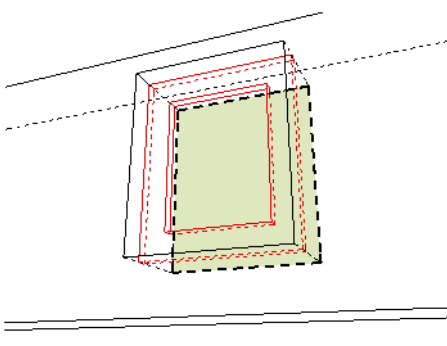
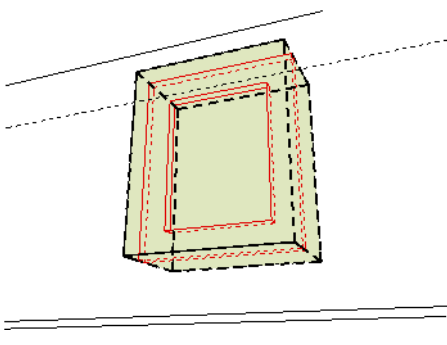
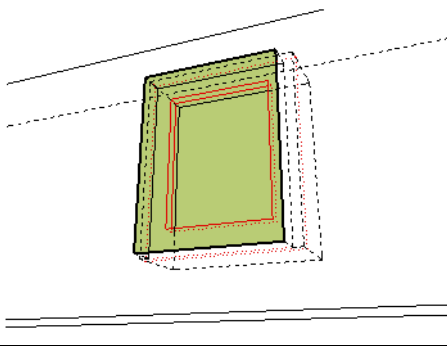
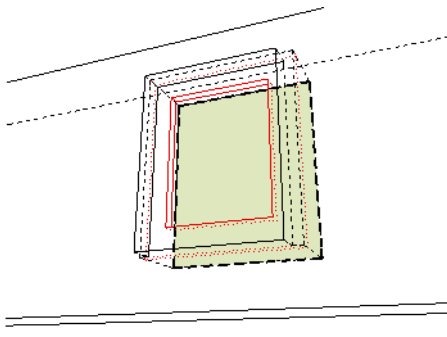
Parameter	Short Description	Long Description	
Egress Dimensions	(For Doors only.) Width and Height of egress, as defined in Door Settings (Custom Settings, Nominal Sizes and Tolerance page).		
Frame Thickness	Frame Thickness, as defined in Window Settings (Custom Settings, Frame and Sash page).		
Leaf Dimensions	(For Doors only.) Width and Height of leaf, as defined in Door Settings (Custom Settings, Nominal Sizes and Tolerance page).		
Nominal W X H Size	Displays dimensions as "Height x Width".	The Height and Width parameters that are "Set as Nominal" in the Settings dialog box (Custom Settings, Nominal Sizes and Tolerance page).	
Nominal W X H x T Size	Displays dimensions as "Height x Width x Wall Thickness".	The Height and Width parameters that are "Set as Nominal" in the Settings dialog box (Custom Settings, Nominal Sizes and Tolerance page), and the thickness of the Wall in which the W/D is located. (Same as the W/D Opening Nominal Volume.)	

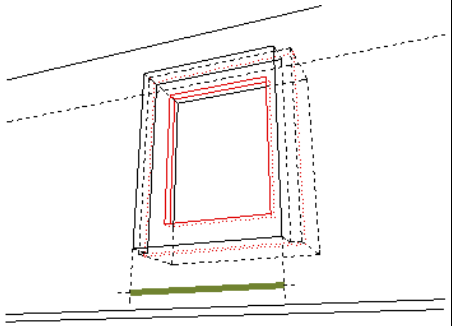
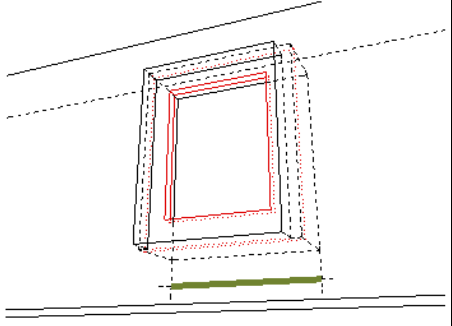
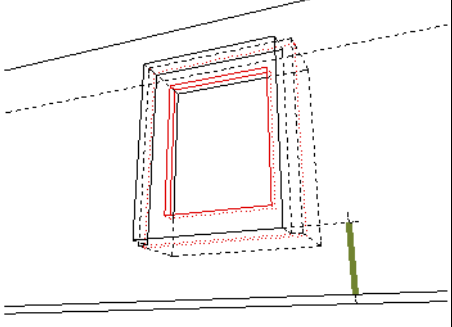
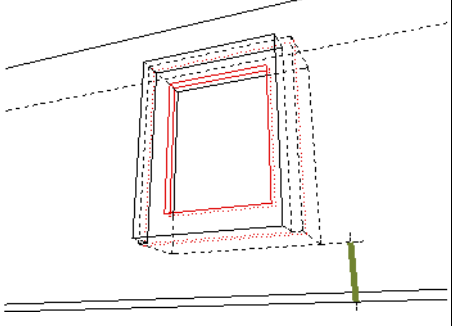
Parameter	Short Description	Long Description	
Nominal W/D Opening Height on the Reveal Side	Wallhole Height, reduced by the head+sill of the Reveal.	Relevant for openings from the Archicad 10 library, if the "Nominal Size = Jamb Inner Size" parameter is ON.	
Nominal W/D Opening Height on the Side Opposite to the Reveal Side	Wallhole Height, reduced by the head+sill of the Reveal.	Relevant for openings from the Archicad 10 library, if the "Nominal Size = Jamb Inner Size" parameter is ON.	
Nominal W/D Opening Surface Area on the Reveal Side	Height between sill and head x Width between Side jambs.	Relevant for openings from the Archicad 10 library, if the "Nominal Size = Jamb Inner Size" parameter is ON.	
Nominal W/D Opening Surface Area on the Side Opposite to the Reveal Side	Wallhole Height x Wallhole Width.	Relevant for openings from the Archicad 10 library, if the "Nominal Size = Jamb Inner Size" parameter is ON.	
Nominal W/D Opening Width on the Reveal Side	Wallhole Width, reduced by the Side jambs of the Reveal.	Relevant for openings from the Archicad 10 library, if the "Nominal Size = Jamb Inner Size" parameter is ON.	

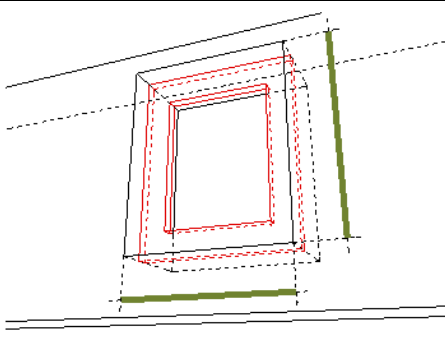
Parameter	Short Description	Long Description	
Nominal W/D Opening Width on the Side Opposite to the Reveal Side	Wallhole Width.	Relevant for openings from the Archicad 10 library, if the "Nominal Size = Jamb Inner Size" parameter is ON.	
Orientation	L for Left, or R for Right.	The direction in which the door or window is opened. Orientation is also indicated by the Opening line on the Floor Plan.	
Reveal Dimensions	Width and Height of the window.	The Height and Width parameters defined for Reveal Dimensions, in the Door/Window Settings dialog box (Custom Settings, Nominal Sizes and Tolerance page)	
Sill/Header Value	Distance from the header (or from the sill) to the Wall Base (or to the current Story), as defined by the user.	Sill or header value as defined in Preview and Positioning panel of the Settings Dialog box.	
Unit Dimensions	Width and Height of door/window unit, as defined in Settings dialog box.	The "unit" is the door/window frame; it is the equivalent of the Wallhole less the tolerance value.	
View from Opening Side	View of the door/window from the Opening Side.		

Parameter	Short Description	Long Description	
View from Reveal Side	View of the door/window from the Reveal Side.		
View from Side Opposite to Opening Side	View of the door/window from the Side Opposite to the Opening Side.		
View from Side Opposite to Reveal	View of the door/window from the Side Opposite to the Reveal.		
W/D Head Height on the Reveal Side	Distance from Wall base to the header.	Defined as the sum of the sill Height plus the door/window Height, taking the Reveal geometry into account.	

Parameter	Short Description	Long Description	
W/D Head Height on the Side Opposite to the Reveal Side	Distance from Wall base to the header on the Opposite to the Reveal Side.	Defined as the sum of the sill Height plus the door/window Height, taking the Reveal geometry into account.	
W/D Nominal Head Height	Distance from Wall base to the header, taking the Reveal geometry into account. Defined as the sum of the sill Height plus the door/window Height, taking the Reveal geometry into account.		
W/D Nominal Sill Height	Distance from Wall base to the top of the sill. Includes the sill value plus the sill Height (if any) of the Reveal.		
W/D Opening Height on the Reveal Side	Wallhole Height, less the head+sill of the Reveal.	Relevant for openings from the Archicad 10 library, if the "Nominal Size = Jamb Inner Size" parameter is OFF.	

Parameter	Short Description	Long Description	
W/D Opening Height on the Side Opposite to the Reveal Side	Wallhole Height.	Relevant for openings from the Archicad 10 library, if the "Nominal Size = Jamb Inner Size" parameter is OFF.	
W/D Opening Nominal Surface Area	Wallhole Height x Wallhole Width.	Relevant for openings from the Archicad 10 library, if the "Nominal Size = Jamb Inner Size" parameter is ON.	
W/D Opening Nominal Volume	Volume, calculated by "Height x Width x Wall Thickness".	These are the Height and Width parameters of the Opening size defined as "Nominal" in the opening's Settings dialog box, and the thickness of the Wall in which the Opening is located. (Same as the W/D Opening Nominal Volume.)	
W/D Opening Surface Area on the Reveal Side	Height between sill and head x Width between Side jambs.	Relevant for openings from the Archicad 10 library, if the "Nominal Size = Jamb Inner Size" parameter is OFF.	
W/D Opening Surface Area on the Side Opposite to the Reveal Side	Wallhole Height x Wallhole Width.	Relevant for openings from the Archicad 10 library, if the "Nominal Size = Jamb Inner Size" parameter is OFF.	


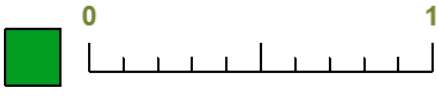
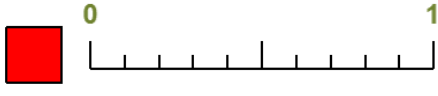

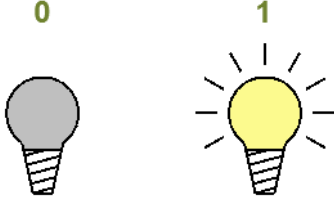
Parameter	Short Description	Long Description	
W/D Opening Width on the Reveal Side	Wallhole Width, less the Side jambs of the Reveal.	Relevant for openings from the Archicad 10 library, if the "Nominal Size = Jamb Inner Size" parameter is OFF.	
W/D Opening Width on the Side Opposite to the Reveal Side	Wallhole Width.	Relevant for openings from the Archicad 10 library, if the "Nominal Size = Jamb Inner Size" parameter is OFF.	
W/D Sill Height on the Reveal Side	Distance from Wall base to the sill, taking the sill jamb depth of Reveal into account.		
W/D Sill Height on the Side Opposite to the Reveal Side	Distance from Wall base to the sill, regardless of sill jamb depth of Reveal.		
Wall Structure	Name of Building Material or Profile of the Wall in which the door/window is placed.		
Wall Thickness	Thickness of the Wall in which the door/window is placed		

Parameter	Short Description	Long Description	
Wallhole Dimensions	Width and Height of the door/window Wallhole as defined in Settings dialog box (Parameters panel or Custom Settings panel).	The "Wallhole" is the Opening that is cut into the Wall, and is composed of the dimensions of the unit (frame) plus any tolerance values.	

Fill Listing Parameters

Parameter	Short Description	Long Description
Fill Category	Used for a 2D Fill element. As set in Fill Settings dialog: - Drafting fill - Cut fill - Cover fill - Building Material (Cut Fill) - Surface (Cover, Color or Texture)	
Fill Holes Perimeter	Perimeter of all holes in the Fill element	
Fill Perimeter	Perimeter of the Fill element	
Fill Type	Type of the Fill pattern (e.g. Solid, Vectorial, Symbol, Image)	To list the name of the Fill type instead: Use the Fill Type parameter in the "General" parameter group.
Holes Surface Area	Surface area of all holes in the Fill element	
Image name	Name of the image-type file (for Fills with Fill type of "Image")	Defined in the Fill Texture panel of the Fills dialog box, if the Fill type is Image
Transparency	For transparent Solid Fills, lists the percentage transparency.	E.g. 25%, 50%, 75%. All other fills have zero Transparency.

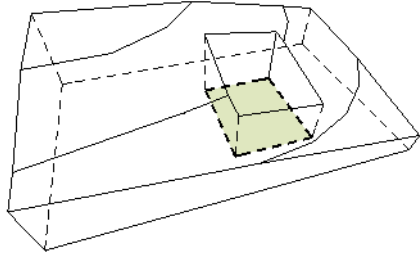
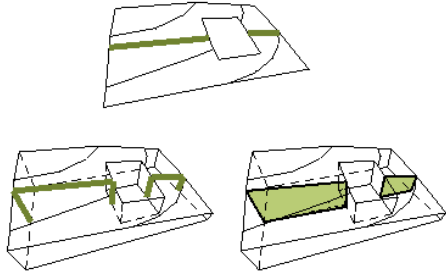
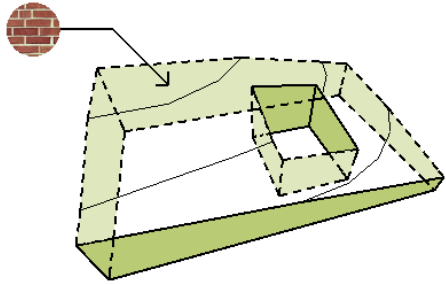
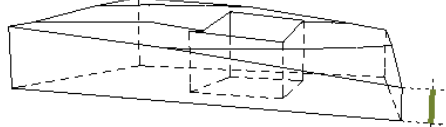
Lamp Listing Parameters

Parameter	Short Description	Long Description	
Color Blue	Blue component of the lamp light.	Based on RGB value in the Color field of the Parameters Panel of Lamp Settings.	
Color Green	Green component of the lamp light.	Based on RGB value in the Color field of the Parameters Panel of Lamp Settings.	
Color Red	Red component of the lamp light.	Based on RGB value in the Color field of the Parameters Panel of Lamp Settings.	
Light Intensity [0-100]	Light intensity value, as set using the Main Intensity slider in Light Settings (Basic Light Parameters).		
Light is on (Yes/No)	Light's on/off setting	Set with the lightbulb icons in Light Settings (Basic Light Parameters). Affects PhotoRenderings.	

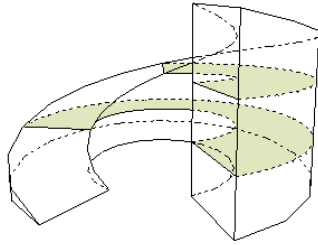
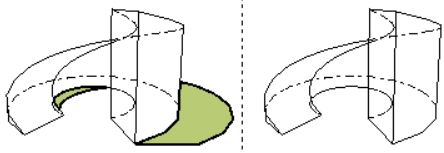
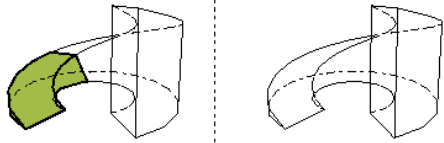
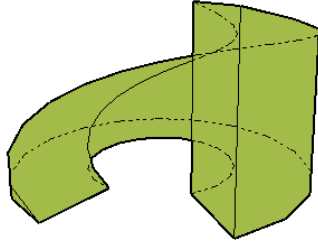
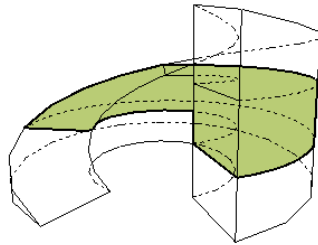
MEP Listing Parameters

Parameter	Short Description	Long Description
Branch Diameter	Diameter of the branch segment.	Relevant for circular branch segments of routes. Defined in the Duct and Pipe Branch Default Settings dialog box under Segment.
Branch Height	Height of the branch segment.	Relevant for rectangular branch segments of routes. Defined in the Duct and Cable Carrier Default Settings dialog box under Segment.
Branch Width	Width of the branch segment.	Relevant for rectangular branch segments of routes. Defined in the Duct and Cable Carrier Default Settings dialog box under Segment.
Diameter	Diameter of the segment.	Relevant for circular segments of routes. Defined in the Duct and Pipe Route Default Settings dialog box under Segment.
Height	Height of the segment.	Relevant for rectangular segments of routes. Defined in the Duct and Pipe Route Default Settings dialog box under Segment.
Insulation Thickness	Thickness of the insulation	
Length	Length of the segment.	
MEP System	Attribute assigned to MEP route (e.g Fresh Air, Heating)	
Slope	The steepness of the segment.	Relevant to the route tools. Defined in the Duct, Cable Carrier and Pipe Route Default Settings dialog box.
Width	Width of the segment.	

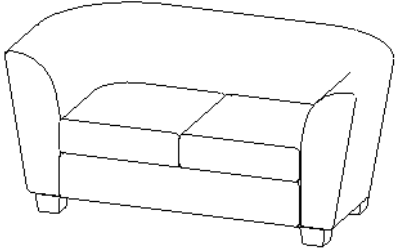
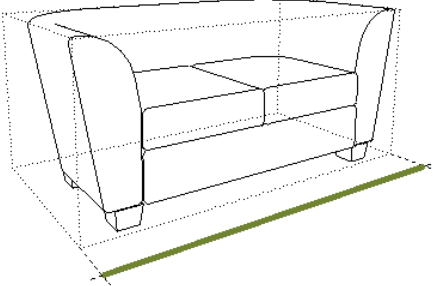
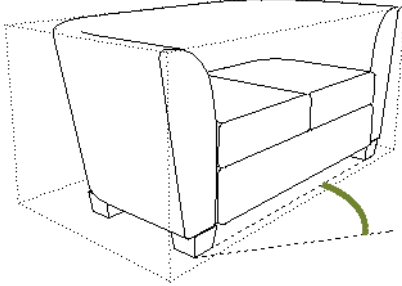
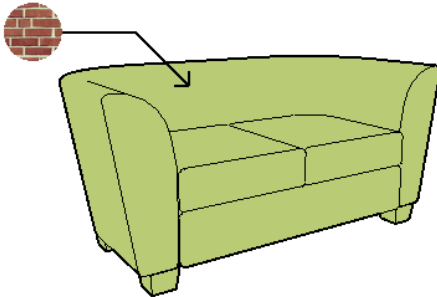
Mesh Listing Parameters

Parameter	Short Description	Long Description	
Holes Surface Area	Total top (2D) surface area of all the holes in the Mesh		
Mesh Type	Construction method used to create Mesh:	Surface only, Surface with Skirt or Solid Body	
Side Surface	Name of the surface assigned to the side of the Mesh.		
Skirt Level	For Meshes with a skirt, the height of the skirt (difference between the base and top values as shown in the Info Box.)		

Morph Listing Parameters

Parameter	Short Description	Long Description	
Area (by Story)	Area of the Morph broken down by Story		
Cast Shadow	On or Off	Set checkbox in Morph Settings Model Panel.	
Classification by DIN	Available only if you are using the Flächenberechnung (Area List) Add-on (used in German-speaking markets).	Set this value in Morph Settings - Classification and Properties Panel	
Receive Shadow	On or Off	Set checkbox in Morph Settings Model Panel.	
Surface	Name of the surface assigned to the entire Morph.		
Volume (by Story)	Volume of the Morph broken down by Story		

Object/Lamp Listing Parameters

Parameter	Short Description	Long Description	
3D Front Axonometry	Shaded 3D Symbol of the Object.		
Length (A)	Length of the Object as defined in the "Dimension 2" field of the Preview and Positioning Panel of Object Settings.		
Rotation Angle	Rotation of the Object from its default position	Rotation either by rotation angle in the Preview and Positioning Panel of Object Settings, or rotating it graphically in the project.	
Surface	Surface of the Object as assigned in the Model tab page of Object Settings.		

Opening Listing Parameters

Parameter	Short Description	Long Description
Center Height to Home Story	For opening in Wall or Beam.	Home Story: same as the Home Story of the opening's associated element.
Center Height to Project Zero	For opening in Wall or Beam.	
Center Height to Wall Bottom	For opening in Wall.	
Center Height to Wall Top	For opening in Wall.	
Cut Symbol Name	Name of the Symbol for the opening's cut display	Opening Settings, Symbol panel
End Surface	Name of the surface applied to the end(s) of a limited Opening body	A limited Opening has one or two ends, where it meets but does not cut through the host element (e.g. niche or recess). Has a value only if you used a surface override for the Opening ends (Opening Settings, Model panel)..
End Surface Area (Net, total)	Surface area of (one or both) ends of a limited Opening.	Reduced by any SEO's or holes.
Extrusion Surface	Name of the surface applied to the sides of the Opening body, where it cuts through the host elements	Has a value only if you used a surface override for the Opening extrusion (Opening Settings, Model panel).
Extrusion Surface Area (Net)	Surface area of the Opening body, where it cuts through the host elements.	Reduced by any SEO's or holes
Header Height to Home Story	For opening in Wall or Beam.	
Header Height to Project Zero	For opening in Wall or Beam.	
Header Height to Wall Bottom	For opening in Wall.	
Header Height to Wall Top	For opening in Wall.	
IDs of Cut Elements	ID(s) of all the elements cut by this opening.	Use " Parent ID " to list the opening's associated element.
Limits	Infinite, or limited on one or both sides	
Number of Cut Elements	Number of elements cut by this opening	
Orientation	Direction of the opening relative to its associated element: Aligned, Vertical, Horizontal or Custom angle	

Parameter	Short Description	Long Description
Shape	Square, Circle or Polygonal	
Sill Height to Home Story	For opening in Wall or Beam.	Home Story is the same as that of the opening's associated element.
Sill Height to Project Zero	For opening in Wall or Beam.	
Sill Height to Wall Bottom	For opening in Wall.	
Sill Height to Wall Top	For opening in Wall.	
Total Thickness	Combined thickness of all elements physically cut by this opening.	
Uncut Symbol Name	Name of the Symbol for the opening's uncut display	Opening Settings, Symbol panel

Railing Listing Parameters

Use these parameters to list data of the Railing element as a whole.

Parameter	Short Description	Long Description
Number of Balusters	The number of Balusters in a Railing.	
Number of Handrails	The number of Handrails in a Railing.	
Number of Inner Posts	The number of Inner Posts in a Railing.	
Number of Panels	The number of Panels in a Railing.	
Number of Posts	The number of Posts in a Railing.	
Number of Rails	The number of Rails in a Railing.	
Number of Toprails	The number of Toprails in a Railing.	
Reference Line Horizontal Length	The horizontal length of the reference line of a Railing.	

Related:

[Railing Segment Listing Parameters](#)

[Railing Members Listing Parameters](#)

Railing Segment Listing Parameters

Use these parameters to list data for a Railing Segment.

Parameter	Short Description	Long Description
Number of Balusters (in a Segment)	The number of Balusters in a Railing Segment.	
Number of Handrails (in a Segment)	The number of Handrails in a Railing Segment.	
Number of Inner Posts (in a Segment)	The number of Inner Posts in a Railing Segment.	
Number of Panels (in a Segment)	The number of Panels in a Railing Segment.	
Number of Rails (in a Segment)	The number of Rails in a Railing Segment.	
Number of Toprails (in a Segment)	The number of Toprails in a Railing Segment.	

Related:

[Railing Listing Parameters](#)

[Railing Members Listing Parameters](#)

Railing Members Listing Parameters

Use these parameters to list index numbers of Railing members (Posts, Balusters, etc.) within a Railing Segment.

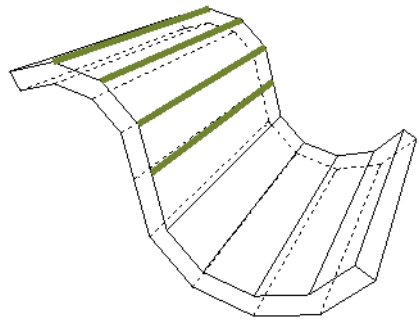
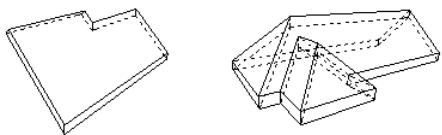
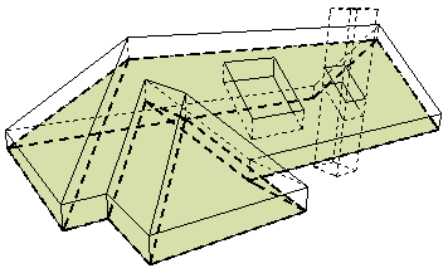
Parameter	Short Description	Long Description
Member Index (Inside a Segment)	The index number of the Railing member in a segment.	
Segment Index	The index number of a Railing Segment.	

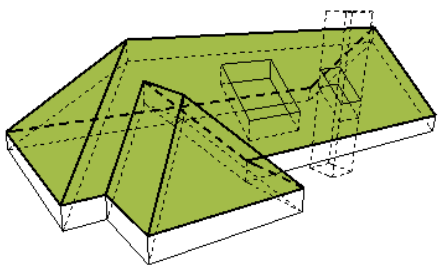
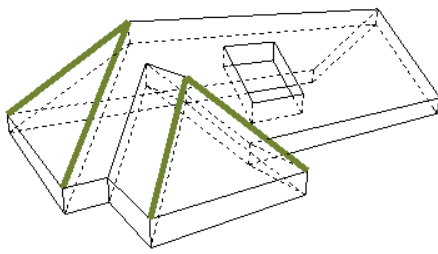
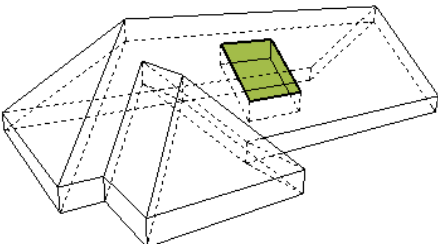
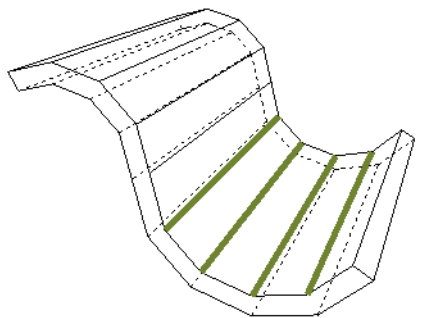
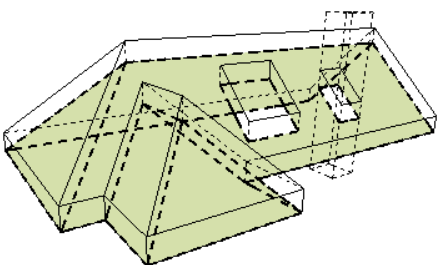
Related:

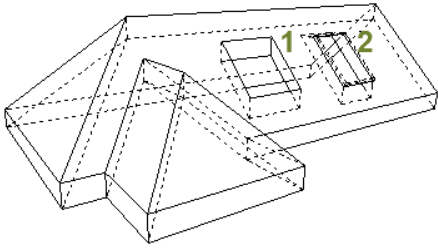
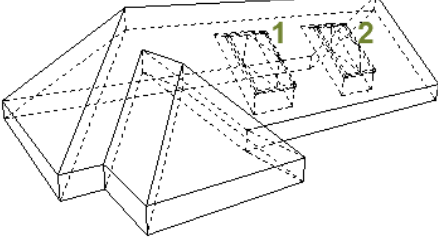
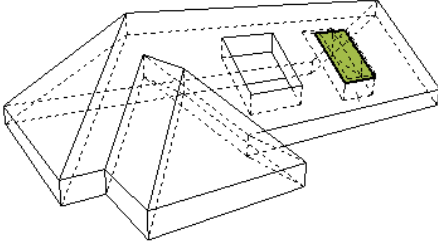
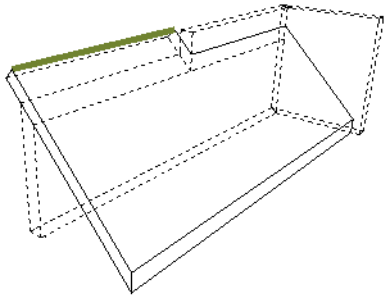
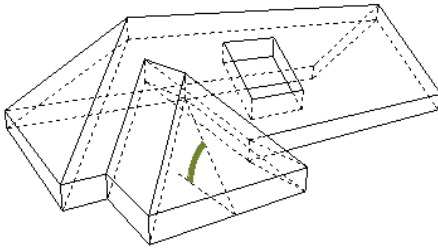
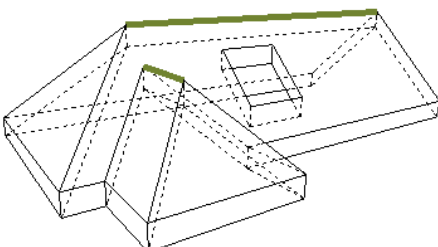
[Railing Listing Parameters](#)

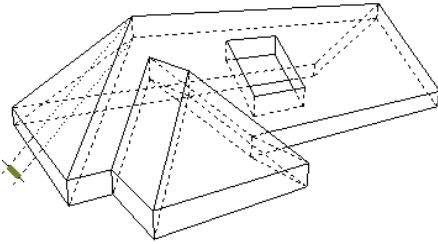
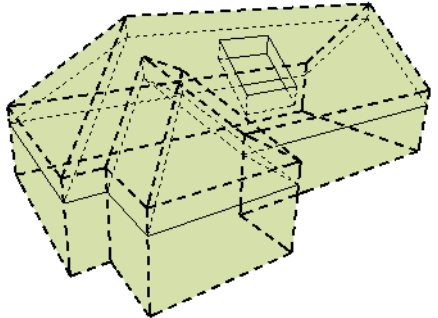
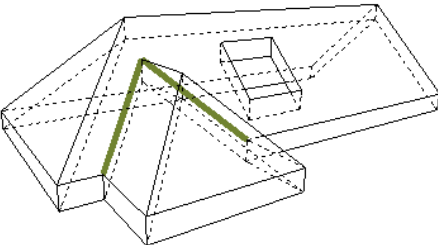
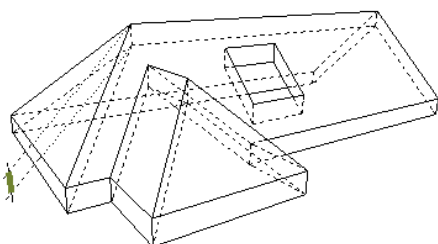
[Railing Segment Listing Parameters](#)

Roof Listing Parameters

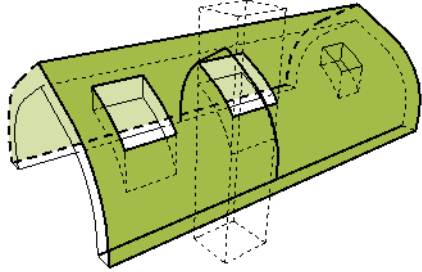
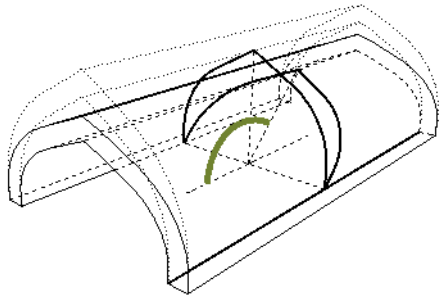
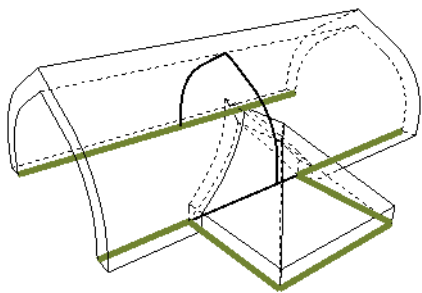
Parameter	Short Description	Long Description	Parameter
Dome Connection length	Length of edges whose Type is set to "Roof Transition - Dome" in the Custom Edge Settings dialog box.	Typically, the edge of two adjacent Roof planes forming a convex angle.	
Eaves Length	Total length of all edges whose Type is set to "Eaves," in the Custom Edge Settings dialog box.		
End Wall Connection Length	Total length of all edges whose Type is set to "End Wall" in the Custom Edge Settings dialog box.	Typically, this is an edge which joins a Wall and slopes down along the Wall.	
Gables Length	Total length of all edges whose Type is set to "Gable" in the Custom Edge Settings dialog box.		
Geometry Method	Single Plane, Multi-plane	Roof Geometry method	
Surface Area of the Roof Bottom (Gross)	Surface area of the bottom of the Roof, including the surface of any holes.	Reduced by trims. SEO's are ignored	
Surface Area of the Roof Edges (Gross)	Surface area of the Roof edges, including the surface of any holes.	Reduced by trims. SEO's are ignored	

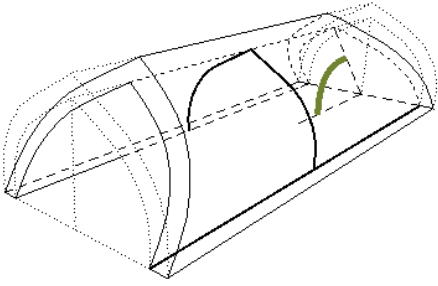
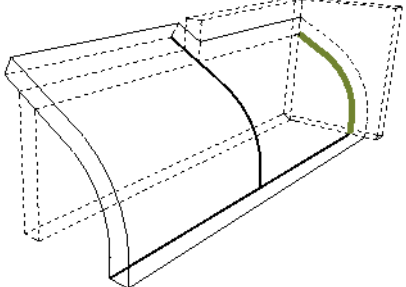
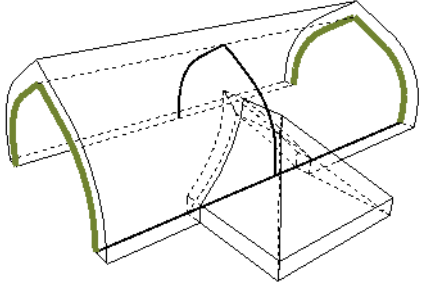
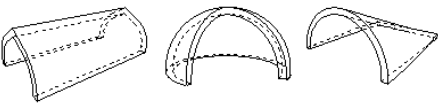
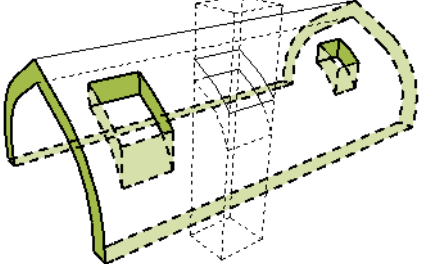
Parameter	Short Description	Long Description	Parameter
Surface Area of the Roof Top (Gross)	Surface area of the top of the Roof, including the surface of any holes.	Reduced by trims. SEO's are ignored	
Hips Length	Length of edges whose Type is set to "Hip" in the Custom Edge Settings dialog box.	Typically, the edge of two adjacent planes that meet as a hip	
Holes Surface Area	Total surface area of all holes in the Roof.		
Hollow Connection Length	Length of edges whose Type is set to "Roof Transition - Hollow" in the Custom Edge Settings dialog box.	Typically, the edge of two adjacent Roof planes forming a concave angle.	
Surface Area of the Roof Bottom (Net)	Surface area of the bottom of the Roof	Reduced by holes, trims and Solid Element Operations.	

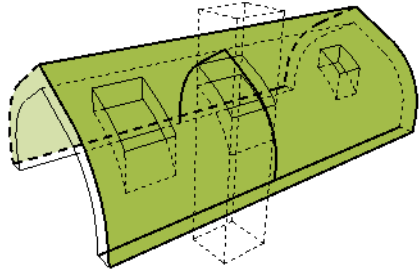
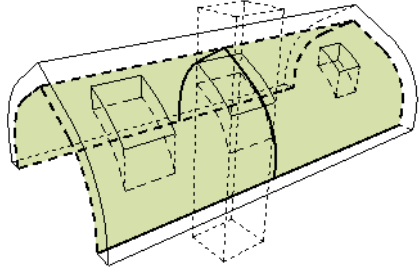
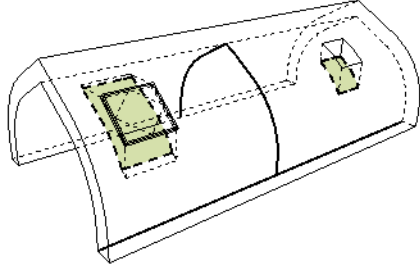
Parameter	Short Description	Long Description	Parameter
Number of Holes	Number of holes in the Roof	Holes are either created in the Roof manually, or they are cut by Skylights placed in the Roof.	
Number of Skylights	Number of Skylights placed into the Roof.		
Openings Surface Area	Total surface area of all Skylights placed in the Roof.		
Peaks Length	Length of edges whose Type is set to "Peak" in the Custom Edge Settings dialog box.	Typically, the edge at the top end of a single-plane Roof.	
Pitch	Pitch of the Roof.	Pitch is set in the Geometry and Positioning Panel but can be customized for any Roof Plane.	
Ridges Length	Length of edges whose Type is set to "Ridge" in the Custom Edge Settings dialog box.	Typically, the edge of two adjacent Roof planes that meet to form a ridge.	

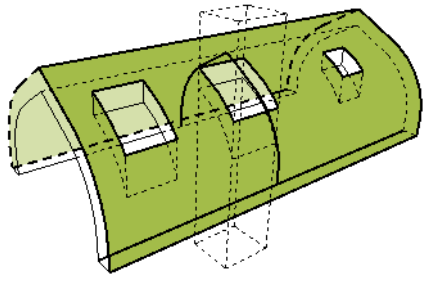
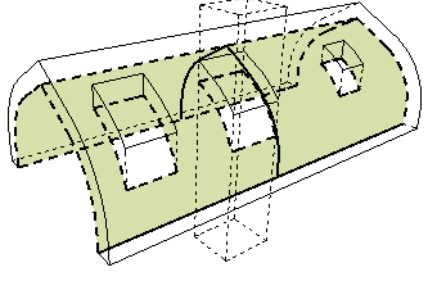
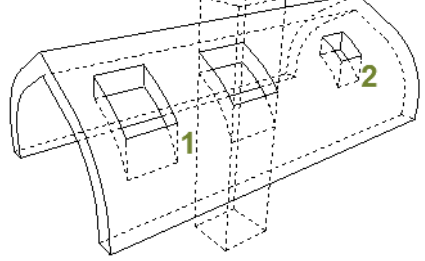
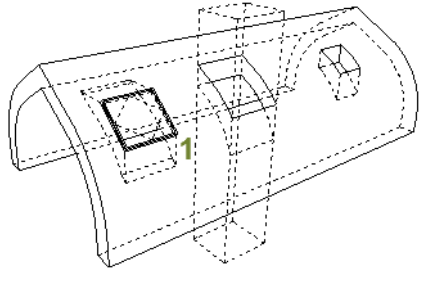
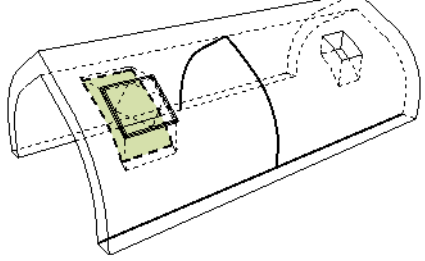
Parameter	Short Description	Long Description	Parameter
Roof Level	In case of a multi-level Roof, indicates the Roof Level to which the listed Roof plane belongs.	Roof Level numbers can be seen in the Geometry and Positioning panel of Roof Settings or the Info box.	
Side Wall Connection Length	Total length of all edges whose Type is set to "Side Wall" in the Custom Edge Settings dialog box.	Typically, a Roof edge which joins a Wall and runs along the Wall horizontally.	
Thickness	Roof thickness, measured perpendicularly from the reference surface.	Thickness value is set in the Geometry and Positioning panel of Roof Settings or the Info box. (Can be displayed either as a perpendicular or vertical measure).	
Trimming Body	Pivot Lines Down or Contours Down	Set in the Model Panel of Roof Settings. Determines the geometry of the Trimming Body which acts upon other elements when they are trimmed by this Roof.	
Valleys Length	Length of edges whose Type is set to "Valley" in the Custom Edge Settings dialog box.		
Vertical Thickness	Roof thickness measured vertically from the reference surface.	Thickness value is set in the Geometry and Positioning panel of Roof Settings or the Info box (Can be displayed either as a horizontal or vertical measure).	

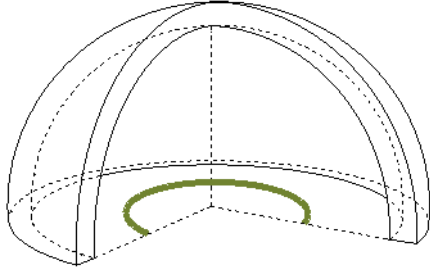
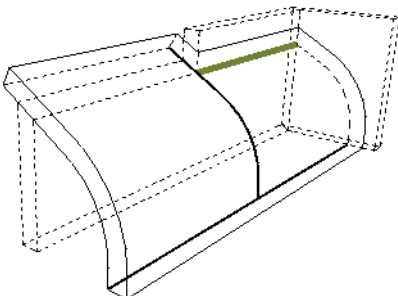
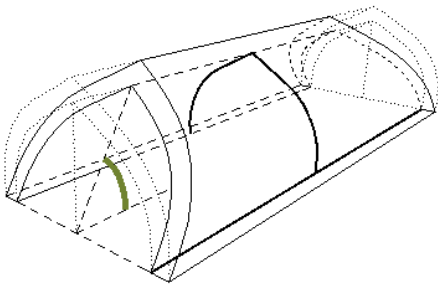
Shell Listing Parameters

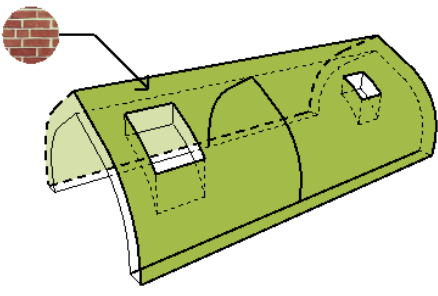
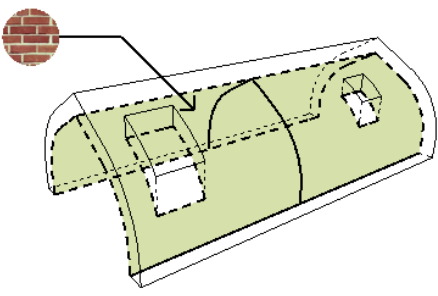
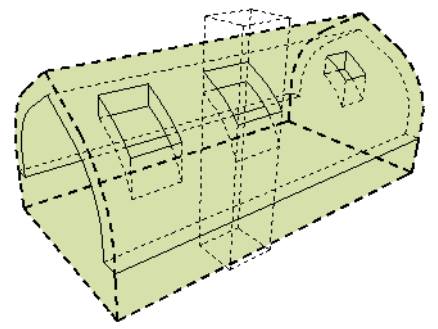
Parameter	Short Description	Long Description	Parameter
Surface Area of the Shell Opposite to Reference Side (Conditional)	Surface area of the side opposite the Reference side of the Shell - this area may be conditioned on a defined Calculation Rule.	Reference side: The Shell membrane (the "top" of the Shell.) For a Composite Shell: The first listed skin of the composite structure, in Composite Settings. Condition: Subtracts the area of holes only if they exceed a given size, as defined in Options > Project Preferences > Calculation Rules Reduced by trims and SEO's.	
Distortion Angle	For an Extruded Shell: Angle of the Shell profile, measured to the Shell plane. By default: 90 degrees, but changing it enables you to extrude the Shell profile at a angle, creating a distorted Shape Shell. For a Revolved Shell: Angle of the axis of the revolved Shell, measured to the Shell plane angle.	Distortion angle value is set in the Extrusion Properties or Revolution Properties panel of Shell Settings or in the Info box.	
Dome Connection length	Length of edges whose Type is set to "Roof Transition - Dome" in the Custom Edge Settings dialog box.	Typically, the edge of two adjacent Shell planes forming a convex angle.	
Eaves Length	Total length of all edges whose Type is set to "Eaves," in the Custom Edge Settings dialog box.		

Parameter	Short Description	Long Description	Parameter
End Angle	For an extruded Shell only. Angle of the Shell profile, measured to the Shell's inclination angle, at the second end of the extruded Shell. End angle value is set in the Extrusion Properties panel of Shell Settings or in the Info box.		
End Wall Connection Length	Total length of all edges whose Type is set to "End Wall" in the Custom Edge Settings dialog box.	Typically, this is an edge which joins a Wall and slopes down along the Wall.	
Gables Length	Total length of all edges whose Type is set to "Gable" in the Custom Edge Settings dialog box.		
Geometry Method	Geometry method used to create the Shell: Extruded, Revolved, or Ruled		
Surface Area of the Shell Edges (Gross)	Surface area of the Shell edges, including the surface of any holes.	Reduced by trims. SEO's are ignored	

Parameter	Short Description	Long Description	Parameter
Surface Area of the Shell Opposite to Reference Side (Gross)	Surface area of the side opposite the Reference side, including the surface of any holes.	Reduced by trims. SEO's are ignored	
Surface Area of the Shell Reference Side (Gross)	Surface area of the Reference side of the Shell, including the surface of any holes.	Reference side: The Shell membrane (the "top" of the Shell.) For a Composite Shell: The first listed skin of the composite structure, in Composite Settings. Reduced by trims. SEO's are ignored.	
Hips Length	Length of edges whose Type is set to "Hip" in the Custom Edge Settings dialog box.	Typically, the edge of two adjacent planes that meet as a hip	
Holes Surface Area	Total surface area of all holes in the Shell.		
Hollow Connection Length	Length of edges whose Type is set to "Roof Transition - Hollow" in the Custom Edge Settings dialog box.	Typically, the edge of two adjacent Shell planes forming a concave angle.	
Surface Area of the Shell Edges (Net)	Total surface of all edges of the Shell	Reduced by holes, trims and Solid Element Operations.	

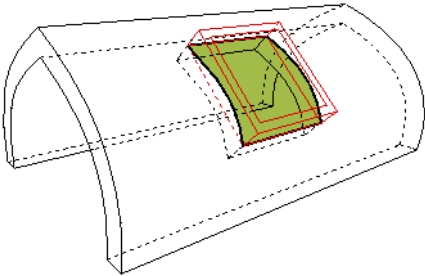
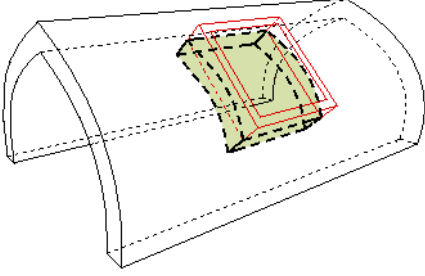
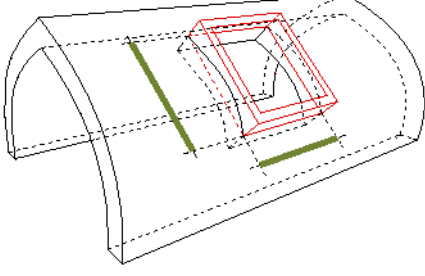
Parameter	Short Description	Long Description	Parameter
Surface Area of the Shell Opposite to Reference Side (Net)	Surface area of the side opposite the Reference side of the Shell	Reduced by holes, trims and Solid Element Operations.	
Net Surface Area of the Reference Side	Surface area of the Reference side of the Shell	Reference side: The Shell membrane (the "top" of the Shell.) For a Composite Shell: The first listed skin of the composite structure, in Composite Settings. Reduced by holes, trims and Solid Element Operations.	
Number of Holes	Number of holes in the Shell	Holes are either created in the Shell manually, or they are cut by Skylights placed in the Shell.	
Number of Skylights	Number of Skylights placed into the Shell.		
Openings Surface Area	Total surface area of all Skylights placed in the Shell.		
Peaks Length	Length of edges whose Type is set to "Peak" in the Custom Edge Settings dialog box.	Typically, the edge at the top end of a single-plane Shell.	

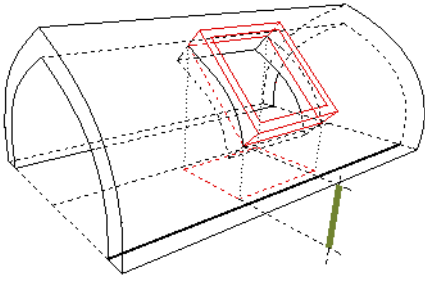
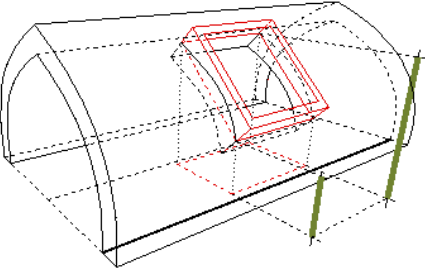
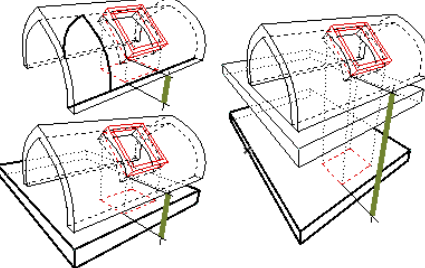
Parameter	Short Description	Long Description	Parameter
Revolution Angle	For a Revolved Shell only. Angle by which the Shell is rotated around its axis. Revolution angle value is set graphically, when creating the Shell, or in the Revolution Properties panel of Shell Settings or in the Info box.		
Ridges Length	Length of edges whose Type is set to "Ridge" in the Custom Edge Settings dialog box.	Typically, the edge of two adjacent Shell planes that meet to form a ridge.	
Side Wall Connection Length	Total length of all edges whose Type is set to "Side Wall" in the Custom Edge Settings dialog box.	Typically, a Shell edge which joins a Wall and runs along the Wall horizontally.	
Start Angle	For an extruded Shell only. Angle of the Shell profile, measured to the Shell's inclination angle, at the starting end of the extruded Shell. Start angle value is set in the Extrusion Properties panel of Shell Settings or in the Info box.		

Parameter	Short Description	Long Description	Parameter
Surface of the Shell on the Opposite to Reference Side	Name of the Surface assigned to the side opposite the Reference side.	Assign surfaces in the Model panel of Shell Settings	
Surface of the Shell on the Reference Side	Name of the Surface assigned to the Reference side.	Reference side: The Shell membrane (the "top" of the Shell.) For a Composite Shell: The first listed skin of the composite structure, in Composite Settings. Assign surfaces in the Model panel of Shell Settings	
Trimming Body	Downward Extrusion, Upward Extrusion, or Editable.	Set in the Model Panel of Shell Settings. Determines the geometry of the Trimming Body which acts upon other elements when they are trimmed by this Shell.	

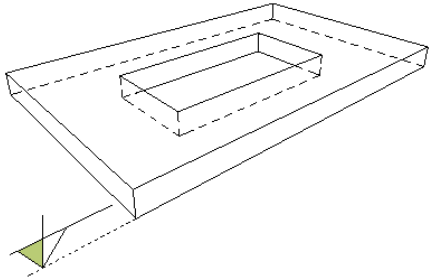
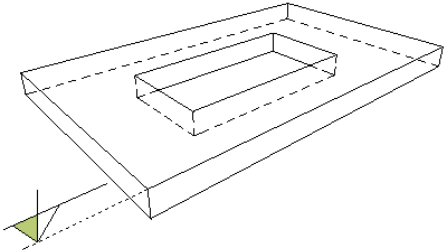
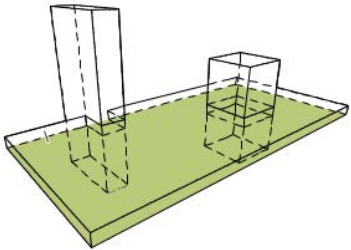
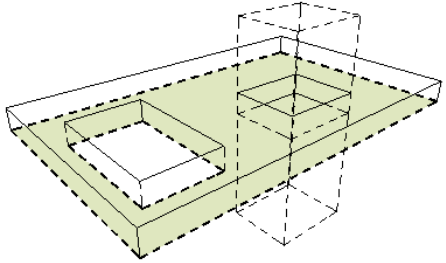
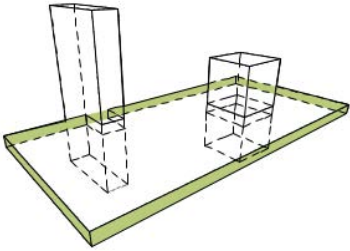
Skylight Listing Parameters

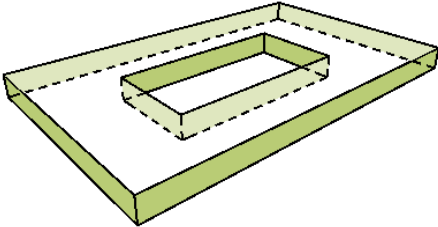
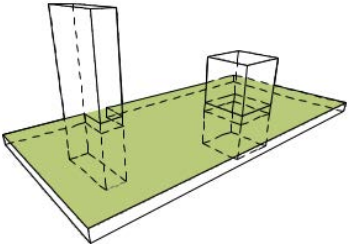
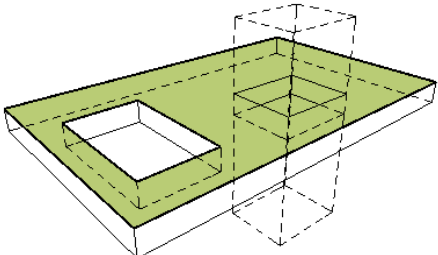
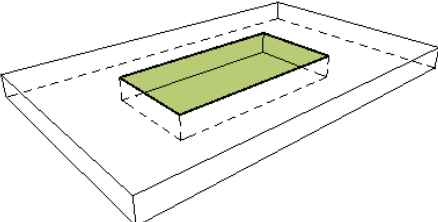
Parameter	Short Description	Long Description	
Acoustic Rating	Acoustic rating value (if any), as defined in Window Settings (Custom Settings, Descriptions page).		
Constraint in Roof	Either "Horizontal" or "Vertical", as set in the Preview and Positioning Panel of Skylight Settings.	Defines how the Skylight will be constrained as it shifts position if the underlying Roof geometry is modified.	
Curb Height	Distance by which Skylight frame extrudes out of the Roof/Shell plane.	A value of 0 means the Skylight lies flat on the plane.	
Fire Rating	Fire rating value (if any), as defined in Window Settings (Custom Settings, Descriptions page).		
Header Height	Vertical Height of the Skylight header	Header or Sill Height is measured from the Anchor (e.g. Story 0), as defined in Preview and Positioning Panel.	
Marker Text	The Marker Text specified in the Settings dialog box, in the Dimension Marker Panel, under Customize Texts-Marker Texts parameter.	This text is equivalent to the element's ID number by default, but a custom text may be assigned here.	

Parameter	Short Description	Long Description	
Shell/Roof Opening Area	Surface area of the opening cut by the Skylight.		
Shell/Roof Opening Dimensions (Archicad 20)	Width x Height of the opening cut by the Skylight.		
Shell/Roof Opening Volume	Volume of the opening cut by the Skylight.	Volume calculation based on hole that results from cutting the GDL openings out of the Shell/Roof.	
Shell/Roof Opening W x H Size			
Shell/Roof Opening W x H x T Size			
Shell/Roof Thickness	Thickness of the parent Roof/Shell in which the Skylight is placed.		

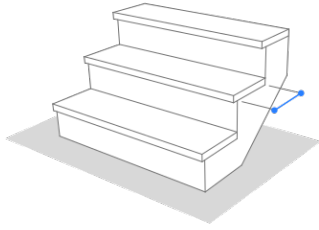
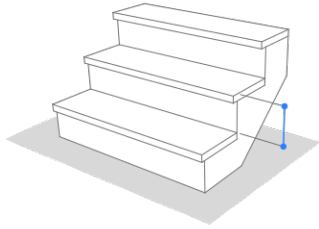
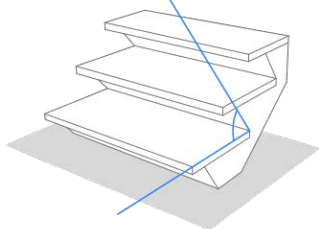
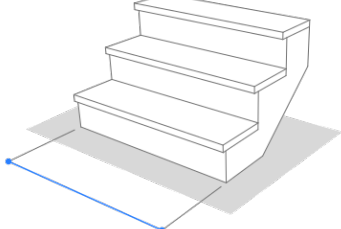
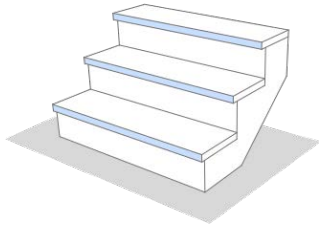
Parameter	Short Description	Long Description	
Sill Height	Height of the Skylight Sill, as shown in the Preview and Positioning Panel	Header or Sill Height is measured from the Anchor (e.g. Story 0), as defined in Preview and Positioning Panel.	
Sill/Header Value	Sill/header height value as defined in Preview and Positioning panel of the Settings Dialog box. (Value measures the distance from the header or from the sill, to the Roof Pivot Lines, to the current Story or another story, as defined by the user.)		
Unit Dimensions (Archicad 20)	Width and height of the Skylight unit, as defined in Settings dialog box (Parameters panel or Custom Settings panel).		
Vertical Anchor	Vertical Anchor as defined in the Preview and Positioning Panel. Header and Sill heights are measured to the vertical anchor.	Not relevant for Skylights placed in Shells.	

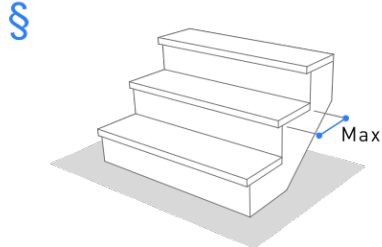
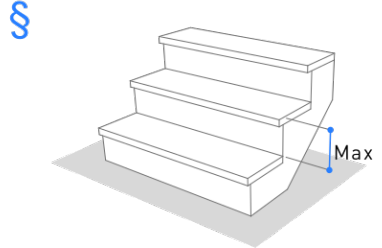
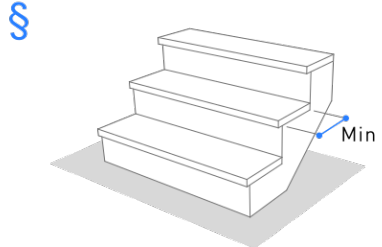
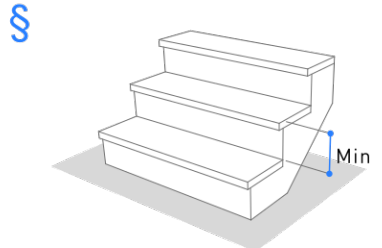
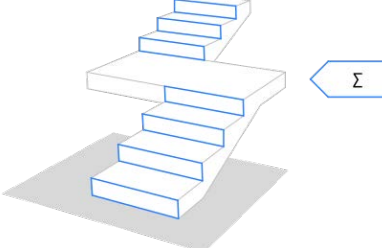
Slab Listing Parameters

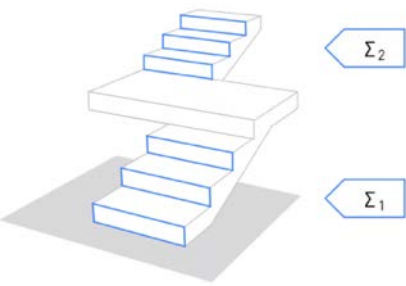
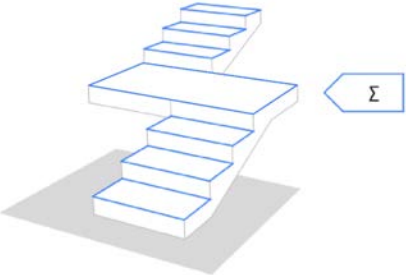
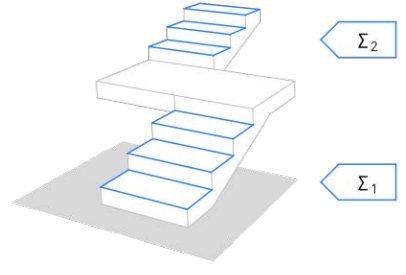
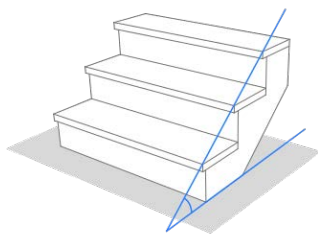
Parameter	Short Description	Long Description	
Elevation Bottom	Elevation of the Bottom of the Slab		
Elevation Top	Elevation of the Top of the Slab		
Surface Area of the Slab Bottom (Gross)	Area of the Surface of the Bottom of the Slab, including the Surface of any holes.	Reduced by trims. SEO's are ignored	
Surface Area of the Slab Bottom (Gross, with holes)	Area of the Surface of the Bottom of the Slab, subtracting the Surface of any holes.	Reduced by trims. SEO's are ignored	
Surface Area of the Slab Edges (Gross)	Area of the Surface of all the Slab edges. Hole edges are ignored.	Reduced by trims. SEO's are ignored	

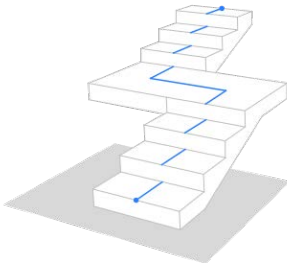
Parameter	Short Description	Long Description	
Surface Area of the Slab Edges (Gross, with holes)	Area of the Surface of all the Slab edges, including the edges of any holes.	Reduced by trims. SEO's are ignored	
Surface Area of the Slab Top (Gross)	Area of the Surface of the Top of the Slab, including the Surface of any holes.	Reduced by trims. SEO's are ignored	
Surface Area of the Slab Top (Gross, with holes)	Area of the Surface of the Top of the Slab, reduced by the Surface of any holes.	Reduced by trims. SEO's are ignored	
Holes Surface Area	Total Top Surface Area of all holes in the Slab, measured according to the Slab's 2D polygon.	SEO's are ignored	
Reference Plane Location	Top or Bottom. For a composite Slab, can also be Core Top or Core Bottom	Reference Plane location as defined in Slab Settings.	

Stair Listing Parameters

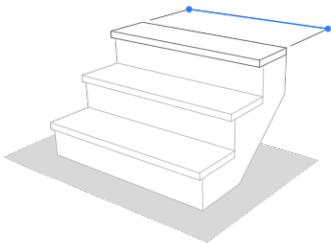
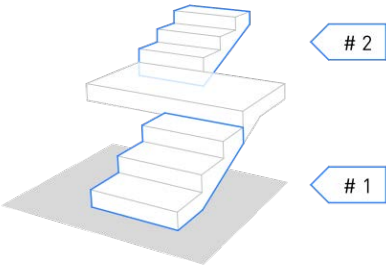
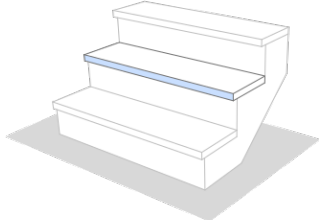
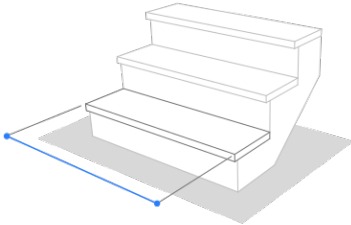
Parameter	Short Description	Long Description	
Default Going Depth	Going depth as defined in Stair Settings.		
Default Riser Height	Riser Height as defined in Stair Settings		
Default Riser Slant Angle	As defined in Stair Settings		
Default Stair Width	Width of the Stair as defined in Stair Settings.		
Front Surface Area	Total Surface area of the Stair from the front		

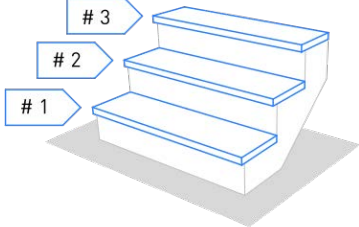
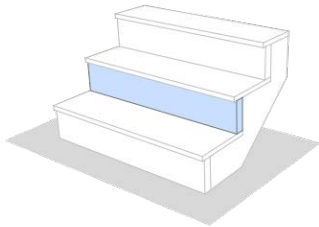
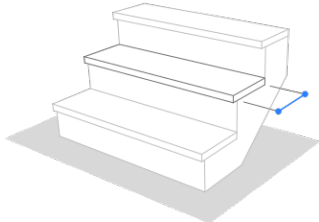
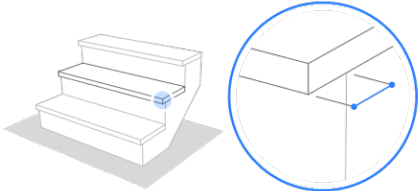
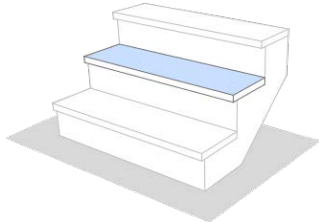
Parameter	Short Description	Long Description	
Maximum Going (by Rule)	As defined in Stair Settings (Rules & Standards Panel)		
Maximum Riser Height (by Rule)	As defined in Stair Settings (Rules & Standards Panel)		
Minimum Going (by Rule)	As defined in Stair Settings (Rules & Standards Panel)		
Minimum Riser Height (by Rule)	As defined in Stair Settings (Rules & Standards Panel)		
Number of Risers (total)			

Parameter	Short Description	Long Description	
Number of Risers by Flights			
Number of Treads (total)			
Number of Treads by Flights			
Required Head-room Height	As defined in Stair Settings (Rules & Standards Panel)		
Stair Pitch			

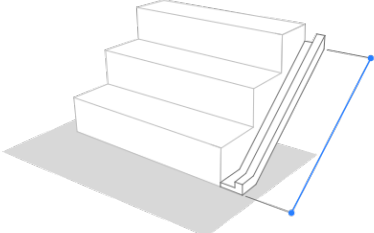
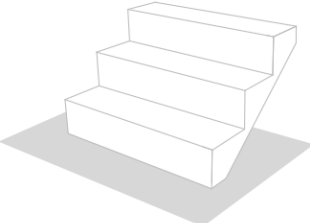
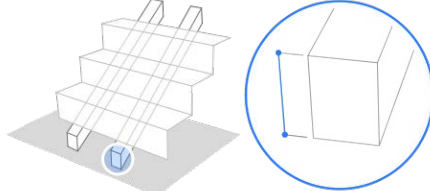
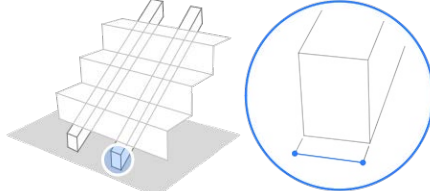
Parameter	Short Description	Long Description	
<p>Walking Line Length</p>	<p>Length of Calculated Walking Line.</p>	<p>Calculated Walking Line: defined in the Geometry and Positioning panel of Stair Settings. (Can differ from length of graphically edited Walking Line.)</p>	

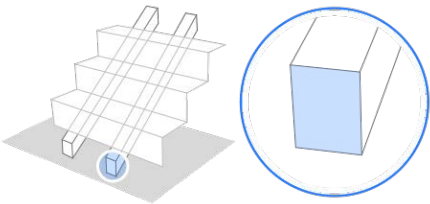
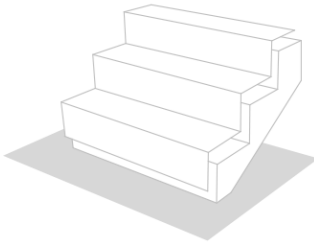
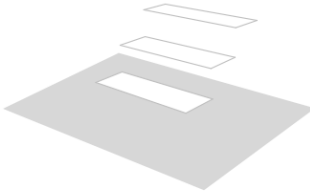
Stair Finish Listing Parameters

Parameter	Short Description	Long Description	
Back Width			
Flight Index	The Flight number to which the Tread/Riser belongs.	(A Tread/Riser on a Landing has a Flight Index of 0.)	
Front Surface Area	Surface area of the front of the tread (based on its vertical projection) The visible part of the Riser viewed from the front		
Front Width			
Segment Type	Landing, Run, Divided Landing, Winder Turn		

Parameter	Short Description	Long Description	
Step Index	The Step number to which the Tread/Riser belongs		
Total Surface Area	Total Surface area of the Riser		
Tread Flight Length	Going depth plus Tread Nosing		
Tread Going	Going depth		
Tread Nosing	Length of nosing's horizontal protrusion		
Tread Top Surface Area	Surface area of the top of the tread		

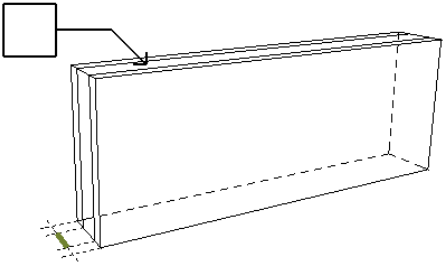
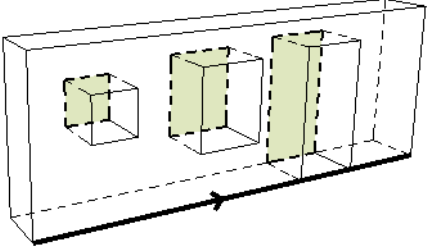
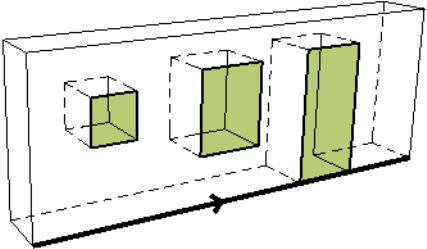
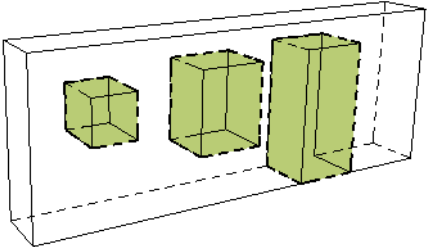
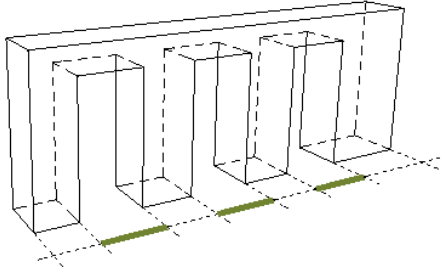
Stair Structure Listing Parameters

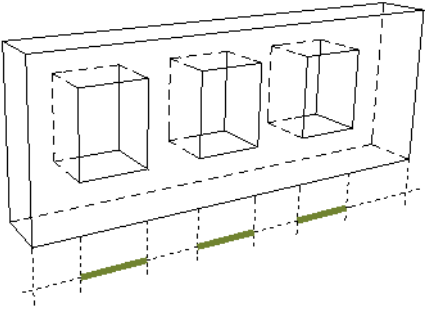
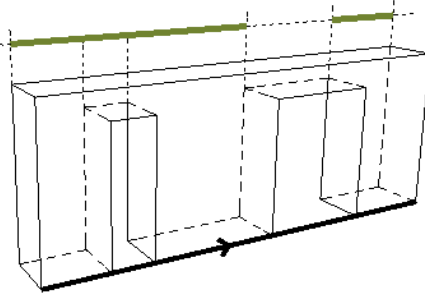
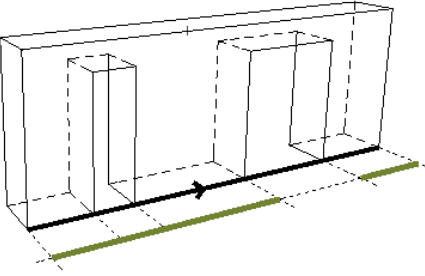
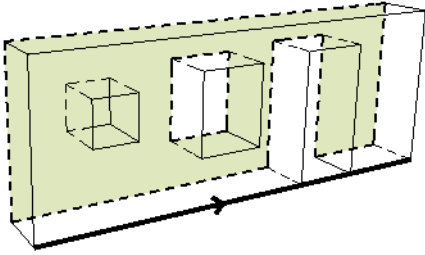
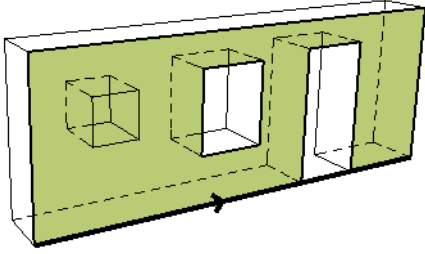
Parameter	Short Description	Long Description	
Construction Type	Type of Stair Structure: Monolith, Beam, Stringers, or Cantilevered		
Drain Length	Length of Draining (Monolith Structure only)		
Drain Position	Position of Draining (Right and/or Left, relative to direction of Stair baseline).	Assign in "Draining on Flight" Panel of Stair Settings (Monolithic Structure only)	
Profile Height	Profile dimension of the supporting Structure.	Not relevant for Monolithic structure.	
Profile Width	Profile dimension of the supporting Structure.	Not relevant for Monolithic structure.	

Parameter	Short Description	Long Description	
Profile WxH	Profile dimension of the supporting Structure.	Not relevant for Monolithic structure.	
Secondary Thickness	As in Settings - relevant for Monolithic Flight Structures using a Stepped or Variable shape.		
Shape Type	Shape of Stair bottom, as defined in Flight Structure Settings.	Flat, Stepped, Variable, or Filled (for Monolithic Flight Structures) Straight or Stepped (for Beam-type Flights)	
Stair Structure 3D Height	Height of Stair structure in 3D	Measured from the flight's lowest to highest z-point	
Stair Structure Flight Index	Flight number	Flight structures are numbered consecutively	
Stair Structure Landing Index	Landing number	Landing structures (if any) are numbered consecutively	
Stair Structure Position	For Stringers: Left or Right position. All other Structures are listed as "Under Stair."		

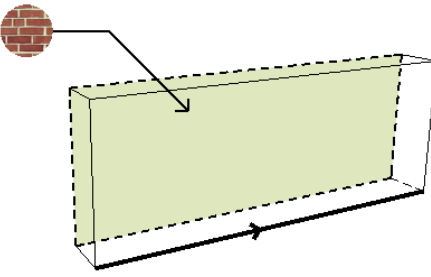
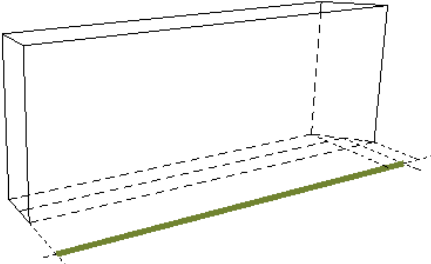
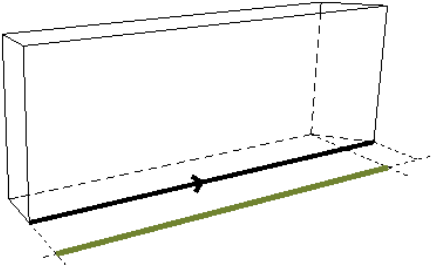
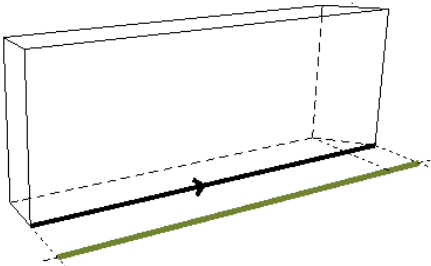
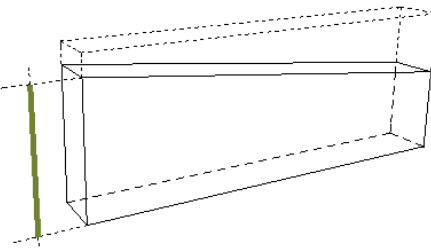
Parameter	Short Description	Long Description	
Thickness	Thickness of Structure element (as in Settings).		

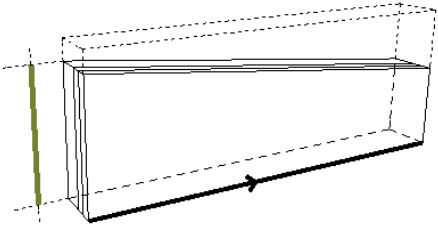
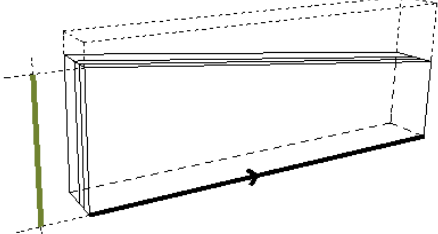
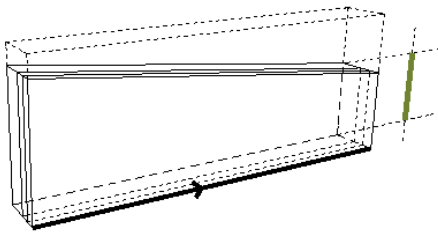
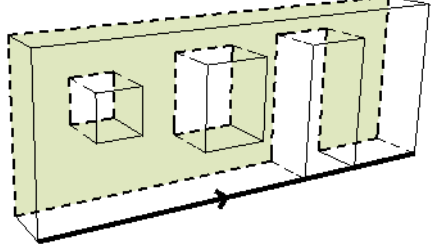
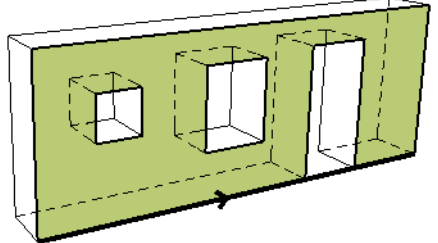
Wall Listing Parameters

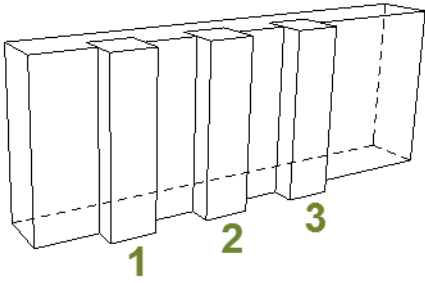
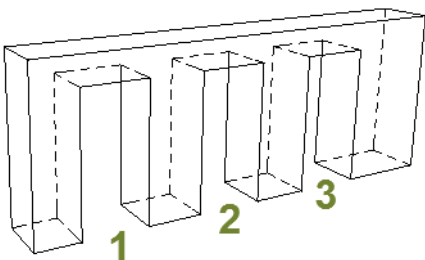
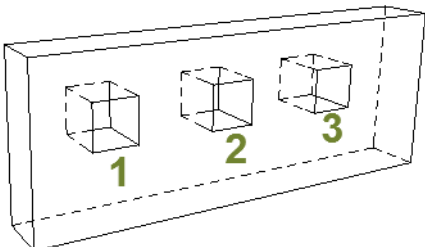
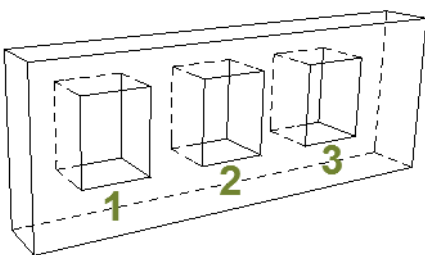
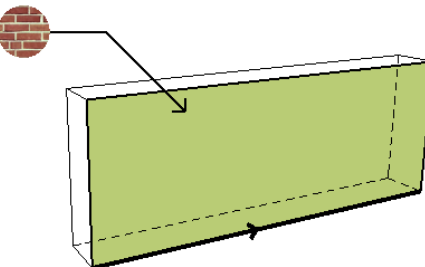
Parameter	Short Description	Long Description	
Air Skin Thickness	Thickness of Wall skin whose structure is defined as "Air skin"	Air Skin is defined at Options > Project Preferences > Calculation Rules.	
Analytic Surface Area of Openings on the Inside Face	Surface Area of all Wall Openings (windows, doors, empty Openings) on the Wall's Inside Face.	Surface calculation is based on hole that results from cutting the GDL Openings out of the Wall.	
Analytic Surface Area of Openings on the Outside Face	Surface Area of all Wall Openings (windows, doors, empty Openings) on the Wall's Outside Face.	Surface calculation is based on hole that results from cutting the GDL Openings out of the Wall.	
Analytic Volume of the Openings	Volume of all Wall Openings (windows, doors, empty Openings).	Volume calculation based on hole that results from cutting the GDL Openings out of the Wall. Does not take SEO's into account.	
Combined Width of Doors	Sum of the nominal widths (as defined in Door Settings) of all doors placed in the Wall.		

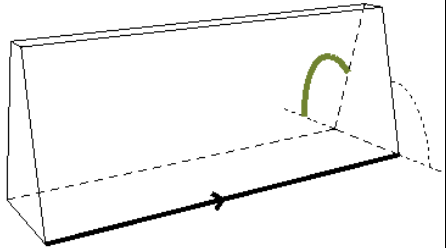
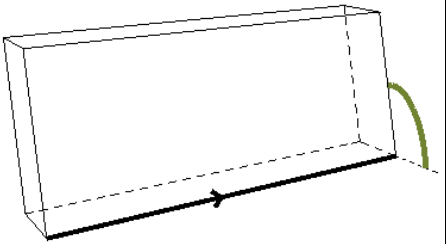
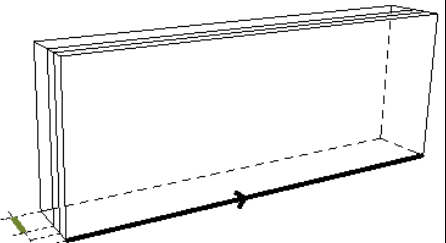
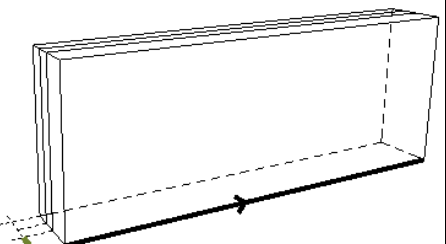
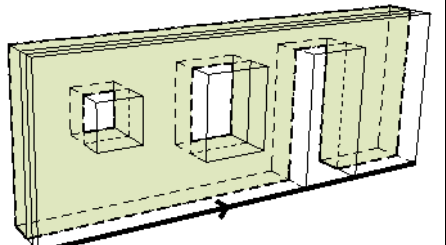
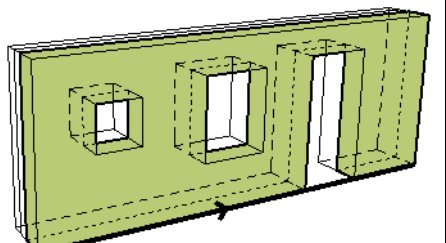
Parameter	Short Description	Long Description	
Combined Width of Windows	Sum of the nominal widths (as defined in Window Settings) of all windows placed in the Wall		
Length of the Wall Inside Face (Conditional)	Length of Wall along its Inside Face - this Length may be conditioned on a defined Calculation Rule.	Condition: Subtracts the Length of any Openings that exceed a given Length, as defined in Options > Project Preferences > Calculation Rules.	
Length of the Wall Outside Face (Conditional)	Length of Wall along its Inside Face - this Length may be conditioned on a defined Calculation Rule.	Condition: Subtracts the Length of any Openings that exceed a given Length, as defined in Options > Project Preferences > Calculation Rules.	
Surface Area of the Wall Inside Face (Conditional)	Surface Area of Wall on its Inside Face - this area may be conditioned on a defined Calculation Rule.	Condition: Subtracts the surface of any Openings that exceed a given area as defined in Options > Project Preferences > Calculation Rules. Reduced by trims and SEO's.	
Surface Area of the Wall Outside Face (Conditional)	Surface Area of Wall on its Outside Face - this area may be conditioned on a defined Calculation Rule.	Condition: Subtracts the surface of any Openings that exceed a given area as defined in Options > Project Preferences > Calculation Rules. Reduced by trims and SEO's.	


Parameter	Short Description	Long Description	
Volume of the Wall Inside Face Component (Conditional)	Volume of skin on the Wall's Inside Face - this Volume may be conditioned on a defined Calculation Rule.	Condition: subtracts the Volume of any Openings that exceed a given size, as defined in Options > Project Preferences > Calculation Rules. Reduced by trims and SEO's.	
Volume of the Wall Outside Face Component (Conditional)	Volume of skin on the Wall's Outside Face - this Volume may be conditioned on a defined Calculation Rule.	Condition: subtracts the Volume of any Openings that exceed a given size, as defined in Options > Project Preferences > Calculation Rules. Reduced by trims and SEO's.	
End Thickness of the Wall	End thickness of trapezoid Wall.	Second thickness, assigned in Wall Settings to a trapezoid Wall.	
Geometry Method	Uniform, Trapezoid or Polygonal.		
Surface Area of the Wall Inside Face (Gross)	Gross Surface Area of the Wall - including the surfaces of any Openings - on the Inside Face.	Reduced by trims. SEO's are ignored.	
Surface Area of the Wall Outside Face (Gross)	Gross Surface Area of the Wall - including the surfaces of any Openings - on the Outside Face.	Reduced by trims. SEO's are ignored.	

Parameter	Short Description	Long Description	
Inside Face Surface	Name of the surface assigned to the Wall's Inside Face.		
Length of Reference Line	Length of the Wall's Reference Line.		
Length of the Wall at the Center	Length of the Wall, as measured from end to end along Wall's centerline.		
Length of the Wall on the Inside Face	Length of the Wall, as measured from end to end along Wall's Inside Face.		
Length of the Wall on the Outside Face	Length of the Wall, as measured from end to end along Wall's Outside Face.		
Maximum Height of the Wall	Height of Wall at its highest point.	Roof crops are considered. Trims and SOE's are ignored.	

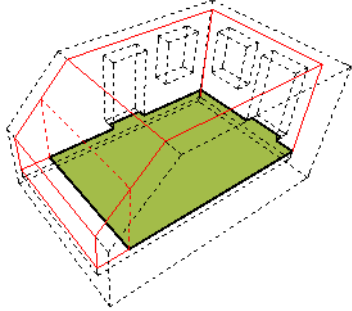
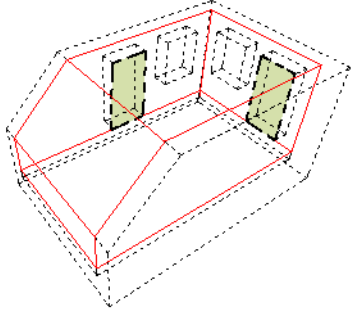
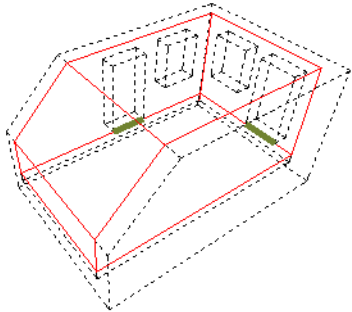
Parameter	Short Description	Long Description	
Maximum Height of the Wall Skin on the Inside Face	Height of Wall skin, at its highest point, on the Wall's Inside Face.	Roof crops are considered. Trims and SOE's are ignored.	
Maximum Height of the Wall Skin on the Outside Face	Height of Wall skin, at its highest point, on the Wall's Outside Face.	Roof crops are considered. Trims and SOE's are ignored.	
Minimum Height of the Wall Skin on the Inside Face	Height of Wall skin, at its lowest point, on Wall's Inside Face.	Roof crops are considered. Trims and SOE's are ignored.	
Minimum Height of the Wall Skin on the Outside Face	Height of Wall skin, at its lowest point, on the Wall's Outside Face.	Roof crops are considered. Trims and SOE's are ignored.	
Surface Area of the Wall Inside Face (Net)	Surface Area of the Wall on the Inside Face.	Reduced by Openings, trims and Solid Element Operations.	
Surface Area of the Wall Outside Face (Net)	Surface Area of the Wall on the Outside Face.	Reduced by Openings, trims and Solid Element Operations.	

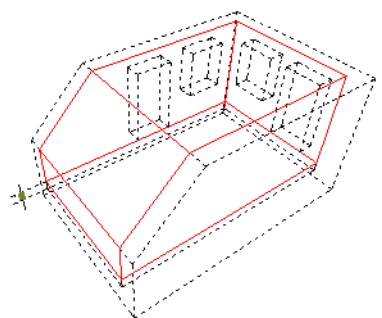
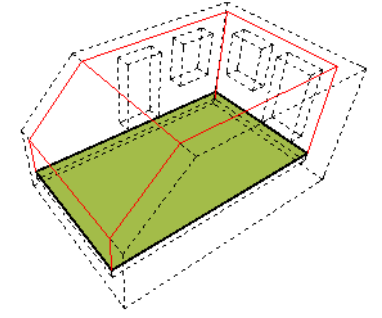
Parameter	Short Description	Long Description	
Number of Columns	Number of Columns that intersect the Wall		
Number of Doors	Number of Door elements in the Wall		
Number of Holes (Number of Empty Openings)	Number of Empty Opening Objects in the Wall		
Number of Windows	Number of Window elements in the Wall		
Outside Face Surface	Name of the surface assigned to the Wall's Outside Face.		
Reference Line location	As defined in Wall Settings (Outside Face, Inside Face or Center).	For composite Walls: can also be Core Outside, Core Inside or Core Center.	

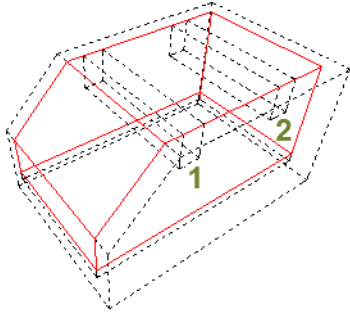
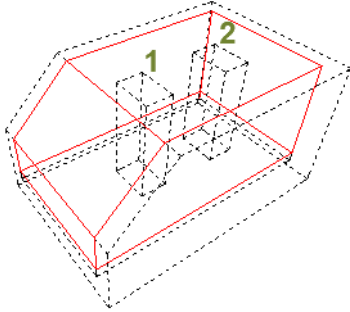
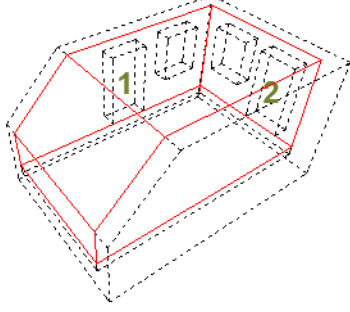
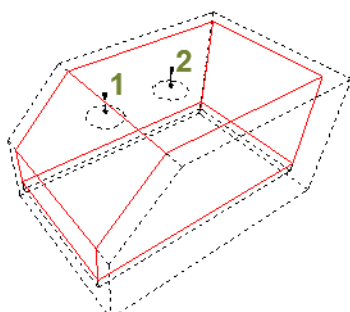
Parameter	Short Description	Long Description	
Wall Inside Slant Angle	Angle of a slanted Wall from the perpendicular to its Inside Face.		
Wall Outside Slant Angle	Angle of a slanted Wall from the perpendicular to its Outside Face.	Same as the General "Slant Angle" parameter	
Wall Component Thickness on the Inside Face	For Composite Walls: Thickness of the skin on the Wall's Inside Face.	Each skin thickness is defined in Options > Element Attributes > Composite Structures.	
Wall Component Thickness on the Outside Face	For Composite Walls: Thickness of the skin on the Wall's Outside Face.	Each skin thickness is defined in Options > Element Attributes > Composite Structures.	
Wall Component Volume on the Inside Face (Net)	Net Volume of the skin on the Wall's Inside Face.	Subtracts Volumes of Openings within the skin Reduced by trims and SEO's.	
Wall Component Volume on the Outside Face (Net)	Net Volume of the skin on the Wall's Outside Face.	Subtracts Volumes of Openings within the skin Reduced by trims and SEO's.	

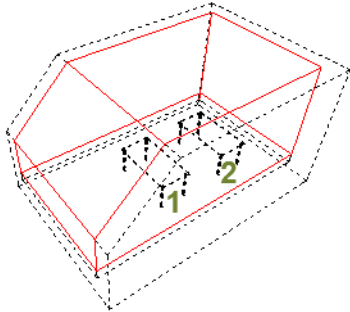
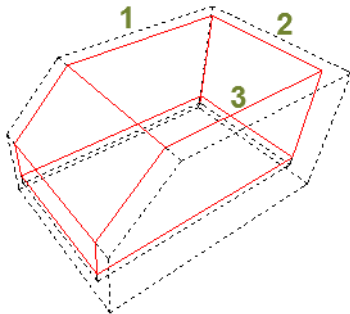
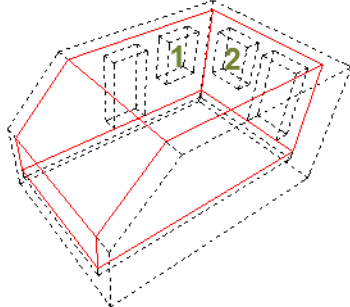
Parameter	Short Description	Long Description	
Wall Slant Type	Straight, Slanted, or Double-slanted.	As defined in Wall Settings, Geometry and Positioning Panel	

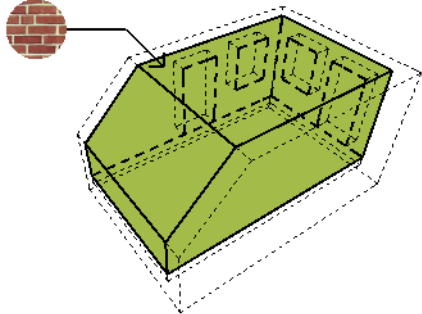
Zone Listing Parameters

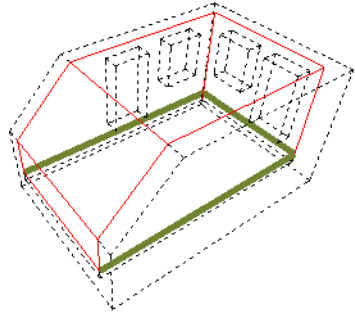
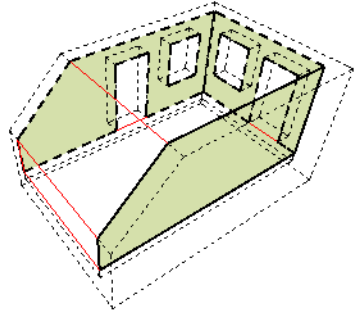
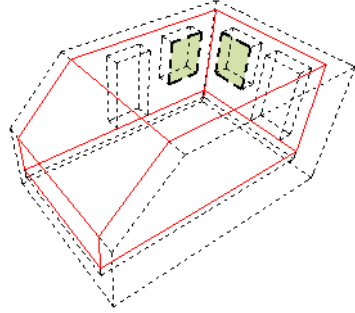
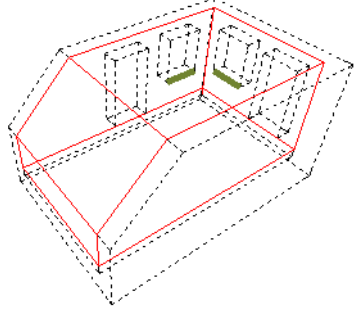
Parameter	Short Description	Long Description	
Area Reduction	The Calculated Area as a percentage of the Reduced Area.	In Zone Settings - Area Calculation Panel: If you check "Reduce by" a certain percentage, the result is the Calculated Area. (For example: If "Reduce by" is 20 percent, the "Area Reduction" is 80 percent.)	
Calculated Area	The Zone's area after the Reduced Area is further reduced by the percentage value set in the Area Calculation panel of Zone Settings.	In Zone Settings - Area Calculation Panel: check "Reduce by", and enter a percentage to subtract that from the Reduced Zone area. The result is the Calculated Area.	
Doors Surface Area	Total surface area of all Doors in the Zone.	A Door is in the zone if any part of its 2D polygon is inside of, or coincides with the edge of, the Zone's 2D polygon.	
Doors Width	Total width of all Doors in the Zone.	A Door is in the zone if any part of its 2D polygon is inside of, or coincides with the edge of, the Zone's 2D polygon.	
Extracted Column Area	Column area subtracted from Measured Zone area.	Relevant if the zone polygon contains Column(s). If Relation to Zones = "Reduce Zone Area Only" AND the Column's area exceeds the minimum defined in Zones Preferences, then the Zone is reduced by the percentage set in Zones Preferences.	

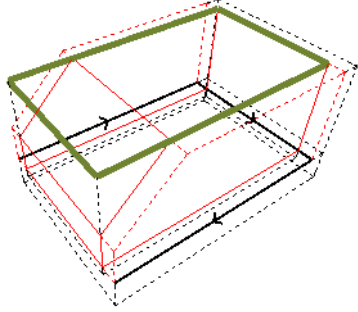
Parameter	Short Description	Long Description	
Extracted Fill Area	Fill area subtracted from the Measured Zone area.	Relevant if the zone polygon contains Fill(s). If Subtract from Zones is enabled for the Fill (see Fill Settings - General Settings), then the Zone area is reduced by the defined percentage.	
Extracted Low Area	Low ceiling area subtracted from Measured Zone area.	If the Zone is trimmed by a Roof or Shell: The zone area is reduced by the "Area reduction under Low Ceiling" as defined in Zones Preferences.	
Extracted Wall Area	Wall area subtracted from Measured Zone area.	Relevant if the zone polygon contains Wall(s). If Relation to Zones = "Reduce Zone Area Only" AND the Wall's area exceeds the minimum defined in Zones Preferences, then the Zone is reduced by the percentage set in Zones Preferences.	
Floor Thickness	Subfloor Thickness value.	Defined in the Name and Positioning Panel of Zone Settings.	
Measured Area	Total area of the Zone polygon. Can include the area of bounding walls up to their reference line.	Bounding Walls are included in Measured Area if the Zone extends to their reference line.	
Measured Net Area	Total area of the Zone polygon. Does not include the area of any bounding walls. Zone Area may be further reduced by area of certain elements within Zone polygon.	The Net Area excludes the footprint of construction elements within the zone whose "Relation to Zones" option is set to "Zone Boundary". Relation to Zones is available for - Walls and Columns: Settings dialog, Model Panel - Curtain Wall: Settings dialog, Member Placement Panel	

Parameter	Short Description	Long Description	
Number of Beams	Number of Beams in the Zone.	A Beam is in the zone if any part of its 2D polygon is inside of, or coincides with the edge of, the Zone's 2D polygon.	
Number of Columns	Number of Columns in the Zone.	A Column is in the zone if any part of its 2D polygon is inside of, or coincides with the edge of, the Zone's 2D polygon.	
Number of Concave Corners	Concave corners of the Zone polygon.		
Number of Corners	Total number of corners of Zone polygon.		
Number of Doors	Number of Doors in the Zone.	A Door is in the zone if any part of its 2D polygon is inside of, or coincides with the edge of, the Zone's 2D polygon.	
Number of Lamps	Number of Lamps in the Zone.	A Lamp is in the zone if any part of its 2D polygon is inside of, or coincides with the edge of, the Zone's 2D polygon.	

Parameter	Short Description	Long Description	
Number of Objects	Number of Objects in the Zone.	An Object is in the zone if any part of its 2D polygon is inside of, or coincides with the edge of, the Zone's 2D polygon.	
Number of Wall Parts	Wall part: a Wall which runs along a Zone border.		
Number of Windows	Number of Windows in the Zone.	A Window is in the zone if any part of its 2D polygon is inside of, or coincides with the edge of, the Zone's 2D polygon.	
Reduced Area	The Zone's area after the Measured Net Area is reduced by certain element-level reductions, subject to Zone Preferences.	<p>Construction elements: if Relation to Zones = "Reduce Zone Area Only" AND the Wall/Column's area exceeds the minimum defined in Zones Preferences, then the Zone is reduced by the percentage set in Zones Preferences.</p> <p>Fills: If Subtract from Zones is enabled for the Fill (see Fill Settings - General Settings), then the Zone area is reduced by the defined percentage.</p> <p>If the Zone is trimmed by a Roof or Shell: The zone area is reduced by the "Area reduction under Low Ceiling" as defined in Zones Preferences.</p>	

Parameter	Short Description	Long Description	
Related Element(s) 3D	The ID of the element(s) with which the Zone collides in 3D	The counterpart (General) listing parameter is " Colliding Zones ," which lists the names of a Zone's colliding elements.	
Surface	Surface assigned to the Zone.		
Total Extracted Area	Total of Wall/Column/Curtain Walls/Fills/Low ceiling area subtracted in calculating the Zone area.	Relevant if the zone polygon contains certain elements. Zone area calculation varies depending on Relation to Zone settings of elements within Zone polygon, and Zones Preferences.	
Uniform Surface	Checkbox On or off indicates whether the Zone uses a uniform surface.	If checkbox is off, then the Zone faces inherit the surfaces of the surrounding Zone boundaries. This value is set in the Model Panel of Zone Settings. Select a surface for the Zone, then choose either: "Use this Surface on All Zone Faces", or "Inherit Boundary Wall and Cropping Element Surfaces."	
Wall Inset Back Side Surface Area	Surface Area of Zone's Wall inset on the back	Wall Insets are created via Opening geometry: Reveals, Wall Inset, and/or Wall Niche. If these geometries affect the Zone (e.g. if the inset size exceeds the minimum as set in Zone Preferences, and/or the opening's reference line coincides with that of the Zone), then these parameters give you the areas of the 3D inset's top, sides and back.	
Wall Inset Side Surface Area	Surface Area of Zone's Wall inset on the side		
Wall Inset Top Surface Area	Surface Area of Zone's Wall inset on the top		

Parameter	Short Description	Long Description	
Walls Perimeter	Perimeter of the Walls bounding the Zone.		
Walls Surface Area	Surface area of Walls in the Zone.	A Wall is in the zone if any part of its 2D polygon is inside of, or coincides with the edge of, the Zone's 2D polygon.	
Windows Surface Area	Surface area of Windows in the Zone.	A Window is in the zone if any part of its 2D polygon is inside of, or coincides with the edge of, the Zone's 2D polygon.	
Windows Width	Total width of all Windows in the Zone.	A Window is in the zone if any part of its 2D polygon is inside of, or coincides with the edge of, the Zone's 2D polygon.	
Zone Category	Name of the Zone Category assigned to this Zone.	Defined in the Name and Positioning Panel of Zone Settings.	
Zone Category Code	Number of the Zone Category assigned to this Zone.	Depends on Zone Category. Zone Category Codes are assigned to Zone Categories at Options > Element Attributes > Zone Categories (click Rename to access the Code field).	

Parameter	Short Description	Long Description	
Zone Gross Perimeter	Perimeter of the Zone, corresponding to the Zone polygon at the bottom of the Zone body.	For a Zone created with the Reference Line method, the perimeter is measured along the Reference Lines of the boundary Walls.	
Zone Name	Name of the Zone as defined in the Name and Positioning Panel of Zone Settings.		
Zone Net Perimeter	Perimeter of the Zone, measured along the inner edges of the surrounding Walls (regardless of the position of the Walls' Reference line).		
Zone Number	Number of the Zone as defined in the Name and Positioning Panel of Zone Settings.		

Component Data in Archicad

What Are Components?

Any construction element or Library part having a Building Material consists of one or more components.

- A simple Slab has one Building Material and one component.
- A composite Slab has as many components as it has skins: each skin has a Building Material and each skin is a separate component (even if several skins happen to use the same Building Material).
- In a profile element: each fill (as seen in Profile Editor) represents a separate component.
- GDL objects work differently. All parts of the object that use the same Building Material are considered a single component. For example, multiple object parts, which all use the same Glass Building Material, are considered as a single component.

Using Component Data in the BIM Model

Component Data can be shown in the following:

- Component List of the Interactive Schedule
- Labels which list Component data
- Exported IFC file

Component Lists

Component Lists are optimized to list information about each component of composite, profile or basic elements: walls, slabs, roofs, morphs, meshes, and/or multi-component profile elements.

Component Area Schedule - ARCHICAD 25				
Building Material	Classification	Component Area (Gross) [m2]	Component Area (Conditional) [m2]	Component Area (Net) [m2]
Concrete Floor	Internal Cladding	6.30	2.31	2.31
Facing Brick	External Cladding	22.02	16.56	16.15
In-situ Reinforced Concrete	Structural	21.12	20.54	17.98
Masonry Block	External Filler	6.75	5.08	5.08
Polystyrol Foam	Insulation Component	21.57	21.57	17.26

A component list can include:

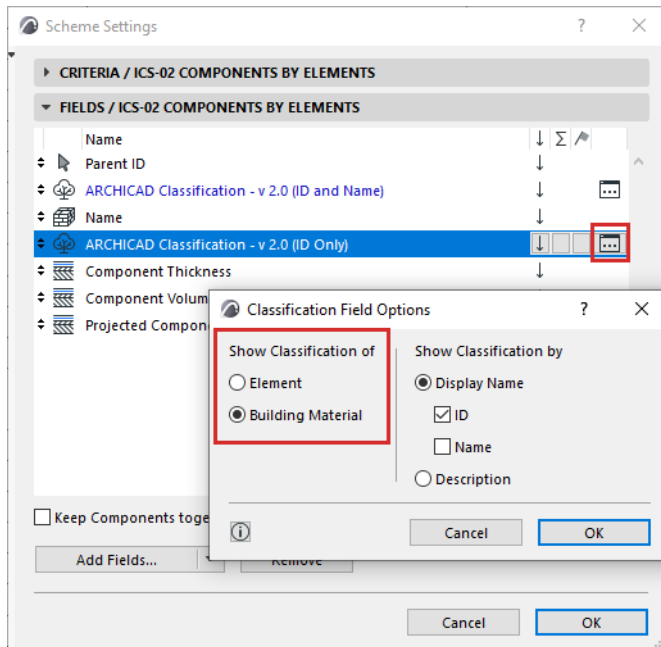
- Properties that are specific to components ([see Component Listing Parameters in the Interactive Schedule](#))
- Any of the Building Material properties (because each component consists of at most one Building Material)
- Naturally, you can add element properties to the list, but these property values will reflect the element-level data (not the component's data)

Show Classification Data in Component List

Components themselves do not have a classification.

If your Component Scheme includes the Classification parameter: click the pop-up to specify which classification data to show:

- by Element (list the classification of the element that contains the component)
- by Building Material (list the classification of the component's Building Material)



Related Topics:

[Component Listing Parameters in the Interactive Schedule](#)

[Calculating Component Areas](#)

[Map Component-Level Properties](#)

[Element Listing Parameters](#)

Component Listing Parameters in the Interactive Schedule

The table below lists the properties that are available to list components in the Interactive Schedule.

Note: Component area properties cannot be calculated for Morphs or basic non-veneer columns.

Use these parameters in the Component lists of the Interactive Element Schedule.

COMPONENT PROPERTY	DESCRIPTION	AVAILABLE FOR ELEMENTS
Component Cross Section Area	Area of the Component cross-section	
Component Fill Outline Pen	Pen number	
Component Thickness	As defined in the component	
Component Type	Core, Finish, or Other	
Component Volume (Net)	Volume of the component. All openings and holes are subtracted.	
Component Volume (Gross)	Openings and holes are ignored.	
Component Volume (Conditional)	Openings and holes are ignored in components that meet the conditions at Options > Project Preferences > Calculation Rules ("Ignore component holes...")	PlaneRoof / PolyRoof Shell Slab Beam Beam Segment Wall
Embodied Carbon		
Embodied Energy		
Mass	Calculated based on component's volume and the Building Material's Density value	
Projected Component Areas:	Area is based on the projection of the virtual plane that runs through the center of the component.	
• Projected Component Area (Net)	Openings and holes are subtracted.	
• Projected Component Area (Gross)	Openings and holes are ignored.	Roof Shell Slab Wall Beam Column

COMPONENT PROPERTY	DESCRIPTION	AVAILABLE FOR ELEMENTS
<ul style="list-style-type: none"> Projected Component Area (Conditional) 	Openings and holes are ignored in components that meet the conditions at Options > Project Preferences > Calculation Rules ("Ignore component holes...")	Roof Shell Slab Wall
Rectangular Component Height	Height of rectangular component in 2D	
Rectangular Component Width	Width of rectangular component in 2D	

Related Topics

[Component Data in Archicad](#)

[Calculating Component Areas](#)

[Element Listing Parameters](#)

Calculating Component Areas

Use the Projected Component Area field to calculate the component areas of basic, composite or profiled elements.

Note:

- Morphs and basic non-veneer columns are an exception: component area cannot be calculated for these elements.

See details here:

[Component Areas of Basic and Composite Elements](#)

[Component Areas of Complex Profiles](#)

Related Topic:

[Component Data in Archicad](#)

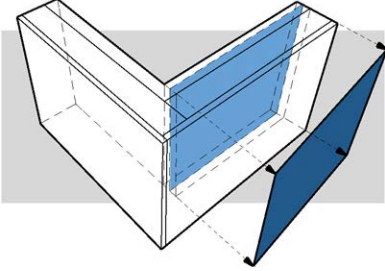
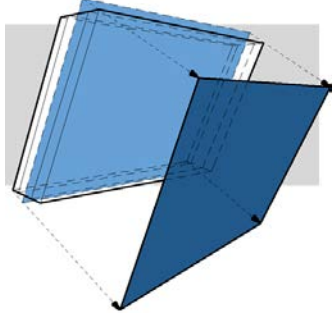
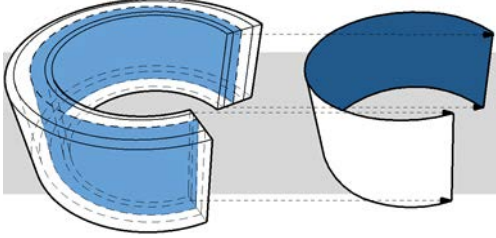
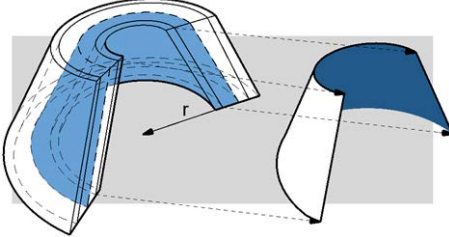
Component Areas of Basic and Composite Elements

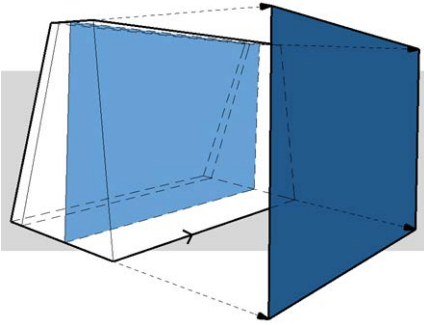
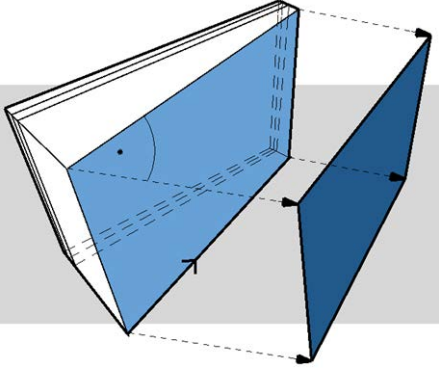
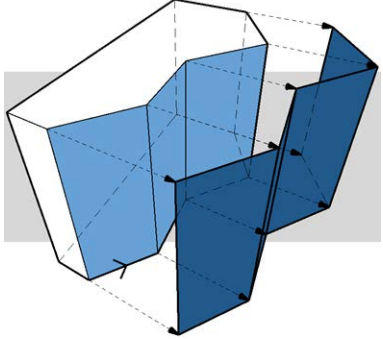
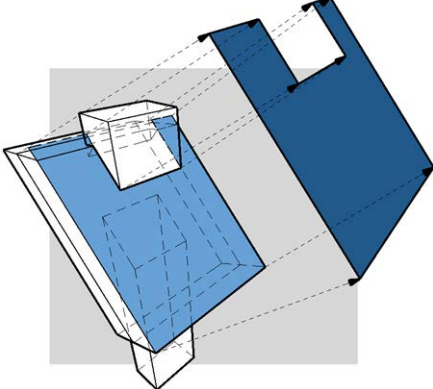
For most elements, the calculated area is the projection onto the virtual plane that runs through the center of the component.

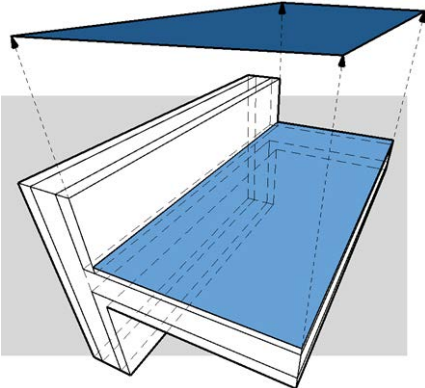
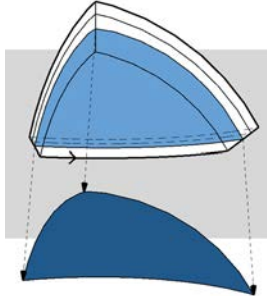
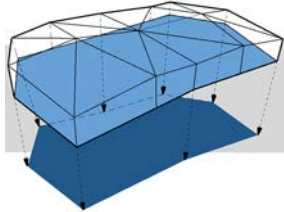
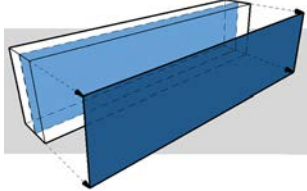
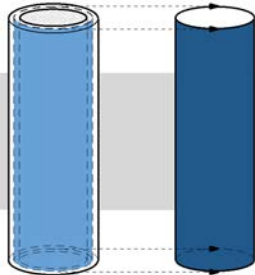
[Watch the Video](#)

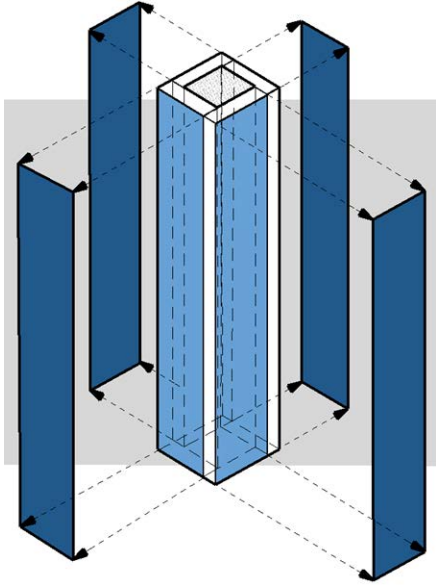
[Watch the Video](#)

The following table gives details about area calculation for each element type.

Basic or Composite Element	Which Plane is used to Calculate Projected Component Area?	
Regular Straight Wall	The vertical plane that runs through the center of the component.	
Slanted Wall	The slanted plane that runs through the center of the component.	
Curved Wall	The plane of the vertical cylinder that runs through the center of the component.	
Curved Slanted Wall	The plane of the cone that runs through the center of the component.	

Basic or Composite Element	Which Plane is used to Calculate Projected Component Area?	
Double Slanted Wall	The plane of the variable-thickness component that is closest to the wall's reference line.	 <p>The diagram shows a 3D perspective of a double slanted wall. A vertical dashed line represents the reference line. A blue shaded vertical plane is shown parallel to the reference line, representing the plane used for area calculation. The wall's thickness is variable, and the slanted surface is highlighted in blue.</p>
Trapezoid Wall	The plane of the variable-thickness component that is closest to the wall's reference line.	 <p>The diagram shows a 3D perspective of a trapezoid wall. A vertical dashed line represents the reference line. A blue shaded vertical plane is shown parallel to the reference line, representing the plane used for area calculation. The wall's thickness is variable, and the trapezoidal surface is highlighted in blue.</p>
Polygon Wall	The plane(s) of the vertical polygon(s) defined by the reference edge(s).	 <p>The diagram shows a 3D perspective of a polygon wall. A vertical dashed line represents the reference edge. A blue shaded vertical plane is shown parallel to the reference edge, representing the plane used for area calculation. The wall's thickness is variable, and the polygonal surface is highlighted in blue.</p>
Roof	Plane of the roof.	 <p>The diagram shows a 3D perspective of a roof. A blue shaded plane is shown parallel to the roof's surface, representing the plane used for area calculation. The roof's thickness is variable, and the slanted surface is highlighted in blue.</p>

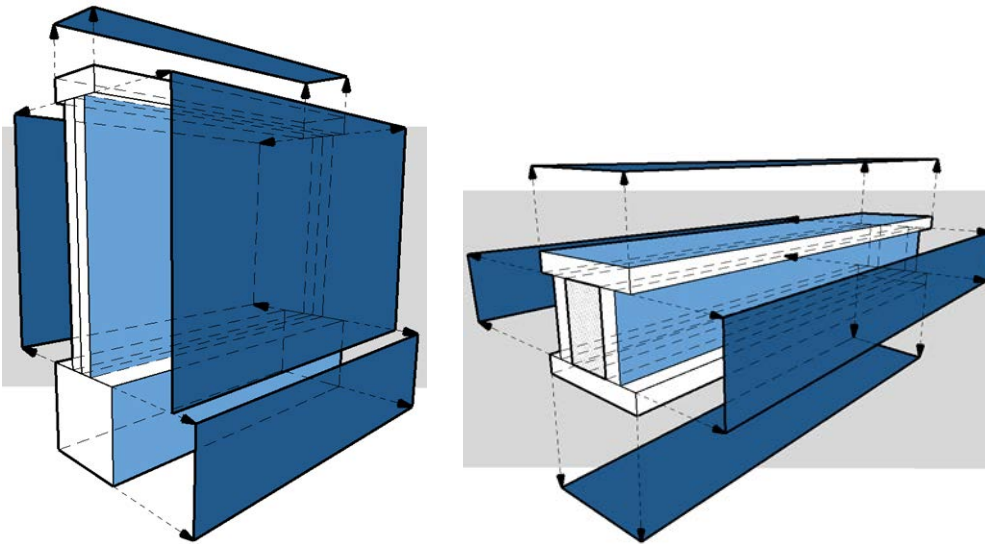
Basic or Composite Element	Which Plane is used to Calculate Projected Component Area?	
Slab	Plane of the slab.	 <p>The diagram shows a 3D perspective of a rectangular slab with a blue top surface. A dashed blue line indicates the projection plane, which is parallel to the top surface of the slab. Dotted lines show the projection of the slab's edges onto this plane.</p>
Shell	The plane of the component that is closer to the Shell reference plane	 <p>The diagram shows a 3D perspective of a curved shell element with a blue top surface. A dashed blue line indicates the projection plane, which is parallel to the top surface of the shell. Dotted lines show the projection of the shell's edges onto this plane.</p>
Mesh	The Mesh reference plane	 <p>The diagram shows a 3D perspective of a mesh element with a blue top surface. A dashed blue line indicates the projection plane, which is parallel to the top surface of the mesh. Dotted lines show the projection of the mesh's edges onto this plane.</p>
Straight Beam	The Beam's vertical plane	 <p>The diagram shows a 3D perspective of a straight beam element with a blue top surface. A dashed blue line indicates the projection plane, which is vertical and parallel to the beam's length. Dotted lines show the projection of the beam's edges onto this plane.</p>
Column without Veneer	No component area is calculated.	
Column with Veneer (circular)	Circular column with veneer: The plane of the centerline of the veneer component.	 <p>The diagram shows two circular columns. The left column is light blue and represents the main column. The right column is dark blue and represents the veneer component. A dashed blue line indicates the projection plane, which is vertical and passes through the centerline of the veneer component. Dotted lines show the projection of the columns' edges onto this plane.</p>

Basic or Composite Element	Which Plane is used to Calculate Projected Component Area?	
<p>Column with Veneer (rectangular)</p>	<p>The program calculates the veneer as if it were divided into four parts as shown, and the vertical plane of each is used.</p>	
<p>Morph</p>	<p>No component area is calculated</p>	

Component Areas of Complex Profiles

When listing components of a profile element, you can choose

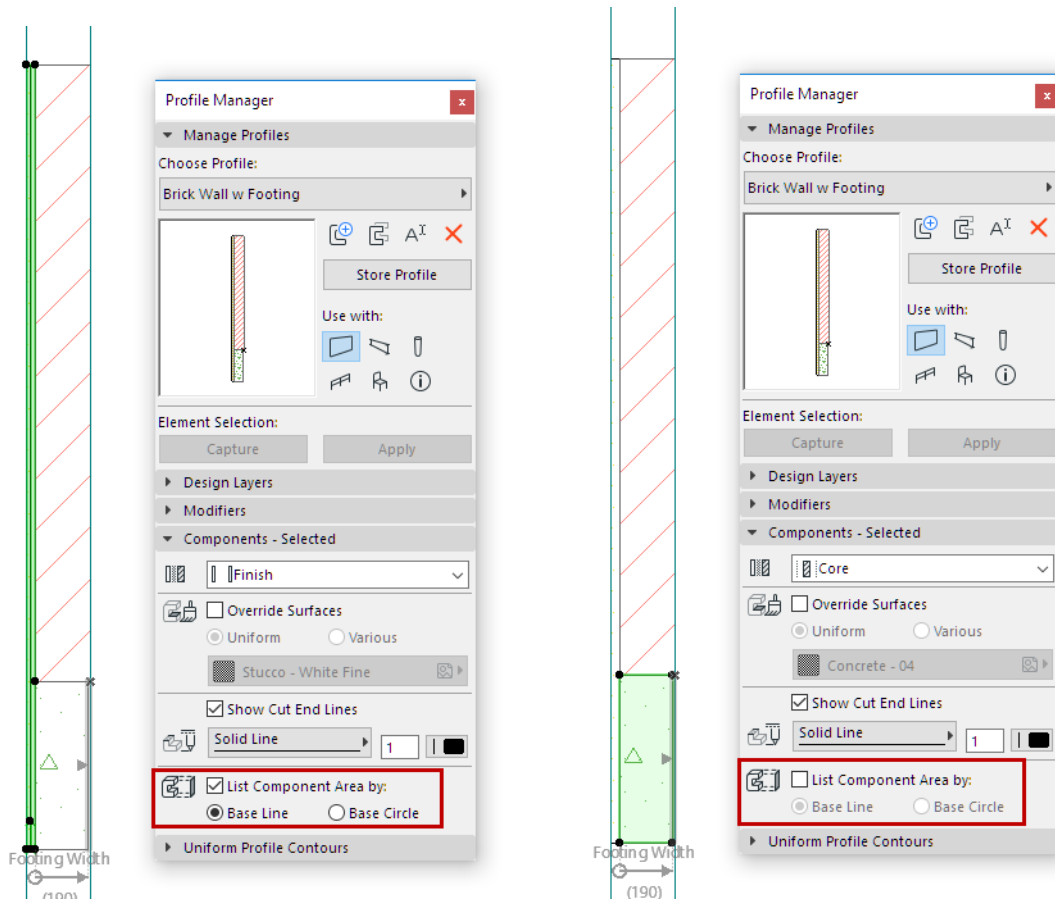
- which Profile component to list
- which of its planes should be used to calculate its area



Choose Profile Components for Area Listing

1. Select a profile element.
2. From the context menu, click **Edit Selected Composite/Profile**.
3. In the Profile Editor window, select a fill (representing a component).
4. In the Profile Manager (Options > Complex Profiles > Profile Manager), activate the **List Component Area** checkbox for each component whose area you want to list.

In this example, we are editing the Brick Wall w Footing element. The veneer's checkbox is on, so the veneer area will be included in the schedule. The footing area, however, is not relevant and its checkbox is off.



List Component Area: On for Veneer, Off for Footing

5. Click **Store Profile** to save changes.
6. The area of the concrete footing component is not included in the schedule.

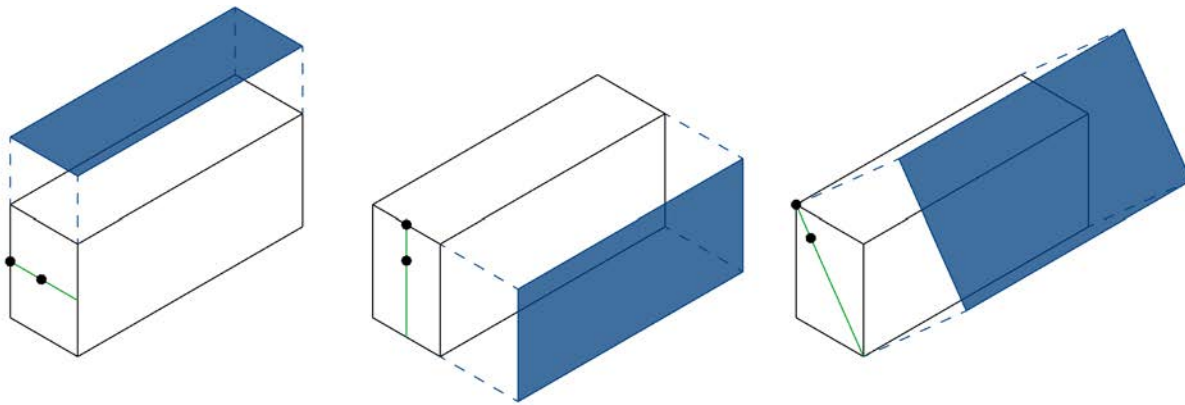
Components by Elements copy 1			
Element ID	Name	Component	
		Volume [m3]	Area [m2]
Partition W 01			
	Brick - Structural	0.28	1.42
	Concrete - Structural	0.07	---
	Gypsum Plasterboard	0.05	1.80

Choose a Plane to Use for Calculating Area

Each component’s projected area is calculated based on a Base Line or Base Circle, shown in the Profile Editor.

Note: To see Base Lines in Profile Editor, make sure you have turned on **View > On-Screen View Options > Component Area Base Line/Circle**.

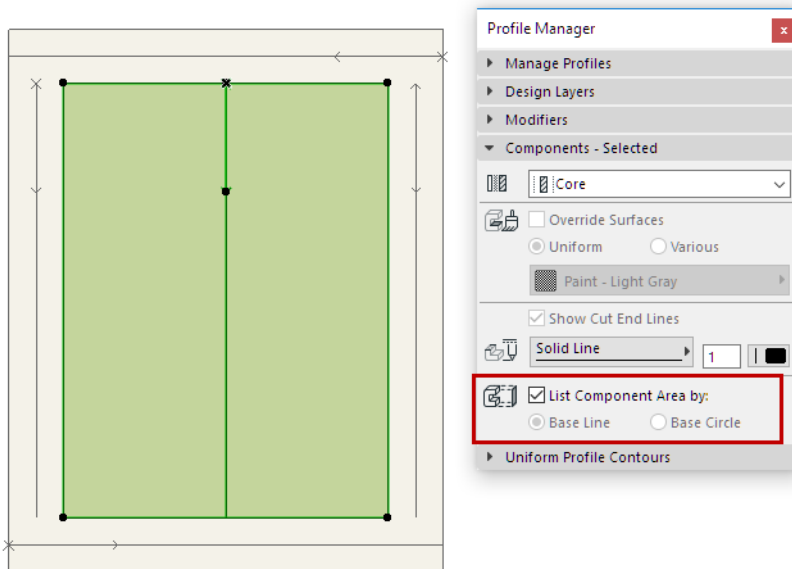
- The **Base Line** is used to calculate the component area as **projected onto the flat surface** represented by the line. By default, the base line is located along the longest segment of the profile.



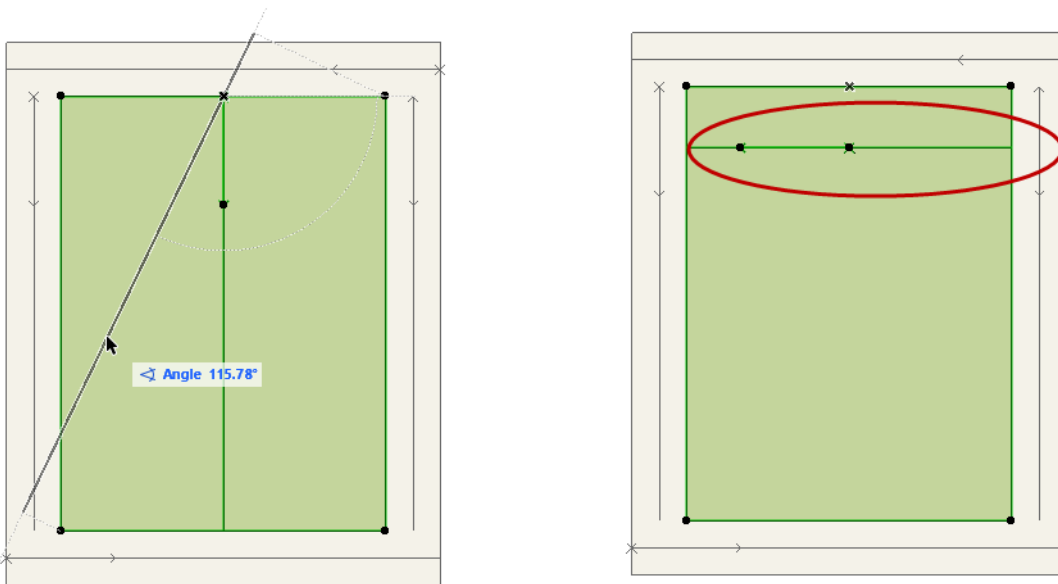
- The **Base Circle** is used to calculate the area as **projected onto an arched component**.
 You can graphically edit this Base Line/ Circle: This changes the plane used to calculate the component area.
 In this example, you need the area of the Generic Column on its horizontal surface.

1. In Profile Editor, select the component.
2. Activate the **List Component Area** checkbox.

You see that the component's base line is vertical, which is not the area you need.



3. In the Profile Editor window, select the base line and rotate it to a horizontal position.



4. Click **Store Profile** to save changes.
5. The horizontal area of the column is included in the schedule.

For general information on editing in the Profile Manager, see [Profile Manager](#).

Functions in Expressions

The constants, operators and functions described in this section can be used to create Expression-defined Properties for Archicad elements.

CONSTANTS

OPERATORS

LOGICAL FUNCTIONS

MATHEMATICAL FUNCTIONS

TRIGONOMETRICAL FUNCTIONS

STATISTICAL FUNCTIONS

TEXT EDITING FUNCTIONS

DATA CONVERSION

Related Topics

[Expression-Defined Element Properties](#)

[Expression-Defined Properties: Examples](#)

[*Read More*](#)

CONSTANTS

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Pi [PI]

True [TRUE] **False** [FALSE]

Pi [PI]**Value**

3.141592654

True [TRUE] False [FALSE]

Returns TRUE or FALSE in a function where the argument requires True or False data.

Example

The Join Strings [TEXTJOIN] function has the option to ignore empty fields.

```
TEXTJOIN ("Delimiter", IgnoreIfEmpty, "Text1", "Text2", ...)
```

In the following `TEXTJOIN` expression, we insert `TRUE` for the `IgnoreIfEmpty` argument, which is a True/False data type.

Expression

```
TEXTJOIN (", ", TRUE, Renovation Status, Related Zone Name)
```

The expression will therefore ignore empty fields.

See [Join Strings \[TEXTJOIN\] for an example](#).

OPERATORS

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Add [+]

Divide [/]

Equals [=]

Greater Than [>]

Greater Than or Equal to [>=]

Less Than [<]

Less Than or Equal to [<=]

Multiply [*]

Not Equal [<>]

Power [^]

Subtract [-]

Add [+]

Adds two arguments (operands).

Syntax

```
[+]
```

Data Types of the Arguments

Integer, Number, Length, Area, Volume, or Angle (Must be identical for all arguments)

Data Type of the Result

Same as Arguments' Data Type

Example

Return the sum of a Stair's Riser Height and Going Depth.

Expression

```
Default Riser Height + Default Going Depth
```

Result

Adds the two length-type values to get a length-type result (e.g. 0.50)

Notes

[See also *Sum \[SUM\]*](#).

Divide [/]

Divides two arguments (operands).

Syntax

```
[ / ]
```

Data Types of the Arguments

Integer, Number, Length, Area, Volume, or Angle

Data Type of the Result

Depends on Arguments' Data Types

Example

Calculate the glazed ratio of a zone by dividing its Windows Surface Area parameter by its Net Area.

Expression

```
Windows Surface Area/Net Area
```

Result

Divides two area-type values for a numerical result (e.g. .31)

Notes

Both Arguments are area-type values. This expression divides them and results in a number with no units (the ratio of the two areas).

Equals [=]

Compares two arguments (operands). Returns True if their values are identical, False if they are not equal.

Syntax

```
[=]
```

Data Types of the Arguments

True/False, Integer, Number, Length, Area, Volume, Angle, or String (Must be identical for all operands)

Data Type of the Result

True/False

Example

Check whether a Zone is classified as Residential Space.

The two arguments being compared (Classification and Residential Space) are both string-type data.

Expression

```
Archicad Classification-25 = "Residential Space"
```

Result

True for Zones classified as "Residential Space", otherwise False

Greater Than [>]

Compares two arguments (operands).

Returns True if the first value is greater than the second, otherwise False.

Syntax

```
[>]
```

Data Types of the Arguments

Integer, Number, Length, Area, Volume, or Angle (Must be identical for all arguments)

Data Type of the Result

True/False

Expression

Identify whether the element's Purchase Price (an existing Number-type property) is greater than 30.

Expression

```
Purchase Price > 30
```

Result

True for elements whose price is over 30, otherwise False

Greater Than or Equal to [>=]

Compares two arguments (operands). Returns True if the first value is greater than or equal to the second, otherwise False.

Syntax

```
[>=]
```

Data Types of the Arguments

Integer, Number, Length, Area, Volume, or Angle (Must be identical for all arguments)

Data Type of the Result

True/False

Example

Check whether the glazed ratio of zones (an existing property) complies with (is greater than or equal to) the mandatory glazed ratio (a fixed value, here .40).

Expression

```
Zone Glazed Ratio >= .40
```

Result

True for zones whose glazing rate is at least 40 percent, otherwise False

Less Than [<]

Compares two arguments (operands). Returns True if the first value is less than the second, otherwise False.

Syntax

```
[<]
```

Data Types of the Arguments

Integer, Number, Length, Area, Volume, or Angle (Must be identical for all arguments)

Data Type of the Result

True/False

Example

Identify whether the element's Purchase Price (an existing Number-type property) is less than 30.

Expression

```
Purchase Price < 30
```

Result

True for elements whose price is less than 30, otherwise False

Less Than or Equal to [\leq]

Compares two arguments (operands). Returns True if the first value is less than or equal to the second, otherwise False.

Syntax

```
[ $\leq$ ]
```

Data Types of the Arguments

Integer, Number, Length, Area, Volume, or Angle (Must be identical for all arguments)

Data Type of the Result

True/False

Example

Identify whether a zone's Subfloor Thickness (an existing Length-type property) is less than or equal to 20 cm.

Expression

```
Subfloor Thickness  $\leq$  20 cm
```

Result

True for Zones with Subfloor Thickness equal or less than 20 cm, otherwise False

Notes

Both arguments must have the same data type - here, they are both Length.

The Length values must have a unit. Choose a Length unit from the Units pop-up of Expression Editor.

Multiply [*]

Multiplies two arguments (operands).

Syntax

```
[*]
```

Data Types of the Arguments

Integer, Number, Length, Area, Volume, or Angle

Data Type of the Result

Depends on Arguments' Data Types

Example

Find the area of a given tile based on its fixed property values, with Length data types: Horizontal Dimension (20 cm) and Vertical Dimension (30 cm).

Expression

```
Tile Dimension H * Tile Dimension V
```

Result

600

Notes

In this example, the arguments are both lengths. Multiply them to get an area: set the resulting property's Data Type to Area.

Not Equal [<>]

Compares two arguments. Returns True if their values are not equal, otherwise False.

Syntax

```
[<>]
```

Data Types of the Arguments

True/False, Integer, Number, Length, Area, Volume, Angle, or String (Must be identical for all arguments)

Data Type of the Result

True/False

Example

Zone classifications include e.g. Office, Hallway, Meeting Room, and Smoking Area.

Smoke alarms are required in all zones except those classified as "Smoking Area".

For each Zone, determine if its Classification is not equal to "Smoking Area".

In this expression, both arguments are strings.

Expression

```
Archicad Classification 25 <> "Smoking Area"
```

Result

True for all Zones that are not Smoking Areas. (Smoke alarm required in all such zones.)

Power [^]

Syntax

```
[^]
```

or use the `Power` [`POWER`] function.

Data Types of the Arguments

Base: Integer, Number, Length, Area, Volume, or Angle

Exponent: Integer or Number

Data Type of the Result

Depends on Arguments' Data Type. (See Note, below).

Expression

```
5^2
```

Result

25

Note

If the Base argument's data type is Length, then

- an Exponent of 2 will return a property whose Data Type must be Area.
- an Exponent of 3 will return a property whose Data Type must be Volume.

Subtract [-]

Syntax

```
[-]
```

Data Types of the Arguments

Integer, Number, Length, Area, Volume, or Angle (Must be identical for all arguments)

Data Type of the Result

Same as Arguments' Data Type

Example

Subtract a Length-type expression from another Length-type expression.

The result is also a Length (its unit depends on Calculation Units for Length: here, meters).

Expression

```
Suspended Ceiling level - Raised Floor level
```

Result

2.2

Note

In this example, the arguments are both lengths. The resulting value is also a length.

LOGICAL FUNCTIONS

And [AND]

Checks whether every item in a series of conditions (logical arguments) is true.

If *all* the arguments are true, then AND returns TRUE.

If *any* of the arguments is false, then AND returns FALSE.

Syntax

```
AND (Logical1, [Logical2],...)
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

True/False

Data Type of the Result

True/False

Example

Determine if both Beam length properties (Length Left and Length Right) exceed a given length.

Expression

```
AND (Length Left > 480 cm, Length Right > 480 cm)
```

Result

True or False. True for all Beams where both Beam lengths exceed the value, otherwise False.

Note

The AND function examines a series of arguments. Within the parentheses, each item must be a logical argument that can be either true or false.

Or [OR]

Checks whether at least one in a series of conditions (Logical arguments) is true.

If *at least one* of the arguments is true, then OR returns TRUE.

If *none* of the arguments is true, then OR returns FALSE.

Syntax

```
OR (Logical1, [Logical2],...)
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

True/False

Data Type of the Result

True/False

Example 1

Determine if either one of the Beam length properties (Length Left or Length Right) exceeds a given length.

Expression

```
OR (Length Left > 480 cm, Length Right > 480 cm)
```

Result

True or False. True for all Beams where at least one Beam length exceeds the value, otherwise False.

Example 2

Determine if a Wall is fire resistant. To be fire-resistant, the wall must meet *at least one* of two possible conditions: its Building Material is "Reinforced Concrete - Structural", and/or it is at least 50 cm thick.

Expression

```
OR (Building Material = "Reinforced Concrete - Structural", Thickness >= 50 cm)
```

Result

True or False. True for all Walls whose Building Material is Reinforced Concrete, and/or whose thickness is at least 50 cm.

Note

The OR function examines a series of arguments. Within the parentheses, each item must be a logical argument that can be either true or false.

Not [NOT]

Changes False result to True and vice versa.

Syntax

```
NOT (Logical)
```

Data Types of the Arguments

True/False

Data Type of the Result

True/False

Example

An existing property returns “True” for all elements whose Model property is undefined. To change these results to False, use NOT.

Expression

```
NOT (ISUNDEFINED (Model))
```

Result

False for every element whose Model property is <Undefined>

If [IF]

Returns a specified value if the given logical argument is true.
and another specified value if the argument is false.

Syntax

```
IF (Logical, ValueIfTrue, ValueIfFalse)
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

Logical: True/False

ValueIfTrue, ValueIfFalse: Any Data Type, but Data Type must be identical for both Values

Data Type of the Result

Same Data type as the Values.

Example

Define the Beam Overhang standard depending on Beam length.

If the Beam is shorter than 480 cm, then the overhang should be 10 cm.

If it is longer than 480 cm, then the overhang should be 12 cm.

Expression

```
IF (Length Left <=480cm, 10 cm, 12 cm)
```

Result

10 for the shorter beams, 12 for the longer beams

Note

To test multiple conditions, it may be easier to use the `IFS` function.

See [If Multiple \[IFS\]](#).

If Multiple [IFS]

Use this function to test a series of logical conditions, then return the value for the first true condition.

IFS can take the place of multiple nested IF statements.

Syntax

```
IFS (Logical1, Value1, [Logical2, Value2]...)
```

You can define up to 127 logical conditions.

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

Logical: True/False

Value: Any Data Type, but Data Type and units must be identical for all Values

Data Type of the Result

Same Data type as the Values.

Example 1

Provide the mandatory percentage of glazed area for each Zone depending on its Classification.

Expression

```
IFS (Archicad Classification - 25 = "Residential Space", "25%", Archicad
Classification - 25 = "Office Space", "30%",
Archicad Classification - 25 = "Storage Space", "10%")
```

Result

Returns the glazed-area percentage required for each Zone depending on its classification. The Values and the Results all have the same data type (here, String).

If None of the Conditions is True

If *none* of the conditions applies, the function returns "Undefined."

However, you can also specify a default result value, if none of the logical conditions is met: enter TRUE for your final logical condition, followed by the default value.

Remember that this default result value must be the same data type as the other result values in the function.

```
IFS (Logical1, Value1, [Logical2, Value2],...[TRUE,DefaultValue])
```

Example 2

Categorize Beams as Long (longer than 6 m) or Short (shorter than 2m). Any Beam that does not fall into those categories is Medium.

Expression

```
IFS (Length Left > 600 cm, "Long", Length Left < 200 cm, "Short", TRUE,
"Medium" )
```

Result

Beams longer than 6 m: Long

Beams shorter than 2 m: Short

Beams with lengths in between: Medium

Exclusive Or [XOR]

Returns True if there is an uneven number of True logical arguments.

In the case of two arguments: either one or the other is true, but not both.

Syntax

```
XOR (Logical1, [Logical2],...)
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

True/False

Data Type of the Result

True/False

Example

Office decor items of various sizes must be boxed up. Three box sizes are available.

Items whose height and width are both more than 1m go in the large boxes.

```
AND (Width > 1m, Height > 1m)
```

Items whose height and width are both less than 1m go in the small boxes.

```
AND (Width < 1m, Height < 1m)
```

Items with **either height or width** is more than 1m go in the medium boxes.

Use a property to determine which items require the medium sized box.

Expression

```
XOR (Width > 1m, Height > 1m)
```

Result

True for all items whose width OR height (but not both) exceeds 1m.

Undefined [ISUNDEFINED]

Checks whether the value of a given Property is Undefined.

Syntax

```
ISUNDEFINED (value)
```

Data Types of the Argument

True/False, Integer, Number, Length, Area, Volume, Angle, or String

Data Type of the Result

True/False

Example

Determine whether an element's string-type "Model" property is undefined. `ISUNDEFINED` is a logical value.

Expression

```
ISUNDEFINED (Model)
```

Result

True for every element whose Model property is <Undefined>

MATHEMATICAL FUNCTIONS

Absolute Value [ABS]

Absolute value.

Returns the absolute value of the number.

Syntax

```
ABS (Number)
```

Data Types of the Arguments

Integer, Number, Length, Area, Volume, Angle

Data Type of the Result

Same as Argument's Data Type

Expression

```
ABS (-4)
```

Result

4

e Exponent [EXP]

Exponential function of e.

Returns "e" raised to the power of the given number.

The base "e" is a constant - the base of the natural logarithm.

Syntax

```
EXP (Number)
```

Data Type of the Arguments

Number, Integer

Data Type of the Result

Number

Expression

```
EXP (2)
```

Result

7.39

Notes

The approximate value of e is 2.71828183.

EXP is the inverse of Natural Logarithm [LN] (the natural logarithm of number).

To calculate powers of other bases, use the operator ^.

See [Power \[^\]](#).

Natural Logarithm [LN]

Returns the natural logarithm of the given number.

Syntax

```
LN (Number)
```

Data Types of the Argument

Number, Integer

Data Type of the Result

Number

Expression

```
LN (86)
```

Result

4.45

Notes

`LN` is the inverse of the `e Exponent [EXP]` function.

This function works on positive numbers only.

Base Logarithm [LOG]

Returns the base logarithm of the given number.

Syntax

```
LOG (Number, Base)
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

Number, Integer

Data Type of the Result

Number

Expression

```
LOG (8, 2)
```

Result

3

Notes

This function works on positive numbers only.

Base 10 Logarithm [LOG10]

Returns the base 10 logarithm of the given number.

Syntax

```
LOG10 (Number)
```

Data Types of the Arguments

Number, Integer

Data Type of the Result

Number

Expression

```
LOG10 (10)
```

Result

1

Notes

This function works on positive numbers only.

Remainder [MOD]

Modulo operation. Finds the remainder after dividing the Number by the Divisor.

Syntax

```
MOD (Number, Divisor)
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

Number, Integer

Data Type of the Result

Number

Expression

```
MOD (3, 2)
```

Result

1

Notes

To return the quotient (without the remainder), use `Quotient [QUOTIENT]`.

[See also *Divide \[I\]*](#).

Nth Root [NTHROOT]

Calculates the degree-th root of the given argument.

Syntax

```
NTHROOT (Number, Degree)
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

Number: Number, Integer, or other data type compatible with root function

Degree: Number, Integer

Data Type of the Result

Depends on Arguments' Data Types

Example

Find the cube root of the given number.

Expression

```
NTHROOT (125, 3)
```

Result

5

Notes

For square root: [See *Square Root \[SQRT\]*](#).

Power [POWER]

Returns the given number raised to the given power.

Syntax

```
POWER (Base, Exponent)
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

Base: Integer, Number, Length, Area, Volume, Angle

Exponent: Number, Integer

Data Type of the Result

Depends on Arguments' Data Types

Expression

```
POWER (5,2)
```

Result

25

Notes

Works the same as Power [^] (Operator).

Quotient [QUOTIENT]

Returns integer portion of a division.

Syntax

```
QUOTIENT (numerator, denominator)
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

Integer, Number, Length, Area, Volume, or Angle (Must be identical for both arguments)

Data Type of the Result

Number

Example

Divide a slab's Top Surface Area property with its Tile Area property.

The Quotient function returns only the whole (integer) portion of the resulting number.

Expression

```
QUOTIENT (Top Surface Area, Tile Area)
```

Result

1974

Both properties are Area-type data. Dividing them returns a number.

Notes

To return the remainder (without the quotient), use `Remainder [MOD]`.

[See also Divide \[/\].](#)

Sign [SIGN]

Returns +1, 0, or -1.

If the Number is positive, the result is +1.

If the Number is zero, the result is 0.

If the Number is negative, the result is -1.

Syntax

```
SIGN (Number)
```

Data Types of the Arguments

Integer, Number, Length, Area, Volume, Angle

Data Type of the Result

Number

Example

For a profile construction element with an editable Base Width offset parameter:

Determine if its Base Width value is positive or negative.

Expression

```
SIGN (Base Width)
```

Result

-1 for a negative Base Width

1 for a positive Base Width

0 for a Base Width of zero.

Square Root [SQRT]

Square root.

Calculates the square root of the given argument.

Syntax

```
SQRT (Number)
```

Data Types of the Argument

Integer, Number, Area

Data Type of the Result

Depends on Argument's Data Type

Expression

```
SQRT (16)
```

Result

4

Notes

To take other roots: [See *Nth Root \[NTHROOT\]*](#).

Sum [SUM]

Calculates the sum of the given arguments.

Syntax

```
SUM (Number1, [Number2], ...)
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

Integer, Number, Length, Area, Volume, Angle (Must be identical for all Arguments)

Data Type of the Result

Depends on Arguments' Data Type

Example

Add the results of two properties together.

Expression

```
SUM (Raised Floor Level, Sub Floor Thickness)
```

Result

.30

Both arguments are length-type data. The added result is also length-type.

Notes

[See also Add \[+\].](#)

Round to Multiple [MROUND]

Rounds the given number to the desired multiple.

Syntax

```
MROUND (Number, Multiple)
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

Integer, Number, Length, Area, Volume, Angle (Must be identical for all Arguments)

Data Type of the Result

Depends on Arguments' Data Type

Expression

```
MROUND (2567, 10)
```

Result

2670

Notes

If the remainder is greater than or equal to half of the multiple, then MROUND rounds up.

To always round up to a certain multiple, use Round Up to Multiple [MROUNDUP].

To always round down to a certain multiple, use Round Down to Multiple [MROUNDDOWN].

To round numbers to specific decimal places: use Round [ROUND], Round Up [ROUNDUP], Round Down [ROUNDDOWN].

Round Down to Multiple [MROUNDDOWN]

Rounds the given number down to the desired multiple.

Syntax

```
MROUNDDOWN (Number, Multiple)
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

Integer, Number, Length, Area, Volume, Angle (Must be identical for all Arguments)

Data Type of the Result

Depends on Arguments' Data Type

Example

The "Beam Length" property calculates the exact length of existing Beam elements in the model (e.g. 2,044).

The "Prefab Beam Length" property is a fixed value (e.g. 300)

Create an expression to round the Beam Length down to the Prefab increment.

Expression

```
MROUNDDOWN ("Beam Length", 300)
```

Result

1,800 (for a sample Beam Length of 2,044)

Notes

[See also: Round to Multiple \[MROUND\], Round Up to Multiple \[MROUNDUP\].](#)

To round numbers to specific decimal places: use Round [ROUND], Round Up [ROUNDUP], Round Down [ROUNDDOWN].

Round Up to Multiple [MROUNDUP]

Rounds the given number up to the desired multiple.

Syntax

```
MROUNDUP (Number, Multiple)
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

Integer, Number, Length, Area, Volume, Angle (Must be identical for all Arguments)

Data Type of the Result

Depends on Arguments' Data Type

Example

The "Beam Length" property calculates the exact length of existing Beam elements in the model (e.g. 2,044).

The "Prefab Beam Length" property is a fixed value (e.g. 300)

Create an expression to round up the Beam Length to the Prefab increment.

Expression

```
MROUNDUP ("Beam Length", 300)
```

Result

2,100 (for a sample Beam Length of 2,044)

Notes

[See also: Round to Multiple \[MROUND\], Round Down to Multiple \[MROUNDDOWN\].](#)

To round numbers to specific decimal places: use Round [ROUND], Round Up [ROUNDUP], Round Down [ROUNDDOWN].

Round [ROUND]

Rounds a number to a specified number of digits.

Syntax

```
ROUND (Number, NumberOfDigits)
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

Number: Integer, Number

Number of Digits: Integer

If the Number of Digits argument is positive, rounding takes place to the specified number of decimal places. [See Expression 1 below.](#)

If the Number of Digits argument is zero, *the number is rounded to the nearest integer.* [See Expression 2 below.](#)

If the Number of Digits argument is negative, *rounding takes place only to the left of the decimal point (decimal values are disregarded).* [See Expression 3 below.](#)

Data Type of the Result

Number

The number of decimal places displayed in the result depends on Preferences > Working Units.

Examples

Expression 1

```
ROUND (46.9855, 2)
```

Result

```
46.99
```

Expression 2

```
ROUND (46.9855, 0)
```

Result

```
47.00
```

Expression 3

```
ROUND (46.9855, -1)
```

Result

```
50.00
```

Notes

To always round up, use Round Up [ROUNDUP]. To always round down, use Round Down [ROUNDDOWN].

To round to a certain multiple, use Round to Multiple [MROUND] (or Round Up to Multiple [MROUNDUP] or Round Down to Multiple [MROUNDDOWN]).

Round Down [ROUNDDOWN]

Rounds a number down to a specified number of digits.

If the Number of Digits argument is positive, rounding takes place downward to the specified number of decimal places.

If the Number of Digits argument is zero, the number is rounded downward to the nearest integer.

If the Number of Digits argument is negative, rounding down takes place to the given number of places, to the left of the decimal point.

Syntax

```
ROUNDDOWN (Number, NumberOfDigits)
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

Number: Integer, Number

Number of Digits: Integer

Data Type of the Result

Number

Example

Expression 1

```
ROUNDDOWN (46.9855, 2)
```

Result

```
46.98
```

Expression 2

```
ROUNDDOWN (46.9855, 0)
```

Result

```
46.00
```

Expression 3

```
ROUNDDOWN (46.9855, -1)
```

Result

```
40.00
```

Notes

[See also: Round \[ROUND\], Round Up \[ROUNDUP\].](#)

To round to a certain multiple, use Round to Multiple [MROUND] (or Round Up to Multiple [MROUNDUP] or Round Down to Multiple [MROUNDSDOWN]).

Round Up [ROUNDUP]

Rounds a number up to a specified number of digits.

If the Number of Digits argument is positive, rounding takes place upward to the specified number of decimal places.

If the Number of Digits argument is zero, the number is rounded up to the nearest integer.

If the Number of Digits argument is negative, rounding up takes place to the given number of places, to the left of the decimal point.

Syntax

```
ROUNDUP (Number, NumberOfDigits)
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

Number: Integer, Number

Number of Digits: Integer

Data Type of the Result

Number

Example

The "Number of Tiles" property divides a Slab's top surface area by the area of a single tile. The result is 371.35.

Create another property which rounds up this result to a whole number.

Expression

```
ROUNDUP (Number of tiles, 0)
```

Result

372 (where the Number of Tiles result is 371.35)

Notes

[See also: Round \[ROUND\], Round Down \[ROUNDDOWN\].](#)

To round to a certain multiple, use Round to Multiple [MROUND] (or Round Up to Multiple [MROUNDUP] or Round Down to Multiple [MROUNDDOWN]).

TRIGONOMETRICAL FUNCTIONS

Arccosine [ACOS]

Returns the arccosine of a number.

Syntax

```
ACOS (Number)
```

Data Type of the Argument

Number, Integer. Must be between -1 and 1.

Data Type of the Result

Angle

Expression

```
ACOS (-0.5)
```

Result

120 deg

Notes

Angle unit of the result (e.g. degrees, radians, grad) depends on what you set in Project Preferences – Calculation Units – Angle Units.

Arccotangent [ACOT]

Returns the arccotangent of a number.

Syntax

```
ACOT (Number)
```

Data Type of the Argument

Number, Integer

Data Type of the Result

Angle

Expression

```
ACOT (2)
```

Result

27 deg

Notes

Angle unit of the result (e.g. degrees, radians, grad) depends on what you set in Project Preferences – Calculation Units – Angle Units.

Arcsine [ASIN]

Returns the arcsine of a number.

Syntax

```
ASIN (Number)
```

Data Type of the Argument

Number, Integer. Must be between -1 and 1.

Data Type of the Result

Angle

Expression

```
ASIN (-0.5)
```

Result

-30 deg

Notes

Angle unit of the result (e.g. degrees, radians) depends on what you set in Project Preferences - Angle Units.

Arctangent [ATAN]

Returns the arctangent of a number.

Syntax

```
ATAN (Number)
```

Data Types of the Arguments

Number, Integer

Data Type of the Result

Angle

Expression

```
ATAN (1)
```

Result

45 deg

Notes

Angle unit of the result (e.g. degrees, radians) depends on what you set in Project Preferences - Angle Units.

Cosine [COS]

Returns the cosine of an angle.

Syntax

```
COS (Angle)
```

Data Type of the Argument

Angle

Data Type of the Result

Number

Expression

```
COS (60 D)
```

Result

0.5

Notes

The argument's angle value must be expressed as a unit (Degree, Radian, Grad). Choose an angle unit from the Units pop-up of Expression Editor.

Cotangent [COT]

Returns the cotangent of an angle

Syntax

```
COT (Angle)
```

Data Types of the Arguments

Angle

Data Type of the Result

Number

Expression

```
COT (30D)
```

Result

1.73

Notes

The argument's angle value must be expressed as a unit (Degree, Radian, Grad).

Choose an angle unit from the Units pop-up of Expression Editor. Here, it is Degree (D).

Sine [SIN]

Returns the sine of an angle.

Syntax

```
SIN (Angle)
```

Data Type of the Argument

Angle

Data Type of the Result

Number

Expression

```
SIN (30D)
```

Result

0.50

Notes

The argument's angle value must be expressed as a unit (Degree, Radian, Grad).

Choose an angle unit from the Units pop-up of Expression Editor. Here, it is Degree (D).

Tangent [TAN]

Returns the tangent of an angle.

Syntax

```
TAN (Angle)
```

Data Types of the Arguments

Angle

Data Type of the Result

Number

Expression

```
TAN (0.785 R)
```

Result

1.00

Notes

The argument's angle value must be expressed as a unit (Degree, Radian, Grad).

Choose an angle unit from the Units pop-up of Expression Editor. Here, it is Radian (R).

STATISTICAL FUNCTIONS

Average [AVG]

Calculates the average of the given arguments.

Syntax

```
AVG (Number1, [Number2], ...)
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

Integer, Number, Length, Area, Volume, or Angle (Must be identical for all arguments)

Data Type of the Result

Same as Arguments' Data Type

Example

Columns are cut by a slanted Roof, so that each Column has both a maximum and a minimum height. For any column, use AVG to find its average height.

Expression

```
AVG (Maximum Height of the Column, Minimum Height of the Column)
```

Result

3.08 (for a given column with a maximum height of 3.11 and a minimum height of 3.04)

Note

Both arguments are Length type data, so the Property's data type must also be set to Length.

Maximum [MAX]

Calculates the maximum of the given arguments.

Syntax

```
MAX (Number1, [Number2], ...)
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

Integer, Number, Length, Area, Volume, or Angle (Must be identical for all arguments)

Data Type of the Result

Same as Arguments' Data Type

Example

A Beam intersects with another Beam, and thus has different lengths on either side.

Use `MAX` to return the longer of the two lengths.

Expression

```
MAX (Length Left, Length Right)
```

Result

3.86 (for a given Beam with side lengths of 3.86 and 3.56)

Minimum [MIN]

Calculates the minimum of the given arguments.

Syntax

```
MIN (Number1, [Number2], ...)
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

Integer, Number, Length, Area, Volume, or Angle (Must be identical for all arguments)

Data Type of the Result

Same as Arguments' Data Type

Example

A Beam intersects with another Beam, and thus has different lengths on either side. (3.86 and 3.56).

Use `MIN` to return the shorter of the two lengths.

Expression

```
MIN (Length Left, Length Right)
```

Result

3.56 (for a given Beam with side lengths of 3.86 and 3.56)

TEXT EDITING FUNCTIONS

Compare [COMPARE]

Compares two strings.

Returns 0 if they are equal

Returns -1 if text1 precedes text2 in alphanumeric order

Returns +1 if text1 comes after text2 in alphanumeric order.

The alphabetical order is case sensitive: a comes before A.

Syntax

```
COMPARE ("text1", "text2")
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

String

Data Type of the Result

Integer

Example

Suppose that an element's FireRating property can have a value from A through F.

If FireRating is A, B or C: it is standard-compliant.

If FireRating is D, E, or F: it is non-compliant.

In this expression, use COMPARE to compare the element's FireRating category to "C".

In addition, use an IF function to return one of two strings based on the result of the COMPARE function.

Expression

```
IF (COMPARE (FireRating, "C") < 1, "Standard-compliant", "Non-Compliant")
```

If the result is 0, this means that FireRating equals C. This is standard-compliant.

If the result is -1, this means that FireRating comes before C alphabetically: it is either A or B. This is also standard-compliant.

If the result is 1, this means that FireRating comes after C alphabetically: it is either D, E or F. This is non-compliant.

Result

"Standard-compliant" for values of 0 or -1, otherwise "Non-compliant"

Merge Strings [CONCAT]

Joins two or more text strings into one.

Syntax

```
CONCAT ("text1", ["text2"], ...)
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

String

Data Type of the Result

String

Example

Create a property called "Door Context ID", consisting of a door's Hotlink Master ID plus its Related Zone Number. (Both are String-type properties.)

The two strings should be separated by a dash.

Expression

```
CONCAT (Hotlink Master ID,"-", Related Zone Number)
```

Result

Room 213-6

Notes

Don't forget that a Property can be applied to an element (including Hotlink elements) only if the Property is available to that element.

The Evaluate function (from Property Manager) does not work on Hotlink Elements. However, the expression involving Hotlink element properties is correctly evaluated in Interactive Schedule, in Element Settings, on the Element Information Palette, and any other interface, provided that the Property is valid and available.

`CONCAT` is similar to `Join Strings [TEXTJOIN]`. Use `Join Strings [TEXTJOIN]` if you want to specify an automatic delimiter between all text strings, and to ignore empty strings.

For another example using `CONCAT`, see [Example 3: Expand Property with Descriptive Text String](#).

Contains [CONTAINS]

Checks whether a specified string is contained within a text.

Returns True if the string is present

Returns False if the string is not present

Case sensitive by default. (Otherwise, enter False for the CaseSensitive argument.)

Syntax

```
CONTAINS ("FindText", "WithinText", [CaseSensitive])
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

All Texts: String

Case sensitive (optional) True/False

Data Type of the Result

True/False

Example

Based on the text of a manufacturers catalog: Display the appropriate length unit (either cm or inches) for a chair's Height value.

Here, the catalog text (a string) is a predefined property value called HeightInString.

This string is transformed into a number (using STRTONUM).

It is then multiplied by the appropriate length unit (either cm or inches, depending on whether the string contains the "cm" tag: this is determined by the CONTAINS function).

The result is a Length-type property.

Expression

```
STRTONUM (HeightInString) * IF (CONTAINS ("cm", HeightInString), 1 cm, 1 in)
```

Result

For a HeightInString property string value of "150 cm", displays a length value of "150 cm".

Find String [FIND]

Returns the position of a text string ("FindText") within another text string ("WithinText").

Search starts at first character by default. (Otherwise, search starts at the character at Position.)

Search is case-sensitive by default. (Otherwise, enter False for the CaseSensitive argument.)

Syntax

```
FIND ("FindText", "WithinText", [Position], [CaseSensitive?])
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

All Texts: String

Starting Position (Optional): Integer

Case-sensitive? (Optional): True/False

Data Type of the Result

Integer

Example

Find the position of the dash character within the Element ID (e.g. "DOO - 007").

Expression

```
FIND ("-", Element ID)
```

Result

5 (for an Element ID of "DOO - 007")

String from Left [LEFT]

Returns a specified number of characters from the start of a text string.

Syntax

```
LEFT ("Text", NumberOfCharacters)
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

Text: String

Number of Characters: Integer

Data Type of the Result

String

Example

Starting at the left end of the given text string, return the first nine characters.

Expression

```
LEFT ("The quick brown fox", 9)
```

Result

The quick

Length of String [LEN]

Returns the number of characters (including spaces) in a string.

Syntax

```
LEN ("Text")
```

Data Types of the Arguments

String

Data Type of the Result

Integer

Expression

```
LEN ("The quick brown fox")
```

Result

19

Lower Case [LOWER]

Converts all characters of a text string to lower case.

Syntax

```
LOWER ("Text")
```

Data Types of the Arguments

String

Data Type of the Result

String

Expression

```
LOWER ("BRICK WALLS")
```

Result

brick walls

String from Position [MID]

Returns a specified number of characters starting from a specified position within a text string.

Syntax

```
MID ("Text", Position, NumberOfCharacters)
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

Text: String

Starting Position: Integer

Number of Characters: Integer

Data Type of the Result

String

Example

From the given text string, start at the 11th position and return the next five characters.

Expression

```
MID ("The quick brown fox", 11, 5)
```

Result

brown

Replace in String [REPLACE]

Replaces a given number of characters within a string with another string, starting from a specified position.

Syntax

```
REPLACE ("OldText", Position, NumberOfCharacters, "NewText")
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

Old and new text: String

Starting Position: Integer

Number of Characters: Integer

Data Type of the Result

String

Example

In the string "2017", start at position 3, and replace two characters with "18"

Expression

```
REPLACE ("2017", 3, 2, "18")
```

Result

2018

Notes

Use `Substitute Strings [SUBSTITUTE]` to replace one string with another, in all instances or a particular instance.

Use `REPLACE` to change a particular string to another string, with a given starting position.

Repeat String [REPT]

Repeats a string a specified number of times.

Syntax

```
REPT ("Text", NumberOfTimes)
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

Text: String

Number of times: Integer

Data Type of the Result

String

Example

Display an asterisk and a dash 3 times.

Expression

```
REPT ("*-", 3)
```

Result

```
*-*-*-
```

String from Right [RIGHT]

Returns a specified number of characters (including spaces) from the end of a text string.

Syntax

```
RIGHT ("Text", NumberOfCharacters)
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

Text: String

Number of Characters: Integer

Data Type of the Result

String

Example

Return the final nine characters from the (right) end of a text string.

Expression

```
RIGHT ("The quick brown fox", 9)
```

Result

```
brown fox
```

Split String [SPLIT]

Splits a string into parts at a given delimiter and returns the specified part.

Syntax

```
SPLIT("Text", "Delimiter", PartIndex)
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

Text and Delimiter: String

Part Index: Integer

Data Type of the Result

String

Example

A Wall's "Brick dimensions" Property is a string: "92mm x 57 mm x 194 mm".

Use SPLIT to get just the second item (the width) from this string: split the string at the "x" delimiters and return the second part.

Expression

```
SPLIT (Brick Dimensions, "x", 2)
```

Result

57 mm

Split String to Left [SPLITLEFT]

Splits a string into parts at a given delimiter.

Returns the specified part and all parts to its left.

Syntax

```
SPLITLEFT ("Text", "Delimiter", PartIndex)
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

Text and Delimiter: String

Part Index: Integer

Data Type of the Result

String

Example

Given an Archicad Element ID, such as TREAD-018 (a string-type data):

Get the part of the string that is on the left of the delimiter.

The delimiter, which separates the Element ID into parts, is a dash.

The part number (index) you want is 1 (the first part).

Expression

```
SPLITLEFT (Element ID, "-", 1)
```

Result

TREAD (for an Element ID of "TREAD-018")

Split String to Right [SPLITRIGHT]

Splits a string into parts at a given delimiter.

Returns the specified part and all parts to its right.

Syntax

```
SPLITRIGHT ("Text", "Delimiter", PartIndex)
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

Text and Delimiter: String

Part Index: Integer

Data Type of the Result

String

Example

Given an Archicad Element ID, such as TREAD-018:

Get the numerical part of the ID.

The delimiter, which separates the Element ID into parts, is a dash.

The part number (index) is 2 (the second part).

Expression

```
SPLITRIGHT (Element ID, "-", 2)
```

Result

018 (for an Element ID of "TREAD-018")

Substitute Strings [SUBSTITUTE]

Replaces a string with another string, in the specified instance (or all instances).

Syntax

```
SUBSTITUTE ("Text", "OldText", "NewText", [InstanceIndex])
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

All Texts: String

InstanceIndex (Optional): Integer

If InstanceIndex value is omitted, then all instances of the OldText are replaced.

Data Type of the Result

String

Example

A Wall's "Brick dimensions" Property is a string: "92mm x 57 mm x 194 mm".

Use `SUBSTITUTE` to extract just the numbers from this string (e.g. for use in another expression).

(For the OldText "mm" strings, substitute a NewText of nothing.)

Expression

```
SUBSTITUTE (Brick Dimensions, "mm", "")
```

The InstanceIndex is not provided. Thus, all instances of the OldText will be substituted.

Result

92 x 57 x 194

Notes

Use `SUBSTITUTE` to replace one string with another, in all instances or a particular instance.

Use `Replace in String [REPLACE]` to change a particular string to another string, with a given starting position.

Join Strings [TEXTJOIN]

Joins two or more text strings into one string, separated by a given delimiter.

If IgnoreIfEmpty is True, the result will ignore empty strings.

Syntax

```
TEXTJOIN ("Delimiter", IgnoreIfEmpty, "Text1", "Text2", ...)
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

Delimiter and Text: String

Ignore if Empty: True/False

Data Type of the Result

String

Example

For each element, list its ID plus Zone Name plus Renovation Status.

The Delimiter consists of a comma and a space.

Expression

```
TEXTJOIN (" , ", TRUE, Element ID, Related Zone Name, Renovation Status)
```

Result

SW-002, Kitchen, To Be Demolished

Notes

`TEXTJOIN` is similar to Merge Strings [CONCAT].

Compared to `CONCAT`, `TEXTJOIN` has the following additional features:

- add a specific delimiter character between each string automatically
- option to ignore empty strings

Trim [TRIM]

Removes duplicate spaces, and the spaces from the start and end of a text string.

Syntax

```
TRIM ("Text")
```

Data Types of the Arguments

String

Data Type of the Result

String

Example

The text string has unneeded spaces at the beginning and the end, and multiple extra spaces between each word.

Use `TRIM` to remove these unneeded spaces.

Expression

```
TRIM (" Q4 2017 Earnings Report ")
```

Result

Q4 2017 Earnings Report

Upper Case [UPPER]

Converts all characters of a text string to upper case.

Syntax

```
UPPER ("Text")
```

Data Types of the Arguments

String

Data Type of the Result

String

Expression

```
UPPER ("client address")
```

Result

CLIENT ADDRESS

DATA CONVERSION

Convert to String [STR]

Converts a non-unit numeric value or a True/False value into a string.
The resulting string shows two decimal places by default.

Syntax

```
STR (Value, [NumberOfDecimals])
```

Note: The required separator (comma or semi-colon) automatically corresponds to your computer's location settings.

Data Types of the Arguments

Value: Integer, Number, True/False

Number of Decimals (optional): Integer

Data Type of the Result

String

Example

Convert the Home Story Number element property (a number-type value) to a string.
Make sure the result is a whole number showing zero decimal places.

Expression

```
STR (Home Story Number, 0)
```

Result

2 (for a Beam with Home Story of 2)

Notes

The result string is displayed with two decimal places by default. If you do not want any decimals (as in the Home Story Number example), add 0 as the Number of Decimals argument.

To convert a unit value (e.g. a Length) to a string, use the `Convert Unit to String [STRCALCUNIT]` function.

For another example using `STR`, see [Example 10: List the Total Cost of a Structure in Dollars](#).

Convert Unit to String [STRCALCUNIT]

Converts a *unit value* to a string, based on the Calculation Units settings in Project Preferences.

Syntax

```
STRCALCUNIT (value)
```

Data Types of the Arguments

Length, Area, Volume, Angle

Data Type of the Result

String

Example

Convert the Insulation Skin Thickness (a Length) into a String

Expression

```
STRCALCUNIT (Insulation Skin Thickness)
```

Result

25

Notes

To convert a *non-unit* value (e.g. Home Story Number) to a string, use the `Convert to String [STR]` function.

For another example using STRCALCUNIT, see [Example 3: Expand Property with Descriptive Text String](#).

Number from String [STRTONUM]

Extracts numerical character(s) from the initial position in a string.

(Only numerical characters at the beginning of the string are considered. Numerical characters within the string are not found.)

Syntax

```
STRTONUM ("Text")
```

Data Types of the Arguments

String

Data Type of the Result

Number

Example

Extract the number which is the first character of the element's Archicad IFC ID (a string-type parameter). (e.g. "3MtfJD8lf96mvn65fX2qo")

Expression

```
STRTONUM (Archicad IFC ID)
```

Result

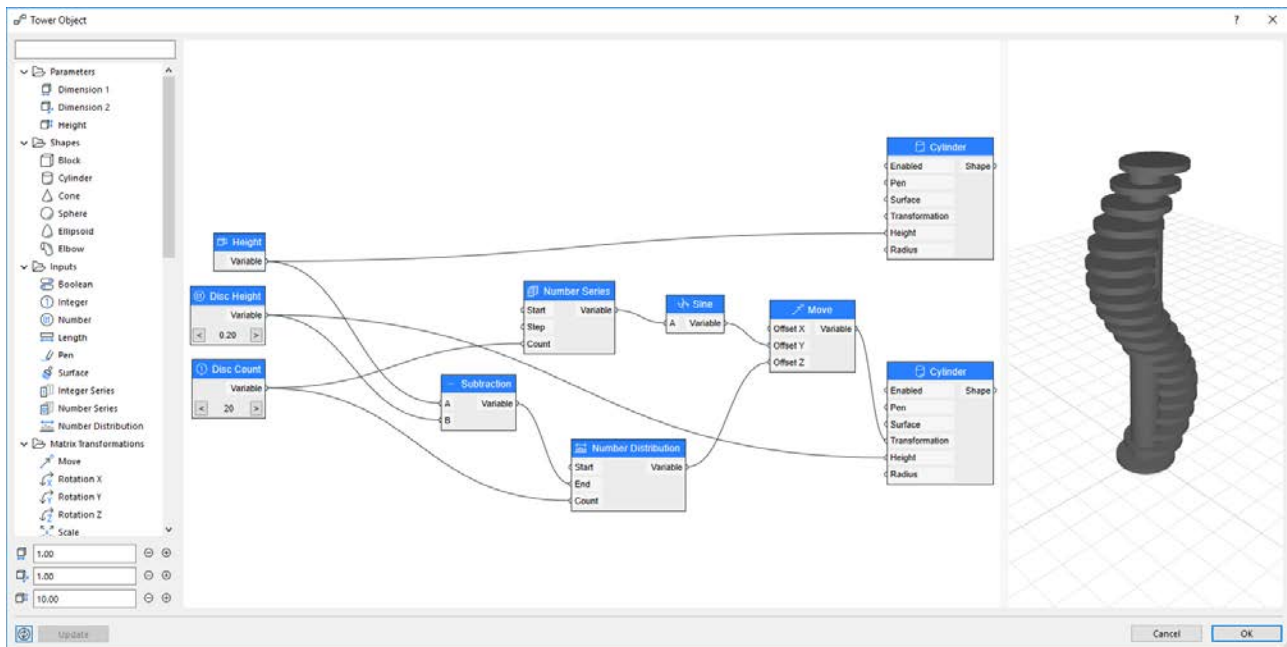
3 (for an element whose ID is "3MtfJD8lf96mvn65fX2qo")

PARAM-O Reference Guide

Note: PARAM-O is available for SSA/Forward customers only.

PARAM-O is a node-based environment to create parametric Archicad objects. You can design your objects in a graphical interface, and then PARAM-O saves it as a regular Archicad object in the embedded library.

The PARAM-O scripts are stored as a section of the library part. The created objects will work in every Archicad - even if PARAM-O is not installed.



Installation

Getting Started

PARAM-O Manager

PARAM-O Editor

Creating an Object

Making an Object Parametric

For details on each node type, see:

PARAM-O Node Documentation

[Watch the Video](#)

[Read More](#)

Installation

Note: PARAM-O is available to SSA/Forward customers only.

No Installation Needed

PARAM-O is part of the Archicad product and does not need to be installed separately. It is updated together with Archicad.

Older PARAM-O Version Must Be Uninstalled

If you have an older PARAM-O version installed, please uninstall it.

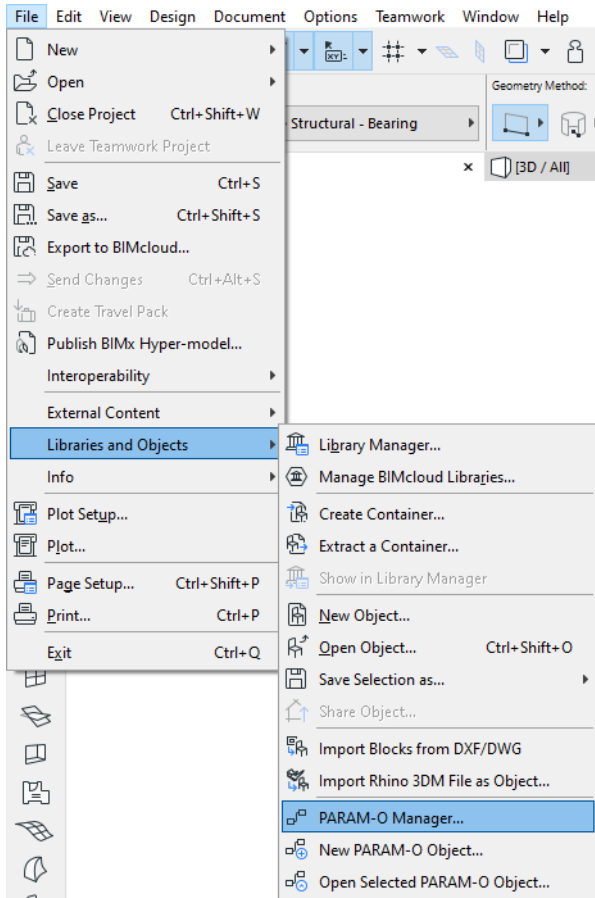
1. Go to the folder where the older PARAM-O.apx file was installed (typically, the Archicad “Add-ons” folder).
2. Delete the .apx file.
3. Restart Archicad.

Note: If your older version was in the Archicad “Add-ons/Standard” folder, it is automatically overwritten by the latest version. There is no need to uninstall it.

Getting Started

PARAM-O registers three menu commands in Archicad's menu structure. You can find these commands in the **File > Libraries and Objects** submenu:

- **PARAM-O Manager:** Opens the manager dialog, where you can manage your existing objects.
- **New PARAM-O Object:** Opens the editor dialog, where you can create a new object without opening the manager dialog.
- **Edit Selected PARAM-O Object:** Opens the editor dialog for the selected library part.

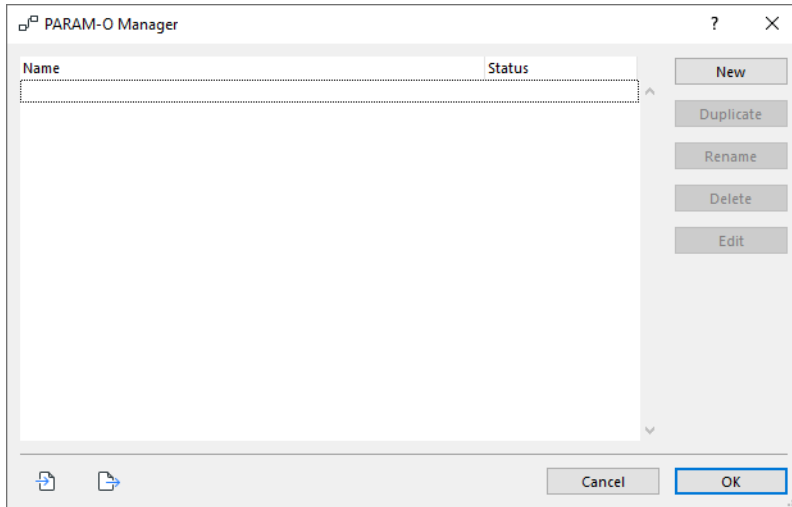


PARAM-O Manager

Use PARAM-O Manager to create and manage your objects.

- Create new objects or edit existing ones.
- Duplicate, rename or delete an existing object.
- Export/Import objects to/from a file on your computer.

Please note that if you delete an object from PARAM-O Manager, it will also be deleted from the embedded library. Renaming an object means that the old object will be deleted and a new one will be created.

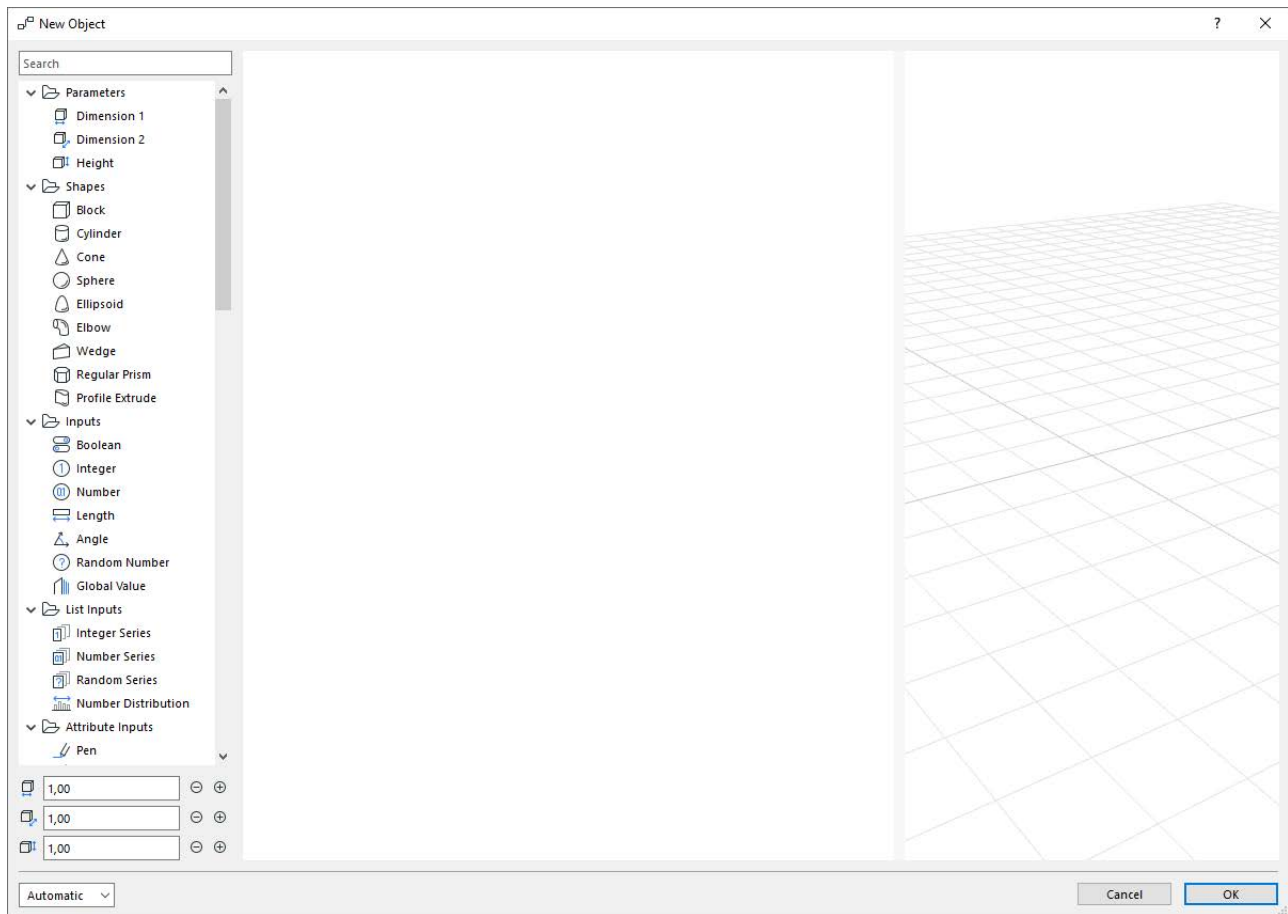


Teamwork

To use any of the functions, you must have the access right to create/modify/delete library parts in the embedded library. If you don't have these rights, you will get an error message.

PARAM-O Editor

PARAM-O Editor is the place where you create or modify your scripts.



Parts of the editor dialog:

- Node Tree
- Editor Canvas
- Model Preview

Node Tree

On the left side of the editor, you can find the available nodes. You can create a new node by double-clicking on it or by dragging and dropping it to the editor canvas.

Editor Canvas

The canvas is the main part of the editor. Here you can edit your script by connecting nodes and setting node parameters.

Model Preview

On the right side of the editor, you see the object preview. Please note that the preview is not exactly the same as what you will see in Archicad. Here are some limitations:

- No texture handling. Every object appears with the base color of the surface.
- No edge rendering. You can set the pen color, but the edges won't be visible in the preview.

- Limited transparency handling. You can see transparent surfaces in the preview, but Archicad has a much better transparency handling method.

You can modify the grid visibility and settings from the preview's context menu.

Automatic and Manual Preview

At the bottom-left corner of the editor, you can switch the preview mode between automatic and manual. In automatic mode, the preview is updated if anything changes in the editor. But with large models, updating the preview may be time-consuming. In this case, you can switch to manual mode and rebuild the model only when needed, using the Update button.

Unit handling

Currently, every number value is in meters. If you use the Length node, then the parameter will appear in Archicad taking the working unit settings into account, but you still have to think in meters during editing.

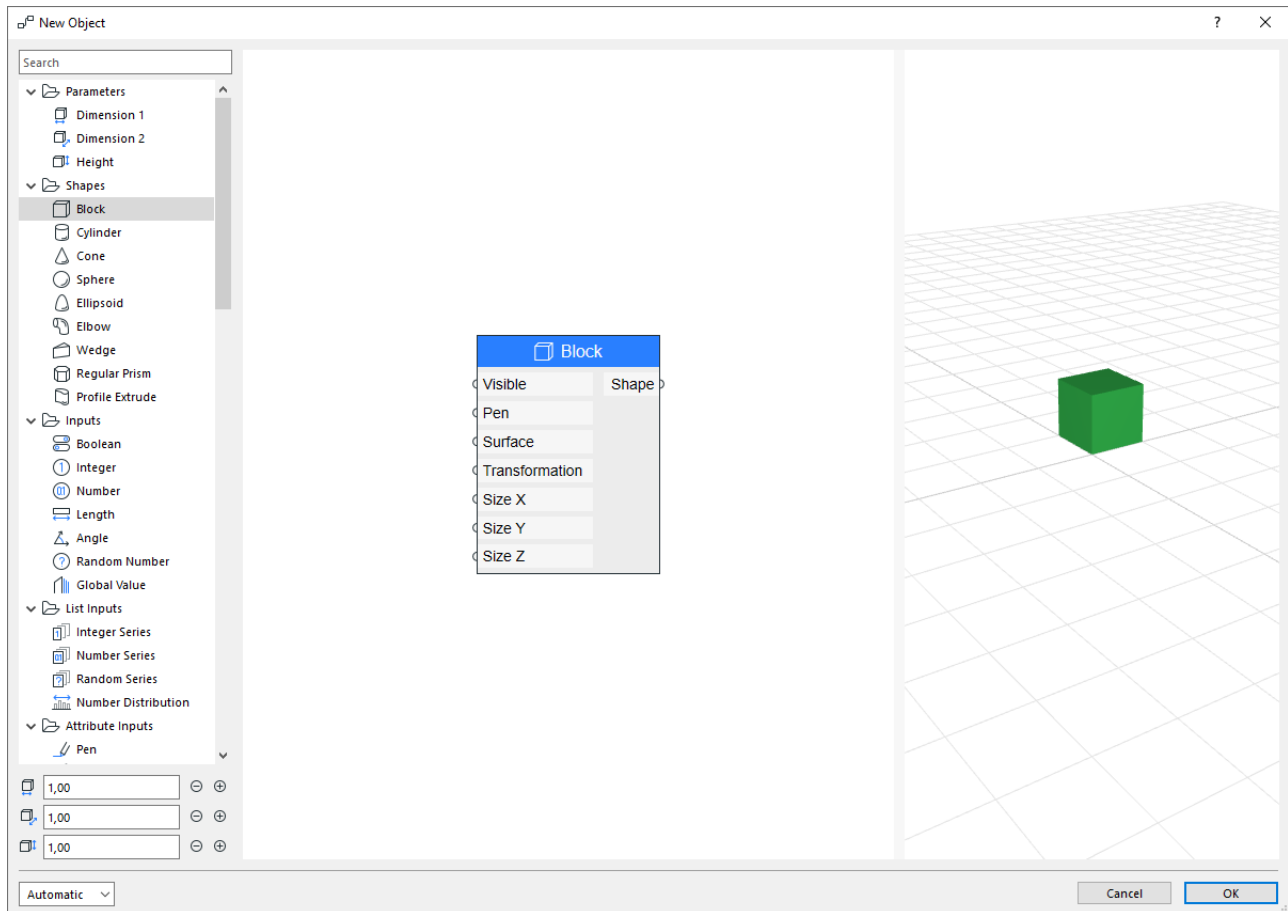
Hotkeys in Editor

Note: Copy-Paste hotkeys work on scripts, too.

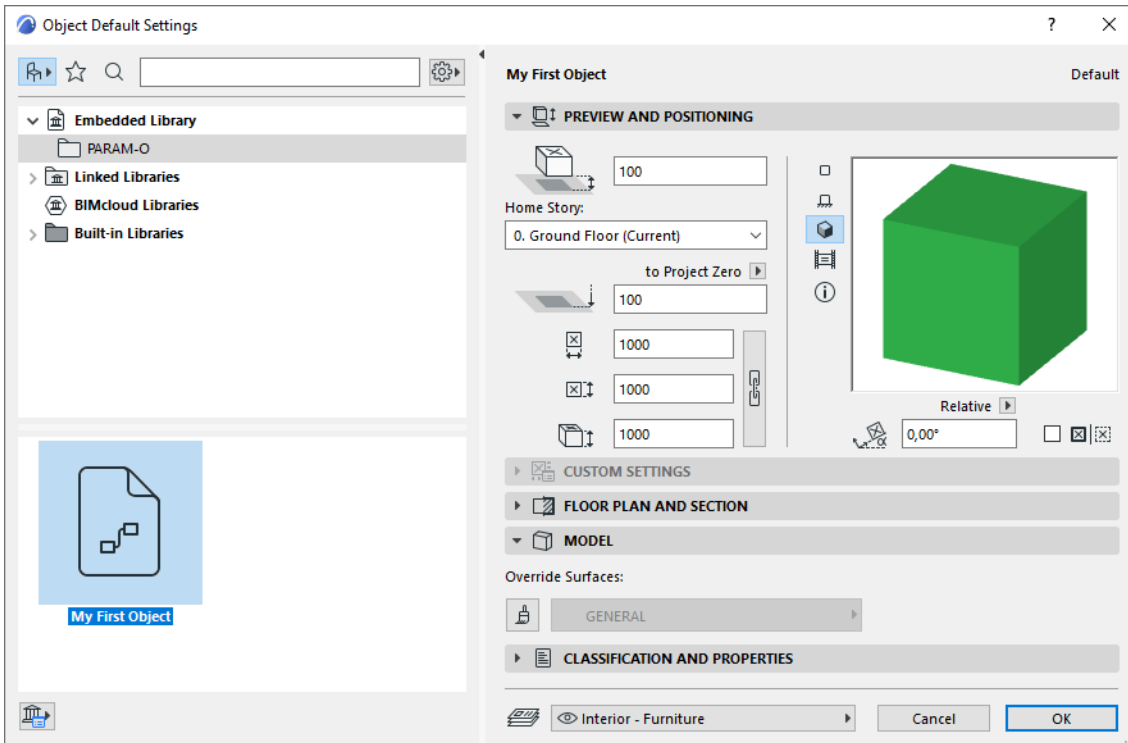
Win	Command	Mac
Esc	Undo selection	Esc
Delete or Backspace	Delete selected nodes	Delete or Backspace
Ctrl+A	Select all nodes	Cmd+A
Ctrl+C	Copy the selected nodes to the clipboard	Cmd+C
Ctrl+V	Paste nodes from the clipboard	Cmd+V
Ctrl+Z	Undo the last modification	Cmd+Z
Ctrl+Shift+Z	Redo the last modification	Cmd+Shift+Z

Creating an Object

Let's see an example. To create a simple block object, double-click on the Block node in the editor. The new node appears on the canvas, and the preview is updated automatically. Set the block's default parameters from the node's context menu.



After clicking OK, give your object a name. The name should be unique among all library objects. After you define the name, the object will be created in the embedded library, in the PARAM-O folder. It is also set as the default for the Object Tool.

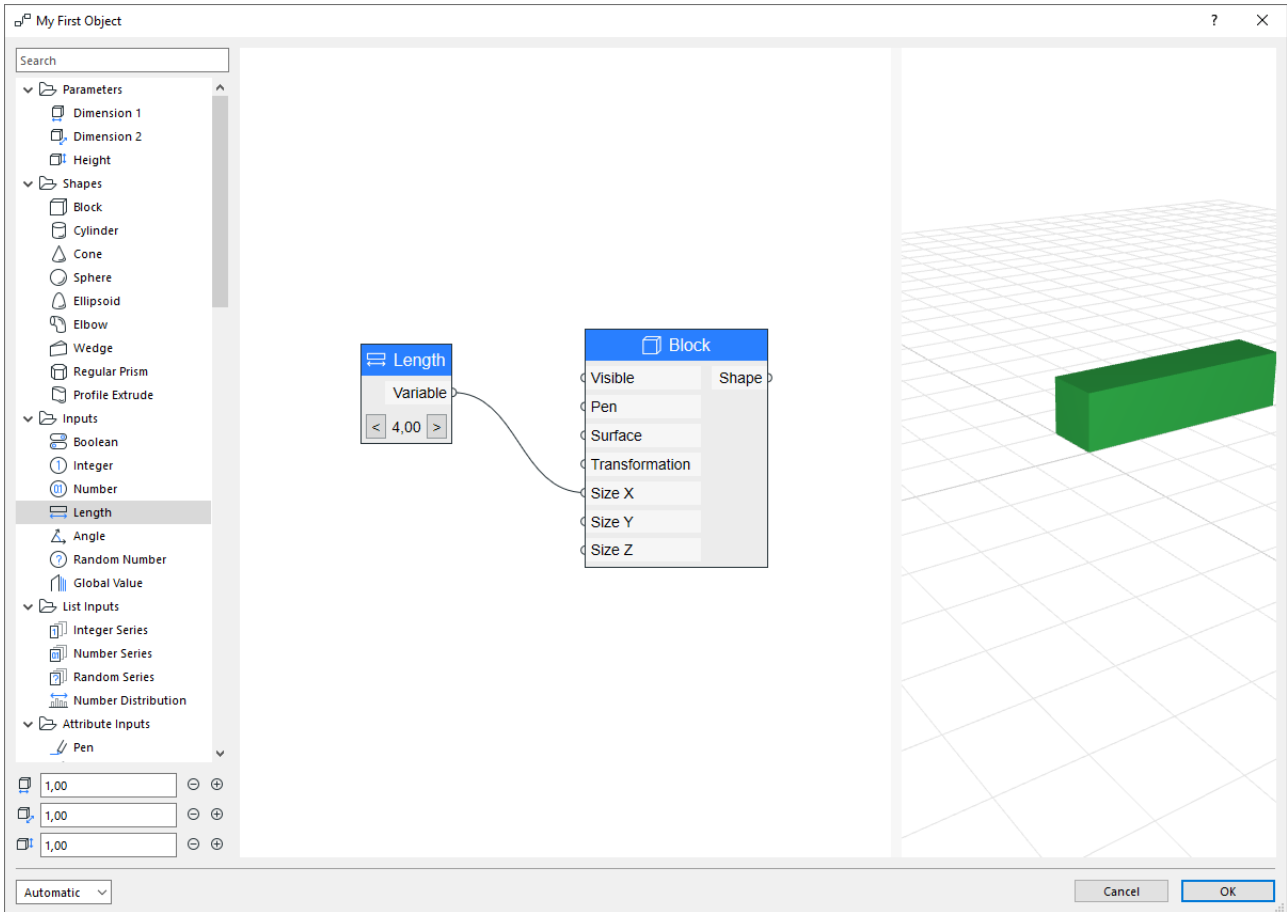


Now you can use your object in Archicad like any other library object. The 2D representation of the object will be the 2D projection of its 3D model.

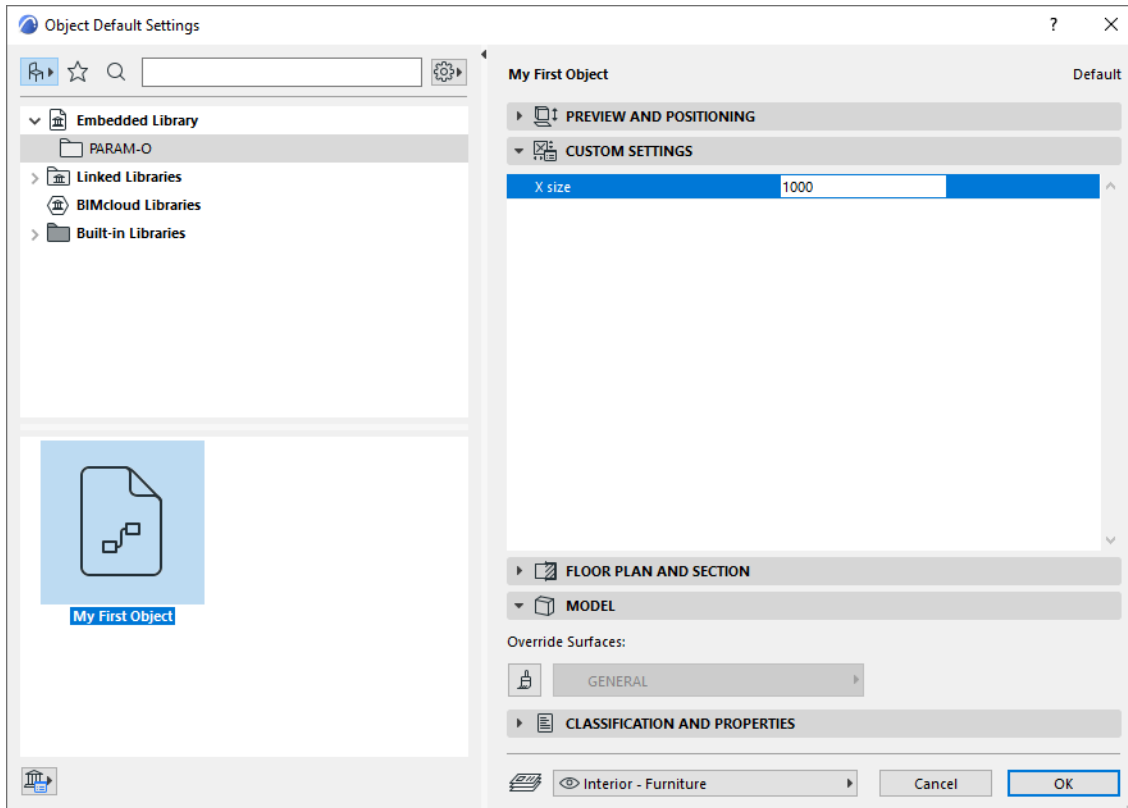
You are probably wondering why the object is green by default. This color comes from the Object Tool's default settings. If no surface is specified, we use the object's default pen to generate the surface.

Making an Object Parametric

To make an object parametric, you have to use an input node. For example, you can create a Length node and connect it to the Size X input of the Block node.



Now the value can be set by modifying the value of the Length node. To make an object parameter from the Length node, open the settings dialog and set the Create Parameter value to true. You can also set a name for this parameter. This name will be visible in the Object Settings Dialog.



You can use as many parameters as you wish for your objects.

Using Standard Parameters

There are some special parameters in Archicad: Dimension 1, Dimension 2 and Height. Set the value of these parameters in the Preview and Positioning page of the Object Settings Dialog. The real size of a well-designed library object is always synchronized with these values.

To use the value of these parameters, use the corresponding nodes from the Parameters group in the editor. The default value of these nodes can be set below the Node Tree.

Available Nodes

See [PARAM-O Node Documentation](#) for details on all node types.

PARAM-O Node Documentation

[Parameters](#)

[Shapes](#)

[Inputs](#)

[List Inputs](#)

[Attribute Inputs](#)

[Point Inputs](#)

[Transformations](#)

[Shape Transformations](#)

[Maths](#)

[Trigonometry](#)

[Solid Operations](#)

[Extras](#)

Parameters

 **Dimension 1**

 **Dimension 2**

 **Height**

Dimension 1



This node outputs the standard Dimension 1 parameter of a GDL object. You can modify this value in Archicad on the Preview and Positioning tab page of the object's settings dialog. During editing you can set the default value in the bottom left corner of the editor.

Outputs

Name	Type	Description
Variable	Number	The value of the parameter.

Dimension 2



This node outputs the standard Dimension 2 parameter of a GDL object. You can modify this value in Archicad on the Preview and Positioning tab page of the object's settings dialog. During editing you can set the default value in the bottom left corner of the editor.

Outputs

Name	Type	Description
Variable	Number	The value of the parameter.

Height



This node outputs the standard Height parameter of a GDL object. You can modify this value in Archicad on the Preview and Positioning tab page of the object's settings dialog. During editing you can set the default value in the bottom left corner of the editor.

Outputs

Name	Type	Description
Variable	Number	The value of the parameter.

Shapes

 **Block**

 **Cylinder**

 **Cone**

 **Sphere**

 **Ellipsoid**

 **Elbow**

 **Wedge**

 **Regular Prism**

 **Profile Extrude**

Block



Creates a block.

Inputs

Name	Type	Description
Visible	Boolean	Defines if the model is visible or not.
Pen	Pen	Pen color of the edges.
Surface	Surface	Surface color of the polygons.
Transformation	Transformation	Transformation of the shape.
Size X	Number	X size of the block.
Size Y	Number	Y size of the block.
Size Z	Number	Z size of the block.

Outputs

Name	Type	Description
Shape	Shape	The resulting shape.

Cylinder



Creates a cylinder.

Inputs

Name	Type	Description
Visible	Boolean	Defines if the model is visible or not.
Pen	Pen	Pen color of the edges.
Surface	Surface	Surface color of the polygons.
Transformation	Transformation	Transformation of the shape.
Height	Number	Height of the cylinder.
Radius	Number	Radius of the cylinder.

Outputs

Name	Type	Description
Shape	Shape	The resulting shape.

Cone



Creates a cone.

Inputs

Name	Type	Description
Visible	Boolean	Defines if the model is visible or not.
Pen	Pen	Pen color of the edges.
Surface	Surface	Surface color of the polygons.
Transformation	Transformation	Transformation of the shape.
Height	Number	Height of the cone.
Bottom Radius	Number	Bottom radius of the cone.
Top Radius	Number	Top radius of the cone.

Outputs

Name	Type	Description
Shape	Shape	The resulting shape.

Sphere



Creates a sphere.

Inputs

Name	Type	Description
Visible	Boolean	Defines if the model is visible or not.
Pen	Pen	Pen color of the edges.
Surface	Surface	Surface color of the polygons.
Transformation	Transformation	Transformation of the shape.
Radius	Number	The radius of the sphere.

Outputs

Name	Type	Description
Shape	Shape	The resulting shape.

Ellipsoid



Creates an ellipsoid.

Inputs

Name	Type	Description
Visible	Boolean	Defines if the model is visible or not.
Pen	Pen	Pen color of the edges.
Surface	Surface	Surface color of the polygons.
Transformation	Transformation	Transformation of the shape.
Height	Number	Height of the ellipsoid.
Radius	Number	Radius of the ellipsoid.

Outputs

Name	Type	Description
Shape	Shape	The resulting shape.

Elbow



Creates an elbow (curved tube).

Inputs

Name	Type	Description
Visible	Boolean	Defines if the model is visible or not.
Pen	Pen	Pen color of the edges.
Surface	Surface	Surface color of the polygons.
Transformation	Transformation	Transformation of the shape.
Radius 1	Number	Radius of the curve of the elbow.
Radius 2	Number	Radius of the tube of the elbow.
Alpha	Number	Curve angle of the elbow in degrees.

Outputs

Name	Type	Description
Shape	Shape	The resulting shape.

Wedge



Creates a wedge (right-angle triangle based prism).

Inputs

Name	Type	Description
Visible	Boolean	Defines if the model is visible or not.
Pen	Pen	Pen color of the edges.
Surface	Surface	Surface color of the polygons.
Transformation	Transformation	Transformation of the shape.
Size X	Number	X size of the wedge.
Size Y	Number	Y size of the wedge.
Height	Number	Height of the wedge.

Outputs

Name	Type	Description
Shape	Shape	The resulting shape.

Regular Prism



Creates a prism with a regular base polygon.

Inputs

Name	Type	Description
Visible	Boolean	Defines if the model is visible or not.
Pen	Pen	Pen color of the edges.
Surface	Surface	Surface color of the polygons.
Transformation	Transformation	Transformation of the shape.
Count	Number	Vertex count of the base polygon.
Height	Number	Height of the prism.
Radius	Number	Radius of the prism.

Outputs

Name	Type	Description
Shape	Shape	The resulting shape.

Profile Extrude



Creates an extruded shape based on a profile attribute. The base polygon of the extrude will be aligned to the origin regardless of the profile's original position.

Inputs

Name	Type	Description
Visible	Boolean	Defines if the model is visible or not.
Pen	Pen	Pen color of the edges.
Surface	Surface	Surface color of the polygons.
Transformation	Transformation	Transformation of the shape.
Profile	Profile	The profile to use.
Height	Number	Height of the extrusion.
End Surface	Surface	Top and bottom surface of the shape.

Outputs

Name	Type	Description
Shape	Shape	The resulting shape.
X Size	Number	X size of the profile's default bounding box.
Y Size	Number	Y size of the profile's default bounding box.

Inputs

 **Boolean**

 **Integer**

 **Number**

 **Length**

 **Angle**

 **Random Number**

 **Global Value**

Boolean



Defines a boolean variable. You can create a GDL object parameter from this variable in the node settings dialog.

Outputs

Name	Type	Description
Variable	Boolean	The resulting variable.

Integer



Defines an integer variable. You can create a GDL object parameter from this variable in the node settings dialog.

Outputs

Name	Type	Description
Variable	Integer	The resulting variable.

Number



Defines a number variable. You can create a GDL object parameter from this variable in the node settings dialog.

Outputs

Name	Type	Description
Variable	Number	The resulting variable.

Length



Defines a length variable. You can create a GDL object parameter from this variable in the node settings dialog. Works the same as the Number node, but the created GDL parameter will have length type.

Outputs

Name	Type	Description
Variable	Number	The resulting variable.

Angle



Defines an angle variable. You can create a GDL object parameter from this variable in the node settings dialog. Works the same as the Number node, but the created GDL parameter will have angle type.

Outputs

Name	Type	Description
Variable	Number	The resulting variable.

Random Number



Defines a random number variable.

Inputs

Name	Type	Description
Min	Number	The minimum value of the random range.
Max	Number	The maximum value of the random range.

Outputs

Name	Type	Description
Variable	Number	The resulting variable.

Global Value



Defines a variable based on a GDL global value.

See the [GDL Reference Guide](#) or more information about the meaning of these values.

Outputs

Name	Type	Description
Variable	Various	The resulting variable.

List Inputs

 **Integer Series**

 **Number Series**

 **Random Series**

 **Number Distribution**

Integer Series



Defines a series of integer variables. For example: [2, 4, 6, 8, 10].

Inputs

Name	Type	Description
Start	Integer	The first element of the series.
Step	Integer	The difference between elements in the series.
Count	Integer	The number of elements in the series.

Outputs

Name	Type	Description
Variable	Integer List	The resulting variable.

Number Series



Defines a series of number variables. For example: [2.0, 4.0, 6.0, 8.0, 10.0].

Inputs

Name	Type	Description
Start	Number	The first element of the series.
Step	Number	The difference between elements in the series.
Count	Integer	The number of elements in the series.

Outputs

Name	Type	Description
Variable	Number List	The resulting variable.

Random Series



Defines a series of random number variables.

Inputs

Name	Type	Description
Min	Number	The minimum value of the random range.
Max	Number	The maximum value of the random range.
Count	Integer	The number of elements in the series.

Outputs

Name	Type	Description
Variable	Number	The resulting variable.

Number Distribution



Defines a series of number variables. The numbers are distributed equally over the given range. For example if you generate 3 numbers between 0 and 10 the result will be [0.0, 5.0, 10.0].

Inputs

Name	Type	Description
Start	Number	The first element of the series.
End	Number	The last element of the series.
Count	Integer	The number of elements in the series.

Outputs

Name	Type	Description
Variable	Number List	The resulting variable.

Attribute Inputs

 **Pen**

 **Surface**

 **Profile**

Pen



Defines a pen variable. You can create a GDL object parameter from this variable in the node settings dialog.

Outputs

Name	Type	Description
Variable	Pen	The resulting variable.

Surface



Defines a surface variable. You can create a GDL object parameter from this variable in the node settings dialog.

Outputs

Name	Type	Description
Variable	Surface	The resulting variable.

Profile



Defines a profile variable. You can create a GDL object parameter from this variable in the node settings dialog.

Outputs

Name	Type	Description
Variable	Profile	The resulting variable.
X Size	Number	X size of the profile's default bounding box.
Y Size	Number	Y size of the profile's default bounding box.

Point Inputs

 **Point**

 **Vector**

 **Deconstruct Point**

 **Deconstruct Vector**

 **Line Points**

 **Line Points**

 **Grid Points**

 **Arc Points**

Point



Defines a point variable.

Inputs

Name	Type	Description
X	Number	The X coordinate of the point.
Y	Number	The Y coordinate of the point.
Z	Number	The Z coordinate of the point.

Outputs

Name	Type	Description
Variable	Point	The resulting variable.

Vector



Defines a vector variable.

Inputs

Name	Type	Description
X	Number	The X component of the vector.
Y	Number	The Y component of the vector.
Z	Number	The Z component of the vector.

Outputs

Name	Type	Description
Variable	Vector	The resulting variable.

Deconstruct Point



Defines three variables based on the input point's coordinates.

Inputs

Name	Type	Description
Point	Point	The point.

Outputs

Name	Type	Description
X	Number	The X coordinate of the point.
Y	Number	The Y coordinate of the point.
Z	Number	The Z coordinate of the point.

Deconstruct Vector



Defines three variables based on the input vector's components.

Inputs

Name	Type	Description
Point	Vector	The point.

Outputs

Name	Type	Description
X	Number	The X component of the vector.
Y	Number	The Y component of the vector.
Z	Number	The Z component of the vector.

Line Points



Defines a series of points along a line.

Inputs

Name	Type	Description
From X	Number	The X coordinate of the start point.
From Y	Number	The Y coordinate of the start point.
From Z	Number	The Z coordinate of the start point.
To X	Number	The X coordinate of the end point.
To Y	Number	The Y coordinate of the end point.
To Z	Number	The Z coordinate of the end point.
Count	Integer	The number of points.

Outputs

Name	Type	Description
Point X	Number List	The X coordinates of the result.
Point Y	Number List	The Y coordinates of the result.
Point Z	Number List	The Z coordinates of the result.

Grid Points



Defines a series of points along a 2D grid.

Inputs

Name	Type	Description
From X	Number	The X coordinate of the start point.
From Y	Number	The Y coordinate of the start point.
To X	Number	The X coordinate of the end point.
To Y	Number	The Y coordinate of the end point.
Count X	Integer	The number of points along the X direction.
Count Y	Integer	The number of points along the Y direction.

Outputs

Name	Type	Description
Point X	Number List	The X coordinates of the result.
Point Y	Number List	The Y coordinates of the result.

Arc Points



Defines a series of points along an arc.

Inputs

Name	Type	Description
Start Angle	Number	The start angle of the arc.
End Angle	Number	The end angle of the arc.
Radius	Number	The radius of the arc.
Count	Integer	The number of points.

Outputs

Name	Type	Description
Point X	Number List	The X coordinates of the result.
Point Y	Number List	The Y coordinates of the result.

Transformations

 **Move**

 **Move By Vector**

 **Rotation X**

 **Rotation Y**

 **Rotation Z**

 **Scale**

 **Combination**

Move



Defines a move transformation.

Inputs

Name	Type	Description
Offset X	Number	Offset along the X axis.
Offset Y	Number	Offset along the Y axis.
Offset Z	Number	Offset along the Z axis.

Outputs

Name	Type	Description
Variable	Transformation	The resulting variable.

Move By Vector



Defines a move transformation.

Inputs

Name	Type	Description
Offset	Point/Vector	The offset of move.

Outputs

Name	Type	Description
Variable	Transformation	The resulting variable.

Rotation X



Defines a rotation transformation around the X axis.

Inputs

Name	Type	Description
Angle	Number	Rotation angle in degrees.

Outputs

Name	Type	Description
Variable	Transformation	The resulting variable.

Rotation Y



Defines a rotation transformation around the Y axis.

Inputs

Name	Type	Description
Angle	Number	Rotation angle in degrees.

Outputs

Name	Type	Description
Variable	Transformation	The resulting variable.

Rotation Z



Defines a rotation transformation around the Z axis.

Inputs

Name	Type	Description
Angle	Number	Rotation angle in degrees.

Outputs

Name	Type	Description
Variable	Transformation	The resulting variable.

Scale



Defines a scale transformation.

Inputs

Name	Type	Description
Scale X	Number	Scale along the X axis.
Scale Y	Number	Scale along the Y axis.
Scale Z	Number	Scale along the Z axis.

Outputs

Name	Type	Description
Variable	Transformation	The resulting variable.

Combination



Defines a combination of some transformations. The result is a transformation with all of the given transformations applied in the same order of the inputs.

Inputs

Name	Type	Description
Transformation 1	Transformation	Transformation to combine.
Transformation 2	Transformation	Transformation to combine.
Transformation 3	Transformation	Transformation to combine.
Transformation 4	Transformation	Transformation to combine.

Outputs

Name	Type	Description
Variable	Transformation	The resulting variable.

Shape Transformations

 **Transform Shape**

 **Move Shape**

 **Move Shape By Vector**

 **Rotate Shape X**

 **Rotate Shape Y**

 **Rotate Shape Z**

 **Scale Shape**

Transform Shape



Transforms a shape with the given transformation matrix.

Inputs

Name	Type	Description
Visible	Boolean	Defines if the model is visible or not.
Shape	Shape	The shape to transform.
Transformation	Transformation	The transformation to use.

Outputs

Name	Type	Description
Shape	Shape	The resulting shape.

Move Shape



Moves a shape.

Inputs

Name	Type	Description
Visible	Boolean	Defines if the model is visible or not.
Shape	Shape	The shape to transform.
Offset X	Number	Offset along the X axis.
Offset Y	Number	Offset along the Y axis.
Offset Z	Number	Offset along the Z axis.

Outputs

Name	Type	Description
Shape	Shape	The resulting shape.

Move Shape By Vector



Moves a shape.

Inputs

Name	Type	Description
Visible	Boolean	Defines if the model is visible or not.
Shape	Shape	The shape to transform.
Offset	Point/Vector	The offset of move.

Outputs

Name	Type	Description
Shape	Shape	The resulting shape.

Rotate Shape X



Rotates a shape around the X axis.

Inputs

Name	Type	Description
Visible	Boolean	Defines if the model is visible or not.
Shape	Shape	The shape to transform.
Angle	Number	Rotation angle in degrees.

Outputs

Name	Type	Description
Shape	Shape	The resulting shape.

Rotate Shape Y



Rotates a shape around the Y axis.

Inputs

Name	Type	Description
Visible	Boolean	Defines if the model is visible or not.
Shape	Shape	The shape to transform.
Angle	Number	Rotation angle in degrees.

Outputs

Name	Type	Description
Shape	Shape	The resulting shape.

Rotate Shape Z



Rotates a shape around the Z axis.

Inputs

Name	Type	Description
Visible	Boolean	Defines if the model is visible or not.
Shape	Shape	The shape to transform.
Angle	Number	Rotation angle in degrees.

Outputs

Name	Type	Description
Shape	Shape	The resulting shape.

Scale Shape



Scales a shape.

Inputs

Name	Type	Description
Visible	Boolean	Defines if the model is visible or not.
Shape	Shape	The shape to transform.
Scale X	Number	Scale along the X axis.
Scale Y	Number	Scale along the Y axis.
Scale Z	Number	Scale along the Z axis.

Outputs

Name	Type	Description
Shape	Shape	The resulting shape.

Maths

- + Addition
- Subtraction
- × Multiplication
- / Division
- < Lower
- ≤ Lower Or Equal
- > Greater
- ≥ Greater Or Equal
- = Equal
- ≠ Not Equal
- {x} Expression
- |x| Abs
- ⌊x⌋ Floor
- ⌈x⌉ Ceil
- √x Sqrt
- (-x) Negative
- ▮ Min
- ▮ Max

Addition



Adds two values.

Inputs

Name	Type	Description
A	Number	First value.
B	Number	Second value.

Outputs

Name	Type	Description
Variable	Number	The resulting variable.

Subtraction

Subtracts two values.

Inputs

Name	Type	Description
A	Number	First value.
B	Number	Second value.

Outputs

Name	Type	Description
Variable	Number	The resulting variable.

Multiplication



Multiplies two values.

Inputs

Name	Type	Description
A	Number	First value.
B	Number	Second value.

Outputs

Name	Type	Description
Variable	Number	The resulting variable.

Division



Divides two values.

Inputs

Name	Type	Description
A	Number	First value.
B	Number	Second value.

Outputs

Name	Type	Description
Variable	Number	The resulting variable.

Lower



Returns true if the first value is lower than the second value, false otherwise.

Inputs

Name	Type	Description
A	Number	First value.
B	Number	Second value.

Outputs

Name	Type	Description
Variable	Boolean	The resulting variable.

Lower Or Equal



Returns true if the first value is lower than or equal to the second value, false otherwise.

Inputs

Name	Type	Description
A	Number	First value.
B	Number	Second value.

Outputs

Name	Type	Description
Variable	Boolean	The resulting variable.

Greater



Returns true if the first value is greater than the second value, false otherwise.

Inputs

Name	Type	Description
A	Number	First value.
B	Number	Second value.

Outputs

Name	Type	Description
Variable	Boolean	The resulting variable.

Greater Or Equal



Returns true if the first value is greater than or equal to the second value, false otherwise.

Inputs

Name	Type	Description
A	Number	First value.
B	Number	Second value.

Outputs

Name	Type	Description
Variable	Boolean	The resulting variable.

Equal



Returns true if the first value is equal to the second value, false otherwise.

Inputs

Name	Type	Description
A	Number	First value.
B	Number	Second value.

Outputs

Name	Type	Description
Variable	Boolean	The resulting variable.

Not Equal



Returns true if the first value is not equal to the second value, false otherwise.

Inputs

Name	Type	Description
A	Number	First value.
B	Number	Second value.

Outputs

Name	Type	Description
Variable	Boolean	The resulting variable.

Expression



Calculates a mathematical expression. You can use any GDL operators or functions in the expression. You have to reference the parameters with a \$ sign prefix.

Inputs

Name	Type	Description
A	Number	Refer to it in expression with \$a.
B	Number	Refer to it in expression with \$b.
C	Number	Refer to it in expression with \$c.
D	Number	Refer to it in expression with \$d.

Outputs

Name	Type	Description
Variable	Number	The resulting variable.

Abs



Returns the absolute value of the given value.

Inputs

Name	Type	Description
A	Number	The value.

Outputs

Name	Type	Description
Variable	Number	The resulting variable.

Floor



Returns the integral part of the given value.

Inputs

Name	Type	Description
A	Number	The value.

Outputs

Name	Type	Description
Variable	Integer	The resulting variable.

Ceil



Returns the smallest integral value that is not smaller than the given value.

Inputs

Name	Type	Description
A	Number	The value.

Outputs

Name	Type	Description
Variable	Integer	The resulting variable.

Sqrt



Returns the square root of the given value.

Inputs

Name	Type	Description
A	Number	The value.

Outputs

Name	Type	Description
Variable	Number	The resulting variable.

Negative

$(-x)$

Returns the opposite value of the given value.

Inputs

Name	Type	Description
A	Number	The value.

Outputs

Name	Type	Description
Variable	Number	The resulting variable.

Min



Returns the minimum value of the given values.

Inputs

Name	Type	Description
A	Number	First value.
B	Number	Second value.

Outputs

Name	Type	Description
Variable	Number	The resulting variable.

Max



Returns the maximum value of the given values.

Inputs

Name	Type	Description
A	Number	First value.
B	Number	Second value.

Outputs

Name	Type	Description
Variable	Number	The resulting variable.

Trigonometry

 **Sine**

 **Cosine**

 **Tangent**

 **Arc Sine**

 **Arc Cosine**

 **Arc Tangent**

Sine



Returns the sine value of the given value.

Inputs

Name	Type	Description
A	Number	The value in degrees.

Outputs

Name	Type	Description
Variable	Number	The resulting variable.

Cosine



Returns the cosine value of the given value.

Inputs

Name	Type	Description
A	Number	The value in degrees.

Outputs

Name	Type	Description
Variable	Number	The resulting variable.

Tangent



Returns the tangent value of the given value.

Inputs

Name	Type	Description
A	Number	The value in degrees.

Outputs

Name	Type	Description
Variable	Number	The resulting variable.

Arc Sine



Returns the arc sine value of the given value.

Inputs

Name	Type	Description
A	Number	The value.

Outputs

Name	Type	Description
Variable	Number	The resulting variable.

Arc Cosine



Returns the arc cosine value of the given value.

Inputs

Name	Type	Description
A	Number	The value.

Outputs

Name	Type	Description
Variable	Number	The resulting variable.

Arc Tangent



Returns the arc tangent value of the given value.

Inputs

Name	Type	Description
A	Number	The value.

Outputs

Name	Type	Description
Variable	Number	The resulting variable.

Solid Operations

 **Union**

 **Difference**

 **Intersection**

Union



Calculates the union of two shapes.

Inputs

Name	Type	Description
Visible	Boolean	Defines if the model is visible or not.
Pen	Pen	Pen color of the edges.
Surface	Surface	Surface color of the polygons.
Transformation	Transformation	Transformation of the shape.
Shape 1	Shape	The first shape.
Shape 2	Shape	The second shape.

Outputs

Name	Type	Description
Shape	Shape	The resulting shape.

Difference



Subtracts the second shape from the first.

Inputs

Name	Type	Description
Visible	Boolean	Defines if the model is visible or not.
Pen	Pen	Pen color of the edges.
Surface	Surface	Surface color of the polygons.
Transformation	Transformation	Transformation of the shape.
Shape 1	Shape	The first shape.
Shape 2	Shape	The second shape.

Outputs

Name	Type	Description
Shape	Shape	The resulting shape.

Intersection



Calculates the intersection of two shapes.

Inputs

Name	Type	Description
Visible	Boolean	Defines if the model is visible or not.
Pen	Pen	Pen color of the edges.
Surface	Surface	Surface color of the polygons.
Transformation	Transformation	Transformation of the shape.
Shape 1	Shape	The first shape.
Shape 2	Shape	The second shape.

Outputs

Name	Type	Description
Shape	Shape	The resulting shape.

Extras

 **GDL Object**

 **GDL Shape**

 **Viewer**

GDL Object



References an Object from the loaded library. You can select four parameters of the object (A, B, C, D) and modify these parameters from the script. You can set the referenced Object and its parameters in the node settings dialog.

This is an experimental node, use it at your own risk.

Inputs

Name	Type	Description
Visible	Boolean	Defines if the model is visible or not.
Pen	Pen	Pen color of the edges.
Surface	Surface	Surface color of the polygons.
Transformation	Transformation	Transformation of the shape.
A	Number	Value of the selected A parameter.
B	Number	Value of the selected B parameter.
C	Number	Value of the selected C parameter.
D	Number	Value of the selected D parameter.

Outputs

Name	Type	Description
Shape	Shape	The resulting shape.

GDL Shape



Runs the user defined GDL script snippet and generates a shape. You can use any GDL command here, but be careful. You have to reset the GDL stack in every snippet. You have to reference the parameters with a \$ sign prefix.

This is an experimental node, use it at your own risk.

Inputs

Name	Type	Description
Visible	Boolean	Defines if the model is visible or not.
Pen	Pen	Pen color of the edges.
Surface	Surface	Surface color of the polygons.
Transformation	Transformation	Transformation of the shape.
A	Number	Refer to it in the script with \$a.
B	Number	Refer to it in the script with \$b.
C	Number	Refer to it in the script with \$c.
D	Number	Refer to it in the script with \$d.

Outputs

Name	Type	Description
Shape	Shape	The resulting shape.

Viewer



Shows a string representation of the given value.

Inputs

Name	Type	Description
Input	Any	The value to show.

Outputs

Name	Type	Description
Output	Any	The same value as the input.

Documentation

This section describes each of the Archicad tools used in project documentation, e.g. Labels, Dimensions, texts. It describes the Layout Book workflow and how to create and update drawings on layouts, and how to publish them in various formats. The optional Revision workflow is also described here - it helps you keep track of changes and Transmittal Sets automatically. Finally, the Calculation feature is introduced for those who prefer this listing method to the Interactive Element Schedule.

Fills

Drafting

Annotation

Drawings in Model Views

Navigator - Layout Book

Working with Layouts

Master Layouts

Subsets

Drawings in the Layout Book

Revision Management

Design Options

Publishing

Element Information

ID Management

Calculation (Legacy)

Fills

Archicad includes a predefined set of default fills.

Fills are used to display elements and as components in Building Materials and Composite structures.

Use the Fill tool to draw freehand fills in the project - for example, to create patches in a Section or Elevation view.

Cover fills can be used to embellish surfaces.



Related Topics:

[Fills](#)

[Fill Categories](#)

[Draw a 2D Fill](#)

[Assign a Cut Fill \(GDL Elements\)](#)

[Assign a Cover Fill](#)

[Add Area Text to a Fill](#)

[Display Vector Fills in project views](#)

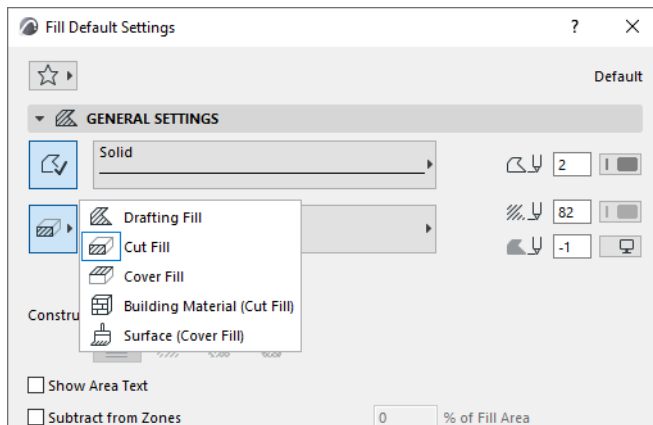
[Set Orientation of Vector or Symbol Fill Patterns](#)

[Using Gradient Fills](#)

Draw a 2D Fill

Use the Fill tool to draw freehand fills in the project - for example, to create patches in a Section or Elevation view.

1. In Fill tool settings, choose a Fill Category.

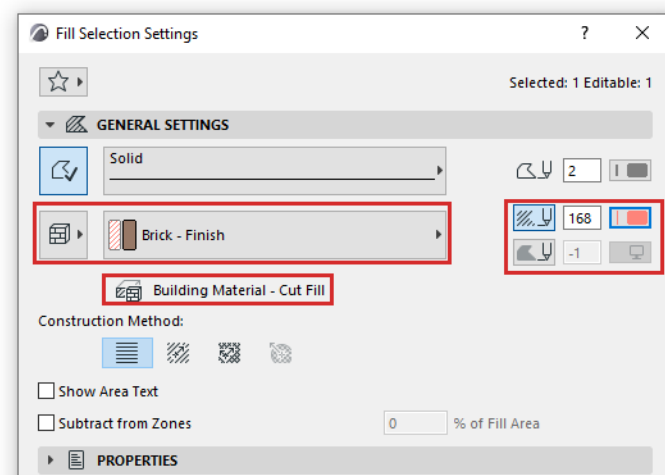
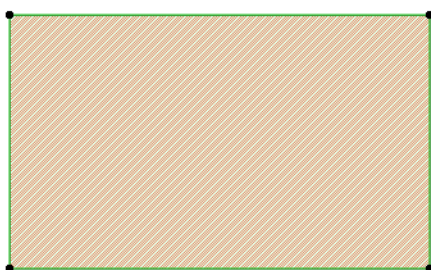


- The first three (Drafting/Cut/Cover) are the same categories defined in the Fill dialog box. [See Fill Categories](#)
 - Use the category that corresponds to the Fill function: if you are patching a section element, then choose Cut Fill; if you are working on an uncut element in elevation, choose Cover Fill.
 - The two other categories (Building Material and Surface) make it easier to match cut fills based on Building Materials, and cover fills based on Surfaces. See descriptions below.
2. The Fill pop-up now contains Fills of that Category. Choose a Fill type (or Surface or Building Material).
 3. Draw a freehand fill.

Building Material (Cut Fill)

Uses the cut fill and colors of the chosen Building Material.

If needed, you can override these background/foreground colors for the Fill element.



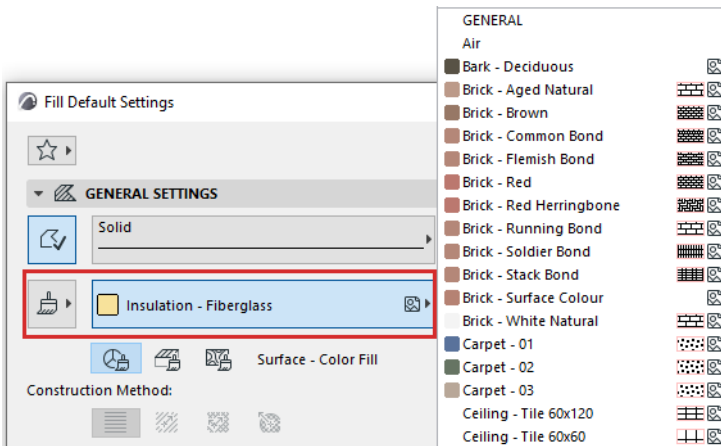
Notes:

- A Fill element in this category is treated as a “Cut Fill” in Graphic Override Rules.

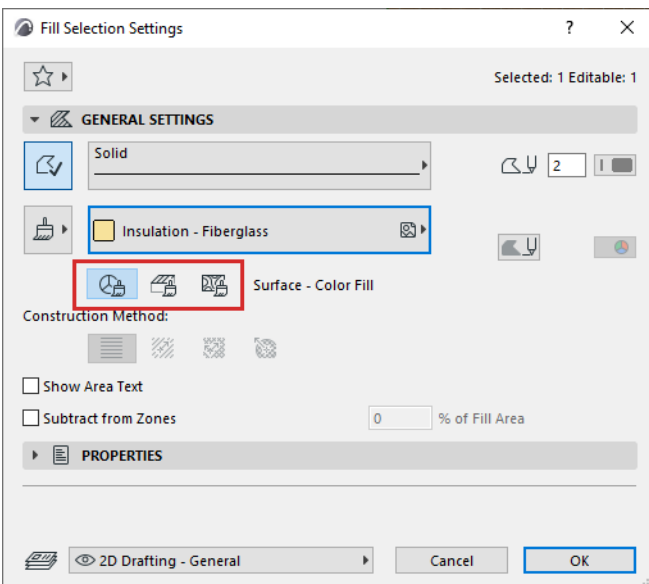
- In criteria (e.g. Find and Select or Schedules), "Building Material - Cut Fill" is a separate, specific criterion.

Surface (Cover Fill)

1. Choose a Surface from the pop-up.



2. Choose to use either the Color, Cover, or Texture of the chosen Surface.



Notes:

- A Fill element in the "Surface" category is treated as a "Cover Fill" in Graphic Override Rules.
- In criteria (e.g. Find and Select or Schedules), the Fill's Surface Color/Cover/Texture are each separate criteria.

Surface - Color Fill

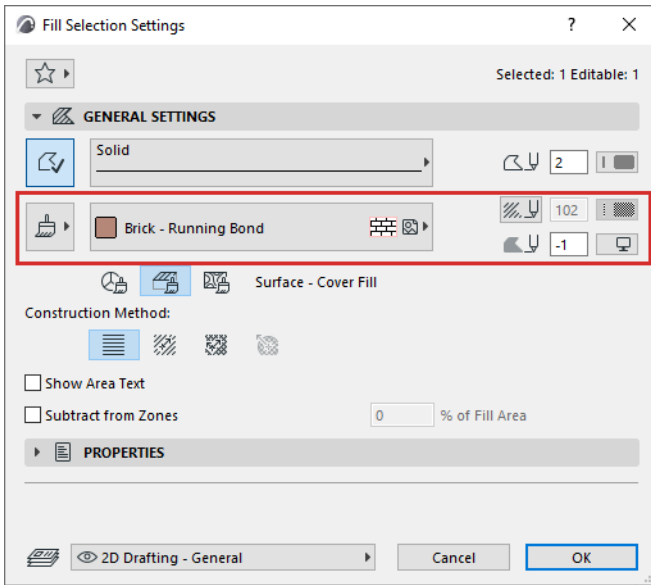
Uses the Surface color's Background Pen.

If needed, you can override this color for the Fill element.

Surface - Cover Fill

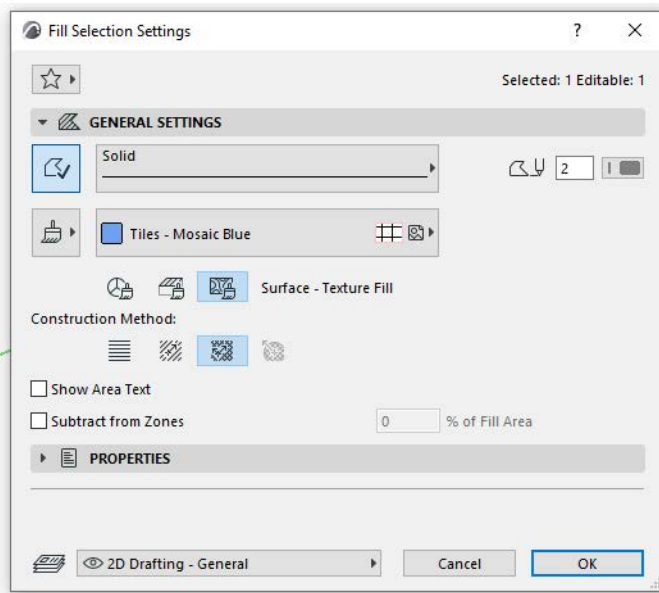
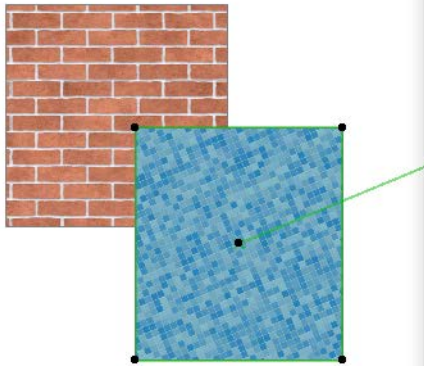
Uses the Surface Cover Fill's Foreground Pen.

Override the Foreground and Background pens of the Fill element if needed.



Surface - Texture Fill

This Fill is based on the Surface Texture. It does not use pens.



Related Topic

[Fill Categories](#)

[Fill Tool Settings](#)

Assign a Cut Fill (GDL Elements)

[See *Building Materials*.](#)

For GDL-based elements (Objects, Curtain Walls), or for a freehand Fill drawn using the Fill Tool, Cut Fills are assigned in their Settings dialog boxes.

Note: Construction elements use the cut fill defined for the assigned Building Material.

To define a default cut fill for a GDL-based element, such as an Object:

1. Activate the Object tool; or select the placed Object.
2. Go to the Floor Plan and Section panel of its Settings (or click Floor Plan and Section button in the Info Box.)
3. Go to the Cut Surfaces controls.
4. Enable the Override Object's Attributes checkbox
5. For the listed Cut Fill parameter, click the fill name or icon to access the list of available fills, and choose one.

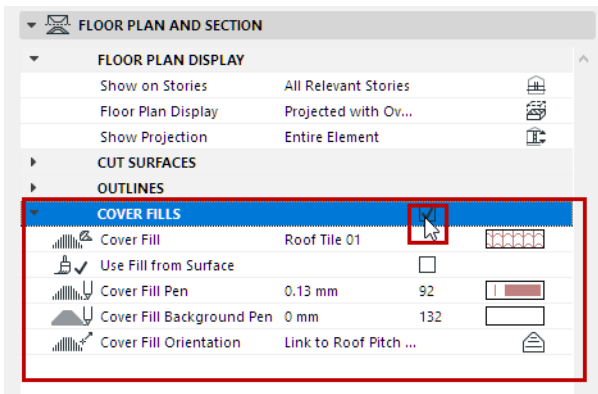
Note: Only Fills defined as Cut Fills in **Options > Element Attributes > Fills** are available here.

Assign a Cover Fill

In Archicad, cover fills can be applied to Beams, Columns, Slabs, Meshes, Roofs, Shells, Openings, Morphs and Zones.

For all of these element types except Zones:

1. Go to the Floor Plan and Section panel of its Settings (or click Floor Plan and Section button in the Info Box.)
2. Check the Cover Fills box.



3. Click the Cover Fill parameter pop-up to access the list of available fills, and choose the desired fill.

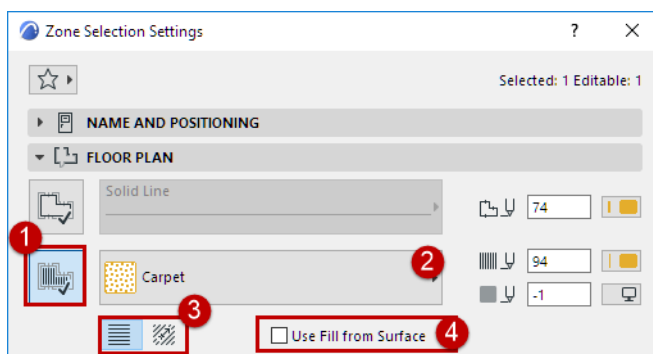
Note: Only Fills defined as Cover Fills in **Options > Element Attributes > Fills** are available here.

 - **Use Fill from Surface:** Check this to use the surface of the element's Building Material instead.
4. Using the cover fill controls that follow:
 - Choose Pen and Background attributes for the Cover Fill.
 - Use the Cover Fill Orientation parameter to define the orientation of the fill pattern.

See [Set Orientation of Vector or Symbol Fill Patterns](#).

For Zones:

1. Activate the “Cover Fill” button in Zone Settings (Floor Plan panel) by clicking it.



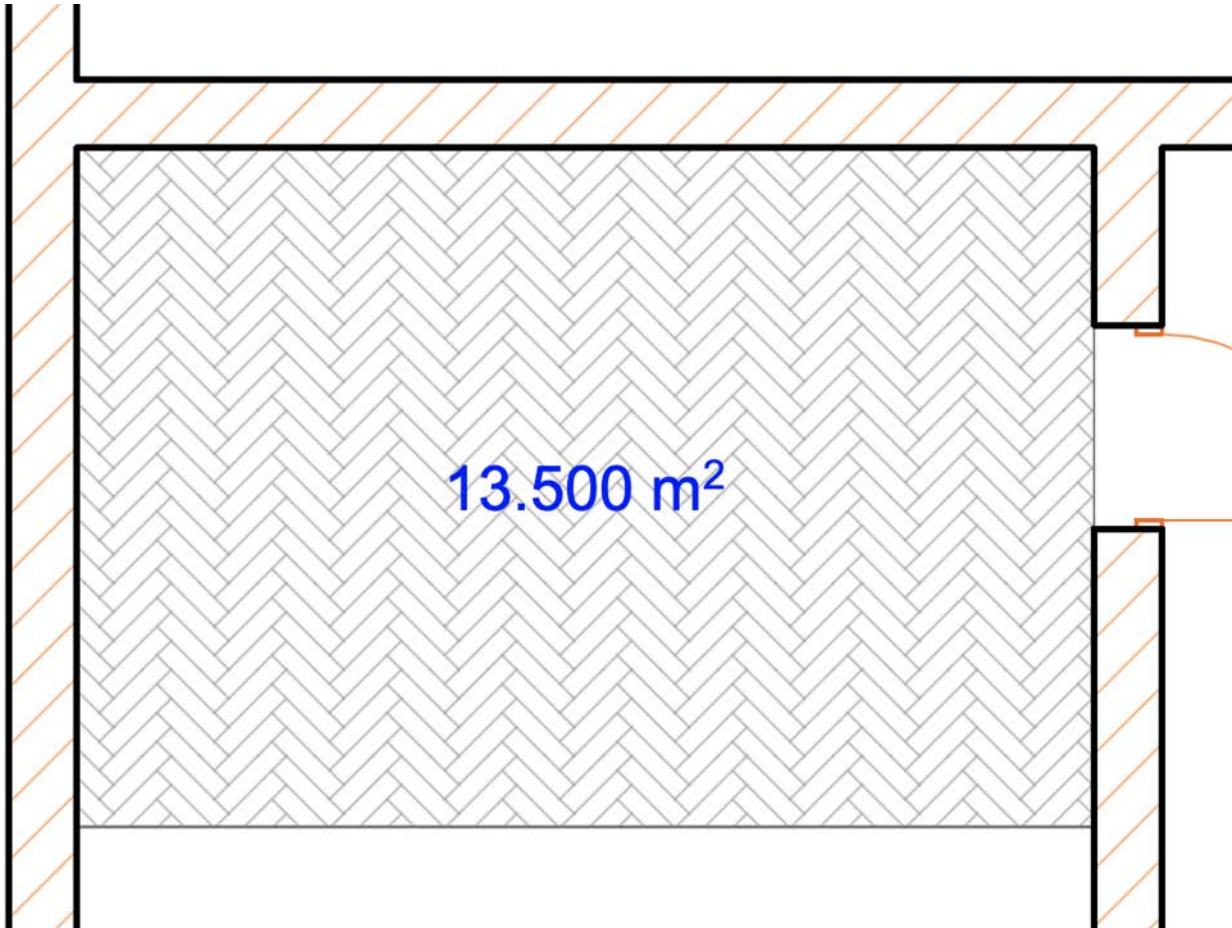
2. Click the fill name or icon to access the list of available fills, and choose the desired fill from the pop-up list.
3. Choose a radio button to define the orientation of the fill pattern.

4. If the Zone's surface has a Cover fill, you have the option to display that instead: check **Use Fill from Surface**.

See [Set Orientation of Vector or Symbol Fill Patterns](#).

Add Area Text to a Fill

Enable the **Show Area Text** checkbox in the General Settings panel of Fill Settings. When you create the Fill, the **Hammer** cursor will appear, prompting you to position the area text with a click.



The text block is created by calculating the area of the Fill. The text settings will be the same as the default values set in Dimension Text Settings. This text will always be displayed horizontally on screen, even if the fill is rotated or mirrored, or the view itself is rotated.

[See *Set Orientation*](#).

Note: If the fill contains holes, they are subtracted from the fill area.

To modify the font settings or even the content of the area text, select the text only (not the Fill itself) and edit the Dimension Text Settings (in this case, the last command of the Edit menu changes to Dimension Text Settings). The style of the Fill Text, if displayed, is stored as part of the Fill's Favorite settings.

[See also *Dimension Text Settings*](#).

Note: Favorites of the Text tool can be applied to the text-type items in Dimensions and Labels, and to Fill Text.

The measurement unit and accuracy of the area value are defined in Dimensions (**Options > Project Preferences > Dimensions**).

Fill Area text will always be displayed horizontally on screen, regardless of the view's orientation.

Display Vector Fills in project views

Vector Fills use a hatching pattern, and can be used to represent the cut surfaces or top covers (cover fills) of construction elements. You can also assign Vector Fills to Surfaces.

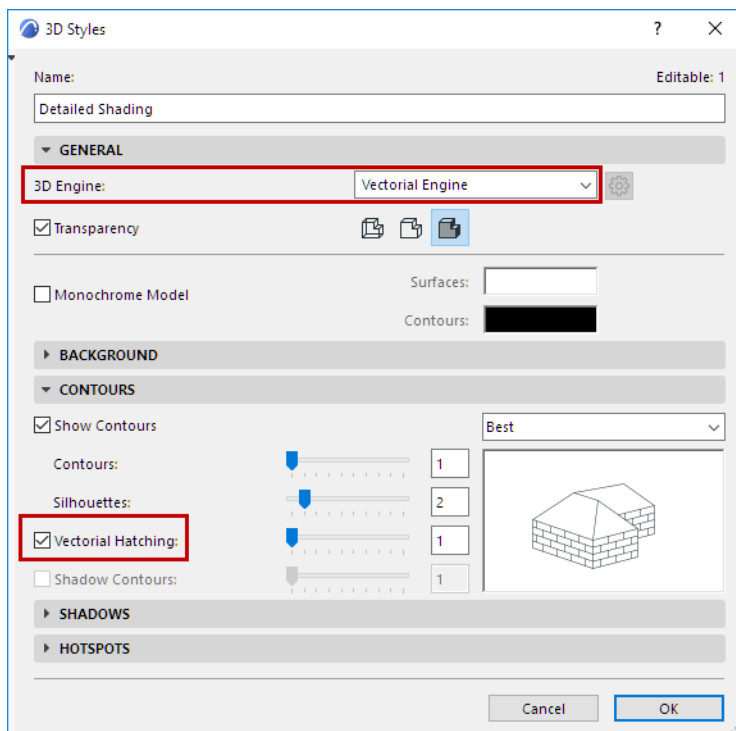
On the Floor Plan

Elements will display the hatching of a vector fill on the Floor Plan only if you have activated **View > On-Screen View Options > True Fill Appearance**. (Otherwise, the on-screen display will show the fill's bitmap face.)

In the 3D Window

Construction elements in the 3D Window are displayed using the Surfaces assigned to them via their Building Material. These Surfaces can be overridden at the element level, using the Model panel of the element's Settings.

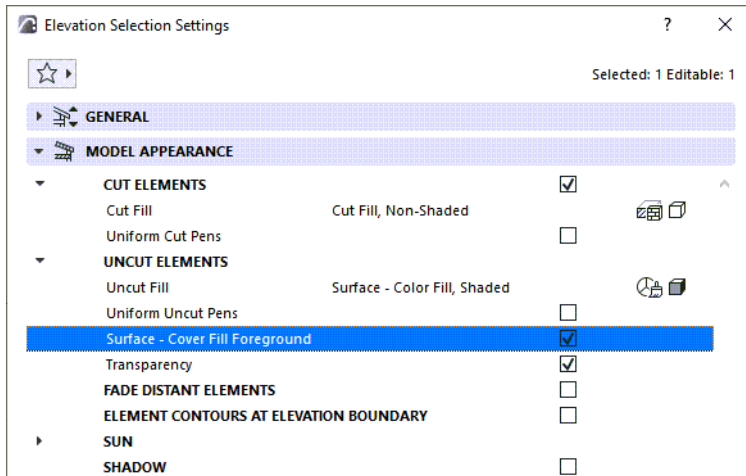
If these Surfaces include a Vectorial Hatching, it will be visible in the 3D Window only if you switch Vectorial 3D Hatching "On" in 3D Styles, and only if you are using the vectorial 3D Engine.



See also [Surface Cover Fill \(Basic Engine\)](#).

In the Section/Elevation/IE and 3D Document Windows

Section/Elevation/IE and 3D Document windows will display vectorial hatching only if you have checked the **Surface - Cover fill foreground** box in the Model Appearance panel of the Settings dialog box.



For more information, see [Model Appearance \(Section-Type Views\)](#) and [3D Document Model Appearance Panel](#).

Vectorial hatching settings have no effect in rendered views.

Related Topics:

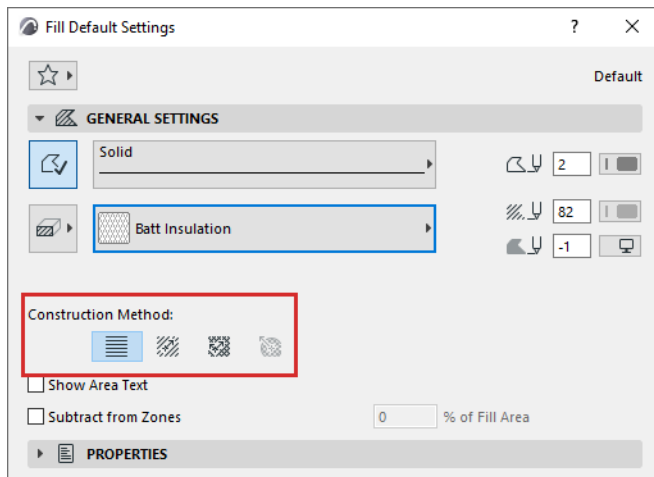
[Fill Display Mode: True Fill Appearance vs. Bitmap](#)
[Set Orientation of Vector or Symbol Fill Patterns](#)

Set Orientation of Vector or Symbol Fill Patterns

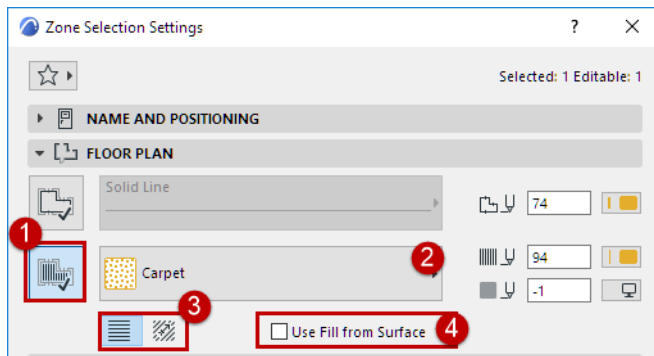
Use the Orientation Method controls to manually define or change the orientation of a vector or symbol fill pattern.

Where to Choose a Fill Orientation Method for an Element

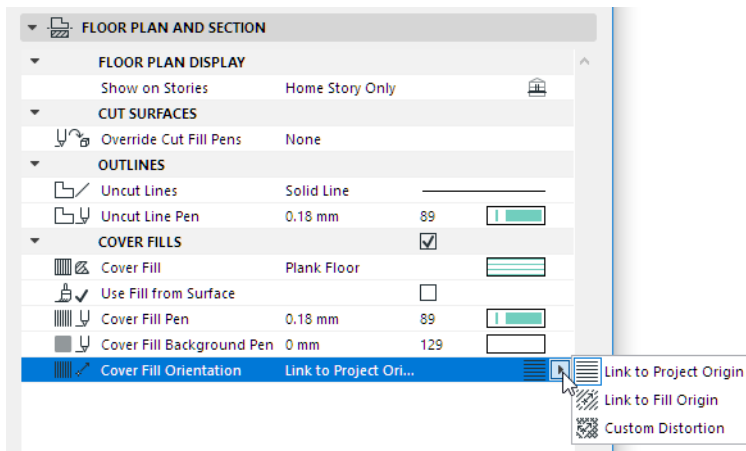
- For **Fill** elements that use a vector or symbolic cover fill, use the “Construction Method” options in the General Settings panel of Fill Settings.



- For **Zones** that use a vector or symbolic cover fill, use the radio buttons in the Floor Plan panel of Zone Settings.



- For Cover Fills assigned to a **Beam, Column, Mesh, Slab, Roof, Opening, Morph or Shell**, use the controls under Cover Fill in the Settings dialog box.



See also [Assign a Cover Fill](#).

- For cut fills used in components of **complex or composite elements**, Fill Orientation settings derive from their Building Material.

Fill Orientation Methods

[Link to Project Origin](#)

[Link to Fill Origin](#)

[Place Vector Fill with Linked Pattern](#)

[3D Distortion](#)

[Custom Distortion](#)

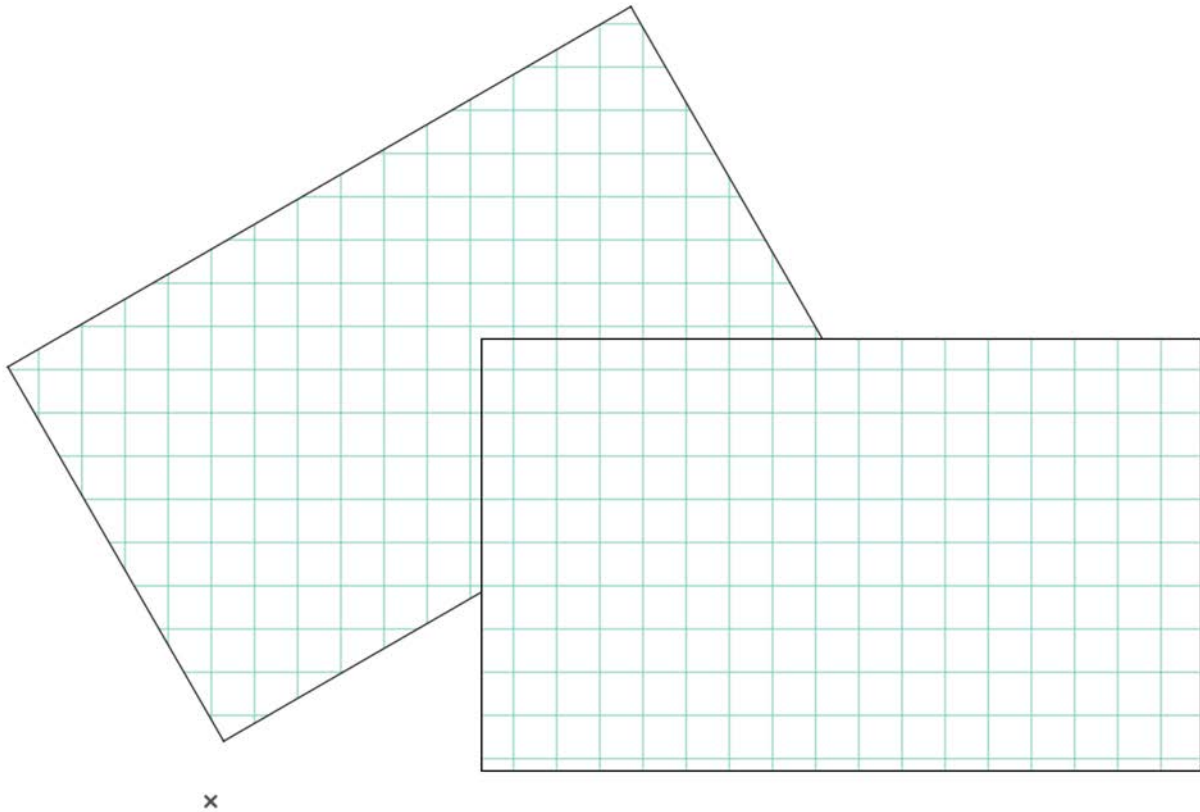
[Distorted Fill for Roofs](#)

[Symbol Fills as Skins of Composite Elements: Fit to Skin](#)

[Radial Distortion \(Symbol Fills only\)](#)

Link to Project Origin

With this option, hatching orientation is always orthogonal from the Project Origin, independently of the transformations made on the element.

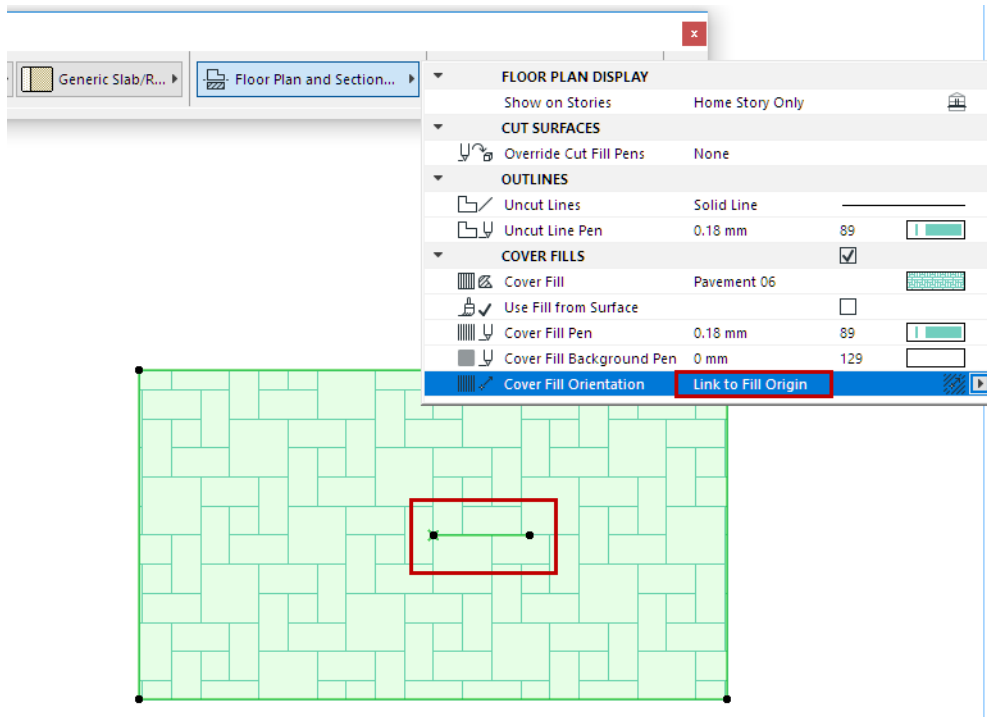


Link to Fill Origin

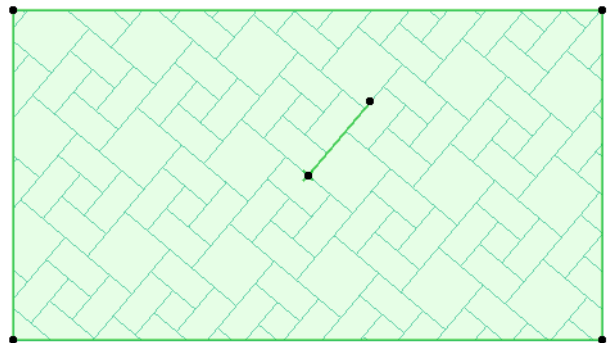
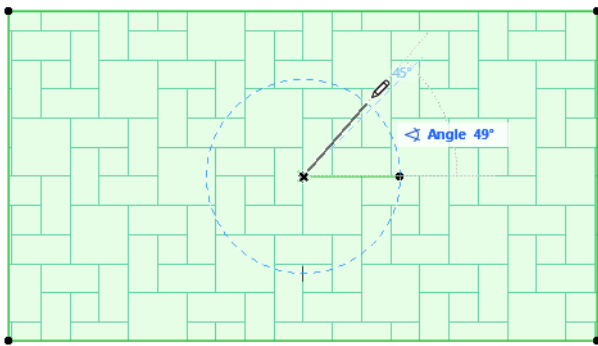
Use the **Link to Fill Origin** to fine-tune the angle of the fill pattern.

A single handle appears on the placed fill to show the orientation of the fill.

Note: Fill handles are visible only if you enable Fill Handles (Drafting and Editing Aids) in **View > On-Screen View Options**.

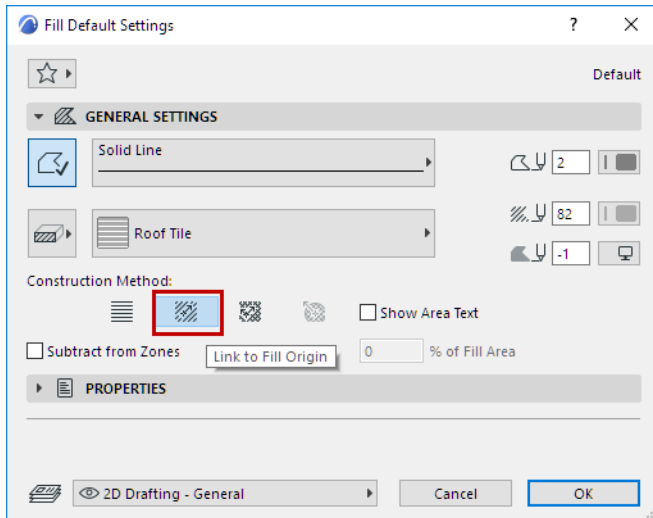


Now drag the Fill Handle to realign the fill pattern as needed. Make sure the Move Sub-Element icon is active on the pet palette.

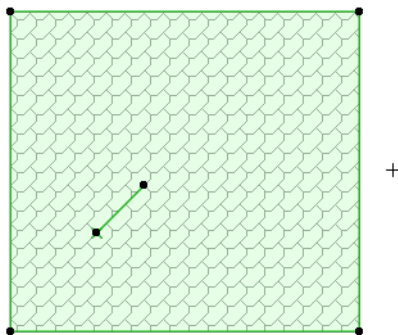


Place Vector Fill with Linked Pattern

1. In Fill Settings, set the **Link to Fill Origin** option.



2. Place the fill polygon.
3. Select the fill.
4. Drag either end of the fill handle to define the origin and direction of the fill pattern.

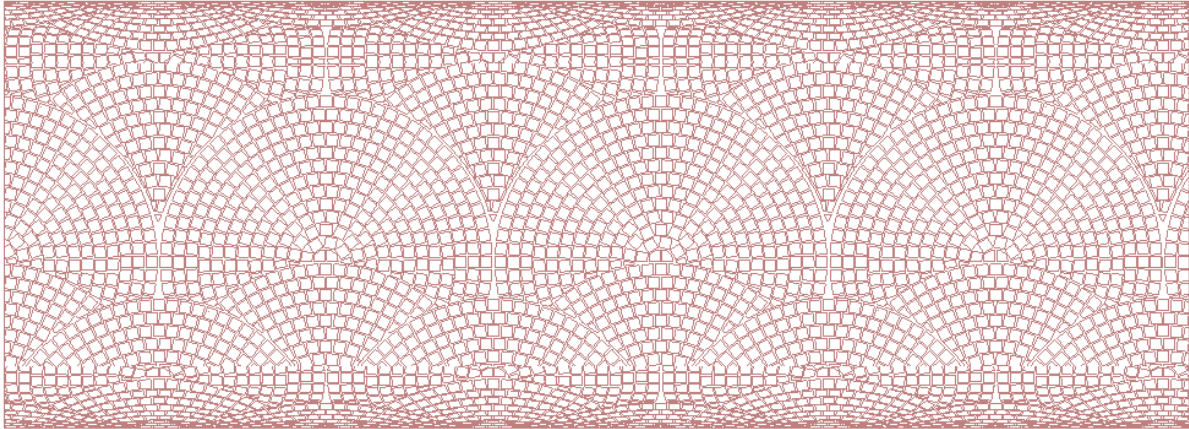


Note that the handle does not necessarily have to be located inside the Fill. You can drag the handle out of the body of the fill after placing it.

3D Distortion

Available for Beam, Column, Shell and Morph.

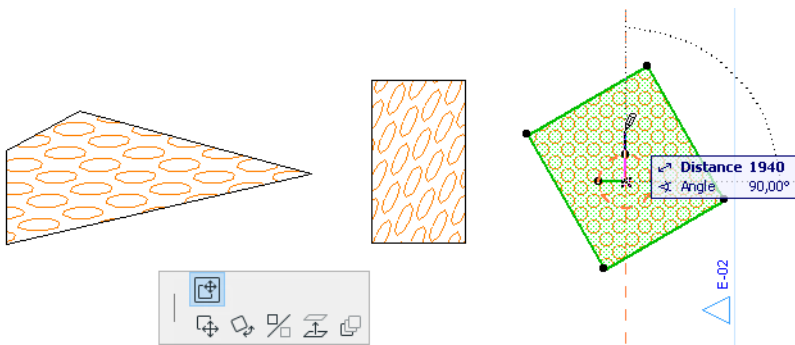
This option distorts the cover fill, simulating the 3D view of a curved surface on the Floor Plan.



Custom Distortion

This option displays a double fill handle at the fill origin.

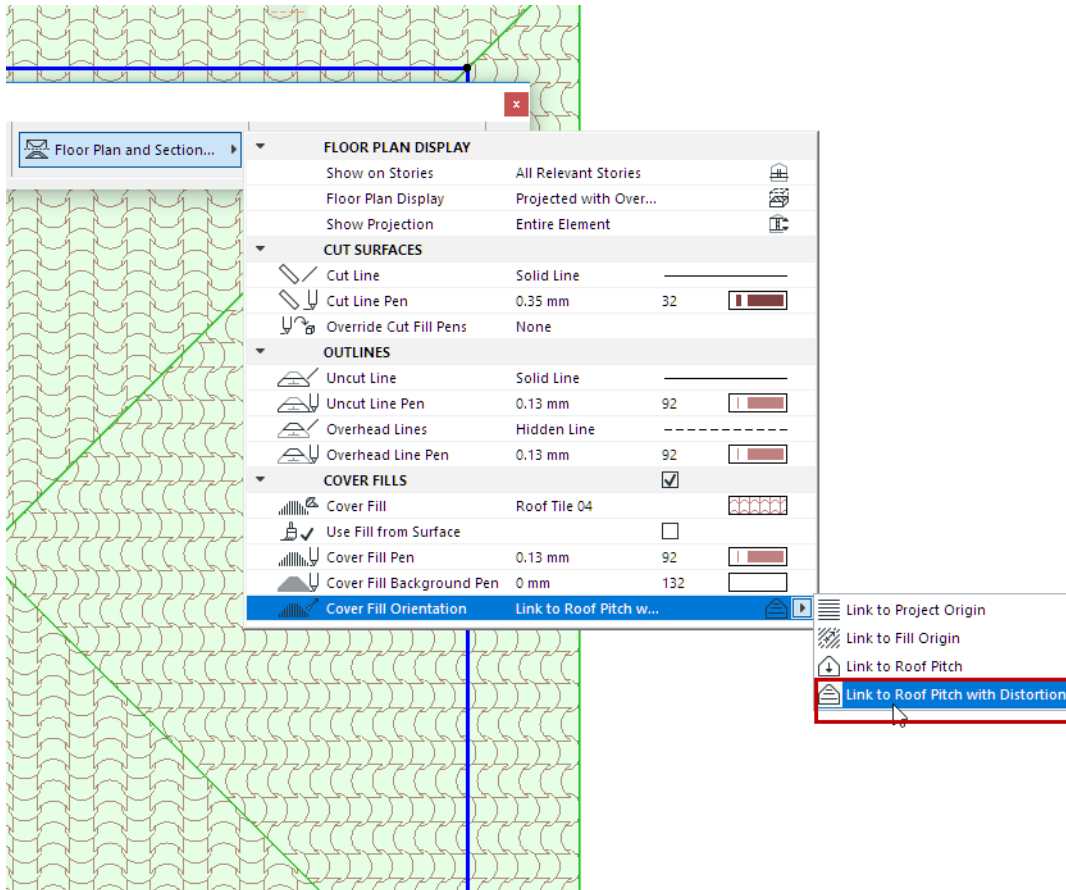
Manipulate the hatching pattern by modifying the length of either handle and the angle closed by the two handles.



Distorted Fill for Roofs

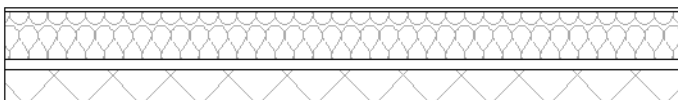
Set the Roof's cover fill Orientation to **Link to Roof Pitch with Distortion**.

The cover fill is automatically distorted to follow the Roof pitch.



Symbol Fills as Skins of Composite Elements: Fit to Skin

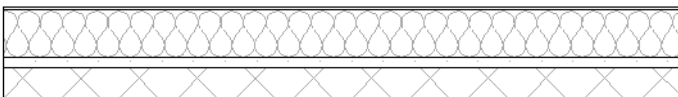
If a composite element includes a symbol fill as part of the Building Material, you can adjust that fill to “fit to skin”, so that the fill pattern is adjusted to the shape of the skin. This is especially useful when modeling soft insulation, plywood, metal decking or any directional fill.



To do this, use the controls in the Building Materials dialog box (**Options > Element Attributes > Building Materials**.)

1. Go to the Fill Orientation control in the dialog box.
2. Choose Fit to Skin.

The symbol fill pattern is now adjusted to fit the contour of the element.



Radial Distortion (Symbol Fills only)

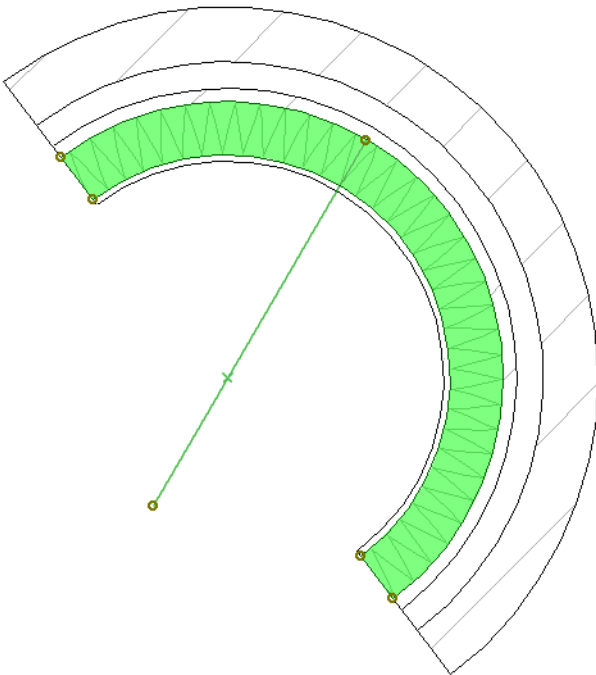
The **Radial Distortion** Fill construction method (Fill Tool) lets you graphically adjust the vector's radial distortion. This is available only for a selected symbol fill and only if:

- the symbol fill was used as a skin of a composite or complex structure; and
- the skin's Fill Orientation was defined as **Fit to Skin** in its Building Material; and
- you have selected this symbol fill after exploding a curved composite/complex wall.

[See Explode into Current View.](#)

Note: The use of many radial distorted fills in your project may decrease project performance.

Here is a curved composite wall with a symbol fill (orientation: Fit to Skin) as one of the skins, after the composite wall has been exploded. Each fill can now be selected separately. The symbol fill displays a fill handle and the Distorted Radial Fill construction method is available. Adjust the fill handle to adjust the fill's distortion or the pattern's origin as needed.



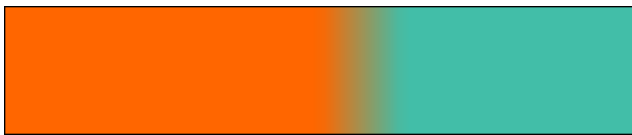
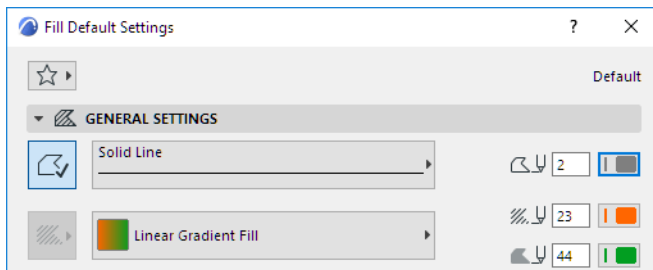
Using Gradient Fills

Gradient Fills are Drafting fills only, and thus are available only from the Fill Tool's pop-up. (Drafting fills are drawn by hand, using the Fill Tool.)

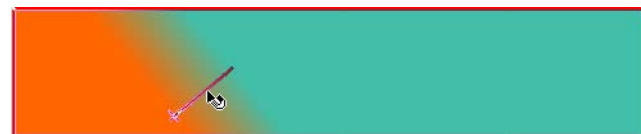
The purpose of Gradient Fills is to enhance architectural graphics; they do not indicate any physical properties, and so they are available for 2D fills created with the Fill tool (Drafting fills) only. Gradient Fills cannot be applied to construction elements (i.e. they are not available as Cut Fills or Cover Fills).

To get a linear gradient fill, choose two different pen colors (foreground and background) on the General Settings panel of Fill Settings.

When placing the Fill, you will see that the pattern is a mix of the two colors with a transition area between them.

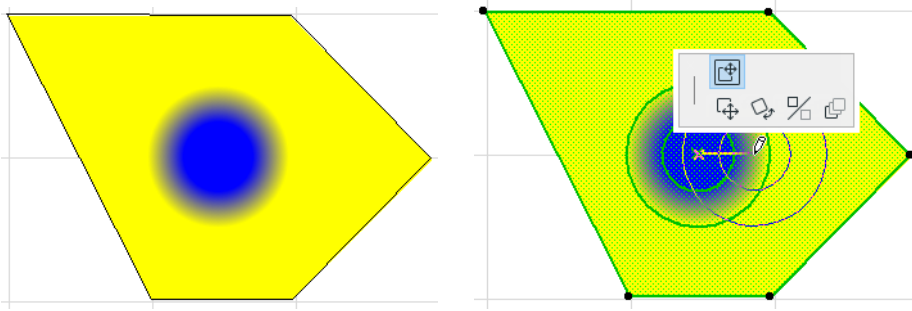


The location, angle and the size of the transition area can be modified with the fill handle provided that you enable Fill Handles (Drafting and Editing Aids) in **View > On-Screen View Options**.



The other type of the gradient fill is radial, that is, of basically circular shape. In this case, the two pen colors (foreground and background) define respectively the inner (next to the core) and outer colors of the gradient fill.

With the fill handle, you can modify the origin of the gradient fill and the extent of both colors as well as that of the transition area.



Note: When moving the fill handle, make sure you select the **Move Sub-element** icon on the pet palette.

Drafting

Drafting Elements are strictly two-dimensional: they include Lines, Hotspots, Figures and Drawings. With the exception of Drawings, they do not appear in list views.

Drafting elements have various uses:

- Details that you did not wish to model fully with construction tools.
- Positioning and drawing aids for the placement of construction elements.
- Their outlines can be used to generate complex 3D shapes using the Magic Wand tool.
- Decoration purposes or elaborating on details, especially in the Section/Elevation/IE and 3D Document windows or on Details and Worksheets.

The Floor Plan and Section/Elevation/Interior Elevation and 3D Document display of drafting elements depends on the attribute choices made for them in their respective Tool Settings dialog boxes. Attribute sets for Drafting Elements (pencolors, lines, fills) are managed from the dialog boxes in **Options > Element Attributes**.

For faster display or output, the 2D appearance of some of the attributes can be modified using the commands at **View > On-Screen View Options**.

Topics in this section:

[Lines, Circles, Polylines, Splines](#)

[Hotspots](#)

[Figures](#)

[Drawings in Model Views](#)

See also:

[Fills](#)

Lines, Circles, Polylines, Splines

The following straight and curved line elements can be created in Archicad using the different Line tools and geometry methods.

- Single straight line segments, series of Chained straight and Curved line segments, Rectangles and Rotated Rectangles.
- Circular Arcs and full Circles
- Elliptical Arcs and full Ellipses
- Straight or Curved Polylines
- Natural Splines, Bézier curves and Freehand curves

Double-clicking any of the **Line** type tools will open a dialog box in which you can adjust the settings of the given tool.

Available Lines are an attribute, defined for the project globally.

[See *Lines*](#).

In each of these dialog boxes, the **Uniform Settings for Line Tools** checkbox allows you to apply the settings made in this dialog box to all line-type tools: Line, Arc/Circle, Spline, Polyline.

Favorites of any of the four Line-type tools (Line, Arc, Polyline, Spline) can be applied to any of the other Line-type elements.

[See *Line-type Tool Settings*](#).

Topics in this section:

[Line Categories](#)

[Drawing a Single Straight Line Segment](#)

[Stretching or Shrinking Lines](#)

[Drawing Arcs and Full Circles](#)

[Stretching Curved Elements](#)

[Editing an Arc Using its Tangent](#)

[Drawing Elliptical Arcs and Full Ellipses](#)

[Convert Ellipse to Circle](#)

[Drawing Splines](#)

[Editing Splines](#)

[Drawing Freehand Curves](#)

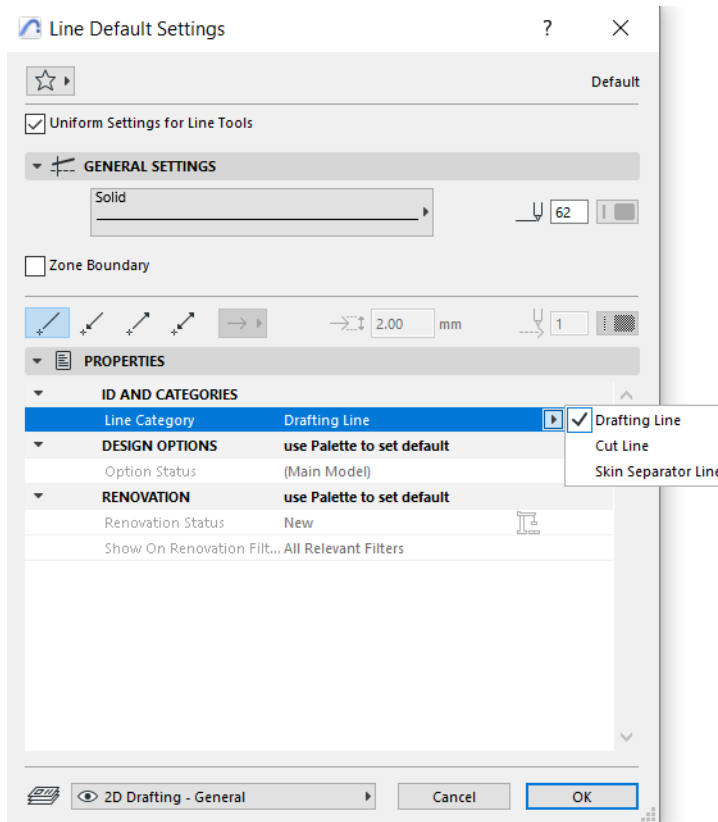
[Drawing Polylines and Chained Lines](#)

[Editing Polylines](#)

Line Categories

Use the Properties Panel of Line Settings to assign line categories.

Use line categories to fine-tune Graphic Overrides and export settings for lines depending on their category.



Archicad has three line categories: Drafting, Cut, and Skin Separator.

By default:

- All contour lines are “Cut lines”
- All skin separator lines (of composite structures) are “Skin Separators”
- All other lines are “Drafting”

Line categories can be useful

- If you have exploded a composite wall into its constituent lines and fills
- If you draw lines by hand into a 2D window or Section-type window

Assign categories to these lines according to their function in the plan. This way, even hand-drawn lines can be considered Cut lines or Skin Separator lines.

Assigning line categories can also be useful if you save your file in DXF/DWG format: the DXF/DWG Translation Setup dialog box allows you to define a separate Custom Layer onto which Skin Separator lines can be exported.

[See Custom Layers.](#)

Drawing a Single Straight Line Segment

To draw a single straight Line segment, choose the **Line** tool in the Toolbox and the **Single Line** geometry method in the Info Box.



The Line segment is defined by clicking at its two endpoints.

Note: This CAD-like drawing method is the default method for drawing segments in Archicad, but you can change it in **Options > Work Environment > Input Constraints & Guides**.

Stretching or Shrinking Lines

To stretch/shrink a straight line with the menu command:

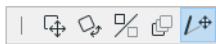
1. Select a line.
2. Choose the **Edit > Reshape > Stretch** command.
3. Click an endpoint of the selected line and move it.
4. Click again to either define the new endpoint as an extension (or reduction) of the previous length. The line is stretched or shrunk and, if needed, rotated according to the new endpoint. The other endpoint will remain at its original position.

Note: Multiple Lines with overlapping endpoints can all be stretched at the same time using the Stretch menu command or with the Marquee tool.

For more information, see [Stretching with the Marquee Tool](#).

When using the pet palette:

1. Select a line.
2. Click one of the line's endpoints. The pet palette appears. Choose the **stretch** icon:



3. Click to define the new endpoint.

Drawing Arcs and Full Circles

To draw an arc or a full circle, choose the **Arc/Circle** tool in the Toolbox and one of the Geometry Methods offered by the first icon in the **Info Box** (Centerpoint, Three Points or Tangent Point).

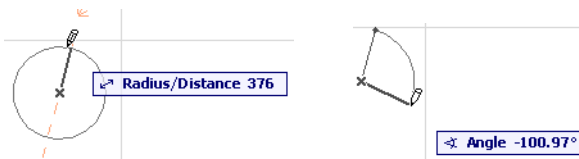


Note: These methods are identical for drawing curved walls.

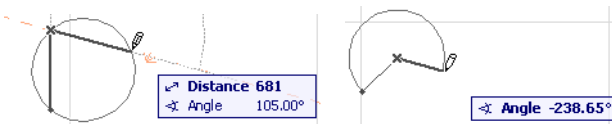
These methods use different basic points to define the arc.

- Centerpoint and radius
 - Click to define centerpoint.
 - Move the cursor to define the radius, then click a second time.
 - Move the cursor to draw the length of the arc, then click to complete.

Note: To draw a full circle, double-click when defining its radius.



- Circumference
 - Click on three points of the arc's circumference (e.g. other elements or intersections, or special snap points).
 - Click a fourth time to define the length of the arc.



- **Tangential:** Define a full circle based on three **tangential edges** or points.
 - Select three initial points: these can be a tangential edge, a node, or a free-floating point.

The next step depends on the geometric situation.

- If there is only one solution, the circle is automatically drawn.
- If there are multiple solutions, the Eyeball cursor appears and the ghost contour of the arc flips from one position to the other as you move the cursor around. Click when it is at the right place to complete the circle.
- If there is no solution, (for example, if you define three parallel edges for tangency), no circle will be made.

Note: Unlike circular Walls, Circles can be fully closed. You will obtain a single element, not two half-circles.

Stretching Curved Elements

You can modify the circumference or radius of Curved Walls, Beams, Arcs and Circles with either the **Stretch** command or the pet palette icon.

Angular Stretch

- Use the **Angular Stretch** icon to change the element's arc length by dragging its endpoint (or reference line endpoint, in case of a curved wall) along the curve.



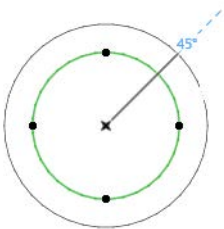
- Alternatively, transform an Arc into a full Circle or the other way around.



Note: A curved Wall stretched into a full circle will be transformed into two half-circular elements.

Radial Stretch/Copy

Use the **Radial Stretch** icon to increase or decrease the element's radius by dragging its edge or reference line.

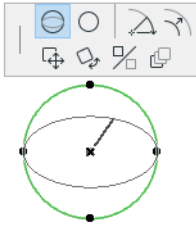


- Make a single copy of circles or ovals: click Option (Mac) or Control (Windows) during radial stretch

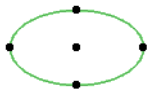
Stretch Ellipse

You can modify the shape of a closed ellipse or full circle by stretching it.

- Select the ellipse or circle.



- Click on a node to bring up the pet palette.
- Choose the **Stretch Ellipse** icon.
- Drag the cursor to stretch the ellipse along its axis or stretch the circle into an ellipse.

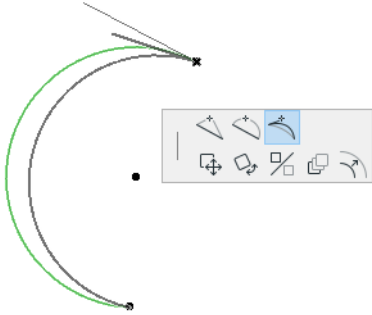


- Click to complete.

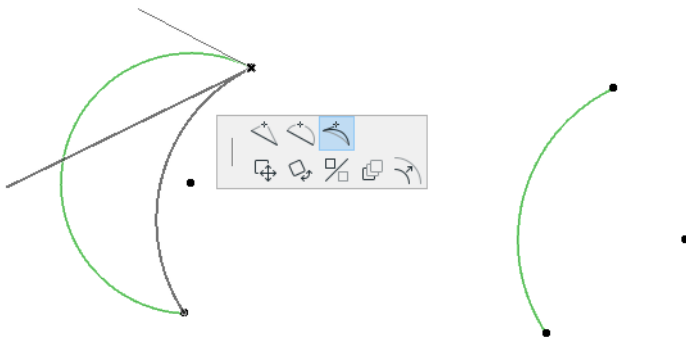
Editing an Arc Using its Tangent

Edit an arc from any point along its edge using the tangent to the nearest arc endpoint.

- Click on the edge of any arc and choose the **Edit segment using tangent** icon from the pet palette.

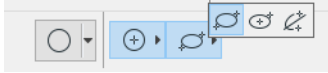


- The tangent to the nearest arc endpoint appears on the plan. Move the cursor to edit the tangent line by choosing a new tangent endpoint. A rubber band line indicates the shape of the resulting new arc. Click to place.

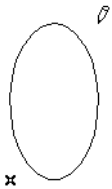


Drawing Elliptical Arcs and Full Ellipses

To draw an elliptical arc or a full ellipse, choose the **Arc/Circle** tool in the Toolbox and one of the Geometry Methods offered by the second icon in the Info Box (Diagonal Ellipse, Semi-Diagonal Ellipse, Ellipse Radii).



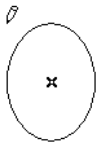
The **Diagonal Ellipse** method creates an Ellipse constrained into a rectangle.



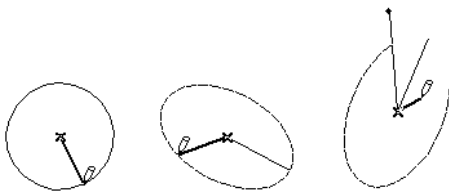
This method stretches the ellipse of an invisible rectangle held by the two points of its diagonal.

1. Click the starting point of the imaginary rectangle.
2. By stretching the imaginary diagonal, you get ellipses of different sizes drawn into the invisible rectangle.
3. With the second click you choose the ellipse of the appropriate size.

The **Semi-Diagonal Ellipse** method works in the same way as the Diagonal method but here you define the imaginary rectangle by its centerpoint and the endpoint of its semi-diagonal.



The **Ellipse Radii** method defines an elliptical arc by the major and another radius of an ellipse and the angle of the arc.

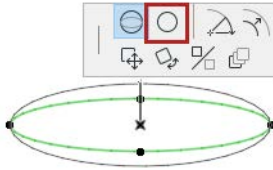


1. Click the centerpoint of the ellipse.
2. Choose the orientation of the major radius, by clicking again.
3. You get the rubber band line of the second radius, which you can stretch to different sizes and angles. Notice the accompanying ellipse that fits the length of the two radii. If you stretch the minor radius too far away or right in line with the major radius, you cannot draw the ellipse.
4. In the final phase, Archicad holds the ellipse while you define the side angle of the arc. First, you use a rubber band line to define the side where the angle starts from. Then you define a different side that closes the angle, while Archicad follows the angle with the Elliptical Arc. Neither side of the angle remains visible when the final arc is drawn.

Convert Ellipse to Circle

To convert an ellipse to a circle:

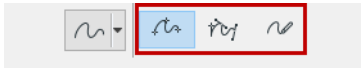
- Select the ellipse.
- Choose one of its nodes to bring up the pet palette.
- Choose the **Convert Ellipse to Circle** icon.



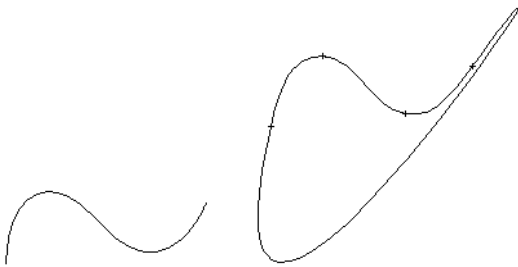
The ellipse will be transformed into a circle, whose radius is equal to that of the ellipse at the clicked node.

Drawing Splines

To draw a natural Spline or a Bézier curve, choose the **Spline** tool in the Toolbox and the first (natural spline) or the second (Bézier curve) Geometry Method icon in the Info Box.



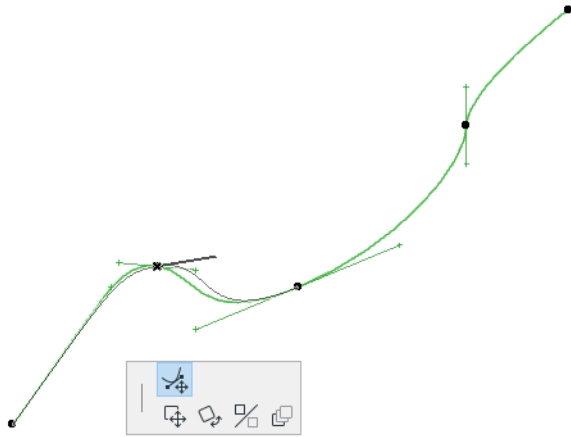
Natural splines can be defined by placing nodes, which the program automatically connects, thereby generating a smooth custom curve. The angle of the tangent and the shape of the spline generated with it is affected by each subsequent node defined. You can conclude this operation by double-clicking on the last node or by clicking the **OK** button in the **Control Box**. Clicking **OK** always results in a closed spline.



Bézier splines are somewhat more complex in nature, but they allow more accurate reproduction of specific custom shapes. Bézier splines are defined by nodes, just like natural splines, but they also have editable tangent handles on either side of these nodes, plus one tangent handle at each end (if it is an open spline). The shape of the Bézier spline is affected by the direction of the tangent and the length of each tangent handle.

When defining Bézier splines, click once to place a node, and keep the mouse button depressed. By moving away from the node in any direction, you are in effect defining the initial tangent, and the length of the handle that defines the curvature. If you release the mouse button, Archicad will assume that you wish to jump to the definition of the next node with your next click. Keep the mouse button depressed, and then define the next tangent and node as above. You can conclude this operation by double-clicking the last node or by clicking the **OK** button in the **Control Box**. Clicking **OK** always results in a closed spline.

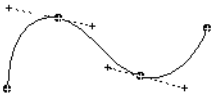
You can edit Bézier splines by varying the length of each handle separately: use the pet palette command for **Move tangent handle**.



Note that there is only one angle associated with both handles of a node, but the length of each handle may be different. The longer the editing handle, the smoother the curve will be at the control point.

You can pull out the handles from sharp nodes to turn them into curved ones.

Note: The visibility of the editing handles of Bézier splines depends on the status of the **View > On-Screen View Options > Spline Handles** toggle. If this toggle is set to “Show,” then all editing handles are visible on the Floor Plan. If the toggle is set to “Hide”, then only the editing handles of the spline last selected will be visible; if the selection is removed, no editing handles will be visible.

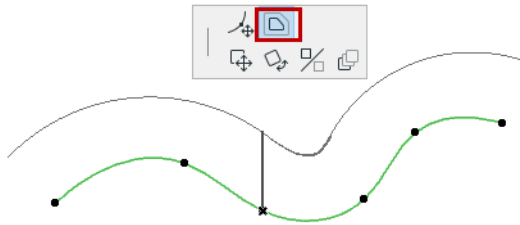


Editing Splines

Select a Spline, then edit it in the following ways.

Offset Spline

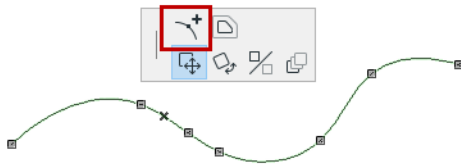
Choose the **Offset** command from the Pet Palette to offset the spline.



See also [Offset All Edges](#).

Insert new node

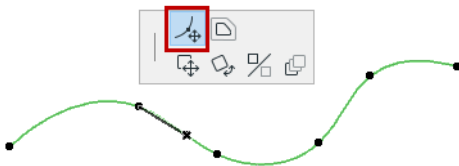
Click a spline with the Mercedes cursor to bring up the Pet Palette; then use the **Add new node to Spline** command from the pet palette.



Move or delete a node

Click a node of the spline to bring up the Pet Palette. Choose the **Modify Spline path** command.

- Drag the node onto its neighboring node to delete it, or
- Drag the node to a new location.

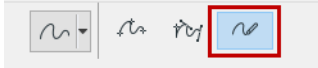


Unify a series of connected splines

- Select the splines.
- Choose the **Edit > Reshape > Unify** command.

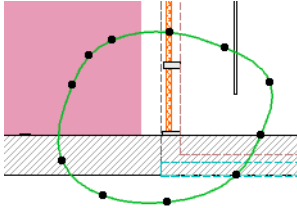
Drawing Freehand Curves

To draw a freehand curve, choose the **Spline** tool in the Toolbox and the third Geometry Method icon in the Info Box.



This method is particularly useful to mark up documents.

The Freehand method always creates natural Splines.

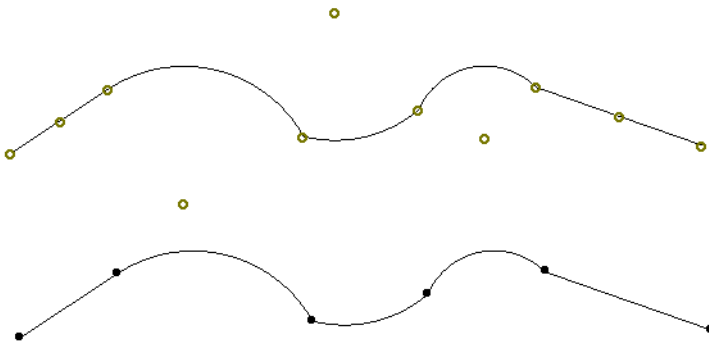


Drawing Polylines and Chained Lines

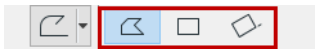
By combining the capabilities of the various line type tools, you can create a series of chained straight and curved line segments using the **Polyline** tool or the **Chained** geometry method of the Line tool.

The difference between the two options is the result they produce.

- **Polylines** are single elements drawn using the **Polyline tool**.
- **Chained lines** are a set of connected line segments drawn with the **Line tool**. Each line segment is a separate element, although they can be grouped before or after they are created. To group a chained line, activate the **Edit > Grouping > Autogroup** command before you start to draw Chained lines, or select the set of Chained lines you have drawn and activate the **Edit > Grouping > Group** command.



To draw a **Polyline**, choose a geometry method for the **Polyline** tool: Polyline method, Rectangle method or Rotated Rectangle method.



To draw a **Chained line**, choose a geometry method for the **Line** tool: Chained method, Rectangle method, or Rotated Rectangle method.



With either method (**Polyline** or **Chained Line**) you produce a series of straight or curved line segments joined at their endpoints.

- Draw a line segment and click to complete the segment.
- As you click to complete one segment, you simultaneously begin the next one.
- Click the **OK** button in the **Control Box** or double-click the last endpoint to complete the polyline or chained line.

For an illustration of this process, see [Create a Chain of Walls](#).

- To cancel the last-drawn segment of a polyline or chained line, use the **Backspace** key.

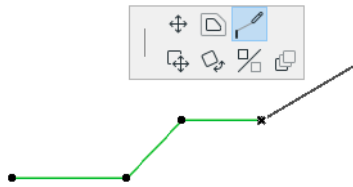
Continue a Polyline

You can extend a completed Polyline from either end:

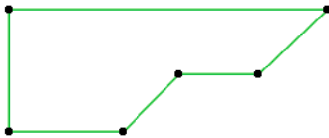
- Select the existing polyline.



- Click on either endpoint of the polyline.
- Choose the **Continue Polyline** icon from the appearing pop palette.



- Draw additional line segments, then double-click (or click OK in the Control box) to complete the polyline.



Note: When extending the original polyline, you can backspace to delete the last drawn segments, but you cannot delete segments of the original polyline.

Create Rectangles with Line or Polyline Tool

Using the **Rectangle** method with the Line tool will produce four individual Line segments, grouped by default. Using the Rectangle method with the Polyline tool will produce a single Rectangle.

With the **Rotated Rectangle** method, you first define a rotation vector for the rectangle's reference line. The rotation vector also defines the length of the two segments parallel to it. By hitting the Shift key once, you can unlock the length component constraint and use only the rotation angle component of the rotation vector.

Editing Polylines

Unify Drafting Elements into Polyline

Connected Lines, Arcs, and Polylines can be transformed into a single Polyline.

- Select all connecting elements.
- Choose the **Edit > Reshape > Unify** command.

The attributes of the last selected element will be applied to the unified set of elements.

You can unify multiple sets of selected, connected items with a single Unify command.

Note: To unify items that are Grouped, you must first Suspend Groups.

[See also *Intersect Two Elements*.](#)

Open a Closed Polyline

To open a closed polyline by deleting one of its segments:

- Activate the Arrow tool.
- **Ctrl-click (Cmd-click)** the line segment you wish to delete.

Note: Use the same method to cut a polyline having at least 3 segments into two independent polylines.

Explode a Polyline

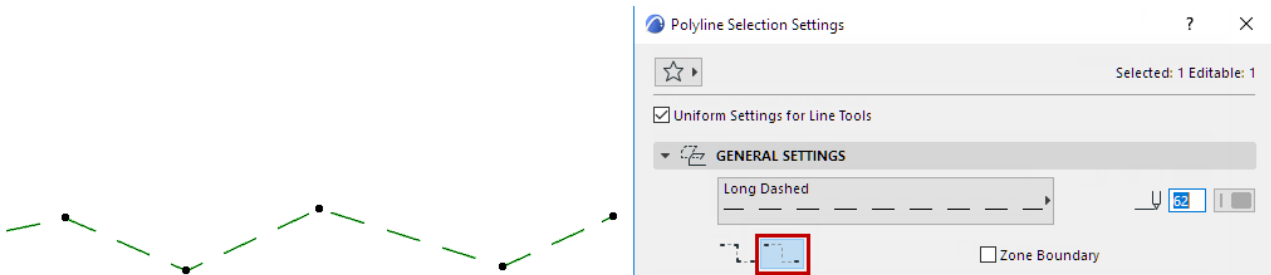
1. Select the Polyline.
2. Use **Design > Explode into Current View**.

The resulting lines and arcs are grouped. Ungroup them as needed.

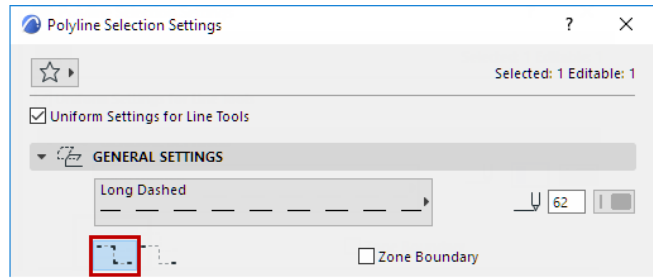
[See also *Explode into Current View*.](#)

Dashed Polylines: by Segment or Continuous

For Polylines only, if you are using a dashed line type, choose a preference in Polyline Settings for how to handle the dashed pattern at polyline nodes.



Distribute Dashed Pattern Over Entire Polyline



Distribute Dashed Pattern By Segment

Hotspots

Hotspots are simple points indicated by a small cross. Their main role is to help position elements in 2D views. Hotspots do not appear on printed and plotted outputs and can be deleted when they are not needed anymore.



For more information, see [Hotspot Tool Settings](#).

Note: The Guide Lines function can also locate intersection points and other useful snap points and directions.

See [Snap Guides and Guide Lines](#).

Placing Hotspots

When you have made your Pen Color and Layer choice in the Hotspot Settings dialog box, you are ready to place Hotspots. Just click at the desired location. In some situations, you may need to continue construction starting exactly on a given point of an element at a location where it has no hotspot or special point. The solution is to generate hotspots that will allow you to snap to this point.

Place Hotspot at Intersection of Two Elements

1. Activate the Hotspot Tool.
2. Select an existing Wall, Slab, Roof, Fill edge or line.
3. Ctrl-click (Cmd-click) the edge of any other existing element.
4. A hotspot will be generated at the intersection of the selected edge and the clicked edge.

Project Perpendicular Hotspot onto Selected Element

1. Activate the Hotspot Tool.
2. Select an existing Wall, Slab, Roof, Fill edge or line.
3. Ctrl-click (Cmd-click) any element hotspot or any empty space.
4. A hotspot will be generated on the selected element at the perpendicular projection of the clicked location.

Place Hotspot at Remote Tangential Point

1. Activate the Hotspot Tool.
2. Select the arc(s) and Ctrl-click (Cmd-click) with the checkmark cursor on the node of any other element.

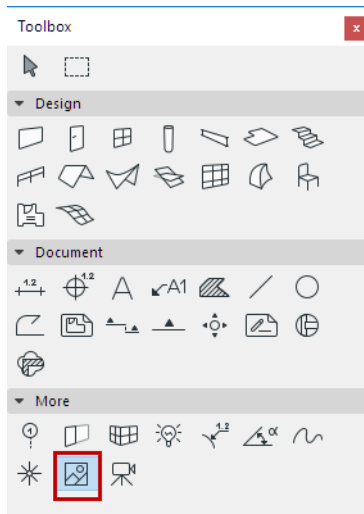
3. A tangential hotspot will be generated on the selected arcs or curved edges, or their extensions.

Place Hotspots Parallel to Existing Element

1. Activate the Hotspot Tool.
2. Select an existing Hotspot.
3. Ctrl-click (Cmd-click) with the Mercedes cursor on any Wall, line or element edge.
4. A second hotspot will be generated, so that the line joining the two hotspots is parallel to the clicked element.

Figures

Figures are image files imported and then embedded into the Archicad project file. They can be inserted in any of the 2D windows (Floor Plan, Section/Elevation/IE and 3D Document window, Detail, Worksheet, Layout). Figures are placed and manipulated by the dedicated Figure tool.



For more information, see [Figure Tool Settings](#).

A placed Figure can serve as a background image to the Archicad model by using the Align View function.

See [Align View](#).

You can also place the cropped content of an Archicad window in the plan as a figure.

Note: Drawings, in contrast to Figures, are assigned ID numbers and (optionally) titles. Drawing frames can be manually adjusted. Unlike Figures, Drawings represent linked files that can be updated.

For more information, see [Drawings in Model Views](#).

Figures can be displayed either with or without handles or as simple placeholders. This setting is global, that is, it affects all placed Figures. The choice can be made using the Figure Handle toggle under **View > On-Screen View Options**.

Note: Regardless of the import method, the images of Figure type elements are embedded in the Project and can make the file quite large. Use sizes and resolutions only as needed to prevent overloading the Project.

Placing Figures

When you have made your choices in Figure Settings, just click with the cursor to place the Figure. If the result does not satisfy you (for example, the image is too small or too large), open the Figure Settings dialog box again, and adjust the size, resolution or other settings before placing the Figure again.

You can also stretch the placed Figure, using the Shift key to constrain the diagonal and keep the image's proportions. If desired, however, you can stretch the Figure's X and Y dimensions independently.

Before you place the Figure, its ghost contour follows the movements of your cursor showing its proportional size. The ghost contour's anchor point is sensitive to element nodes and edges. To make this option inactive, go to **Options > Work Environment > On-Screen Options > Show Ghost Bounding Box of Objects, Columns, Figures and Drawings**.

You can also place Figures in Archicad directly, without using the Figure tool, by pasting bitmaps originating either from another Archicad window (for example a PhotoRendered Model picture) or another image-processing software. The pasted element will be handled as a Figure by Archicad.

Note: Images imported by this method are always pasted pixel by pixel in the current zoom.

Drawings in Model Views

Drawings placed **into Model views** are external files (such as DWG, PDF files and image files).

Drawings can be placed into any model view except a 3D Window or a Schedule view.

A placed drawing retains its source file information and can be updated. You can customize the size, title and framing options for your drawings. (Figures, in contrast, do not have these features.)

Placing a Drawing into a model can be useful if, for example, you receive a standard roof structure detail drawing from a manufacturer in PDF format. You can place this PDF document as a Drawing into your Detail window, add your own alterations or markups as needed, then place it as a modified detail onto a Layout.

Similarly, an external DWG file can be placed as a Drawing into your model as a vectorial drawing. This is a simpler solution for accessing DWG files in Archicad, as opposed to attaching the DWG file as an XREF.

Topics in this section:

[Place Drawing in Model View](#)

[Delete Drawing](#)

Related Topics

[Drawing Settings](#)

[Master Layout Settings](#)

[Arranging Multiple Drawings on the Layout](#)

[Drawings in the Layout Book](#)

[Working with PDF Documents](#)

[Managing and Updating Placed Drawings](#)

[Embedded Layers](#)

[Drawing Frame](#)

Place Drawing in Model View

Do one of the following:

Drag and Drop

Drag drawing files (one at a time) from your file system.

With Drawing tool

1. Activate the Drawing tool.
2. Click in the Archicad view.
3. The **Relink Drawing** dialog appears. Choose the source of the Drawing: either local file system or BIMcloud.

If needed, log in to BIMcloud. [See *Log in to BIMcloud*](#).

4. Click Browse, then select the desired Drawing file.

Notes:

- You can place one file at a time and one PDF page at a time using the Drawing tool. To place multiple drawings, use **File > External Content > Place External Drawing** as described below.

With File menu commands

Use one of the following depending on the source of the Drawing:

- **File > External Content > Place External Drawing**
- **File > External Content > Place External Drawing from BIMcloud**

Choose the BIMcloud, then the project you need.

If needed, log in to BIMcloud. [See *Log in to BIMcloud*](#).

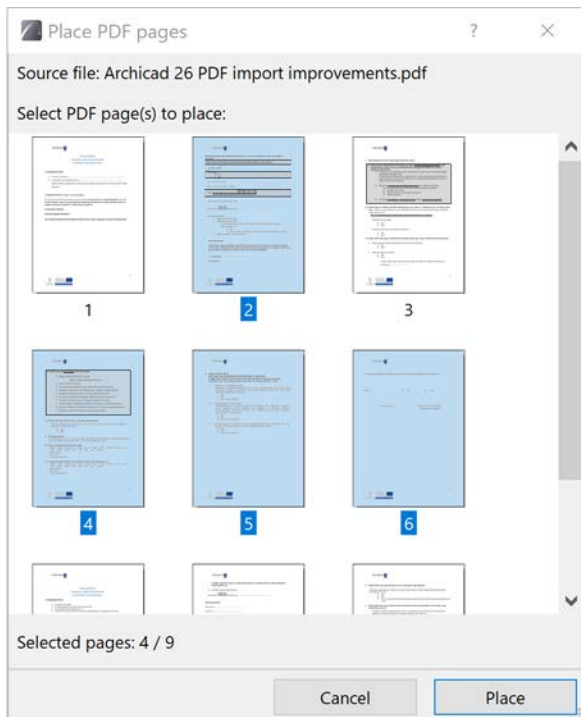
Note: To link a drawing from a Teamwork project, the source Teamwork project must have the same version number as the host project.

Place selected pages from a PDF

This works if the PDF is placed using drag-and-drop or a File menu command.

If the imported drawing is a multi-page PDF:

1. A chooser dialog appears. Select one or more or all pages.
2. Click **Place**.



Related Topics

To import only certain layers of DWG/DXF or DWF drawings:

[See "Enable Partial Open" at *Open Options \(DXF/DWG Translation Setup\)*.](#)

To show/hide the source layers of imported DWG drawings placed in Archicad:

[See *Embedded Layers*.](#)

Delete Drawing

To delete a drawing, select it and delete it like any other Archicad element. Note that deleting a drawing is not undoable EXCEPT if the active window is the same type (e.g. Floor Plan or Layout window) as the window which contained the deleted drawing.

Annotation

Annotation Elements display numeric or textual data in 2D views, either about the elements they refer to or as standalone informative elements.

- **Dimensions** are added to construction elements and drafting elements in the Floor Plan and the Section/Elevation/IE, 3D Document, and Worksheet/Detail windows using Dimension tools.
- **The Grid Tool** allows you to place Grid elements individually or in a Grid System. Grid elements are part of the model and serve as an editable framework for placing multiple elements or for identifying locations in the model.
- **Texts** are inserted using the Text tool, in text blocks accompanied by a flexible formatting options.
- **Autotext** is a text element containing a definition filled in automatically by the program. Autotext can be placed in both Model Views and Layouts.
- **Labels** allow you to link text information or a symbol to construction elements and 2D Fills. Independent (non-associative) Labels are also available.

Topics in this section:

[Dimensioning](#)

[The Grid Tool](#)

[Text Blocks](#)

[Autotext](#)

[Labels](#)

[Pointer Line](#)

[Find and Select Text Blocks/Labels](#)

[Text Editing in Archicad](#)

Dimensioning

See this video for an overview: [Dimension Editing](#)

Dimensions can be placed in the Floor Plan, the Section/Elevation/IE, the 3D Document, and the Detail and Worksheet windows.

Many dimensions are associative, which means that dimension values will be updated automatically if the element they are associated to is modified.

Note: “Static” Dimensions are not associative.

See [Static Dimensions](#).

The 3D Document window has some unique dimensioning options.

See [Linear Dimensions in the 3D Document Window](#).

The following dimensioning tools are available:

- **Linear Dimensions** display element lengths, both curved and straight.



- **Elevation Dimensions** (a construction method of the Linear Dimension tool) display height values in Section/Elevation/IE and 3D Document views.



- **Level Dimensions** display height values on the Floor Plan.



- **Radial Dimensions** display the radius of curved elements.



- **Angle Dimensions** display the angle in degrees between pairs of lines or edges.



Related Topics:

[Dimensioning Standards](#)

[Glossary of Dimensioning Terms](#)

[Static Dimensions](#)

[Linear Dimensions](#)

[Dimension Tool Settings](#)

[Dimension Text Settings](#)

[Pointer Line](#)

[Dimension Wall or Slab Thickness](#)

[Linear Dimensions in the 3D Document Window](#)

[Elevation Dimensions](#)

[Radial Dimensions](#)

[Level Dimensions](#)

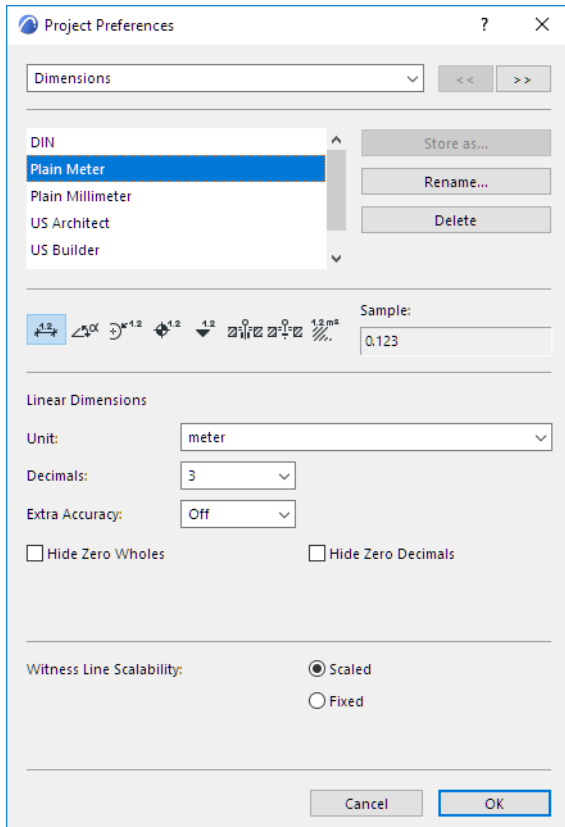
[Angle Dimensions](#)

[Edit Dimensions](#)

Secondary Dimensions (Add-on)
Automatic Exterior Dimensioning
Automatic Interior Dimensioning

Dimensioning Standards

Use **Options > Project Preferences > Dimensions** to apply a Dimensioning standard or to create a customized standard.



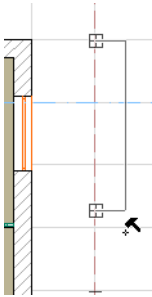
For more information, see [Dimensions Preferences](#).

Glossary of Dimensioning Terms

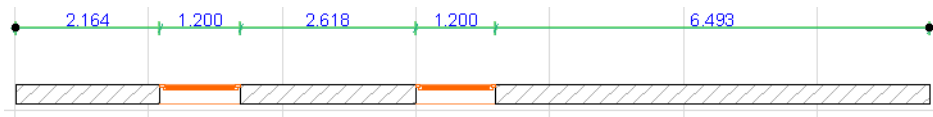
Reference points are temporary nodes that you create on Archicad elements during the dimensioning process; they disappear after you click to place the dimension chain.

- Round reference points: Resulting dimension points will be associative.
- Rectangular reference points: Resulting dimension points will be static (dimension values are not updated automatically).

See also [Static Dimensions](#).



The Dimension **Chain** appears after you click the hammer cursor at the end of the dimensioning process.



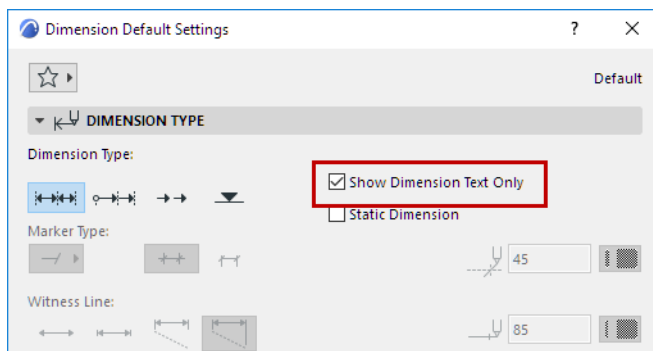
A dimension chain consists of

- a **dimension line** along the length of the element;
- **witness lines** (perpendicular to the dimension line), if Dimension Settings are set to display them;
- **dimension points** (where the dimension line intersects with the witness lines).

A **dimension segment** connects two adjacent dimension points and includes **dimension text**.

Show Dimension Text Only

With Linear, Radial and Angle dimensions, you have the option to **Show Dimension Text Only**.

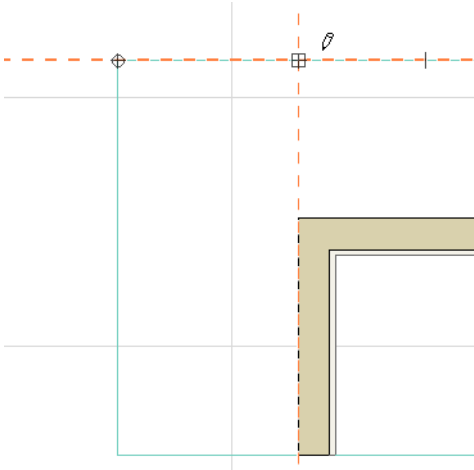


For such dimensions, you can display a reference line (on-screen only) as an aid in editing: turn it on at **View > On-Screen View Options > Dimension Reference Lines**.

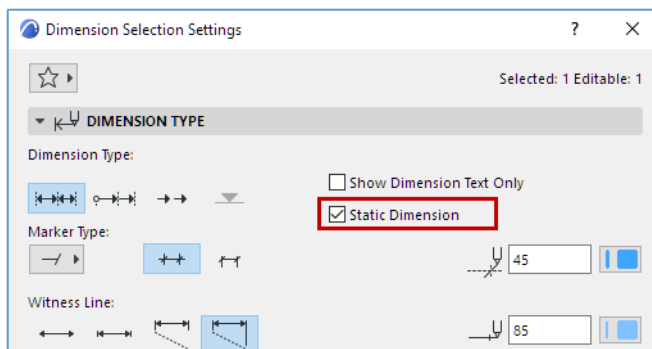
Static Dimensions

Static Dimensions are not associative.

The reference point of a static linear dimension is rectangular.



To make an existing dimension into a static dimension, select the dimension and enable the Static Dimension checkbox in Dimension Settings.

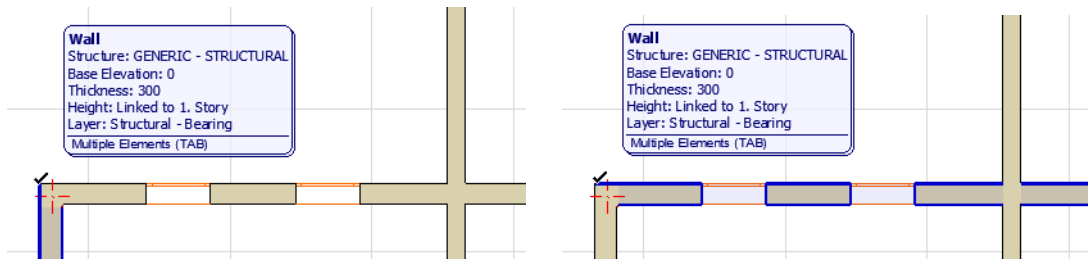


Once a dimension has been made static, it cannot be made associative again.

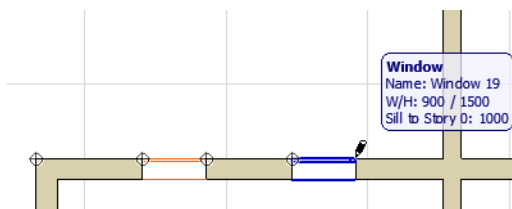
Linear Dimensions

1. Activate the Linear Dimension tool.
2. Move your cursor to the element you wish to dimension. Highlighted feedback lets you know which element will be dimensioned.

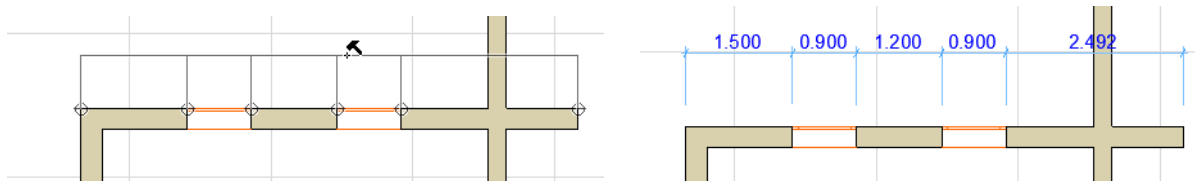
If your cursor is hovering over multiple elements, the Info Tag prompts you to click Tab to cycle the highlight through the elements.



3. Click along the element you want to dimension to place temporary reference points where dimension segments should begin and end. As you proceed, the element to which the reference point belongs is highlighted.
4. If needed, revoke any reference point by clicking it again.



5. Double-click after placing the final reference point.
6. Click the black hammer cursor where you want the dimension chain to appear.



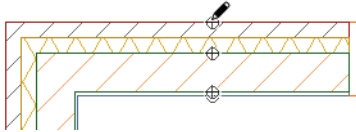
Related Topics

- [Dimension Wall or Slab Thickness](#)
- [Linear Dimension Geometry Methods](#)
- [Linear Dimension Tool Settings](#)

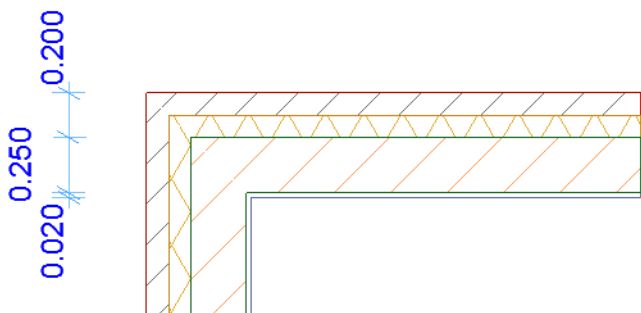
Dimension Wall or Slab Thickness

For Walls in Floor Plan/Section, and for Slabs in Section:

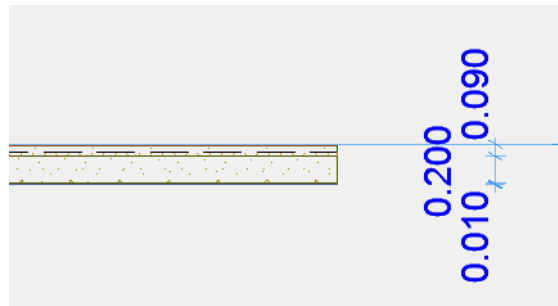
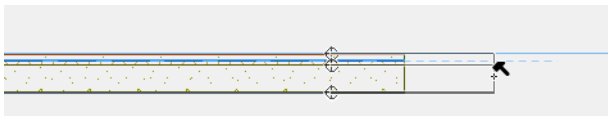
1. With the Linear Dimension tool active, click the contour line of the element to be dimensioned.
2. Reference points appear on the faces, skins and/or reference line (depending on the settings at Dimension Details - see below).



3. Double-click to complete the dimension chain and place it with the Hammer cursor.



In Section, you can dimension Slab thickness the same way:



If you place a series of wall-thickness dimension markers along a chained wall, only the dimension points located on walls perpendicular to the dimension chain will be displayed.

Related Topic:

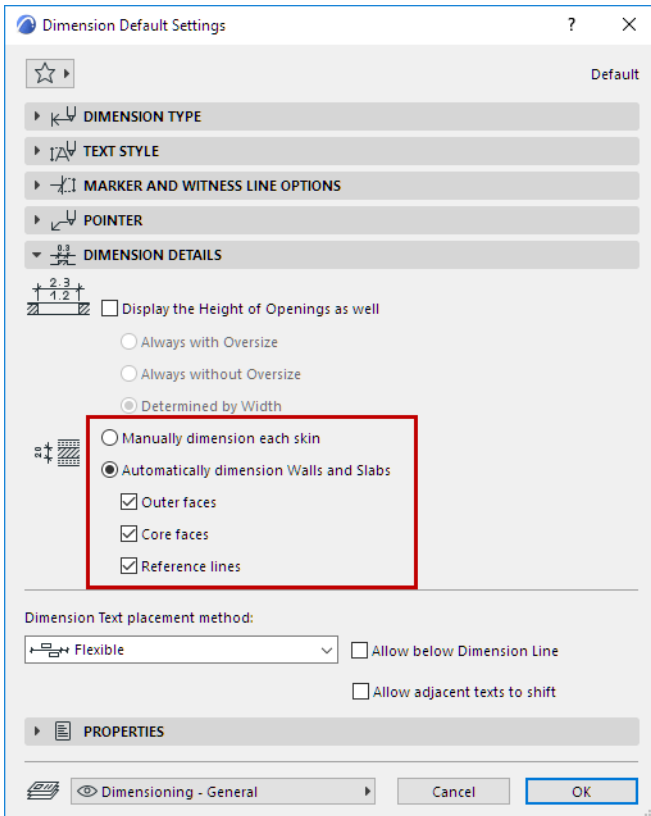
[Single-Click Thickness](#)

Dimension Skins of Composite/Profile Element Automatically

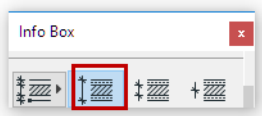
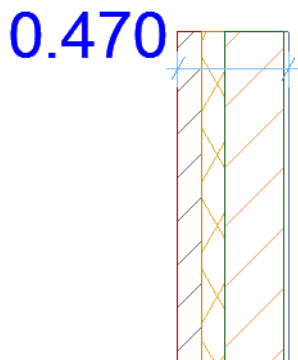
To dimension the individual skins and/or reference lines of composite or profile elements automatically:

1. Go to the Dimension Details panel of Dimension Settings (or the corresponding controls in the Info Box).
2. Choose the Automatic Method, then check options to be dimensioned.

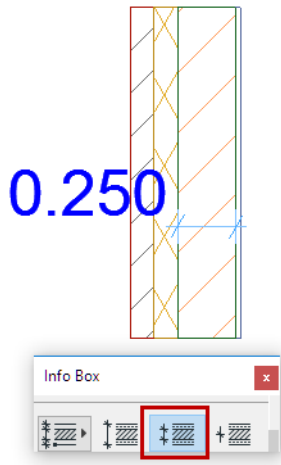
Use one or more checkboxes in combination to get all the dimensions you need.



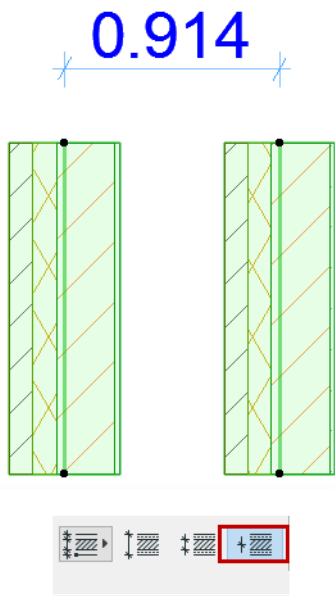
Dimension Outer Faces



Dimension Core Faces



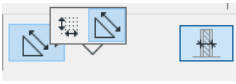
Dimension Reference Lines



[Watch the Video](#)

Linear Dimension Geometry Methods

Geometry Methods for Linear Dimensions are set in the Info Box.



The Geometry Method you choose applies to the entire dimension chain, and cannot be set unit by unit.

Note: There are additional Geometry Methods for the 3D Document window.

See [Linear Dimensions in the 3D Document Window](#).

Topics on this page:

[Any Direction Geometry Method](#)

[X-Y Only Geometry Method](#)

[Arc Length Geometry Method](#)

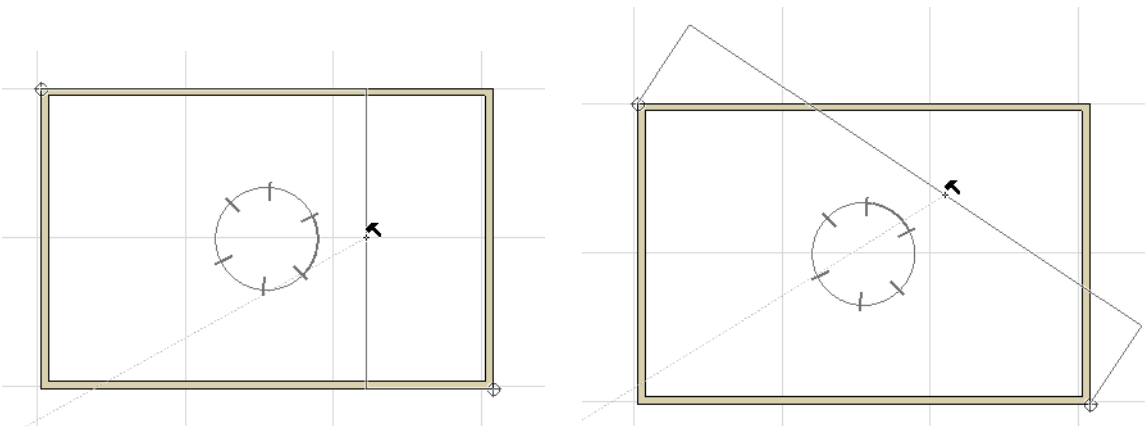
[Single-Click Thickness](#)

Any Direction Geometry Method

The default **Any Direction** method enables you to create dimension chains at a variety of positions relative to the element's position: either parallel to the first two reference points placed, or horizontal/vertical, or parallel to another edge/surface of your choice.



Move the cursor around to get feedback on the possibilities, which vary depending on the dimension points' position.



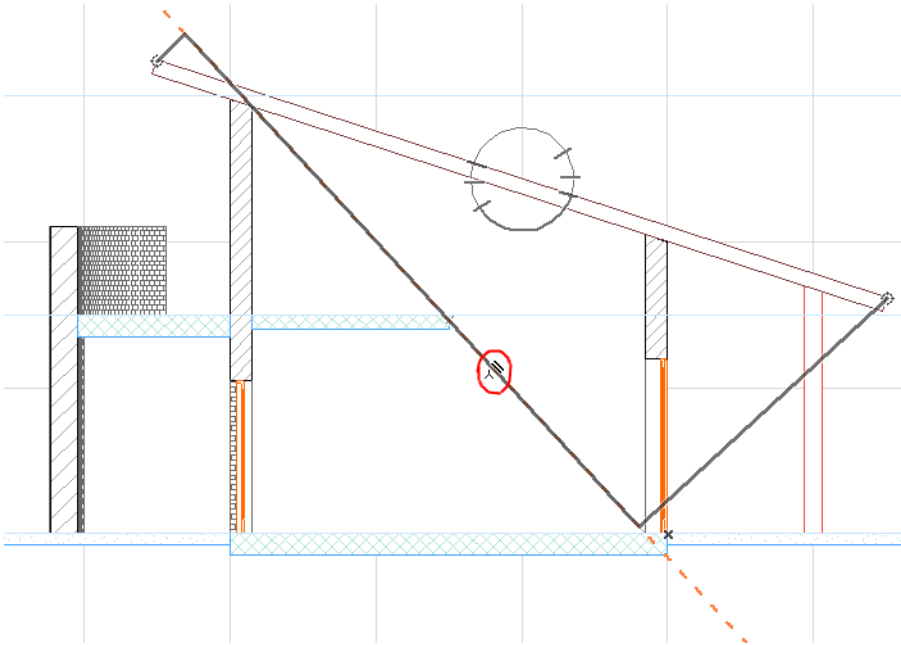
An additional dimension line vector is also available: After the black hammer cursor appears, you can move this cursor **to any straight line/edge** in the project.

Note: In the 3D Document window, you must align the dimension to the edge of a construction element - line segments do not work.

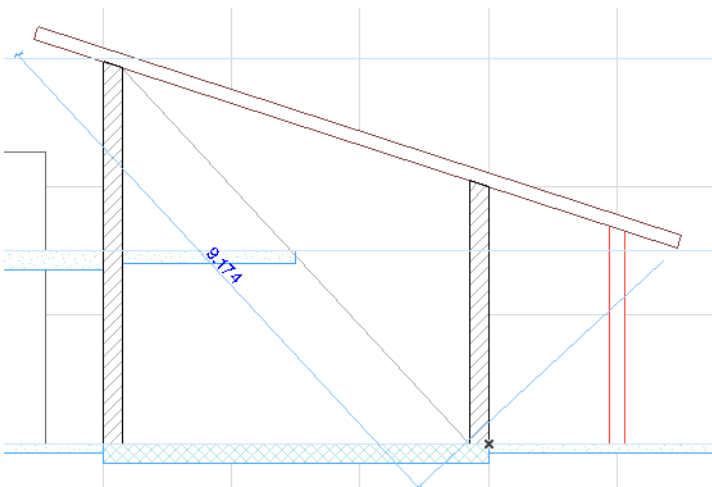
Example

Suppose you want to dimension the roof using the vector parallel to the diagonal, instead of parallel to the roof.

1. Click the two dimension nodes on either end of the roof.
2. Move the cursor over the diagonal. The cursor then changes to the “Mercedes + parallel dimension” cursor shape, to indicate that it has found an edge, to which the dimension line can be parallel:



3. Click to constrain the dimension line to be parallel to this line/edge.
4. Drag the dimension line to its final position and click with the black hammer to place the dimension chain.



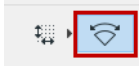
X-Y Only Geometry Method

The **X-Y Only** method restricts the dimension line zones to horizontal and vertical only, relative to the screen.



Arc Length Geometry Method

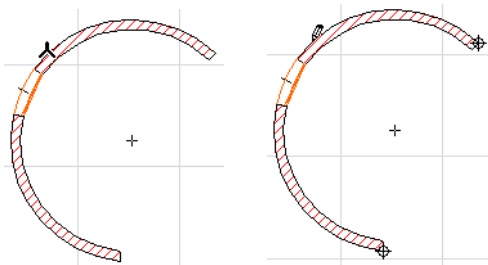
Use this method to dimension curved elements.



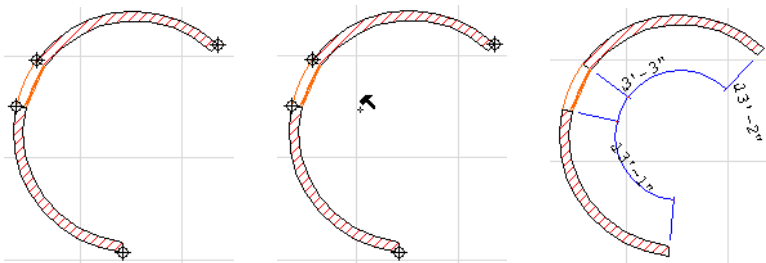
1. With the Linear Dimension tool active, choose the Arc Length Geometry Method from the Info Box.



2. Click with the Mercedes cursor on a circular arc or a curved edge - you don't have to find the end points, you can click anywhere on the curve.

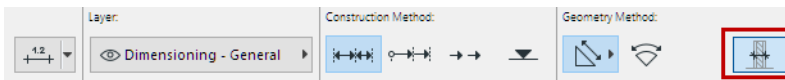


3. Archicad automatically marks the end points of the arc or the edge. This is the edge that arc length will be measured on. You may click any other points you want to add to the dimension chain.
4. Double-click anywhere in the workspace with the **Empty Pencil** cursor or click the **OK** button in the **Control Box** to finish selecting arc points for dimensioning.
5. Click with the **Hammer** cursor to place the dimension chain.



Single-Click Thickness

Enable the **Single-Click Thickness** control in the Linear Dimension Tool Info Box, to dimension Wall and Slab thickness with a single click.



1. With the Linear Dimension tool active, move the cursor to the Wall or Slab contour line.
If you click any other element, the one-click option does not work, and you have to place the dimension line as usual.
2. Click once to place the thickness dimension(s), depending on the settings at Dimension Details:
 - In Automatic Mode, depending on the Outer/Core/Reference Line Settings, this will result in multiple dimension points in the chain.

Note: If *only* the Reference Line option is active, the one-click option does not work.

- In Manual Mode, this will result in a chain dimensioning all skins of the walls and slabs.

For more information, see [Dimension Wall or Slab Thickness](#).

Linear Dimensions in the 3D Document Window

In the 3D Document window, use the Linear Dimension tool to place dimensions as needed. All the options for Linear Dimensions are available except Arc Length.

In the 3D Document window only, you can constrain the plane in which the dimension line is to be placed. (see below)

Choose Dimension Plane in 3D Document Window

To choose a plane constraint for linear dimensions in the 3D Document window, choose an option in the Linear Dimension Tool Info Box:



- horizontal plane
- vertical plane
- any plane (default option, which gives you the most freedom)

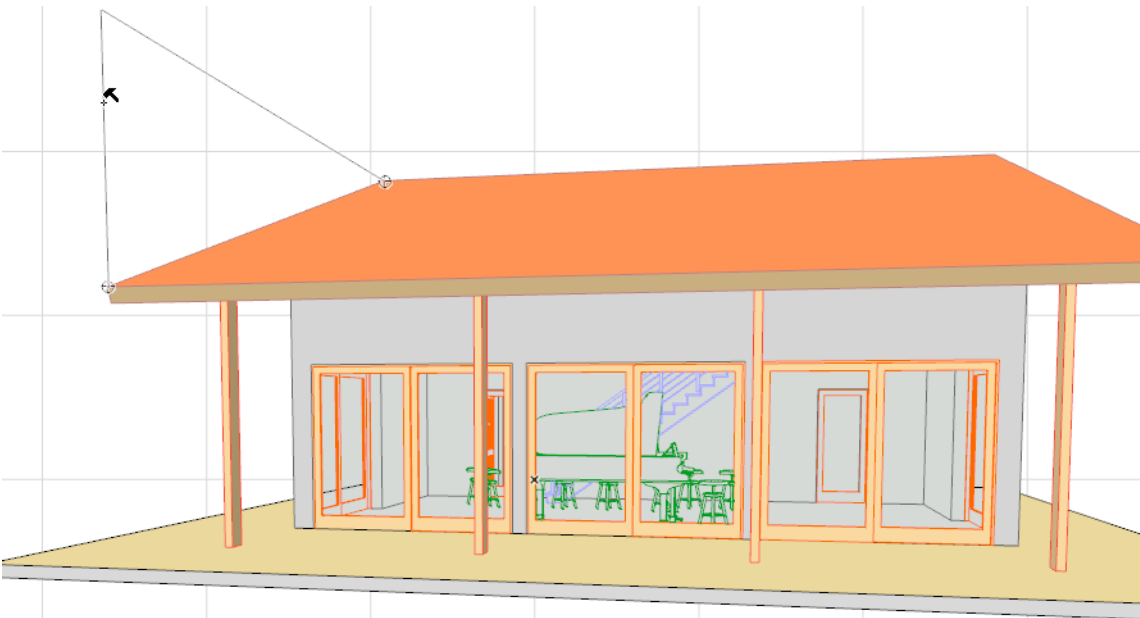
Example

To dimension the length of a roof's gable end in a 3D Document:

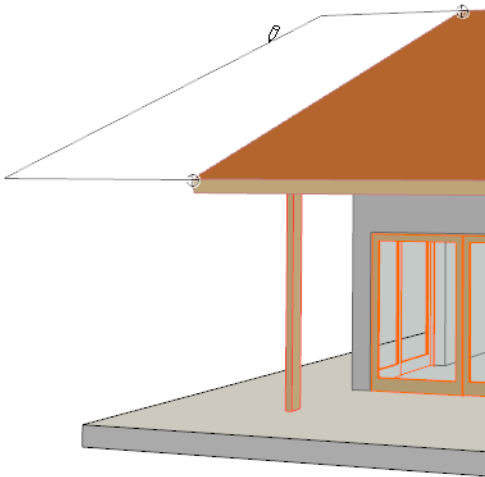
1. Activate the Linear Dimension tool.
2. In the Info Box, select the "Any Plane" geometry method and the "Any Direction" option.



3. Click on both ends of the roof, then double-click to finish the input.

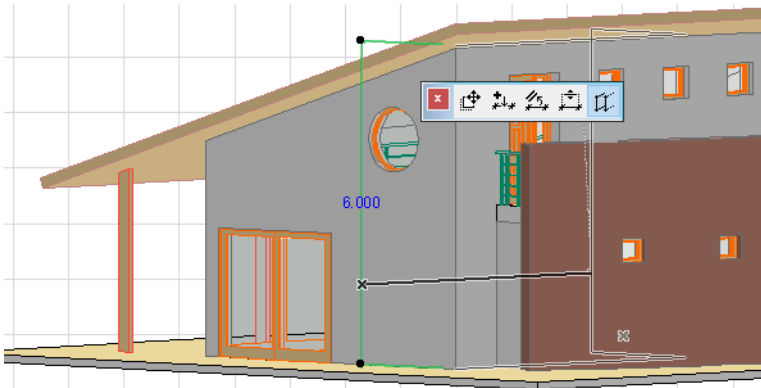


4. Move the cursor over the roof plane you wish to align to (feedback will indicate this alignment).
5. Click to define the plane. (If there is no plane detected, this step is skipped and the alignment will default to vertical or horizontal).
6. Click once more to place the dimension in its final location.



Offset Dimension Plane

Once you have placed a dimension line in the 3D Document window, you can offset it within its current plane. This option is available for dimensions in the 3D Document only:



The usual dimension line editing options are also available from the pet palette (insert/merge dimension point, drag or align dimension line.)

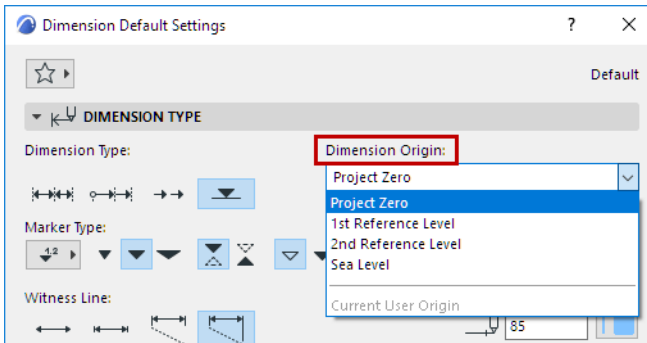
See [Edit Dimensions](#).

Elevation Dimensions

A special case of the linear Dimension tool is the **Elevation Dimensioning** construction method. Elevation Dimensions allow you to place height markers in Section/Elevation/IE and 3D Document windows.

Note: On the Floor Plan or Worksheet/Detail windows, use Level Dimensions. [See *Level Dimensions*](#).

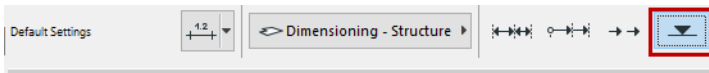
Elevation Dimensioning is calculated based on the Dimension Origin, which you can set as needed in Elevation Dimension Settings.



A series of Elevation Dimensions behaves as an associative dimension chain.

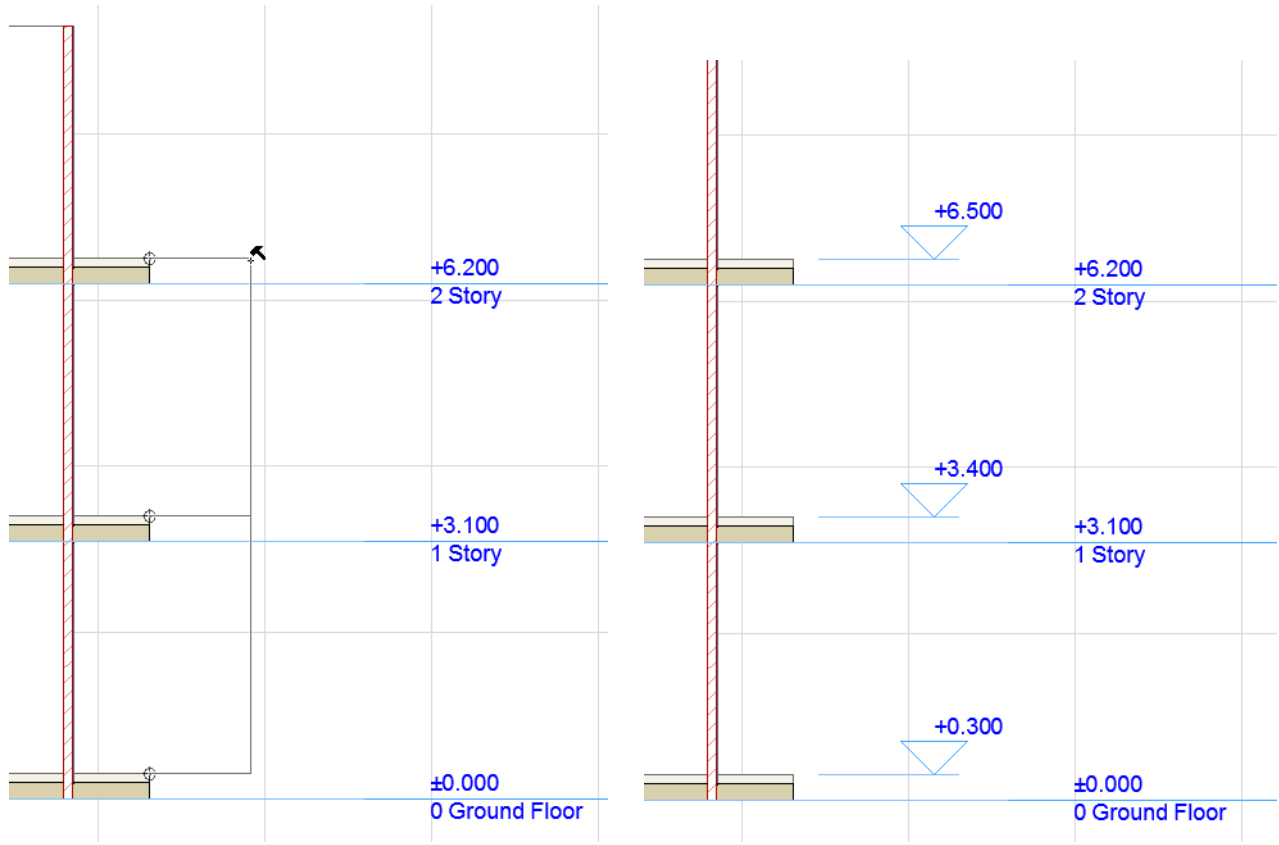
Place Elevation Dimension Chain

1. Activate the Dimension tool, with the Elevation Dimension construction method.



2. In the Section/Elevation/IE or 3D Document window: click the series of points where you need Elevation Dimensions.

3. Double-click (or click OK in the Control Box) and use the **Hammer** cursor to place the chain.

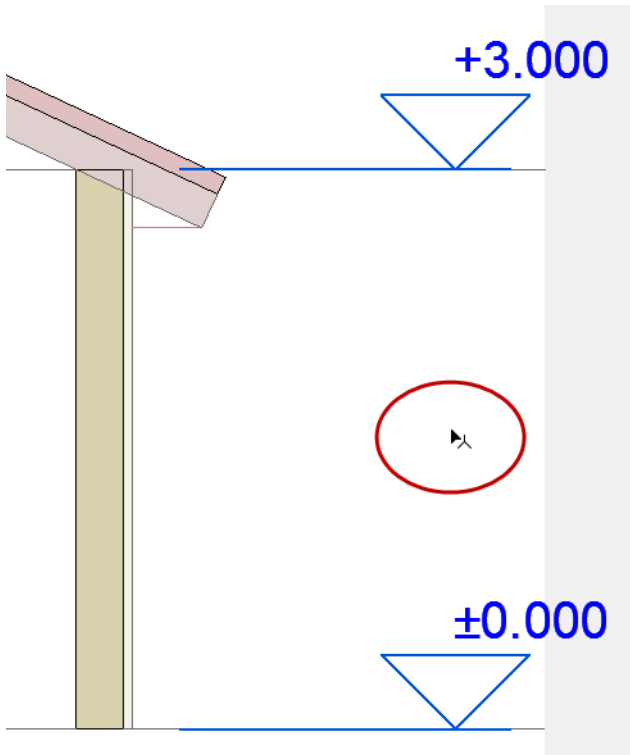


In Section/Elevation/IE windows, you can dimension all **Story Level Lines** in one step.

See [Dimension Story Level Lines](#).

Select Elevation Dimensions

Shift-click on the invisible axis of the chain (where the cursor changes to Mercedes with Arrow).

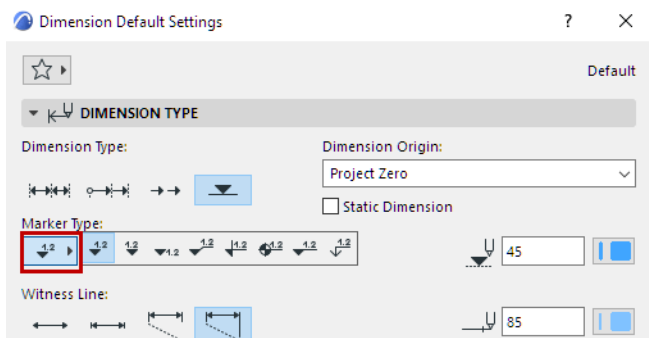


Elevation Dimension Settings

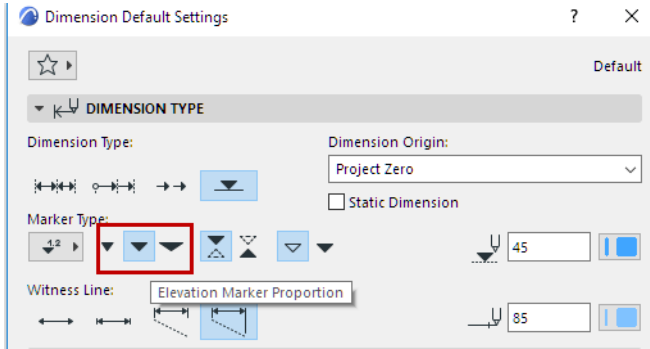
Most of the settings in this dialog box are identical to those at: [Dimension Tool Settings](#)

The following describes the controls which are unique to **Elevation Dimension Settings**.

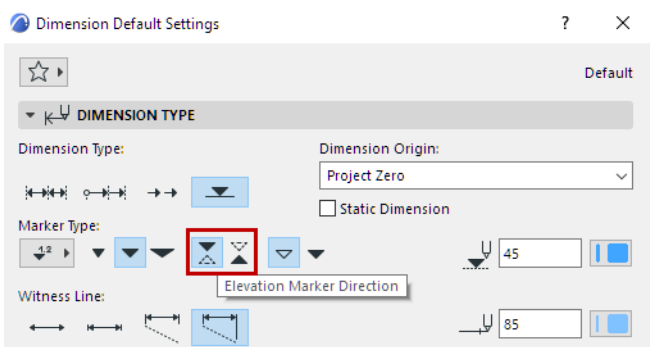
Marker Type: Click the pop-up to choose a Marker/Dimension text arrangement for the Elevation Dimension.



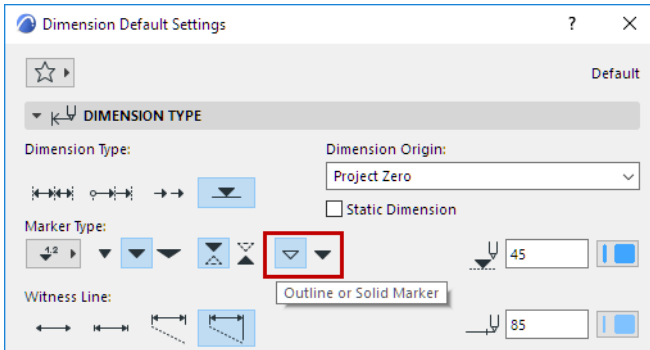
Click one of these icons to choose proportions for the Elevation Dimension Marker.



Click one of these icons to choose a direction for the Elevation Dimension Marker.



Click one of these buttons to choose outline or solid Marker.

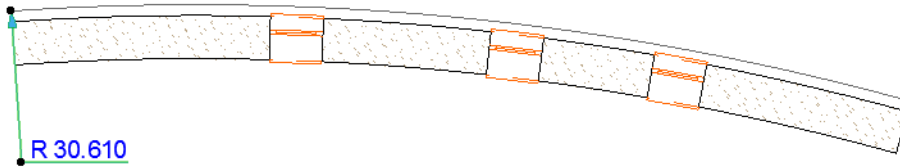


Choose a **Dimension Origin** from the pop-up list:

- Project Zero
- Sea Level (defined at **Options > Project Preferences > Reference Levels**)
- A Reference Level (defined at **Options > Project Preferences > Reference Levels**)
- Current User Origin: this option only appears for selected Elevation Dimension chains and indicates that the User Origin has been changed since the dimension chain was placed and the dimension chain has not been updated.

Radial Dimensions

Radial Dimensions display the value of the radius of a curved element.



Radial dimensions have two main parts: the Dimension (pointer) Line and the text.

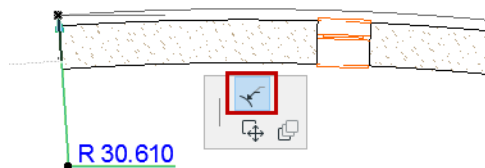
1. Click on a point of the curved element.
2. Draw the dimension line, either toward the centerpoint of the curve or in the opposite direction. The radial dimension must always originate from a curved element.
3. Click again to place the text, which shows the value of the radius.

Text orientation can be defined in Radial Dimension Settings. (See below.)

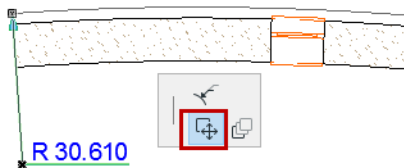
Stretch or Move Radial Dimension

You can adjust a placed radial dimension with the aid of the pet palette:

Use the **Stretch Radial Dimension** icon to stretch the dimension.



Use the **Move Radial Dimension** icon to move it.

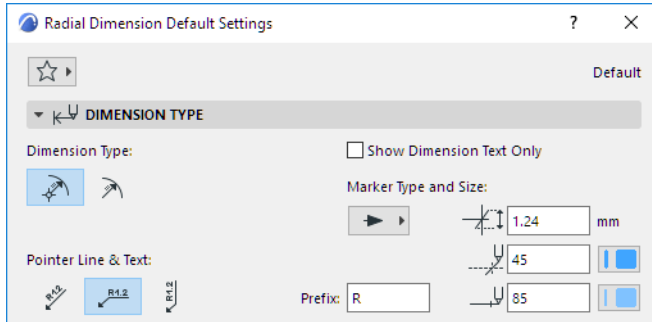


Radial Dimension Settings

The following controls are unique to **Radial Dimensions**:

Radial Dimension Type Panel

Dimension Type: Specify the type of the radial dimension here.



- Dimensions with centerpoints: Mark the centerpoint of the arc with a crosshair.
- Dimensions without centerpoints.

Pointer Line & Text:

Choose an option for orienting the pointer line and the associated text.

Prefix: Enter an optional prefix for the dimension text here.

Level Dimensions

Level Dimensions are point-level markers that display the story level, or an element's elevation as measured from a chosen reference level.

Available in Floor Plan and Worksheet/Detail windows.

Note: In Section/Elevation/IE windows, use Elevation Dimensions. [See *Elevation Dimensions*](#).

Related topics:

[Place Level Dimension on element](#)

[Level Dimensions on Tread, Landing, or Landing Structure](#)

[Place static dimension](#)

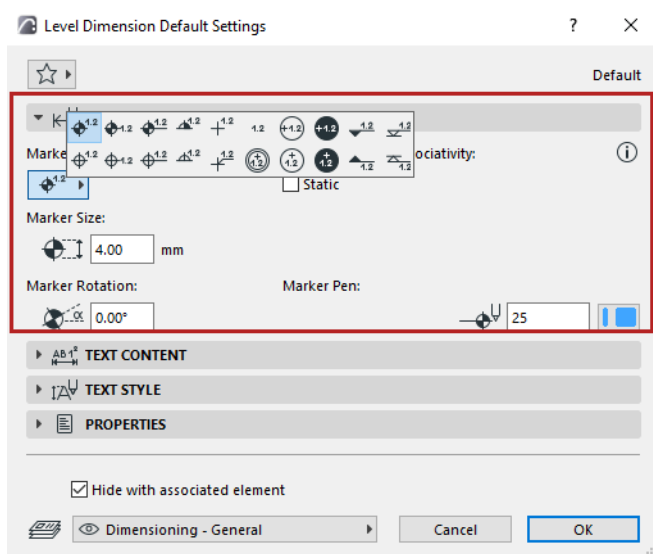
Place Level Dimension on element

You can place a Level Dimension on any of the following elements:

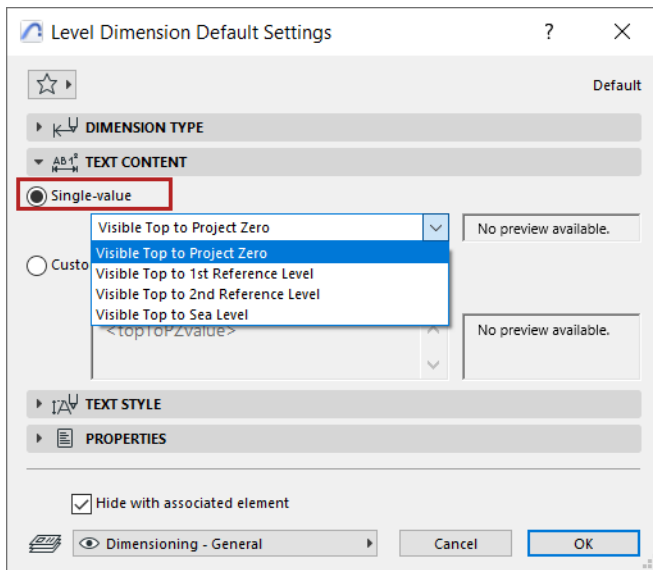
- Slab
- Mesh
- Roof (single-plane and multi-plane)
- Shell
- Stair (and Stair sub-components)

Note: A Level Dimension placed in empty space (instead of an element) will be associated to the story level.

1. Open Level Dimension settings.
2. In the Dimension Type panel: Choose marker type, size, rotation, and pen.

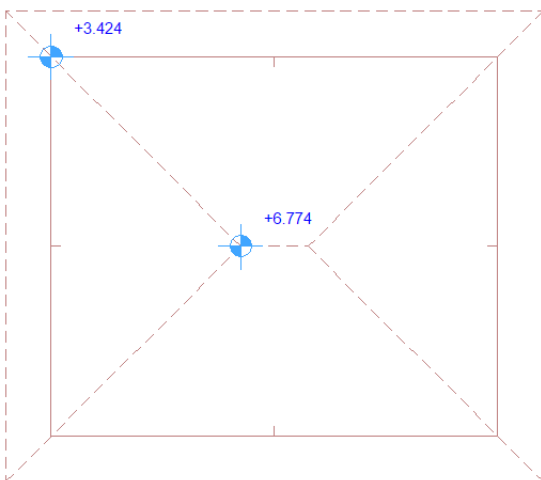


3. In the Text Content panel: Choose how to display the elevation value.
 - “Single value” is a basic option to display the elevation of the element’s visible top
 - “Custom” gives you more options via Autotext to choose different measured levels and reference levels. You can also add custom text and superscript.



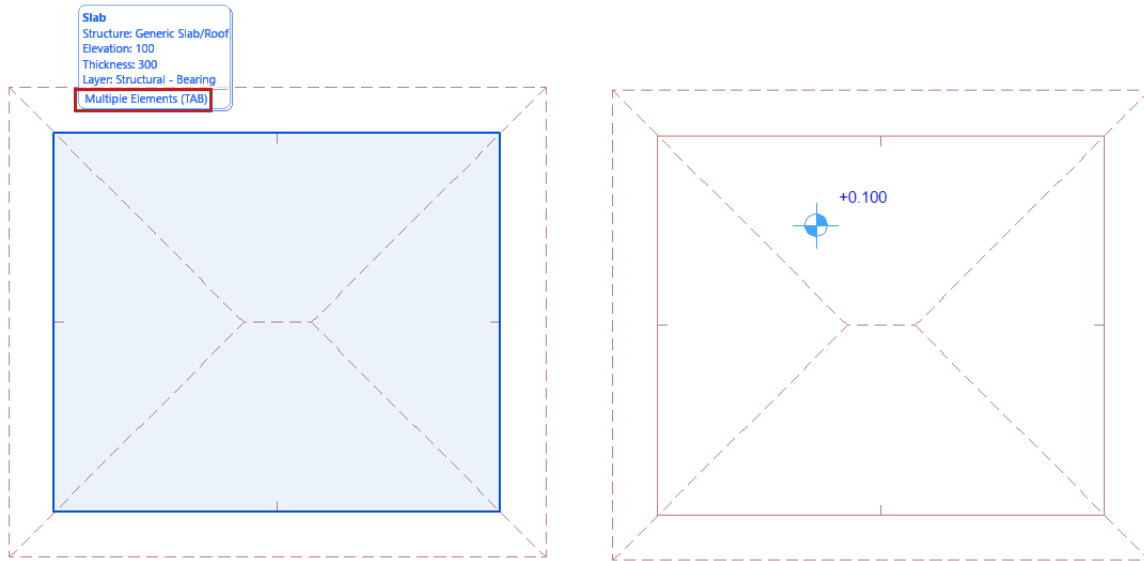
See details at [Text Content](#).

4. Hover your cursor over the element, then click to place. You must click on visible, selectable parts of the target element.



Level Dimensions on a Roof

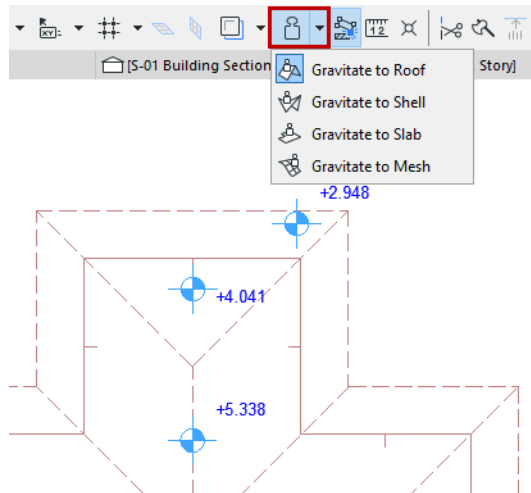
If the cursor is over multiple elements: click Tab until the desired element is highlighted, then click to place the dimension.



Level Dimension on a Slab

Use Gravity to filter target elements

If Gravity is on, the Level dimension can be placed only on the element type using gravity (either Slab, Roof, Shell or Mesh). [See Gravity.](#)

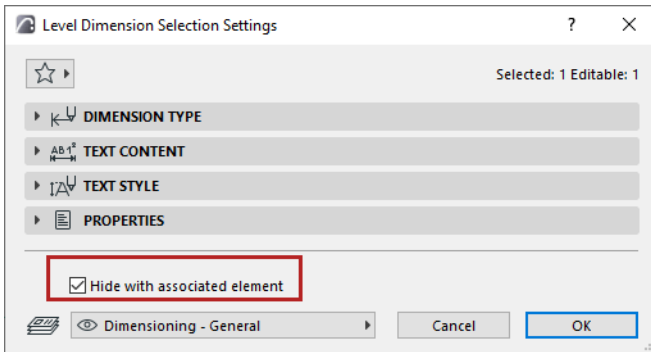


Edit dimension text

To edit the content and style of dimension text: see [Text Style.](#)

Hide with associated element

By default, a Level Dimension is hidden if its associated element is hidden (e.g. on a hidden layer). To display the dimension anyway, uncheck this box.

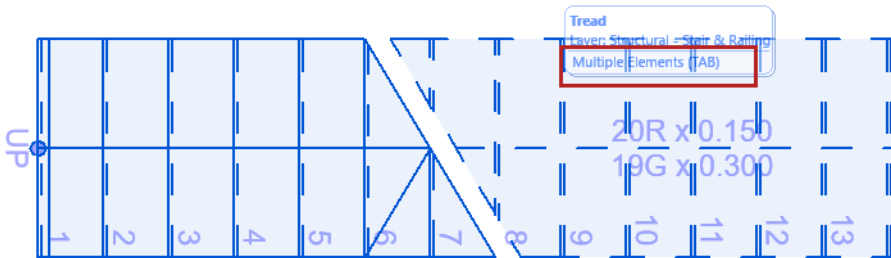


Place static dimension

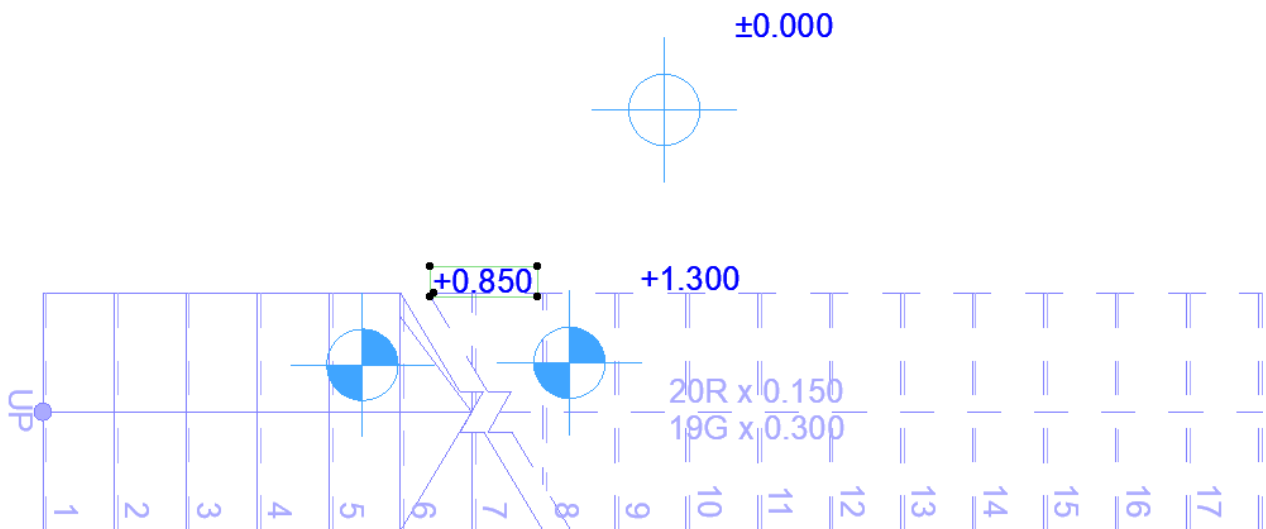
To place a static dimension (not associated to any element), go to Level Dimension Settings and check the **Static** box.

Level Dimensions on Tread, Landing, or Landing Structure

1. Make sure the Stair symbols (Grid Going and Grid Fill) are turned on (visible) in Model View Options or Stair Settings.
2. Hover your cursor over the Stair. Use Tab to switch the highlight between the Stair and its components (e.g. a tread).



3. Click to place.



Angle Dimensions

Angle Dimensions display angle values

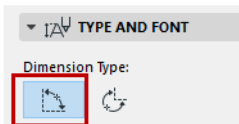
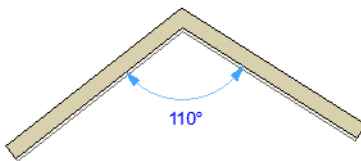
- between a pair of lines or linear edges
- or the angle of a curved element

Note: Angle dimensions are not available in the 3D Document Window.

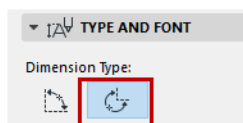
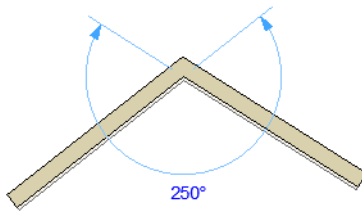
Angle Dimension Construction Methods

The Angle Dimension has two construction methods (choose from the Info Box or Angle Dimension Settings):

- **Inner Dimension** (acute) construction method:



- **Outer Dimension** (obtuse) construction method:

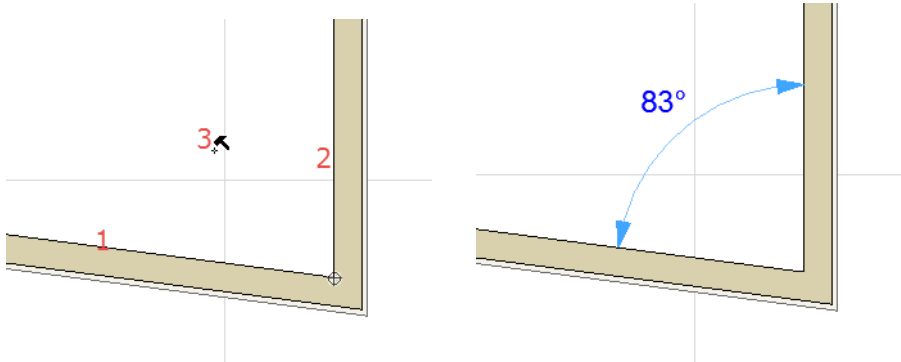


Place Angle Dimension Between Two Edges

To dimension an angle between two edges:

1. Click an edge or line.
2. Click a second edge or line.

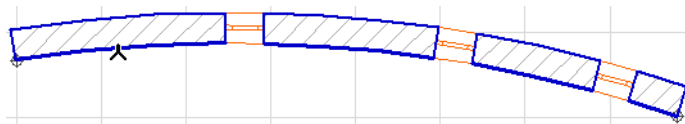
3. Click a third time to place the dimension of the angle.



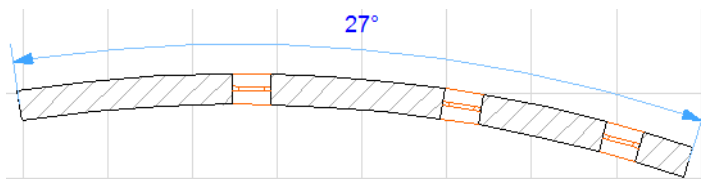
You can also click on any four locations in the window or on a Guide Line, to define two vectors. Click again to place the angle dimension between the two vectors: in this case, the angle dimensions is not associative.

Place Angle Dimension on Curved Element

1. Select the **Angle Dimensioning** tool in the **Toolbox**.
2. Click the curved element. The two endpoints of the arc will be automatically marked.



3. Click with the Hammer cursor to place the dimension.

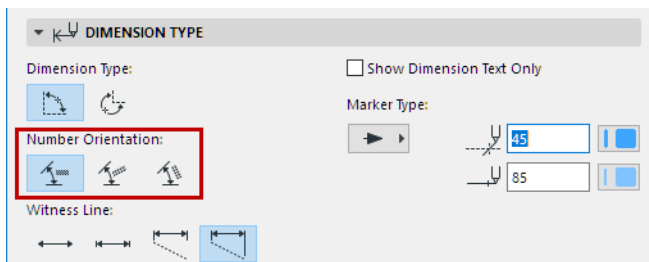


Angle Dimension Settings

Most of the settings in this dialog box are identical to those at: [Dimension Tool Settings](#)

Number Orientation

Choose a position for the dimension numbers: horizontal, perpendicular or parallel.



Edit Dimensions

Topics in this section:

[Select Dimension](#)

[Insert New Dimension Point](#)

[Delete Dimension Point](#)

[Move Dimension Point](#)

[Delete Dimension Segment](#)

[Move Dimension Line Segment](#)

[Break and Offset Dimension Line](#)

[Split Dimension Line](#)

[Merge Dimension Chains](#)

[Drag/Rotate/Mirror Dimension Chain](#)

[Align Dimension Line to an Edge](#)

[Stretch or Move Radial Dimension](#)

See also:

[Edit Text Box](#)

[Modify the Witness Line](#)

[Dimension Text Settings](#)

Select Dimension

Select Dimension Chain

Click at any “free spot” on the dimension line with the Checkmark with Mercedes cursor.

Select Dimension Segment

Click at its midpoint with the Checkmark with Arrow cursor.

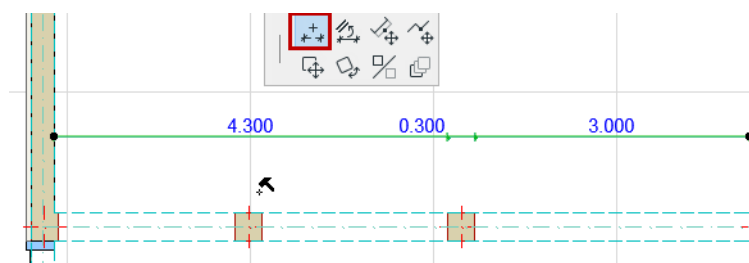
Select Dimension Text

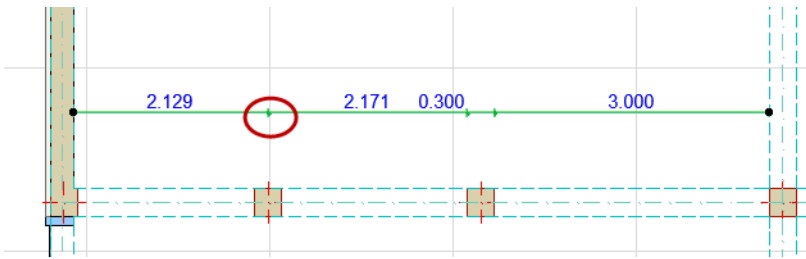
- Use the Quick Selection cursor.
- Click on the lower left corner of the text with the Checkmark with Arrow cursor.

[See *Dimension Text Settings*](#). [See also *Edit Text Box*](#).

Insert New Dimension Point

1. Select the whole dimension chain (click at a “free spot” on the dimension line).
2. Do one of the following:
 - From the pet palette, choose the **Insert/Merge Dimension Point** command, then click on the element whose dimension you want to add, or anywhere outside the line.





- Select the Dimension Tool, then **Ctrl/Cmd-click** on the element whose dimension you want to add, or anywhere outside the line.

Edit Dimension Point

Click the point. Then do any of the following:

- Click again to access Pet Palette.
- Access Dimension Settings for the selected point: the only available option is to edit the length of the witness line.

Delete Dimension Point

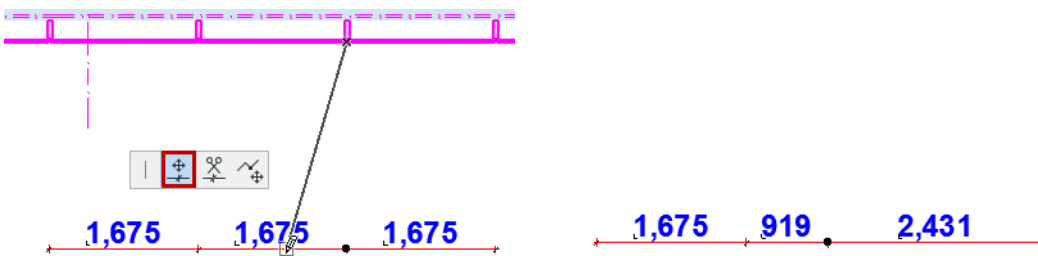
Select the Dimension point, then delete it.

Alternatively:

1. Select the entire dimension chain (click at a “free spot” on the dimension line).
2. Ctrl (Cmd)+Click any individual **point** to delete it.

Move Dimension Point

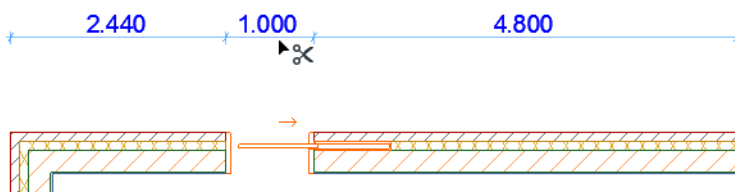
1. Select the dimension point.
2. Do one of the following:
 - From the pet palette, choose **Move Dimension Point**. Drag the selected dimension point to its new location.
 - Ctrl (Cmd)+Click at a new location to move the dimension point there.



Delete Dimension Segment

Do one of the following:

- **Ctrl + click** on a segment to delete it:

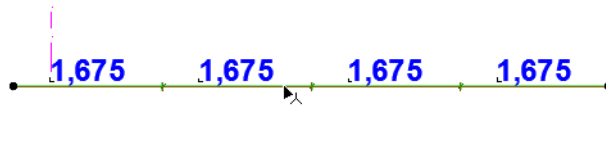




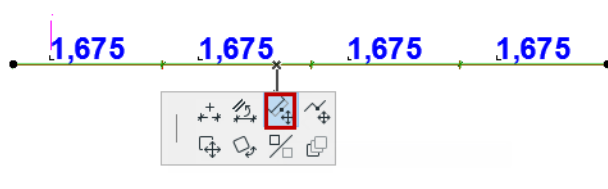
- Select a Dimension Text and delete it.

Move Dimension Line Segment

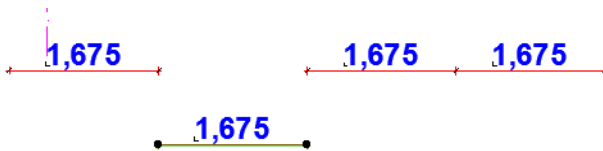
1. Select the whole dimension chain (click at a “free spot” on the dimension line).
2. Click on the segment you want to move.



3. From the pet palette, choose **Move Dimension Line Segment**.



4. Drag the segment to its new position. The original dimension chain is split accordingly.

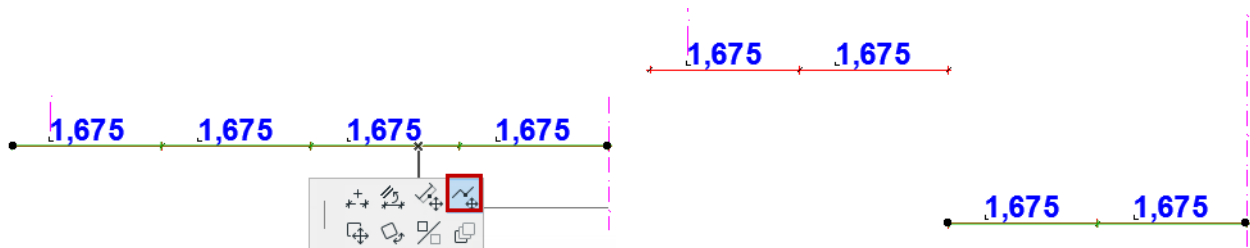


[Watch the Video](#)

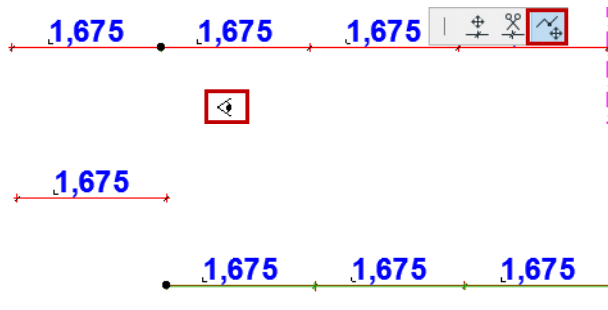
Break and Offset Dimension Line

This command is available on an internal point or segment of a linear dimension line.

1. Click at the point or segment where you want to break the line.
2. From the pet palette, choose **Break Dimension Line**.
 - If you clicked a **segment**: The dimension line is broken at the closest endpoint of the segment you clicked. You can drag the newly split line to a new position.



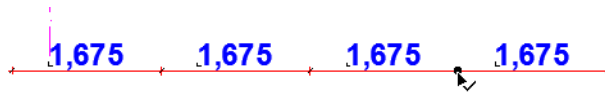
- If you clicked on a **point**: The dimension line is broken at the point. Click with the eyeball cursor on the side that you wish to offset.



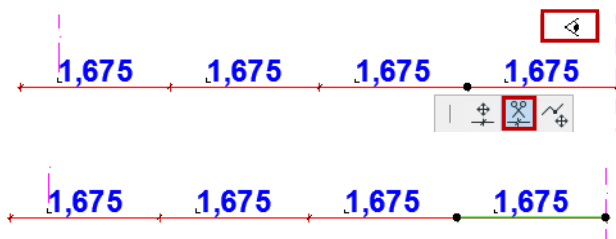
Split Dimension Line

This command is available on an internal point of a linear dimension line.

1. Click on the point at which you want to split the dimension line.



2. From the pet palette, choose **Split Dimension Line**.
3. Click with the eyeball cursor on the side of the line that you want to keep selected.



[Watch the Video](#)

Merge Dimension Chains

To merge two separate dimension chains (whether straight or curved), do one of the following:

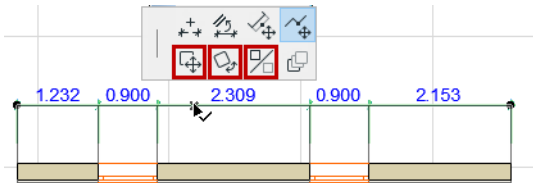
- Select one of the chains (click at a “free spot” on the dimension line), then Ctrl/Cmd-click on the other dimension chain.
- Select one of the chains (click at a “free spot” on the dimension line), bring up the **Insert/Merge Dimension Point** command from the Pet Palette, then click on the other dimension chain to merge it.



[Watch the Video](#)

Drag/Rotate/Mirror Dimension Chain

1. Select the entire dimension chain (click at a “free spot” on the dimension line).
2. Use the **Drag**, **Rotate** or **Mirror** command from the pet palette.

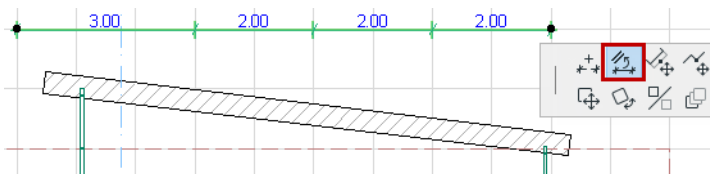


Notes:

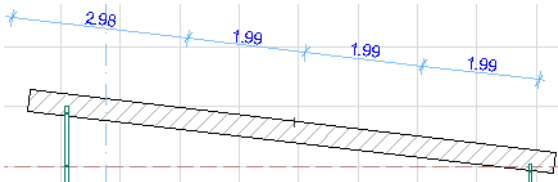
- You can drag or rotate a dimension chain as a whole. Units or points cannot be dragged, rotated or mirrored away from the parent dimension chain.
- Associative Dimension chains can only be dragged along the direction of their witness lines.
- After a rotation, all the reference points will automatically be redimensioned and projected along the new direction.

Align Dimension Line to an Edge

1. Select the entire dimension chain (click at a “free spot” on the dimension line).
2. Bring up the **Align dimension line** command from the Pet Palette



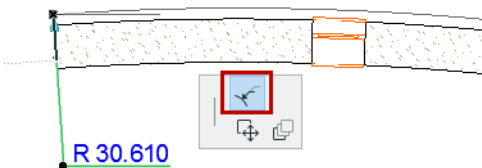
3. Click on a straight edge or line. The selected Dimension chain will align itself with the clicked edge.



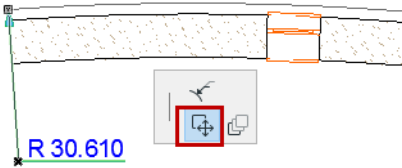
Stretch or Move Radial Dimension

You can adjust a placed radial dimension with the aid of the pet palette:

Use the **Stretch Radial Dimension** icon to stretch the dimension.



Use the **Move Radial Dimension** icon to move it.



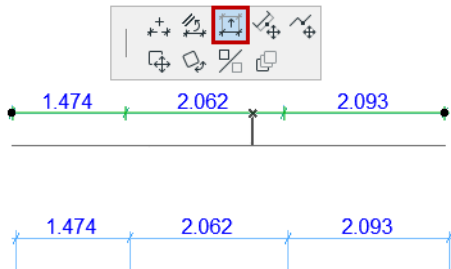
Modify the Witness Line

To choose a Witness line style, use Linear Dimension Settings.

See [Witness Line Format](#).

Edit Length of All Witness Lines

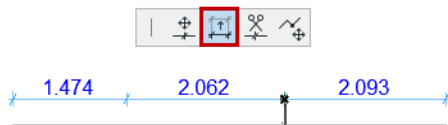
1. Select the entire dimension chain (click at a “free spot” on the dimension line).
2. Bring up the **Edit Length of Witness Line** command from the Pet Palette
3. Drag the cursor to the desired spot.



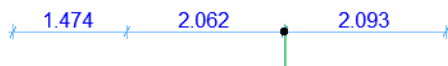
Dynamic witness lines will never extend beyond the reference node of the dimensioned element.

Edit Length of Single Witness Line

1. Select the dimension point.
2. From the pet palette, choose **Edit Length of Witness Line**.



3. Drag the selected witness line of the selected point to its new location.



Edit Text Box

The following applies to positioning text boxes and pointer lines (if any) used in Dimensions, Fill Area texts, and Labels.

Select Text Box

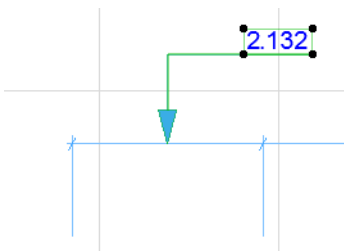
Select Text Box Plus Pointer Line

Copy-Paste Dimension Text

Select Text Box

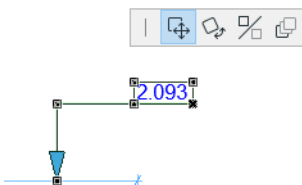
To select a Dimension or Label text box, do one of the following:

- Use the Quick Selection cursor
- Click on a corner of the text with the Checkmark with Arrow cursor



Select Text Box

Once you select a text box, you can change its position and/or orientation using **Edit** functions (e.g., **Drag**, **Rotate**, **Mirror**).



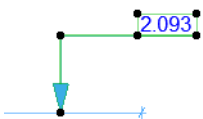
Note: If the Text box also has a Pointer Line, the Lock Angle control affects the range of graphical editing. If it is ON, the Pointer Line's starting angle remains fixed, limiting the edit possibilities.

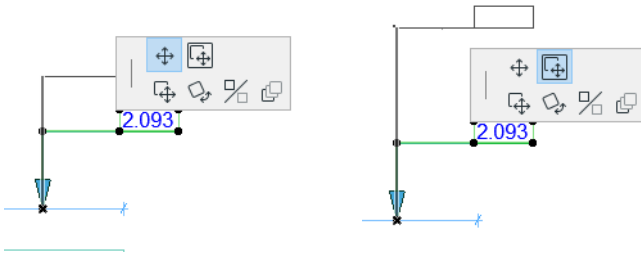
Delete Text Box

Deleting the Dimension Text deletes the corresponding Dimension segment too.

Select Text Box Plus Pointer Line

If you click the Pointer line, both the line and the text box are selected, enabling additional editing options.





[Watch the Video](#)

[Watch the Video](#)

[Watch the Video](#)

Automatic Dimension Text Placement in Linear Dimensions

Archicad automatically adjusts crowded Dimension Texts to enhance readability. The default method is usually satisfactory. For more control over how dimension texts are distributed, use the options at in Dimension Settings:

[See *Dimension Text Placement Method*.](#)

Copy-Paste Dimension Text

Only simple text items can be copied; superscripts will not be copied.

1. Select the dimension text only (not the dimension line itself)
2. Use the Copy command (**Edit > Copy** or **Ctrl/Cmd + C**)
3. Paste the dimension text (**Edit > Paste** or **Ctrl/Cmd + V**) - for example, to a text box or onto a Layout.

Related Topics:

[Dimension Text Settings](#)

[Pointer Line](#)

[Text/Autotext Label Content](#)

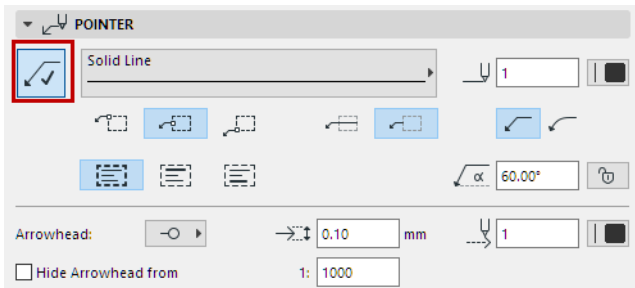
[Show Dimension Text Only](#)

Pointer Line

Available for Text Label and Dimension Texts.

[Watch the Video](#)

Click the Pointer button to assign a pointer line and activate the rest of the settings on this panel.

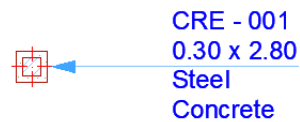
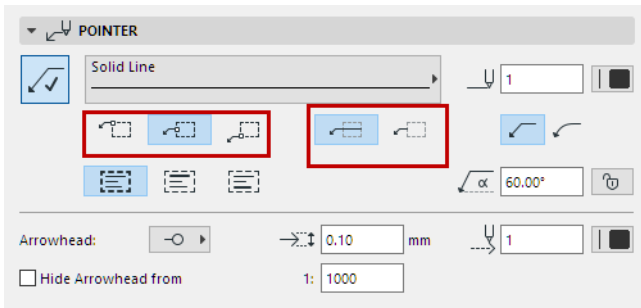


Define a **Line type** and **Pencolor** for the pointer line.

Connect Pointer to Text Box

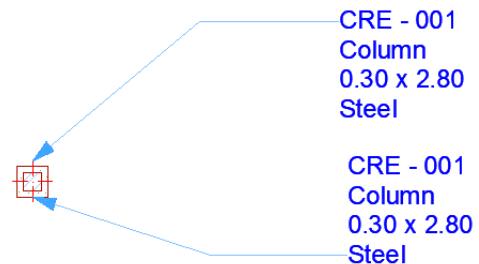
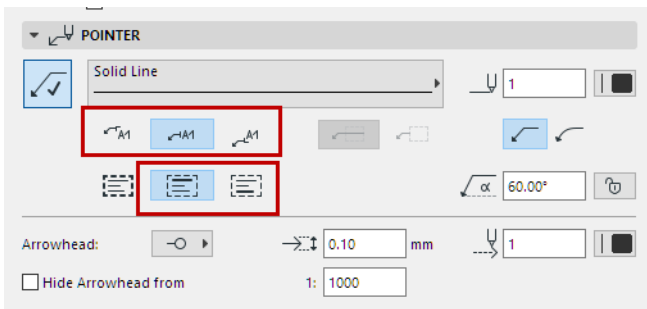
Use these controls to connect the Pointer Line to the Text Box (top, middle, or bottom).

If needed: Extend the pointer line to run underneath the text row.



Connect Pointer to First/Last Text Row

Use these controls to connect the Pointer Line to the First or last row of text (aligned with the top, middle or bottom of the row):



Choose whether the pointer should have a straight, curved or inverted pointer line.

Enter the **Pointer Starting Angle** using a value from 0 to 90 (the default is 90 degrees). The angle is measured from the dimension line. Click the lock icon to prevent the angle from being modified graphically (e.g. by accident).

Note: The Lock Angle control affects the range of graphical editing. If it is ON, the Pointer Line's starting angle remains fixed, limiting the edit possibilities.

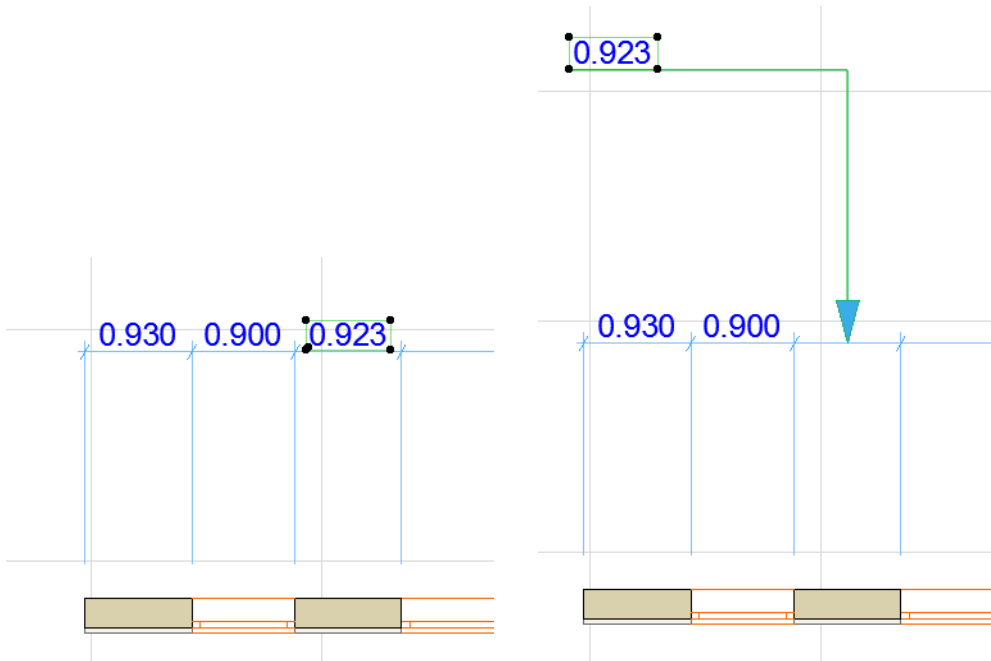
Choose a style and pencolor, and enter the size of the **Arrowhead**.

Hide Arrowhead from: Check this box if you want to hide the pointer line arrowheads on views which exceed a certain scale (enter the desired scale in the field to the right).

[See also *Edit Text Box*.](#)

Automatically Appearing Pointer Line (Dimension Text Only)

Even if you have not turned on a Pointer Line for a Dimension text, a Pointer Line will automatically appear if you drag the Dimension text box beyond a certain range.



This automatically added Pointer line will disappear if you do any of the following:

- Turn the Pointer Line off in Dimension Text Settings
- Drag the dimension text back within range
- Check **Revert to Automatic Position** in Dimension Text Settings

Related Topic:

[Custom Settings for Symbol Labels](#)

Automatic Exterior Dimensioning

Use this function to add automatic associative linear Dimensions to selected Walls or Curtain Walls on the Floor Plan.

1. On the Floor Plan, select the Walls or Curtain Walls to be dimensioned.

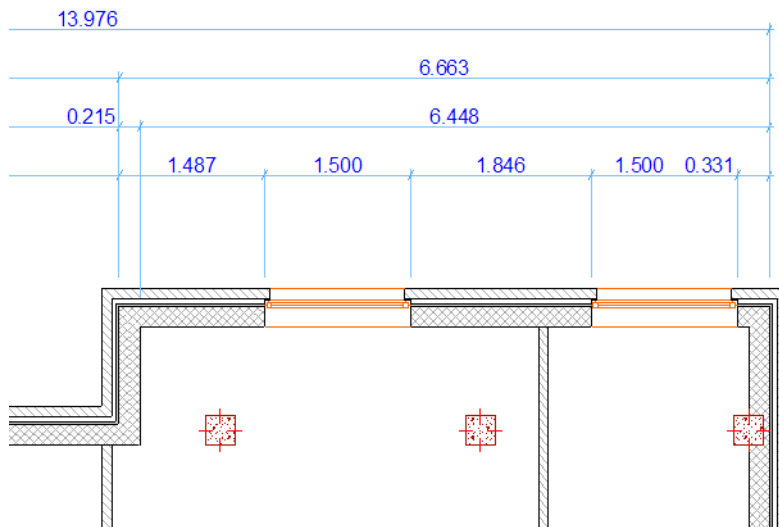
Go to **Document > Annotation > Automatic Dimensioning > Exterior Dimensioning**. Set the desired options.

See [Automatic Exterior Dimensioning Dialog Box](#).

2. Define the direction of the dimension lines. Do this in one of two ways:

- Click the side edge of any element (the dimension direction will follow the element direction); or
- Click in an empty area, then click a second time to define the dimension direction.

3. Click with the hammer cursor to place the dimension line(s).

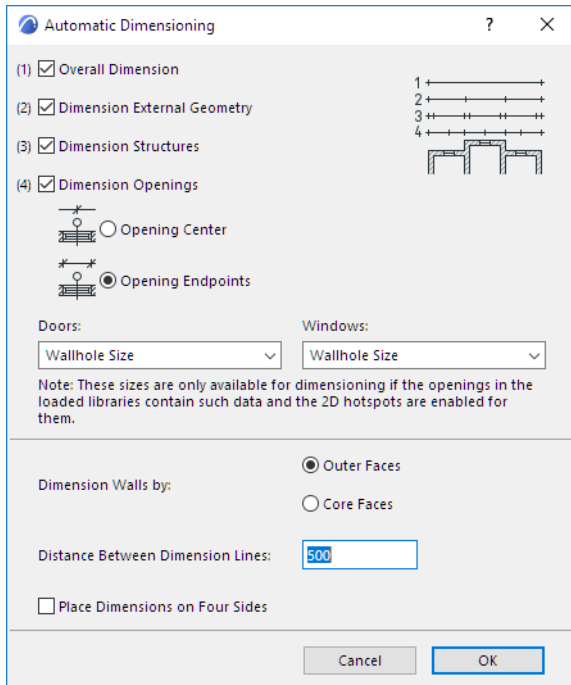


Automatic Dimensioning will automatically place the dimension lines (up to four types) depending on the options you have chosen in the dialog box. If any of these dimension lines are unnecessary (e.g., there are no openings, or the facade is straight), it is ignored.

The display of automatic dimensions depends on the options chosen in Dimension Tool Settings.

Automatic Exterior Dimensioning Dialog Box

Open the dialog box from **Document > Annotation > Automatic Dimensioning > Exterior Dimensioning**.



Check one or more Dimension options to use:

Overall Dimension

A single dimension line that runs the length of the selected element set.

Dimension External Geometry

The linear dimensions of extruding elements, such as a facade.

Dimension Structures

The linear dimension of the element itself, such as wall thickness

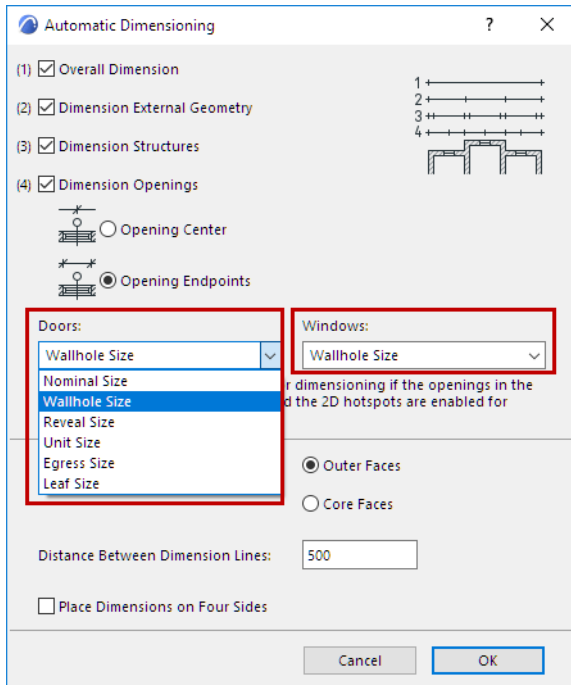
Dimension Openings

Check this box to include the dimensions of openings (doors and windows).

Choose where to place the dimension for openings:

- **Opening Center**
- **Opening Endpoints**

Choose separate dimensioning preference for doors and for windows.



Note: If you automatically dimension doors placed from the Archicad 12 or Archicad 11 libraries, and if you choose Leaf or Egress size as the preferred door dimension, the door's height value will not be shown unless you choose Leaf or Egress as the Nominal value in Door Settings. (Door widths will be dimensioned without any problem.) If you automatically dimension doors placed from earlier Archicad versions, only the Wallhole and Nominal Size options will work.

Dimension Walls by

Choose a preference for dimensioning composite Walls.

- Outer Faces
- Core Faces

Distance Between Dimension Lines

Enter a distance using the working units currently set in Archicad.

Place Dimensions on Four Sides

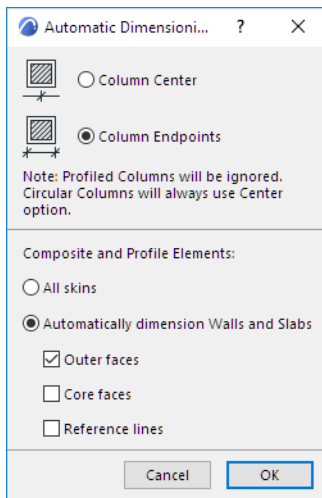
Check this box to automatically place dimension lines on all four sides of the bounding box of the selected elements.

Automatic Interior Dimensioning

Use this function to quickly dimension a set of selected Columns and composite or profile (complex) elements on the Floor Plan.

1. Select one or more Columns or Composite/Profile elements to be dimensioned.

Note: Profiled Columns will be ignored.
2. Open the dialog box from **Document > Annotation > Automatic Dimensioning > Interior Dimensioning**.
3. Choose preferred dimensioning options.



Column Dimension Preference

- **Column Center:** Dimension the axis of the column
- **Column Endpoints:** Dimension the endpoints of the column.

Note: A circular column will always be dimensioned at its center regardless of this checkbox.

Composite and Profile Elements

Choose an option:

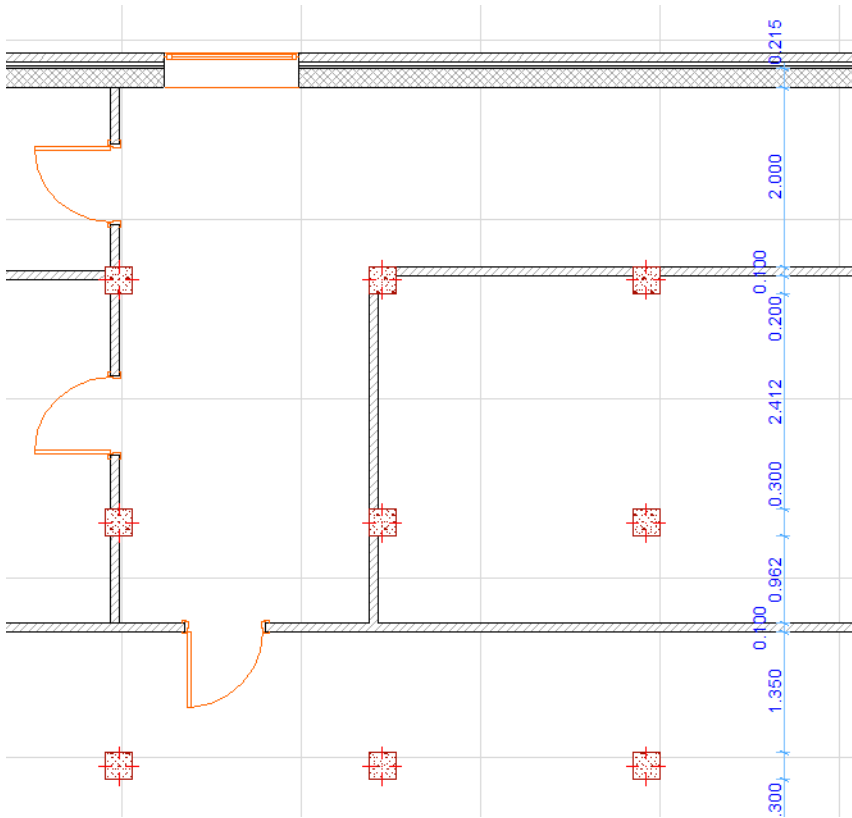
- **All Skins:** Each skin of the composite/profile element is dimensioned separately. (If this is unchecked, a single dimension is applied to the total width of all the skins.)
- **Automatically dimension Walls and Slabs:** Dimension points are placed at the faces and/or reference lines depending on the options checked below (at least one category must be checked):
 - Outer faces
 - Core faces
 - Reference lines

Note: These options have the same effect as Automatic Method of Dimension Details (Dimension Tool Settings).

See [Dimension Details Panel](#).

4. Drag the cursor to draw a temporary line across the selected elements to be dimensioned.
 - Walls, Columns, Beams, and Slab, Roof and Mesh edges perpendicular to or crossing the drawn line will be dimensioned. The line can consist of several segments.

- 5. Double-click to finish drawing the line.
- 6. Click to place the dimension chain.



Secondary Dimensions (Add-on)

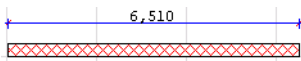
The **Secondary Dimensions** function allows you to add metric unit equivalents to dimension values displayed in feet and inches, and vice versa. The Secondary Dimensions menu is located at **Document > Annotation**.

Note: If your program does not display the Secondary Dimensions menu, you can add it to your work environment.

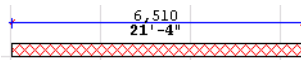
For more information, see [Menu Customization Dialog Box](#).

Use the **Document > Annotation > Secondary Dimensions > Convert Dimensions** dialog box to set the unit and the number of decimals of the converted dimension value.

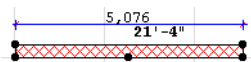
Click **OK**, and the secondary values will appear on the opposite side of the dimension line.



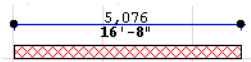
The new values will be placed as a text block, and can be edited as any other text block.



- If nothing is selected on the Floor Plan, all placed dimensions will be converted.



- If there are selected dimensions, only their values will be converted.



Changes made to dimensioned elements do not automatically update the converted dimensions.

Choose **Document > Annotation > Secondary Dimensions > Refresh converted dimensions** to update both the value and the location of the dimension. Again, this command will work on either all dimensions (if nothing is selected) or selected ones.

If you do not need equivalents any more, simply choose **Remove converted dimensions** from the Secondary Dimensions menu.

For more information, see [Convert Dimensions Dialog Box \(Add-on\)](#).

The Grid Tool

The Grid Tool allows you to place Grid elements individually or in a Grid System; it is a tool in the Archicad Toolbox.



Grid elements are part of the model and serve as an editable framework for placing multiple elements or for identifying locations in the model.

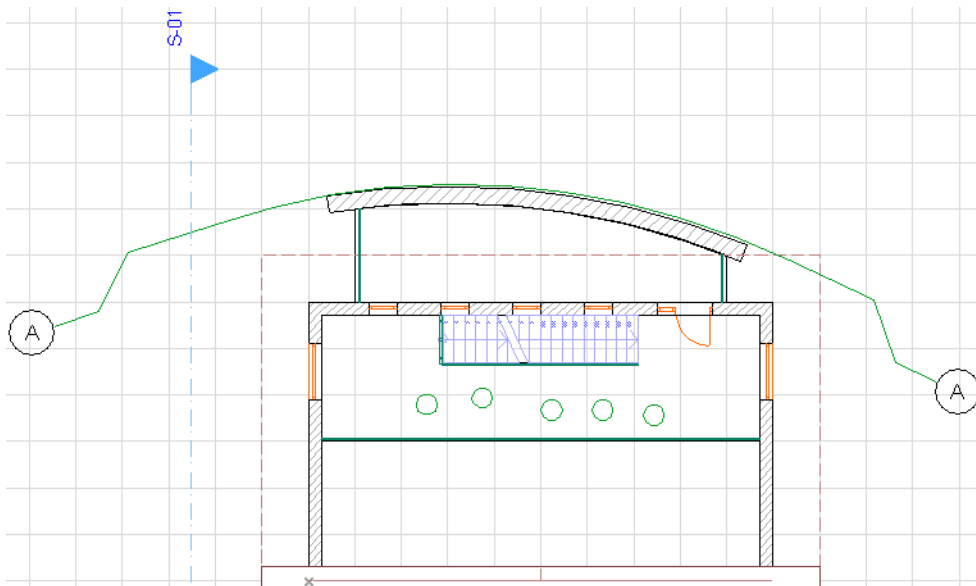
The Grid Tool creates a custom local structural grid. (In contrast, the Construction and Snap Grid set in the **View > Grid Options > Grids and Background** is a global, equally distributed net covering the whole 2D window, and is used as an input and editing aid.)

The grid created by this tool is interactive, so you can edit grid elements even after placing them. Grid elements can be edited using Grid Settings.

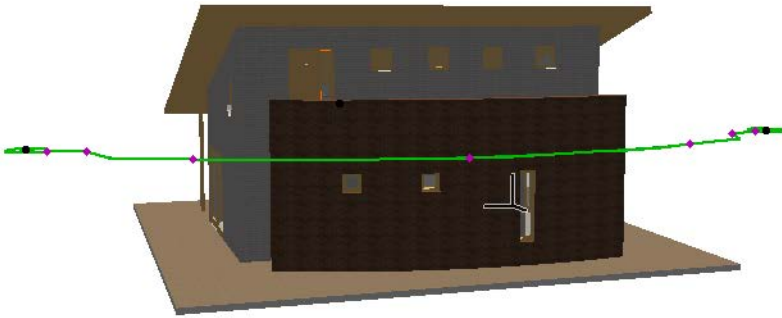
[See Grid Tool Settings.](#)

Grid elements can be placed in either the Floor Plan or the 3D window, but they can be displayed in Sections, Elevations, Interior Elevations and the 3D Document as well.

A Grid element marks a vertical location in the model. Like other model elements, Grid elements are saved along with views.



Grid element on the Floor Plan

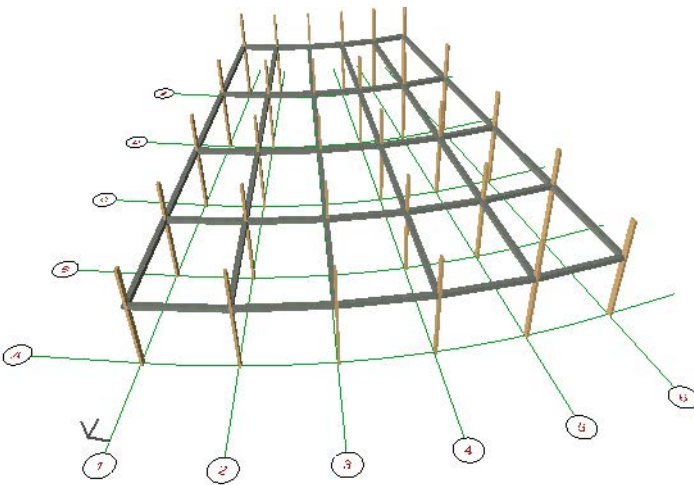


Grid element in the 3D window

To place a whole scheme of horizontal and vertical gridlines in one step, use the **Design > Grid System** command. The follow-up Grid System Settings dialog allows you to configure the system of Grid Elements and to place dimensions, beams, columns or optional library parts at the grid intersection points.

[See Grid System Settings.](#)

This image shows a Grid System in 3D with Beams and Columns placed at intersections:



Related Topics:

[Components of a Grid Element](#)

[Visibility of a Grid Element](#)

[Creating a Straight Grid Element](#)

[Creating a Curved Grid Element](#)

[Editing a Grid Element](#)

[Editing Grids in Teamwork](#)

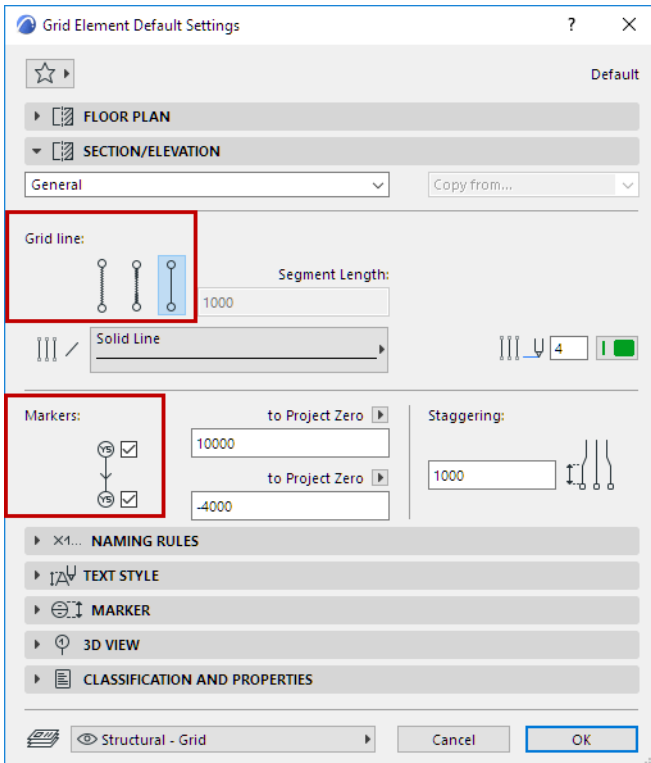
[Place a Grid System](#)

Components of a Grid Element

A Grid Element has two main components: Grid Marker and Grid Line.

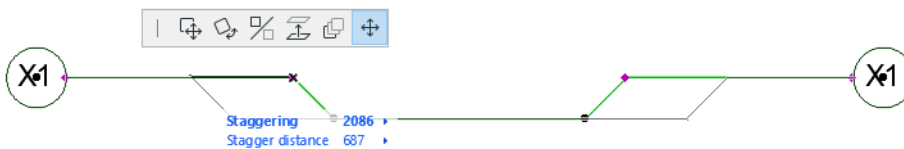
The Grid line can be displayed in three different formats to suit your documentation needs.

Grid Markers can be switched on and off optionally at both ends of the Grid Line, in the Grid Settings Floor Plan panel.



You can stagger grid markers away from the Grid line graphically, using a pet palette command.

See [Editing a Grid Element](#).



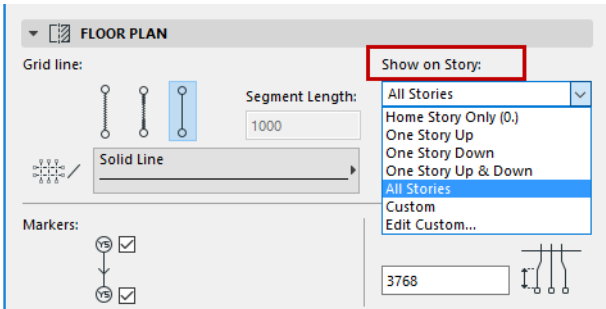
When you stagger the grid markers sideways, a certain line segment will be staggered together with the markers. The length of this line segment is the **Staggering** value.

The default Staggering length for any Grid element can be set in both the Floor Plan and Section/Elevation panels of Grid Tool Settings.

Visibility of a Grid Element

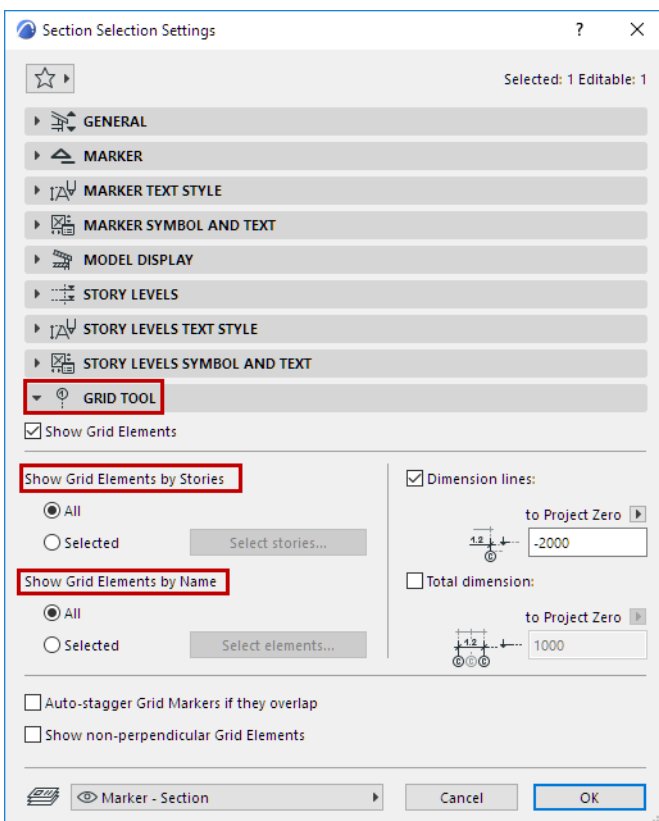
Provided that their layer is visible, Grid elements are visible on:

- the Floor Plan, on any or all stories: Use the Show on Story pop-up in the Grid Settings Floor Plan panel.

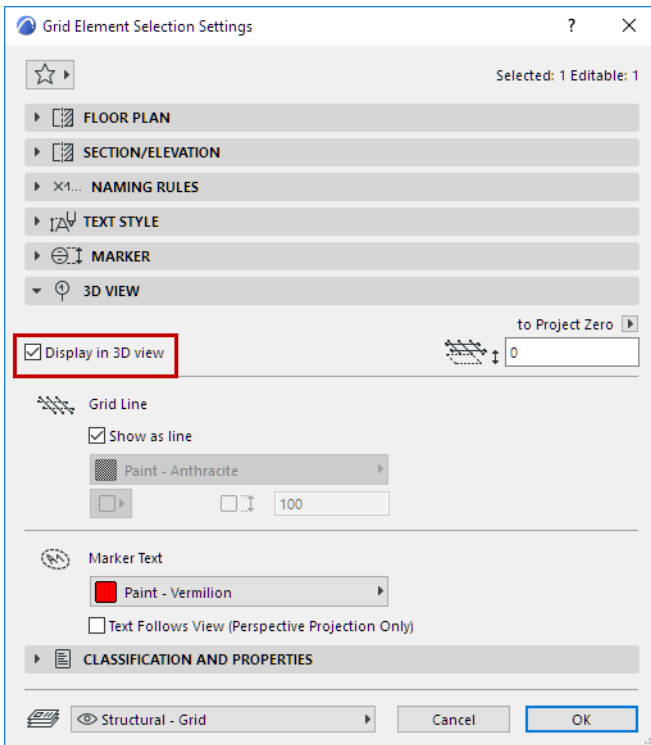


- the Section/Elevation windows, if the Section or Elevation settings allow: Use the **Show Grid Elements** checkbox on the Grid Tool Panel of Section Settings.

There are two sets of filtering criteria for display of Grid elements on this Section: You can filter by Grid element Story, and/or you can filter by Grid element Name.

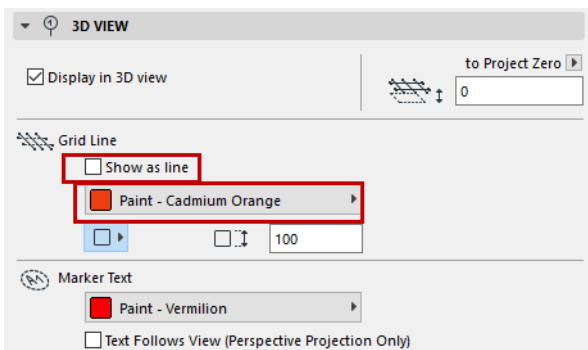


- 3D window and 3D Documents, if the Grid element is set to appear in 3D: Use the **Display in 3D view** checkbox in the 3D View panel of Grid Tool Settings.



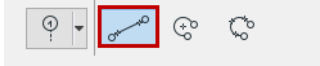
Detail and Worksheet windows will not display the grid elements, just their exploded views.

By default, Grid elements are displayed in 3D as lines only, and are not displayed in renderings. However, you can choose to display Grid elements as model elements in 3D so that they appear in renderings, with a cross-section and surfaces: to do so, adjust the options in the 3D View panel of Grid Settings.



Creating a Straight Grid Element

Grid elements can be created in the Floor Plan or 3D window. The Straight Grid Element method produces one straight Grid Element at a time.



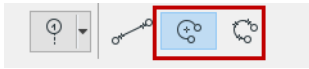
After setting up the element settings or choosing a Favorite, select the Grid Tool. Choose the Straight Grid Element geometry method from the Info Box. Click to place the endpoints and to define the length of Grid Element.

The moment you start drawing the Grid Element, a ghost contour appears and it follows the cursor until you click at the second endpoint.



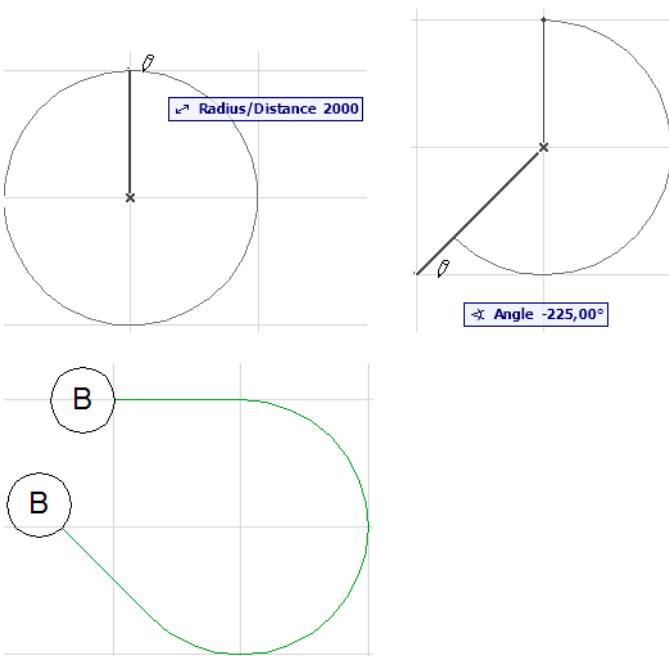
Creating a Curved Grid Element

After setting up the element settings or choosing a Favorite, select the Grid Tool. Choose one of the two Curved Grid Element Geometry methods from the Info Box.

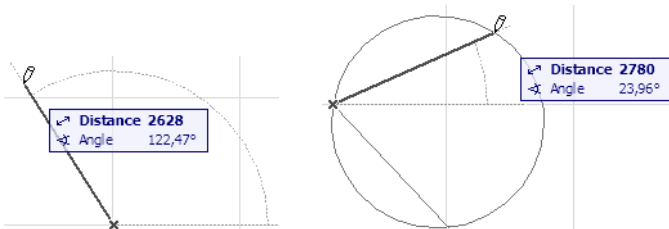


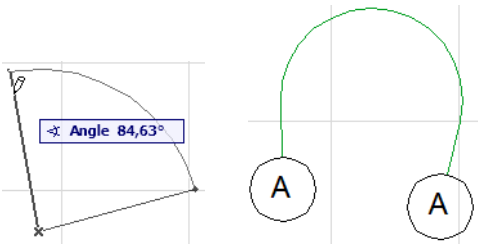
The first option defines the curved Grid by the arc's centerpoint and radius. Your first click defines the centerpoint.

A ghost contour of the circular grid follows your cursor until you click a second time to define the radius. After that, only a partial ghost curved segment is shown, until you click a third time to define the length of the arc grid's circumference. When all three points are defined, the ghost contour is replaced by the fully displayed Grid Element.



The second option defines the curved Grid by three points on the arc's circumference. You will probably place first two points so that they connect other elements, grid intersections or special snap points. Next, a ghost circle follows the cursor until you click a third time to define the third point.



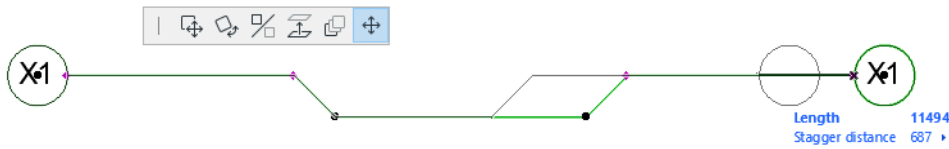


A fourth click is needed to define the length of the arc segment. The ghost arc is now replaced by the fully displayed Grid Element.

Editing a Grid Element

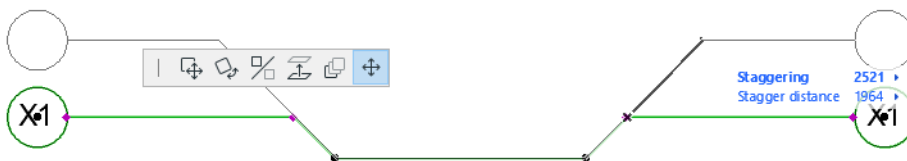
Edit Length of Grid Line

Any Grid elements can be stretched with Pet palette commands: select the **Grid Element**, then choose the **Stretch** icon.



Stagger Markers Away from Grid Line

Select a Grid element. Click on the editing hotspot which represents the staggered part of the Grid marker, and then drag the Marker, together with the staggering length, to a new position.



Edit Grid Position

The horizontal position of a grid element can be edited on the Floor plan and 3D window only. Select the Grid Element and use the familiar pet palette commands and shortcuts to **Drag**, **Mirror** or **Rotate the Grid element**, or copies of it.



The vertical position (Z-value) of a grid element can be edited

- in the 3D window
- at the 3D View Panel of Grid Settings, or
- in the Info box.

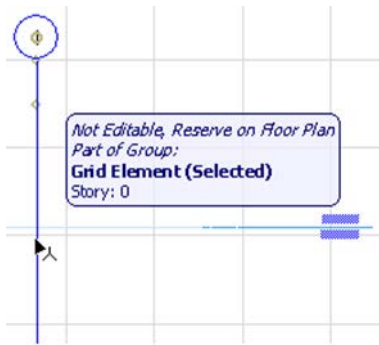
Editing Grids in Teamwork

Note: The word “Section” in this document is understood to include Sections, Elevations and Interior Elevations.

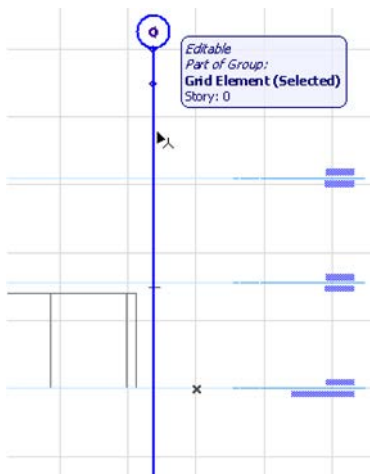
When editing Grids in the Section window in Teamwork, certain procedures will require an extra reservation. Context-based pop-up bubbles will inform you of what to do as you work. The extra reservation is required in three cases, detailed below.

Editing Grids in Section

You can reserve a Grid element in Section with the usual reservation methods. However, even after reserving the Grid element, you cannot edit it until you also reserve it on the Floor Plan. The Info Tag notifies you of this if you try to edit a reserved Grid element in Section:



You must go to the Floor Plan and reserve the Grid element in question. Now you may edit it in Section as usual.



Note: If you reserve and edit a Grid Element on the Floor Plan, the Section will be rebuilt as expected, including the Grid elements; no additional steps are necessary.

Place a Grid System

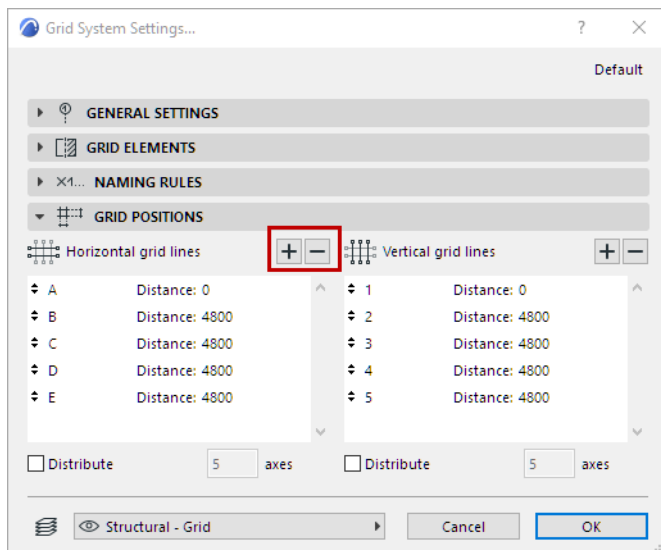
To lay out a complete orthogonal or curved grid mesh, use the Grid System function. The **Grid System** command can save you a lots of time by automating the task of placing multiple Grid Elements in one step, following a defined scheme, and even placing elements on dedicated positions. You can also place dimensions throughout the Grid system in one step.

A Grid System is a collection of Grid Elements; each Grid element continues to act as a distinct element, even though they are placed as part of a system.

Select **Design > Grid System**. Adjust the settings.

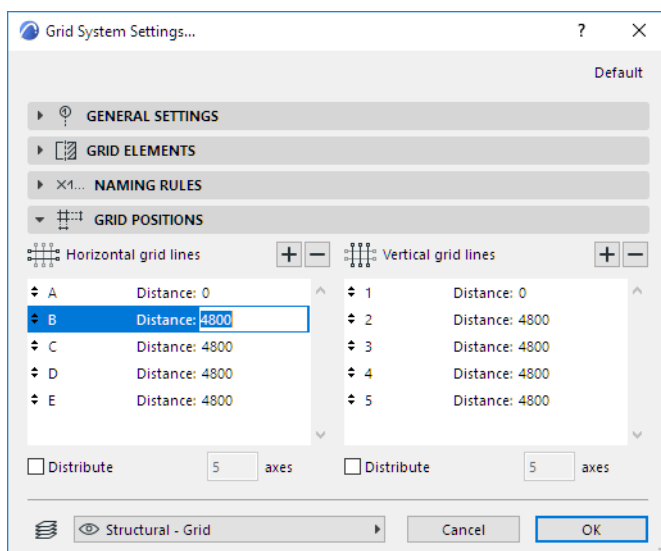
See [Grid System Settings](#).

The number of horizontal and vertical grid lines to be placed in this Grid System depends on what you set in the Grid Positions panel: click the plus and minus signs to add or remove individual Grid lines from the Grid System.



The number of Grid lines is shown in the “axes” field below.

Click in the Distance field of any Grid line to define its distance from the previous Grid line:

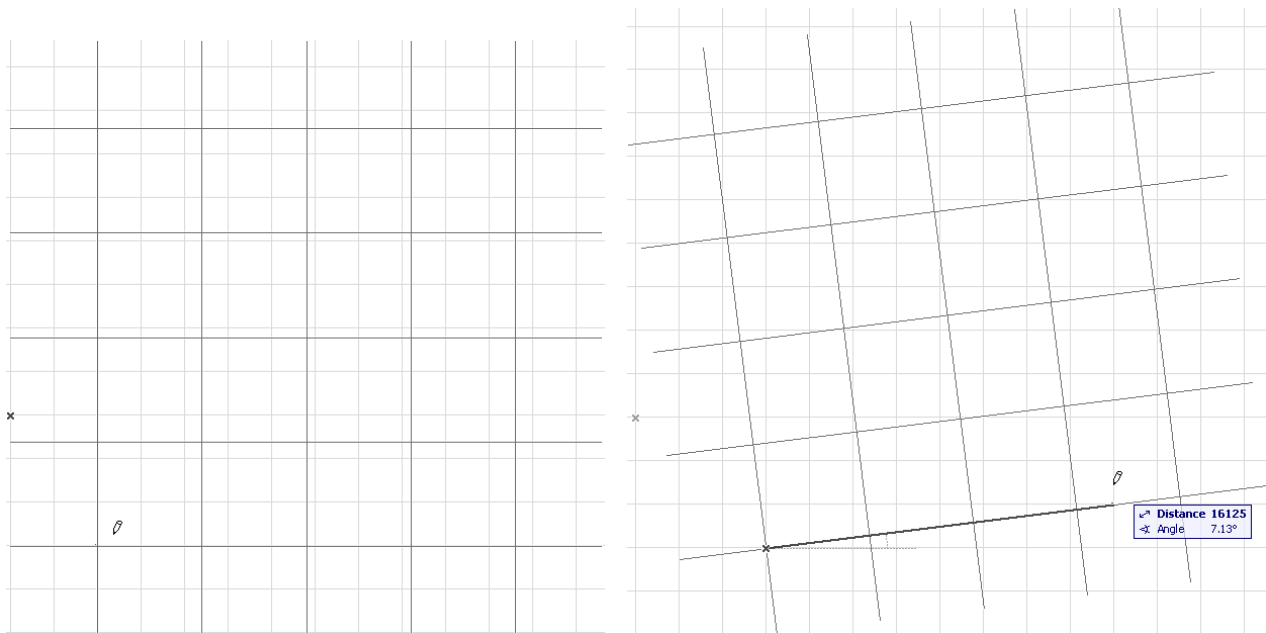


If you do not know the distance you will need between your Grid lines, check the **Distribute** box for either or both Grid line directions (i.e. horizontal and/or vertical grid lines). This will enable you, when you place the Grid system, to set the beginning and endpoints of the Grid system in the given direction, divided into the number of Grid lines defined in the list.

When you are satisfied with Grid System Settings, click OK.

You will see a ghost grid system at the cursor, which you can place with two to four clicks:

- the first click defines the location of the anchor point you selected in the settings
- the second click defines the rotation of the grid system
- if you selected the Distribute function as well, then one or two additional clicks are required to define the full distances for distribution in one or both directions, depending whether you checked Distribute for one or both directions.



Note: You can select and edit all or selected elements of a placed grid system by selecting them using any of the Archicad selection methods.

Place Object, Column or Beams with Grid System

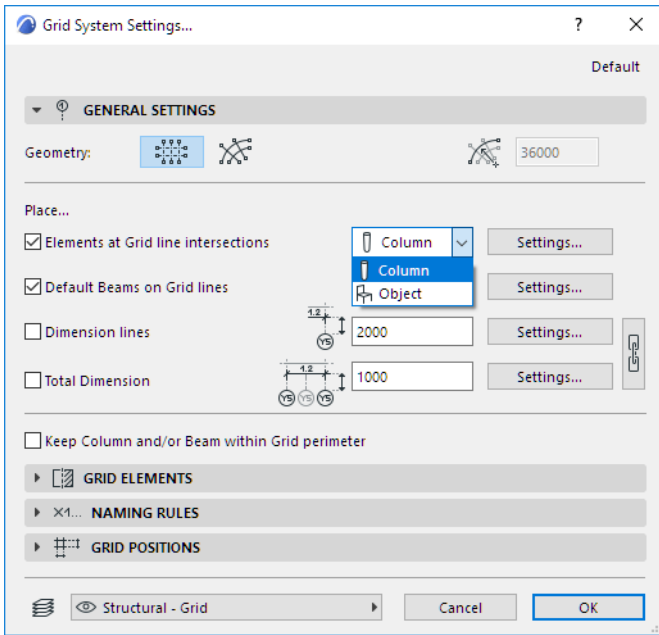
Use the General Settings panel of Grid System Settings to place an Object or Column automatically at every Grid intersection point; and/or to place a Beam along every Grid line of the Grid System.

Note: If you are placing a curved grid system, you have the additional option to place either straight or curved beams along the gridlines.

The Settings button next to each checkbox will take you to the relevant Tool Settings dialog box (Column, Object, Beam).

For details on these controls, see [Grid System General Settings Panel](#).

Although these checkboxes mean that Beams, Columns or objects are placed at each Grid intersection, these elements are independent of the Grid and will not be moved or rotated along with the Grid System.



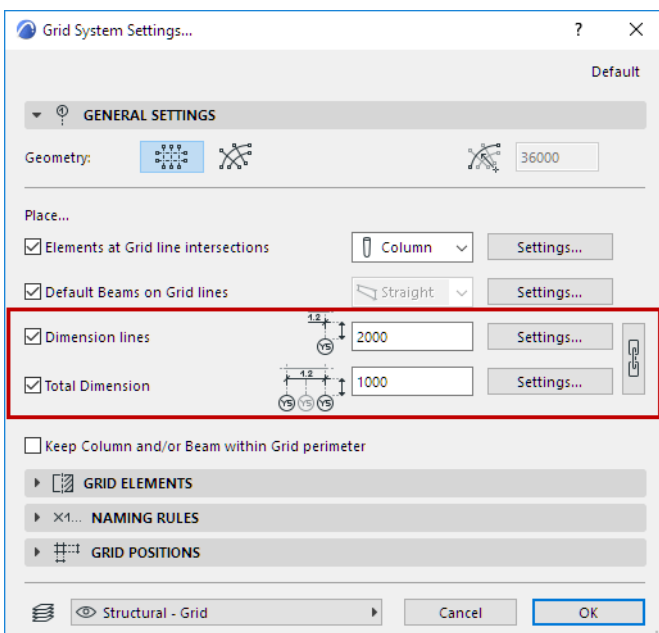
[Watch the Video](#)

Dimension Grid System

Two additional checkboxes in the General Settings panel of Grid System Settings enable you to automatically dimension each Grid line segment in the system, and/or to place a single total dimension for each Grid line. These dimensions are associated to the Grid system.

The Settings button to the right of the Dimension checkboxes take you to Dimension Default Settings, where you can set the options for the chosen Dimension type for your Grid System. For example, you can set all of the individual Grid line dimension text to be displayed in italic, while the Total Dimension values are displayed in bold type.

Note that if you would like to use uniform Dimension settings for both dimension lines, then you can use the Chain button to link them together, in the General Settings panel of Grid System Settings.



Text Blocks



Using the **Text** tool, you can create multi line texts with full-scale font options, multiple styles and alignment in any direction. Text blocks can be created in the Floor Plan, in Section/Elevation/IE and 3D Documents, and in Detail and Worksheet windows, and on Layouts. You can format the content of text blocks as a whole (using Tool Settings or Info Box), and you can also individually format characters within the text block (using the Text Editor).

Favorites of the Text tool can be applied to the text-type items in Dimensions and Labels, and to Fill Text.

Text blocks are always readable from left to right even after a mirroring.

The advantages of freely editable text elements are available for the Label tool as well as the Text tool. (Rich text functions are not available for other textual items in Archicad.)

Depending on the state of the Textbox Handles toggle in **View > On-Screen View Options**, all text blocks can be framed with brackets at each corner handle.

This makes text blocks easy to locate and edit, while keeping the text readable.

Favorites of the Text tool can be applied to the text-type items in Dimensions and Labels, and to Fill Text.

Related Topics:

[Placing Text Blocks](#)

[Formatting Text Blocks As a Whole](#)

[Resizing Text Graphically](#)

[Applying Favorite Text Settings](#)

[Formatting Individual Components of Text Blocks](#)

[Formatting Palette and Text Editor Controls](#)

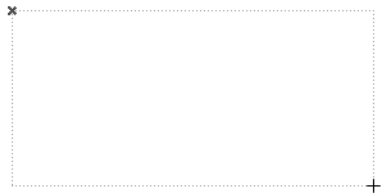
Placing Text Blocks

Before starting to type into a text block, you can determine whether it will be a “breaking” or “non-breaking” text block.

- A **breaking** text block’s width remains fixed, with the line of text “breaking” automatically to the next line when it reaches the limit during input.
- A **non-breaking** text block’s width depends on how much text you enter. Use a non-breaking text block if you are not sure in advance how long the text will be.

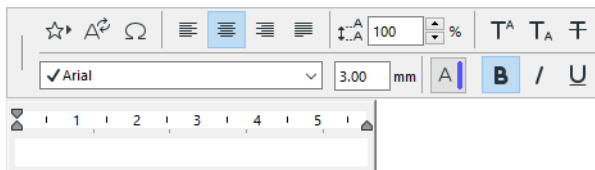
To place a breaking text block:

1. Choose the Text tool.
2. Draw a rectangle to define the text block width.



A **Text Editor** window and a **Formatting Palette** appear on the screen.

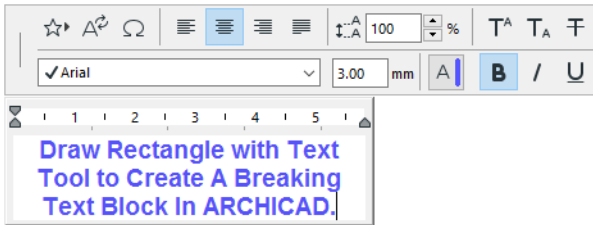
For more information, see [Formatting Palette and Text Editor Controls](#).



3. Type the desired text into the Text Editor.

Note: The style and formatting of a newly placed Text Block depend on the settings in the Text Settings dialog box and Info Box. If you want different style or formatting for particular characters or paragraphs you are typing, use the Formatting Palette.

- In addition to typing in text, you can use the controls of the Text Editor to add the following predefined items to the text block
 - Autotext
 - Favorites
 - Symbols: Click this button to access a Character Map (Character Palette on Mac) to choose and insert a symbol.
- Once you reach the end of the text block, additional text will be automatically entered on the next line.



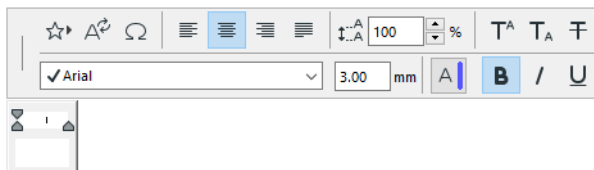
- You can start a new line at any time by hitting the Return key on your keyboard.
- Click **Cancel** (Control Box) to cancel the text editing operation.

4. To complete the Text box, do one of the following

- click on an empty space in the window
- click **OK** in the Control Box
- press Ctrl-Enter (Cmd-Enter).

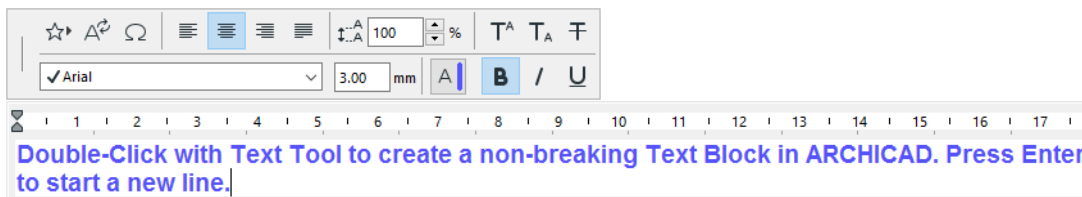
To place a non-breaking text block:

- 1.** Choose the Text tool.
- 2.** Double-click in the view. An editing window and a formatting palette appear on the screen.



3. Type the desired text into the Text Editor.

- The text block will extend indefinitely as long as you keep typing.
- Start a new line by pressing Enter.



–

4. To complete the Text box, do one of the following

- click on an empty space in the window
- click **OK** in the Control Box
- press Ctrl-Enter (Cmd-Enter).

Formatting Text Blocks As a Whole

Use Text Settings and the Info box to format the Text Block as a whole.

See [Text Tool Settings](#).

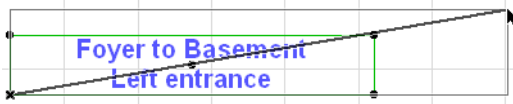
To open Text Tool Settings: Double-click the **Text** tool. The dialog box has two panels:

- [Text Style](#): These settings apply to the appearance of your text characters.
- [Text Block Formatting Panel](#): These settings apply to the shape and appearance of the text block.

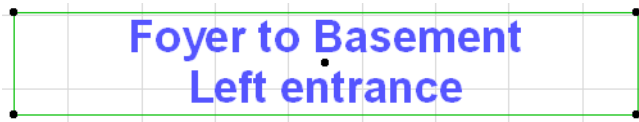
To **stretch a Text Block**, select it and reshape it using any of its corner nodes. A rubber band box provides feedback as you stretch; the text in the block will automatically rearrange itself according to the new outline.

Resizing Text Graphically

You can change the size of the text inside the text block using a Stretch technique. Select the text block and stretch it with either the **Stretch** menu command or the pet palette. Move the cursor along the text block's diagonal, and press the **Shift** key to constrain it to the diagonal.



With Shift still pressed, click to define the new size. The font size will grow or shrink proportionally.



Applying Favorite Text Settings

To apply Favorite settings to the entire text block:

Select the text block, then do one of the following:

- Switch to a Favorite in Text Settings, and click OK
- Double-click the desired Favorite from the Favorites palette

Note: Favorites of the Text tool can be applied to the text-type items in the following elements:

- Dimensions
- Labels
- Fill Text
- Section/Elevation, Interior Elevation
- Detail, Worksheet
- Story Level Markers
- Grid
- Door/Window Markers
- Drawing Title
- Zone Stamp

See also [Favorites](#).

Formatting Individual Components of Text Blocks

To format individual characters, lines and paragraphs that have been typed into your text block, or to switch styles while typing, use the **Formatting Palette** and the indent/tab controls of the **Text Editor**.

The Text Editor and Formatting Palette appear as soon as you click on the Floor Plan (with the Text tool active) and begin typing into the text box.

The controls of the Formatting Palette and Text Editor apply only to the text you are entering in the Text Editor, or to selections inside the Text Editor.

See [Formatting Palette and Text Editor Controls](#).

Note: Favorites of the Text tool can be applied to the text-type items in Dimensions and Labels, and to Fill Text.

Autotext

An **Autotext** is a text element containing a definition filled in automatically by the program. After you insert an Autotext, the program will fill in the data that apply in the current context. This data - which can be text, a numerical value, or element-specific parameters - is automatically updated to follow any changes in the model, layout or drawing.

You can add Autotext:

- in any window where the Text tool is active
- as part of the Text/Autotext Label
- in GDL objects, including Markers
- in the folder/item name fields of Publisher items
- as part of the ID of Interior Elevation viewpoints

The style of the Autotext can be formatted the same way as for any manually typed text.

Related Topics:

[How to Insert Autotext](#)

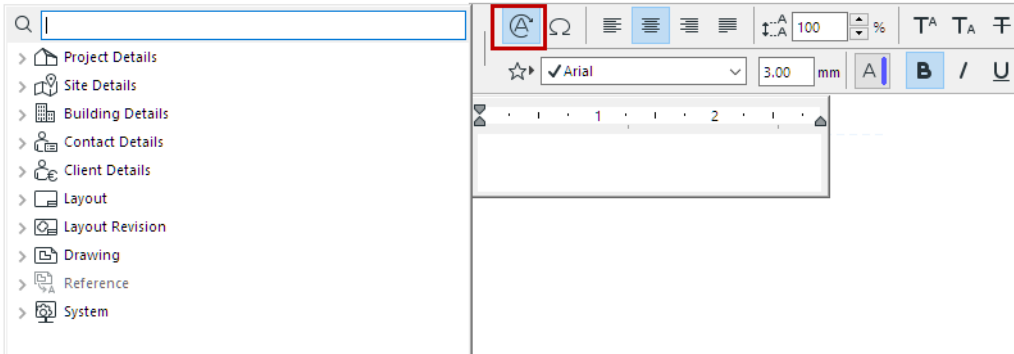
[Autotext Reference Drawing](#)

[Autotext Keywords](#)

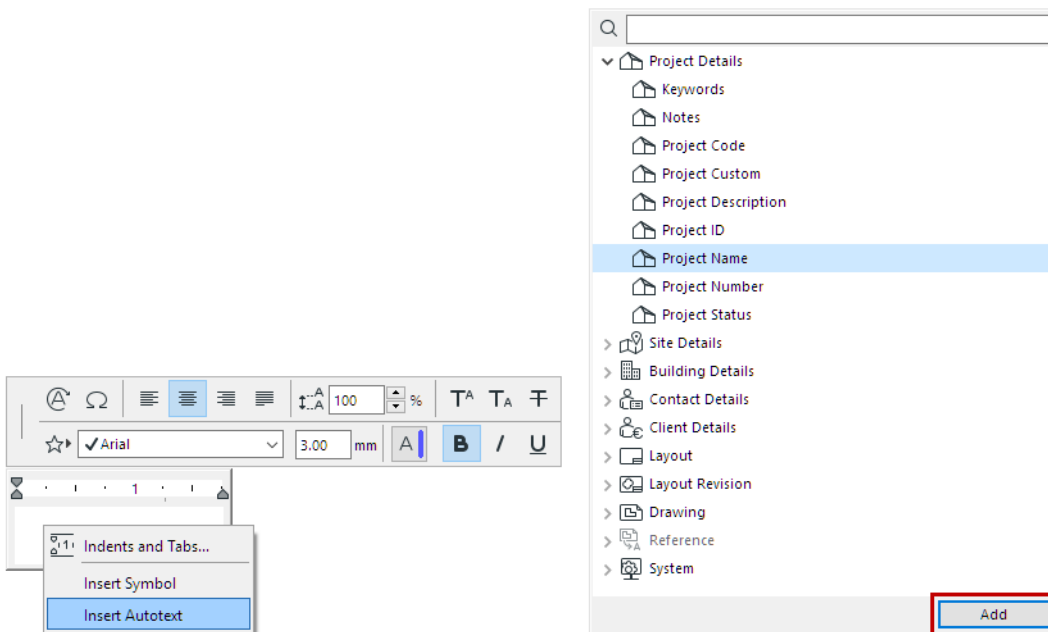
[Text/Autotext Label Content](#)

How to Insert Autotext

1. Activate the **Text** tool
2. Click on screen. The Formatting Palette and Text Editor appear.
3. Do one of the following:
 - Click the **Insert Autotext** button in the Formatting Palette

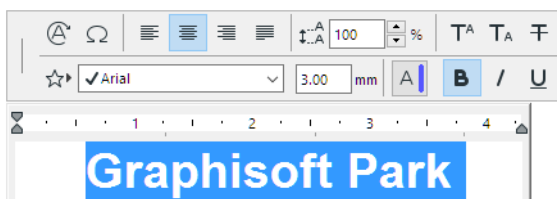


- Right-click at the insertion point in the Text Editor and choose the appropriate command from the **Insert Autotext** hierarchical menu:

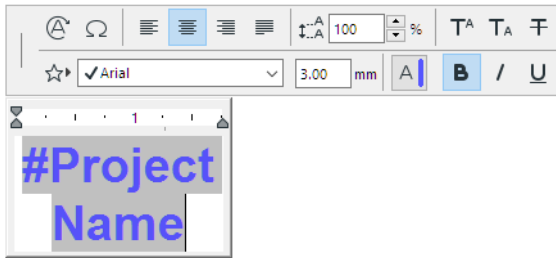


4. Choose one or more Autotext items. Double-click, or click **Add** to add them to the Text Editor.

If the information is already available, the appropriate text will appear in the Text Editor after you have selected it.



If the referenced information is not available, you will see a placeholder text.



When the missing information becomes available, the placeholder text is immediately updated with the relevant data.

You can add any number of Autotexts, and type additional static text, in the same text block.

The current Text Settings will be applied. To format individual characters or parts of text within a text block, use the controls of the Formatting Palette and Text Editor.

[See *Formatting Palette and Text Editor Controls*.](#)

Autotext Sources

There are several sources of Autotext:

Project-specific information such as the name of the project (not necessarily the same as the file name), the name of the architect, the date of Transmittal Set or any other piece of information entered in the **File > Info > Project Info** dialog box.

[See also *Project Info*.](#)

- Layout-specific information such as the name or ID of the layout, the name or ID of the subset, the number of layouts in the layout book.
- Drawing-specific information including the name, drawing scale and magnification factor of the drawing.
- Information that comes from a specified drawing (one which you define as an Autotext Reference Drawing).

[See also *Autotext Reference Drawing*.](#)

- System-dependent information (file name, path, date of creation and modification, etc.).

Adding Autotext to GDL Objects

You can use Autotext when scripting GDL objects.

For example, if you use a GDL object-type Title block placed on a Master Layout, you can script the Title block to show the Project Name.

In a GDL script window, the Text Editor interface (with its lists of Autotext entries) is not available, so you must use the corresponding formatting codes.

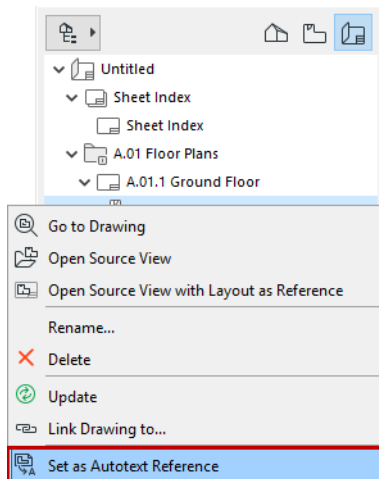
[For a list of Autotext keywords for use in GDL scripts, see *Autotext Keywords*.](#)

Autotext Reference Drawing

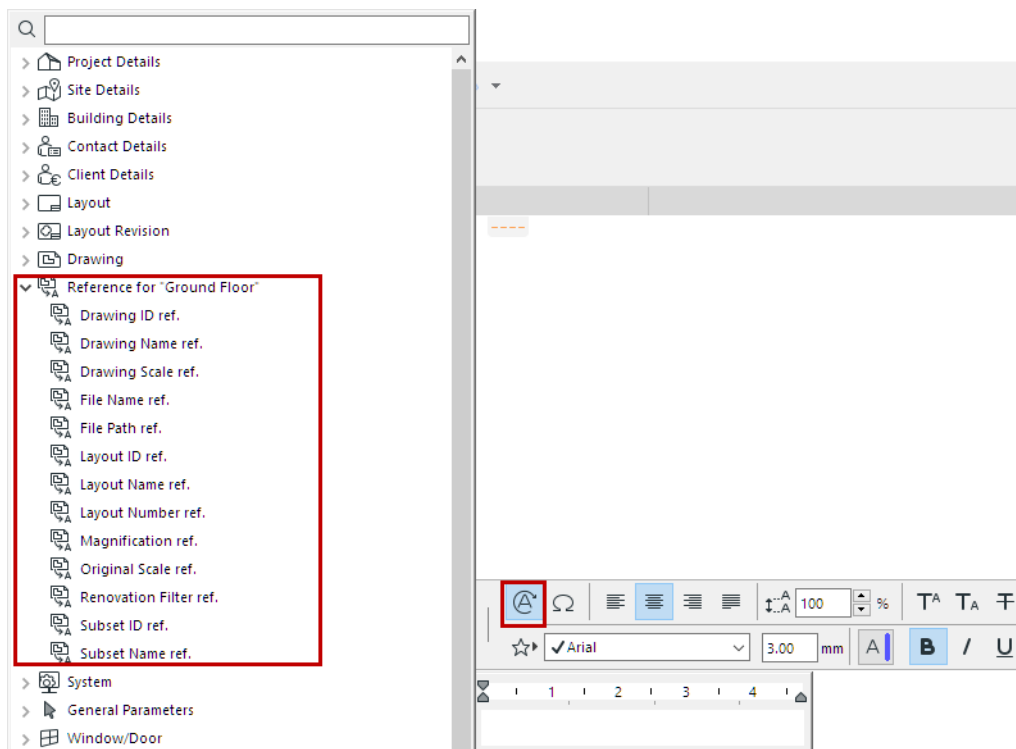
Archicad lets you set a particular drawing as an Autotext reference. This means that your Autotext does not necessarily have to refer to the view or layout onto which you are placing the text.

To set a drawing as Autotext reference:

1. Select a drawing in the Navigator or Organizer or on the layout.
2. Right-click to display a context menu and choose the **Set as Autotext Reference** command. (The same command is also available from the Drawing Manager.)



Once defined as an Autotext Reference, this drawing will appear as a choice in the Insert Autotext list.



[See How to Insert Autotext.](#)

The Autotext entries that use the Reference as source will then show values based on the reference drawing. If you later define a different drawing as Autotext Reference, earlier Autotexts referring to the original drawing will remain unchanged.

Autotext Keywords

The following **Autotext Keywords** are available when using the **Autotext** commands.

Use the keywords in the right-hand column to insert Autotext items into GDL scripts.

To show this Autotext:	Type this Keyword
Project Details	
Project Name	<PROJECTNAME>
Project Description	<PROJECT_DESCRIPTION>
Project ID	<PROJECT_ID>
Project Code	<PROJECT_CODE>
Project Number	<PROJECTNUMBER>
Project Status	<PROJECTSTATUS>
Keywords	<KEYWORDS>
Notes	<NOTES>
Project Custom	created dynamically
Site Details	
Site Name	<SITE_NAME>
Site Description	<SITE_DESCRIPTION>
Site ID	<SITE_ID>
Site Full Address	<SITEFULLADDRESS>
Site Address1	<SITEADDRESS1>
Site Address2	<SITEADDRESS2>
Site Address3	<SITEADDRESS3>
Site Postal Box	<SITEPOSTBOX>
Site City	<SITECITY>
Site State	<SITESTATE>
Site Postcode	<SITEPOSTCODE>
Site Country	<SITECOUNTRY>
Site Gross Perimeter	<SITEGROSSPERIMETER>
Site Gross Area	<SITEGROSSAREA>
Site Custom	created dynamically
Building Details	
Building Name	<BUILDING_NAME>
Building Description	<BUILDING_DESCRIPTION>
Building ID	<BUILDING_ID>
Building Custom	created dynamically

To show this Autotext:	Type this Keyword
Contact Details	
Contact Full Name	<CONTACT_FULLNAME>
Contact Prefix Title	<CONTACT_PREFIXTITLE>
Contact Given Name	<CONTACT_GIVENNAME>
Contact Middle Name	<CONTACT_MIDDLENAME>
Contact Family Name	<CONTACT_FAMILYNAME>
Contact Suffix Title	<CONTACT_SUFFIXTITLE>
Contact ID	<CONTACT_ID>
Contact Role	<CONTACT_ROLE>
Contact Department	<CONTACT_DEPARTMENT>
CAD Technician Full Name	<CAD_TECHNICIAN_FULLNAME>
CAD Technician Prefix Title	<CAD_TECHNICIAN_PREFIXTITLE>
CAD Technician Given Name	<CAD_TECHNICIAN_GIVENNAME>
CAD Technician Middle Name	<CAD_TECHNICIAN_MIDDLENAME>
CAD Technician Family Name	<CAD_TECHNICIAN_FAMILYNAME>
CAD Technician Suffix Title	<CAD_TECHNICIAN_SUFFIXTITLE>
Contact Company	<CONTACTCOMPANY>
Contact Company Code	<CONTACT_COMPANYCODE>
Contact Full Address	<CONTACTFULLADDRESS>
Contact Address1	<CONTACTADDRESS1>
Contact Address2	<CONTACTADDRESS2>
Contact Address3	<CONTACTADDRESS3>
Contact Postal Box	<CONTACTPOSTBOX>
Contact City	<CONTACTCITY>
Contact State	<CONTACTSTATE>
Contact Postcode	<CONTACTPOSTCODE>
Contact Country	<CONTACTCOUNTRY>
Contact E-mail	<CONTACTEMAIL>
Contact Phone Number	<CONTACTPHONE>
Contact Fax	<CONTACTFAX>
Contact Web	<CONTACTWEB>
Contact Custom	created dynamically
Client Details	
Client Full Name	<CLIENT_FULLNAME>
Client Prefix Title	<CLIENT_PREFIXTITLE>
Client Given Name	<CLIENT_GIVENNAME>

To show this Autotext:	Type this Keyword
Client Middle Name	<CLIENT_MIDDLENAME>
Client Family Name	<CLIENT_FAMILYNAME>
Client Suffix Title	<CLIENT_SUFFIXITITLE>
Client Company	<CLIENTCOMPANY>
Client Full Address	<CLIENTFULLADDRESS>
Client Address 1	<CLIENTADDRESS1>
Client Address 2	<CLIENTADDRESS2>
Client Address 3	<CLIENTADDRESS3>
Client Postal Box	<CLIENTPOSTBOX>
Client City	<CLIENTCITY>
Client State	<CLIENTSTATE>
Client Postcode	<CLIENTPOSTCODE>
Client Country	<CLIENTCOUNTRY>
Client E-mail	<CLIENTEMAIL>
Client Phone Number	<CLIENTPHONE>
Client Fax	<CLIENTFAX>
Client Custom	created dynamically
Layout	
Layout ID	<LAYOUTID>
Layout Name	<LAYOUTNAME>
Layout Number	<LAYOUTNUMBER>
Layout Number in Current Subset	<LAYOUTNUMBERINCURRENTSUBSET>
Layout Number in Current Transmittal Set	<LAYOUTNUMBERINCURRENTISSUE>
Layout Number in Subset	When inserting Autotext: choose which subset. No keyword available
Layout Number in Top Subset	<LAYOUTNUMBERINTOPSUBSET>
Master Layout - Height	<MASTERLAYOUTHEIGHT> Value based on Working Units for Layout (Options > Project Preferences).
Master Layout - Name	<MASTERLAYOUTNAME>
Master Layout - Width	<MASTERLAYOUTWIDTH> Value based on Working Units for Layout (Options > Project Preferences).
Number of Layouts	<NUMOFLAYOUTS>
Number of Layouts in Current Subset	<NUMBEROFLAYOUTSINCURRENTSUBSET>

To show this Autotext:	Type this Keyword
Number of Layouts in Current Transmittal Set	<NUMBEROFLAYOUTSINCURRENTISSUE>
Number of Layouts in Subset	When inserting Autotext: choose which subset. No keyword available.
Number of Layouts in Top Subset	<NUMBEROFLAYOUTSINTOPSUBSET>
Subset ID	<SUBSETID>
Subset Name	<SUBSETNAME>
Layout Revision	
Current Revision ID	<CURRENTREVISIONID>
Current Revision Transmitted By	<CURRENTTRANSMITTEDBY>
Current Revision's Transmittal Set Date	<CURRENTTRANSMITTALSETDATE>
Current Revision's Transmittal Set ID	<CURRENTTRANSMITTALSETID>
Current Revision's Transmittal Set Name	<CURRENTTRANSMITTALSETDESCRIPTION>
Drawing	
Drawing ID	<DRAWINGID>
Drawing Name	<DRAWINGNAME>
Drawing Scale	<DRAWINGSCALE>
Magnification	<MAGNIFICATION>
Original Scale	<ORIGINALSCALE>
Renovation Filter	<RENOVATIONFILTER>
Reference	Available if a Drawing has been defined as the Autotext Reference Drawing. See Autotext Reference Drawing.
Drawing ID ref.	<DRAWINGID_R>
Drawing Name ref.	<DRAWINGNAME_R>
Drawing Scale ref.	<DRAWINGSCALE_R>
File Name ref.	<FILENAME_R>
File Path ref.	<FILEPATH_R>
Layout ID ref.	<LAYOUTID_R>
Layout Name ref.	<LAYOUTNAME_R>
Layout Number ref.	<LAYOUTNUMBER_R>
Magnification ref.	<MAGNIFICATION_R>
Original Scale ref.	<ORIGINALSCALE_R>
Renovation Filter ref.	<RENOVATIONFILTER_R>
Subset ID ref.	<SUBSETID_R>
Subset Name ref.	<SUBSETNAME_R>
System autotexts	Note: File-dependent system autotexts will be undefined until you save your project file at least once.

To show this Autotext:	Type this Keyword
Date (Long)	<LONGDATE>
Date (Short)	<SHORTDATE>
Day of Month	<DAYOFMONTH>
Day of Week	<DAYOFWEEK>
File Name	<FILENAME>
File Path	<FILEPATH>
Last saved at	<LASTSAVEDAT>
Last saved by	<LASTSAVEDBY>
Month (name)	<MONTHNAME>
Month (number)	<MONTHNUMBER>
Time	<TIME>
Year (short)	<YEARSHORT>
Year (long)	<YEARLONG>
Other Autotexts	These are available in the Rename Publisher Items dialog box only. See <i>Rename Publisher Items</i> .
Folder Name	<FolderName> (Renames the item using the folder name in which it is located)
Folder Structure Name	<FolderStructureName> (Renames the item using the entire folder path of the item's location in the Publisher Set)
Publisher Set Name	<PublisherSetName>
Source Item ID	<SourceID> (Renames the item using View ID or Layout ID, depending on the source of the Publisher item.)
Source Item Name	<SourceItemName> (Renames the item using View Name or Layout Name, depending on the source of the Publisher item.)

Note: Your **Drawing Scale**, if you customize it in Drawing Settings, is distinct from its **Original Scale** (the scale of the Drawing's source view).

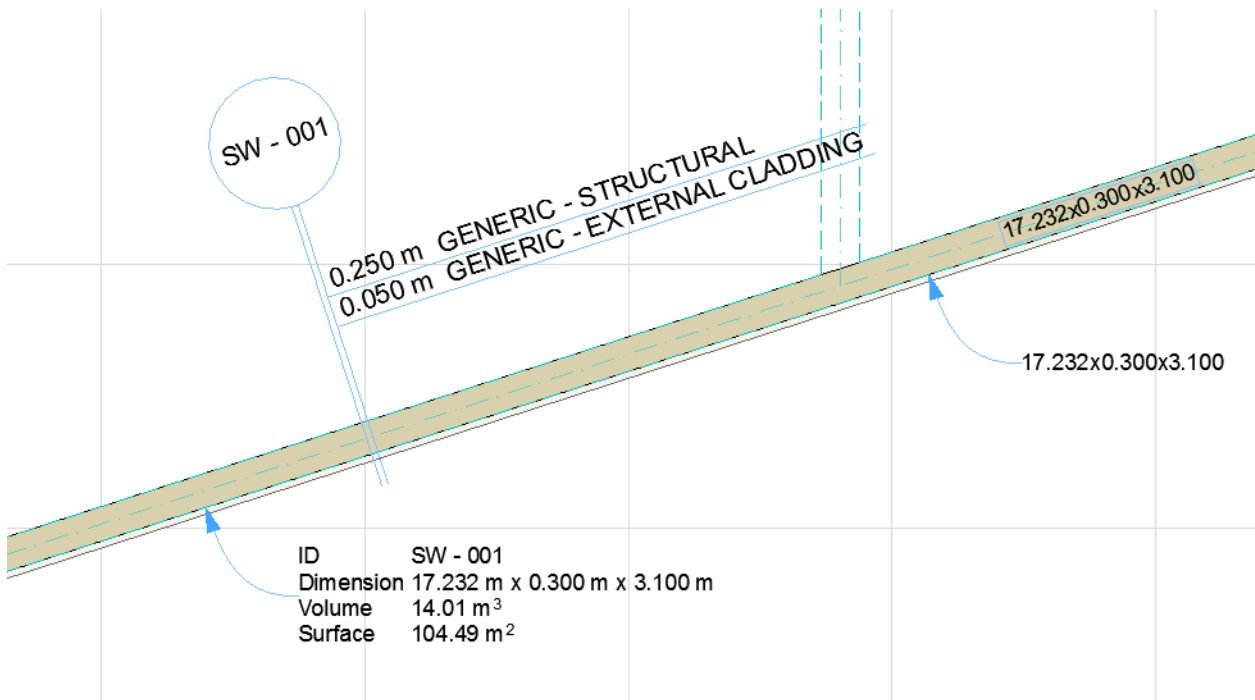
Labels

An element can have any number of Labels. Even if the element is part of a Hotlink Module or reserved by another user - you can place the Label.

Archicad comes with a set of built-in Labels, with highly customizable content.

You can also create your own Label by saving 2D drawing elements, including Autotext.

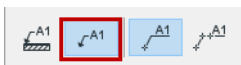
See [Save 2D Elements as Label](#).



Label Geometry Methods: Associative and Independent

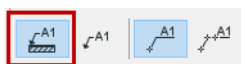
There are two ways to place a label with the Label tool:

- **Independent** labels are not associated to any element. Typically, you will use an independent label to display information that does not come from the model (e.g. "install snow picks here").



Independent Label

- **Associative** labels are placed by clicking an element. The label remains associated to the element even after you move or modify the element. As applicable, any parameter values are automatically updated in the label.



Associative Label

Notes:

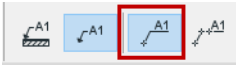
- If you delete an element, its associated labels are also deleted.

- By default, a Label is hidden if its associated element is hidden (e.g. on a hidden layer). To display the Label anyway, go to Label Tool Settings and uncheck “Hide with Associated Element”.
- Multi-story elements must be labeled on each story separately.
- Associated Zone Labels can be placed in one step at the same time the Zone is created: [see *Place Zone Label Automatically*](#).

[Watch the Video](#)

Label Placement Methods: Simple and Detailed

- The **Simple** method places the Label with one click, using the arrowhead and pointer line type/starting angle (if activated) as defined in the Info Box or Label Settings.



Simple Placement

- The **Detailed** method requires three clicks (or two, if the pointer line's starting angle is locked), giving you more control over pointer line placement.



Detailed Placement

See [Pointer Line](#) for details on Pointer Line settings.

See also [Edit Text Box](#).

Related Topics:

[Place an Associative Label](#)

[Place an Independent Label](#)

[Convert Associative Label to Independent Label](#)

[Label Tool Settings](#)

[Text/Autotext Label Content](#)

[Symbol Labels with Element Information](#)

[Custom Settings for Symbol Labels](#)

[Label Multiple Selected Elements](#)

[Labeling Curtain Walls](#)

[Labeling Stairs](#)

[Find and Select Text Blocks/Labels](#)

[Save 2D Elements as Label](#)

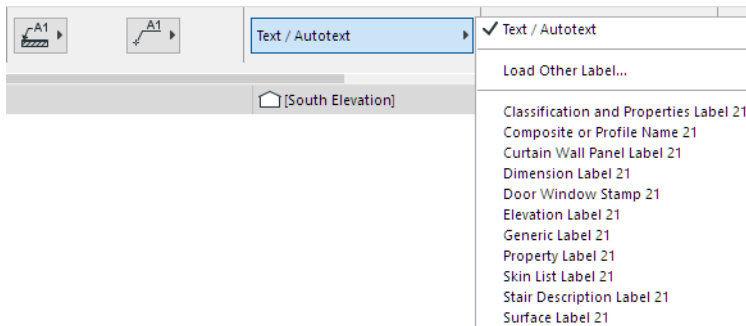
[Watch the Video](#)

Place an Associative Label

1. Activate the Label Tool.
2. Make sure its Geometry Method is set to **associative** in the Info Box (this is the default setting):

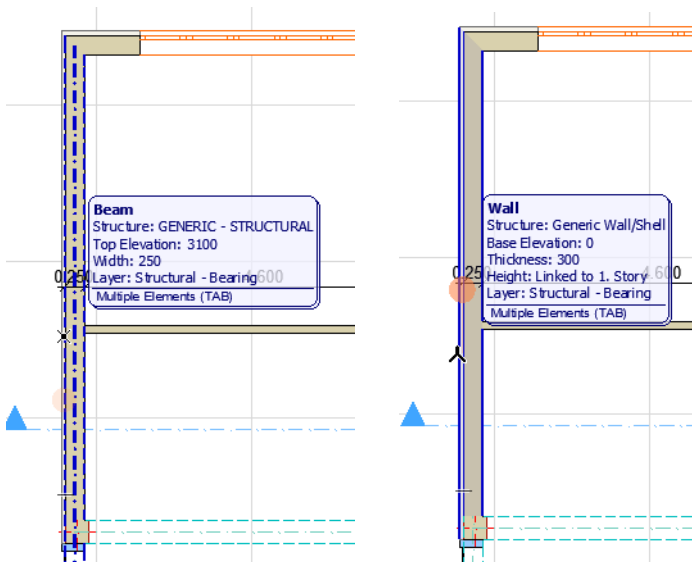


3. In Label Settings or the Info Box, choose the label type you wish to use.



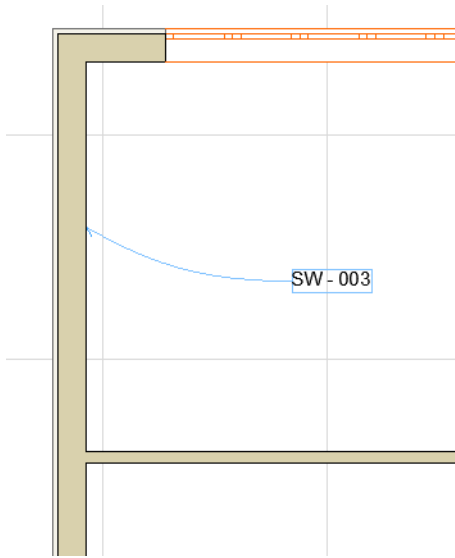
4. Move your cursor to the element you wish to label. Highlighted feedback lets you know which element will be labeled.

If your cursor is hovering over multiple elements, the Info Tag prompts you to click Tab to cycle through the elements.



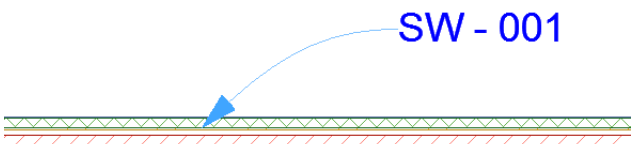
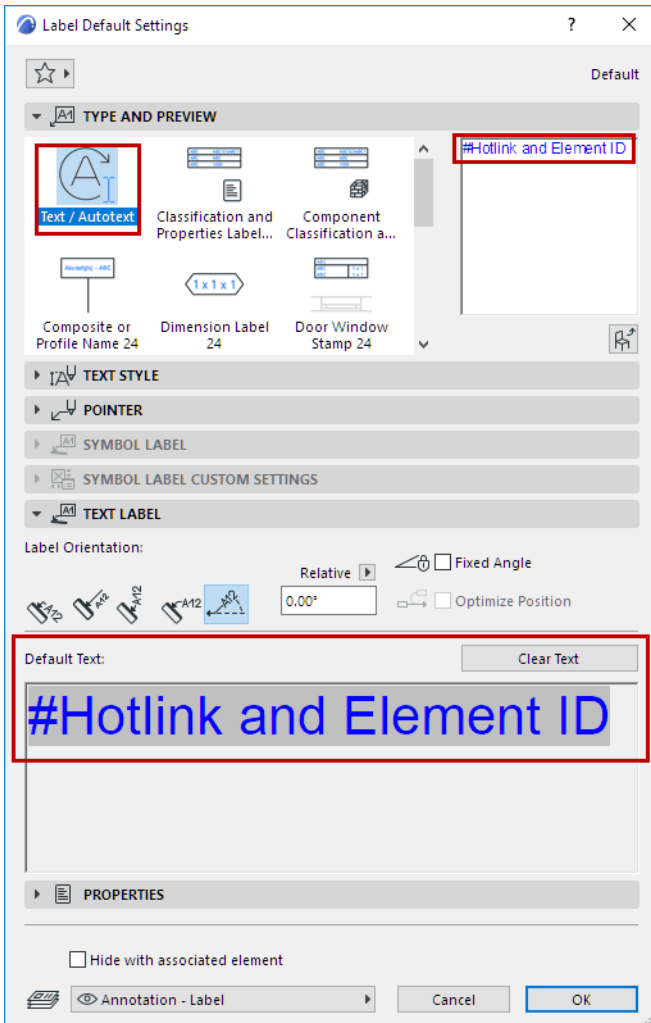
5. When the desired element is highlighted, click to place the label.

Note: Depending on the Label type and its Placement method, this will require one or more clicks.

**Related Topics:**[Text/Autotext Label Content](#)[Label Multiple Selected Elements](#)[Symbol Labels with Element Information](#)[Custom Settings for Symbol Labels](#)

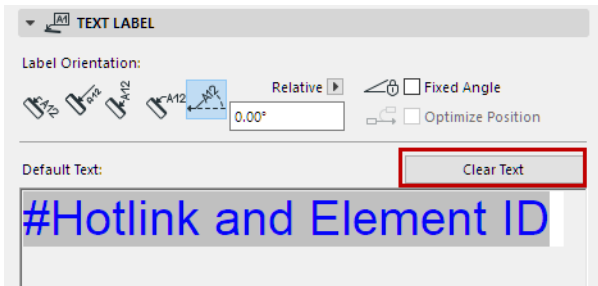
Text/Autotext Label Content

In this example, the default Text/Autotext Label contains the labeled element's Hotlink ID (if any) and Element ID.



Change Default Content of Text/Autotext Label

1. In Default Label Settings, go to the Text Label panel and click **Clear Text** to delete whatever text is currently the default.

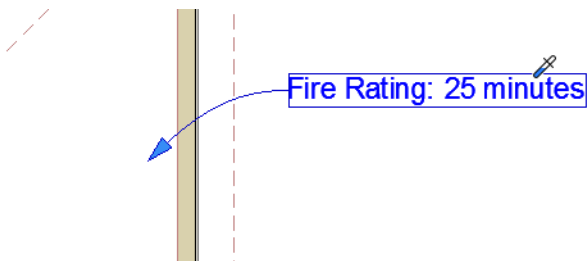


2. Place the (empty) Text/Autotext Label in the project.
3. In the Text editor, type in the desired label text, including Autotext as needed. For example, type in “Fire Rating:”, followed by the Autotext entry for Fire Resistance Rating. This way, you can create simple text labels that display associative data about the element they are attached to.

See [How to Insert Autotext](#).



4. Click to place the Label.
5. Do one of the following:
 - Use the **Pick Up Parameters** command or shortcut (Alt+Click) on the Label to make it the default.
 - Save the Label as a Favorite: the text in the Label Text field is saved as part of the Favorite. Use the Favorite to place Labels with a predefined text. (See [Create New Favorite](#).)



To set up the Pointer or Leader line and its connection to the Text box, use the Pointer Panel of Label Settings.

See [Pointer Line](#).

To format individual characters, lines and paragraphs in your label text block, or to insert text symbols, use the controls of the Formatting Palette and Text Editor.

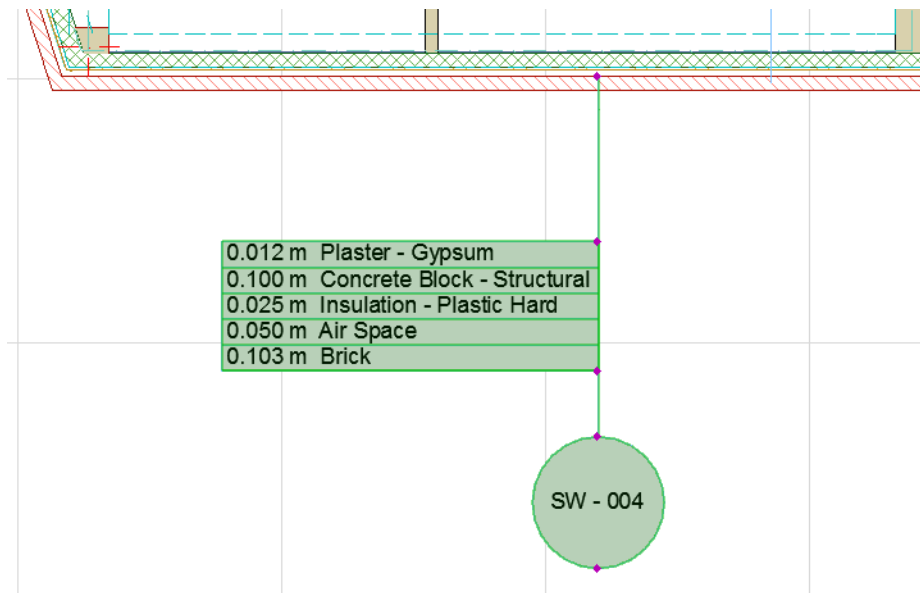
Note: Favorites of the Text tool can be applied to the text-type items in Dimensions and Labels, and to Fill Text.

See [Formatting Palette and Text Editor Controls](#).

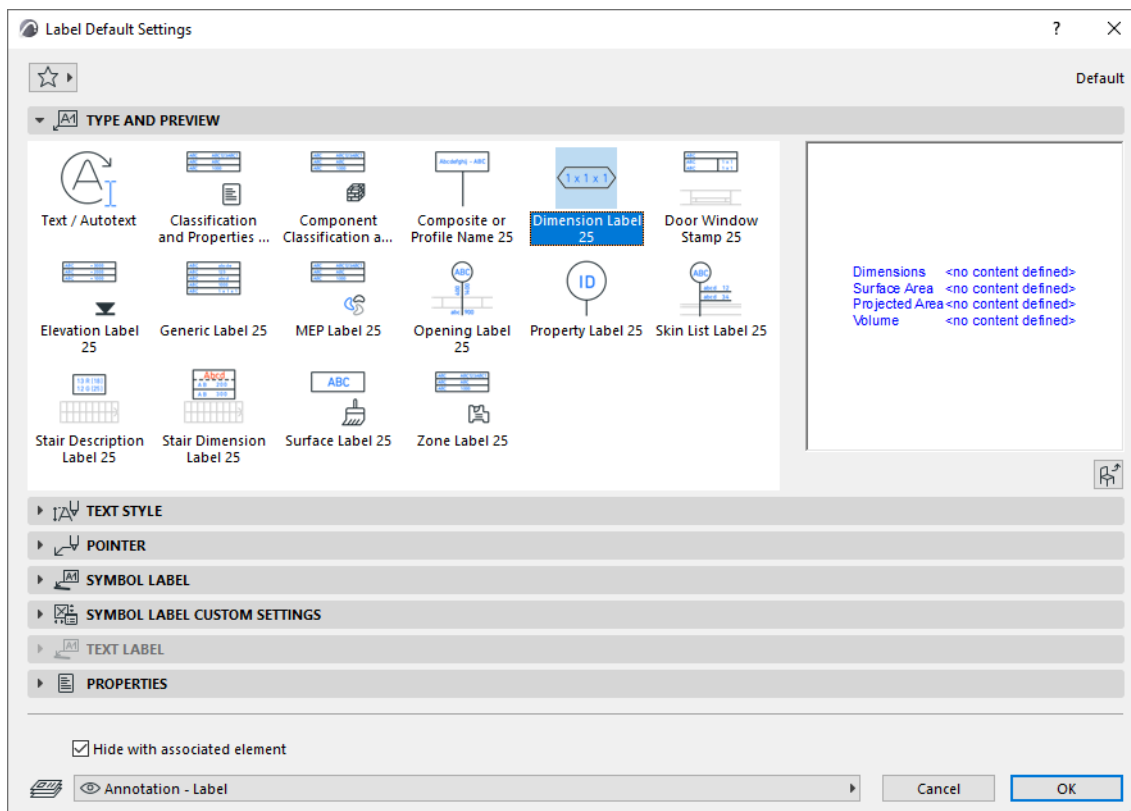
[Watch the Video](#)

Symbol Labels with Element Information


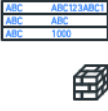


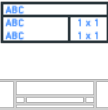

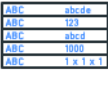
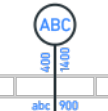

Symbol Labels can display all kinds of element information on the drawing, for example the skins of a composite Wall.

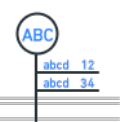
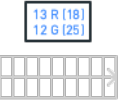
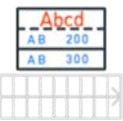

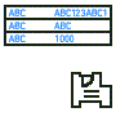


Symbol Labels are GDL objects. Along with Text/Autotext Labels, they are available from the Label Tool pop-up and Label Tool Settings.



The following are the standard Symbol Labels available in the Archicad INT Library. Labels provided in other localized Archicad libraries may vary, to meet local standards.

Symbol Labels and Displayed Data		
	<p>Classification and Properties Label</p>	<p>Element parameters Classification and Properties IFC Properties</p>
	<p>Component Classification and Properties Label</p>	<p>Classification and Properties of Components</p>
	<p>Composite or Profile Name</p>	<p>Element / Hotlink ID</p>
	<p>Dimension Label</p>	<p>Dimensions Area Volume</p>
	<p>Door Window Stamp</p>	<p>Element/Hotlink ID Dimensions Surfaces</p>
	<p>Elevation Label</p>	<p>Elevation dimensions</p>
	<p>Generic Label</p>	<p>Element / Hotlink ID Name Elevation dimensions Dimensions Area Volume Skin list Composite / Profile / Bldg Material name Surfaces</p>
	<p>Opening Label</p>	<p>Door/Window ID Dimensions Properties</p>
	<p>Property Label</p>	<p>Element / Hotlink ID Name Property values</p>

Symbol Labels and Displayed Data		
	<p>Skin List Label</p>	<p>Building Material properties Element / Hotlink ID Dimensions Skin list</p>
	<p>Stair Description Label</p>	<p>Dimensions Parameters</p>
	<p>Stair Dimension Label</p>	<p>Stair-Specific Dimensions (Width, Riser Height, Going Depth, Number of Risers)</p>
	<p>Surface Label</p>	<p>Surfaces</p>
	<p>Zone Label</p>	<p>Zone Name, Number, Calculated Area</p>

See also [Labeling Curtain Walls](#).

[Watch the Video](#)

[Watch the Video](#)

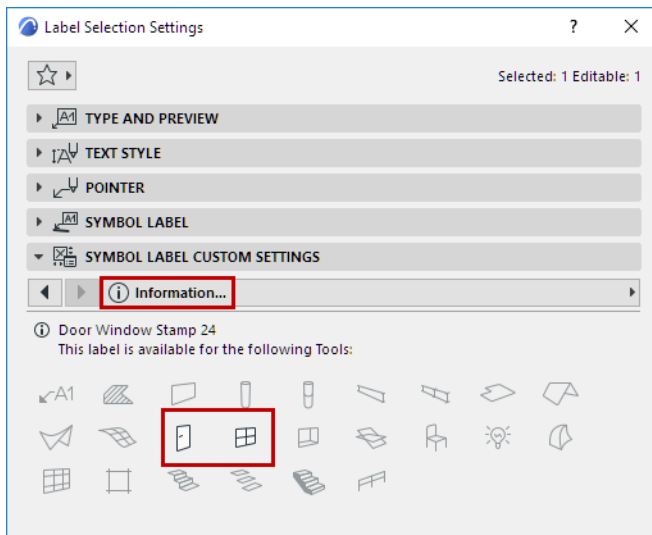
[Watch the Video](#)

See also [Select Surface \(for Surface Label\)](#).

Not every label can be placed on every element type. For example, if you inadvertently attach a Door Window Stamp to a Column, a “placeholder” text will inform you of the mistake.



Note: For each label, Custom Label Settings provides information on which element types it can be applied to.



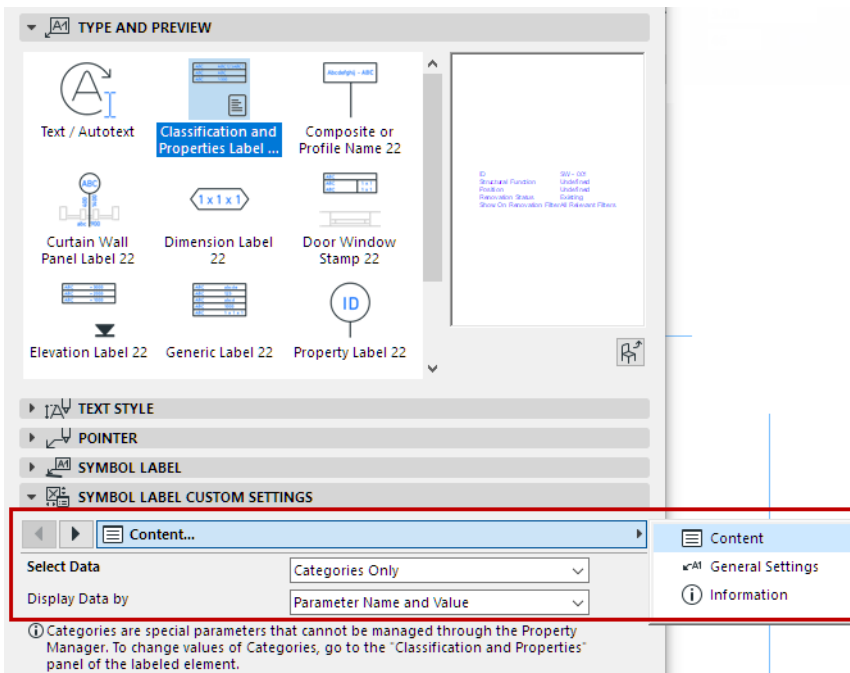
For more details, see [Custom Settings for Symbol Labels](#).

You can also create your own Label by saving 2D drawing elements, including Autotext.

See [Save 2D Elements as Label](#).

Custom Settings for Symbol Labels

Each Symbol Label has a Custom Settings panel in the Label Tool Settings dialog box. The parameters are divided among 3-5 tab pages, as shown here:



Use these tab pages to set preferences for label content, positioning and format.

Note: Not every tab page is available for every label type.

- To make repositioning easier, many labels have a reference point, which remains in place even after you change the label size or content.
- Many labels have several header formats available, and the leader line can be turned off, where applicable.

Most of the settings are self-explanatory. The following sections contain some additional tips for using these controls:

Define Content of Symbol Label

Defining Dimension Units in Labels

Select Surface (for Surface Label)

Label Frame Shape

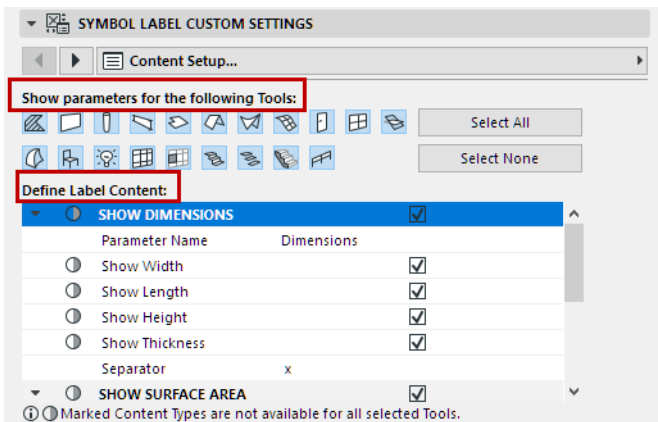
Pointer Line vs. Leader Line

Information Tab Page

Define Content of Symbol Label

Use the **Content Setup** tab page to define the information displayed in the label.

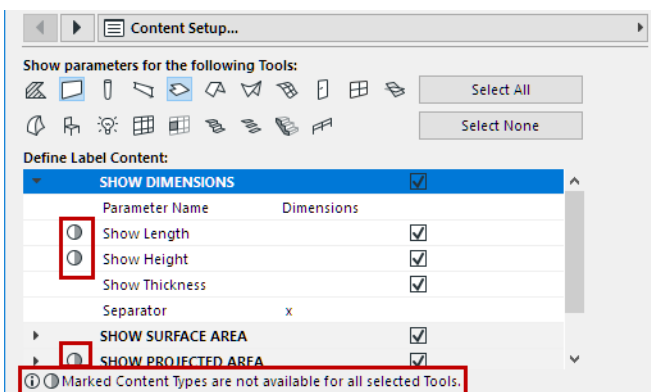
For the **Generic** and **Dimension** labels, the available element types - and therefore the list of possible parameters - is very extensive, and the Content Setup tab page looks like this:



Since you can attach the label to different element types, the label content may vary, depending on which element it is attached to.

- Show parameters for the following tools:** This control filters the content list in the bottom part of the dialog box. Select the tool(s) whose element parameters you want to choose from. (By default, all tools are selected.) The more tools you select, the greater the number of available parameters.

Note: If a particular tool is not selected in the “Show parameters for” section, you can still place this label on such elements, provided the element has a value that can be displayed by the label.
- Define Label Content:** Select the parameters whose values you want to display in the label.
 - If you don’t see a tool-specific parameter in the list, make sure that the corresponding tool icon is selected in the “Use with” section above.
 - Parameters shown with the circle marking are not relevant for one or more of the selected element types (in the “Use With” section). Thus, label content might vary, depending on the element type to which it is attached.



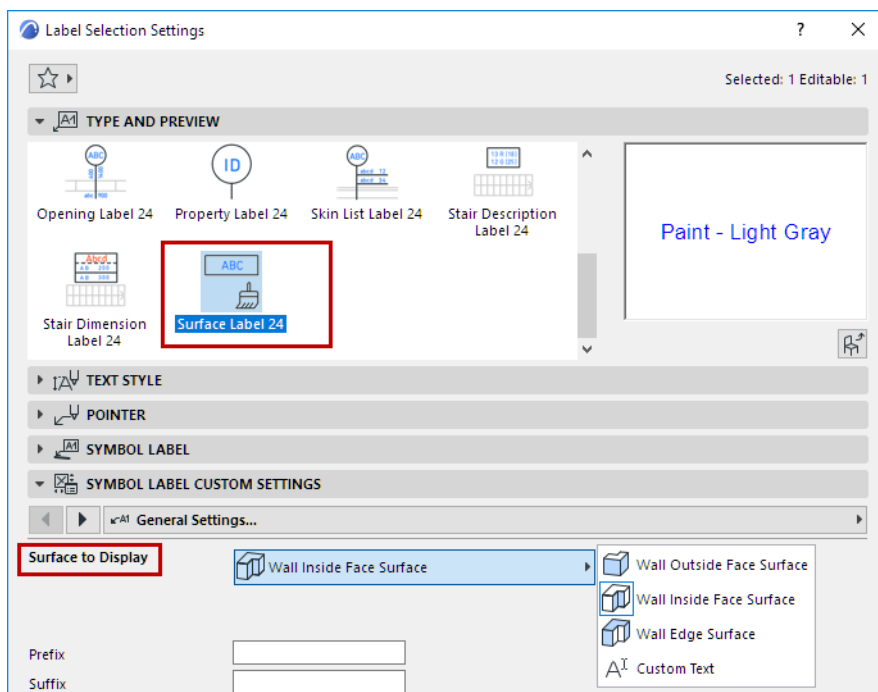
Defining Dimension Units in Labels

When specifying dimension units for label display, many labels include the option “By Project Preferences”. This means the dimension will be displayed in the units defined for it at **Options > Project Preferences**.

- For volume dimension data shown in labels, set the units at **Options > Project Preferences > Calculation Units**.
- For all other dimension data shown in labels, set the units at **Options > Project Preferences > Dimensions**, using the relevant dimension type options. [See Choose Dimension Type](#).

Select Surface (for Surface Label)

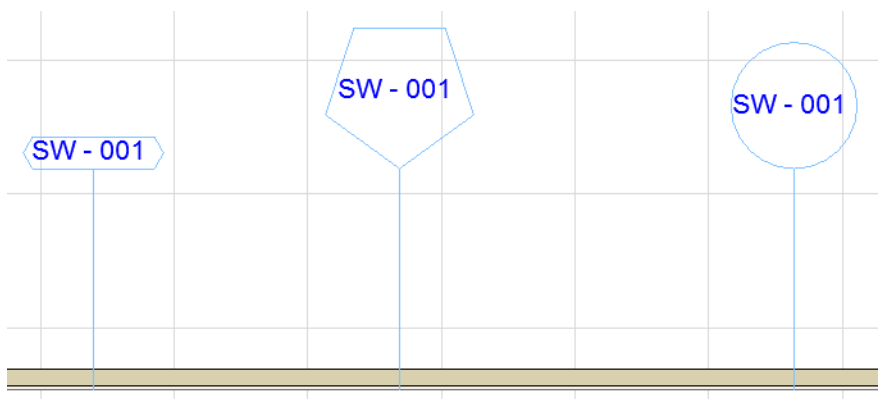
The Surface Label displays the name of the element’s selected surface. You must first select the particular element surface in Label Settings:



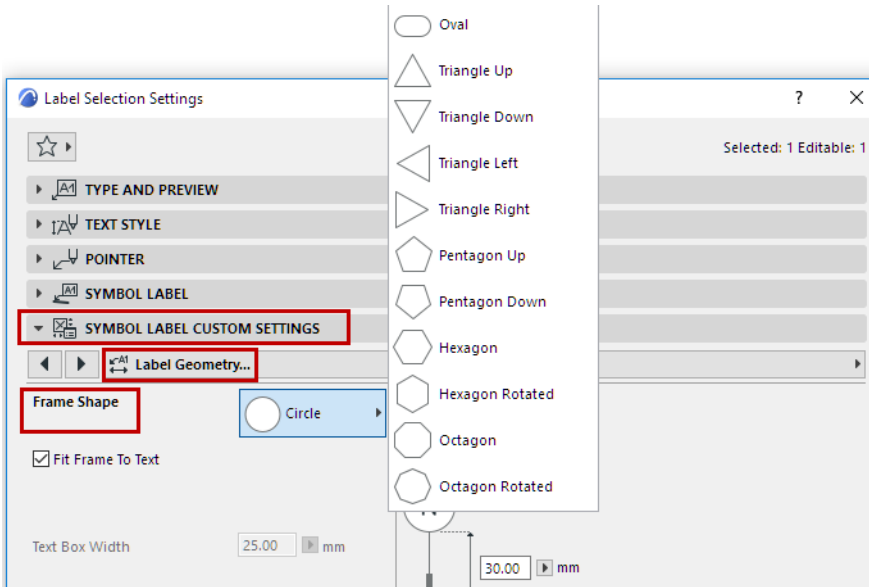
[Watch the Video](#)

Label Frame Shape

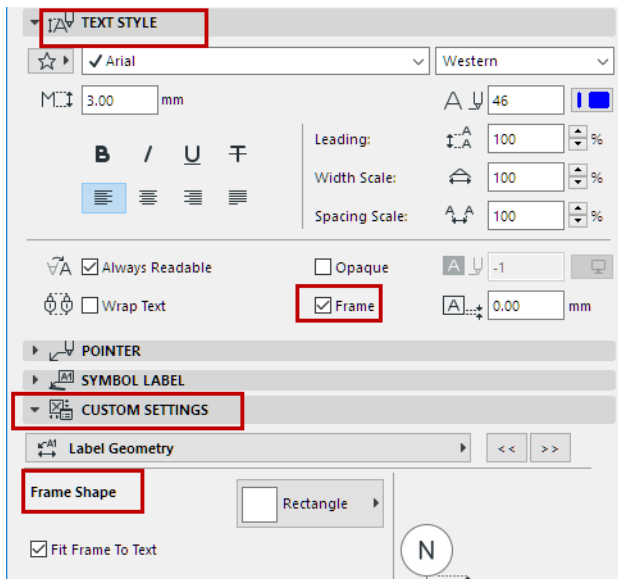
For labels having a frame, you can select among different shapes.



Set this option in the Custom Settings panel, in the **Label Frame Style** or the **Label Geometry** tab page.



The Frame Shape control is only available if you have activated a Frame, in the Text Style panel (as shown below).

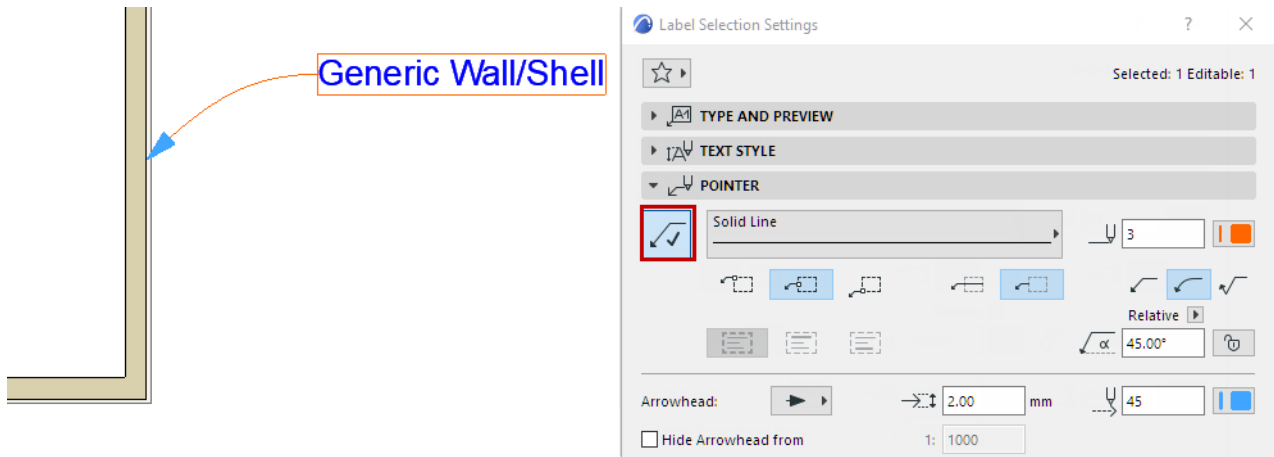


[Watch the Video](#)

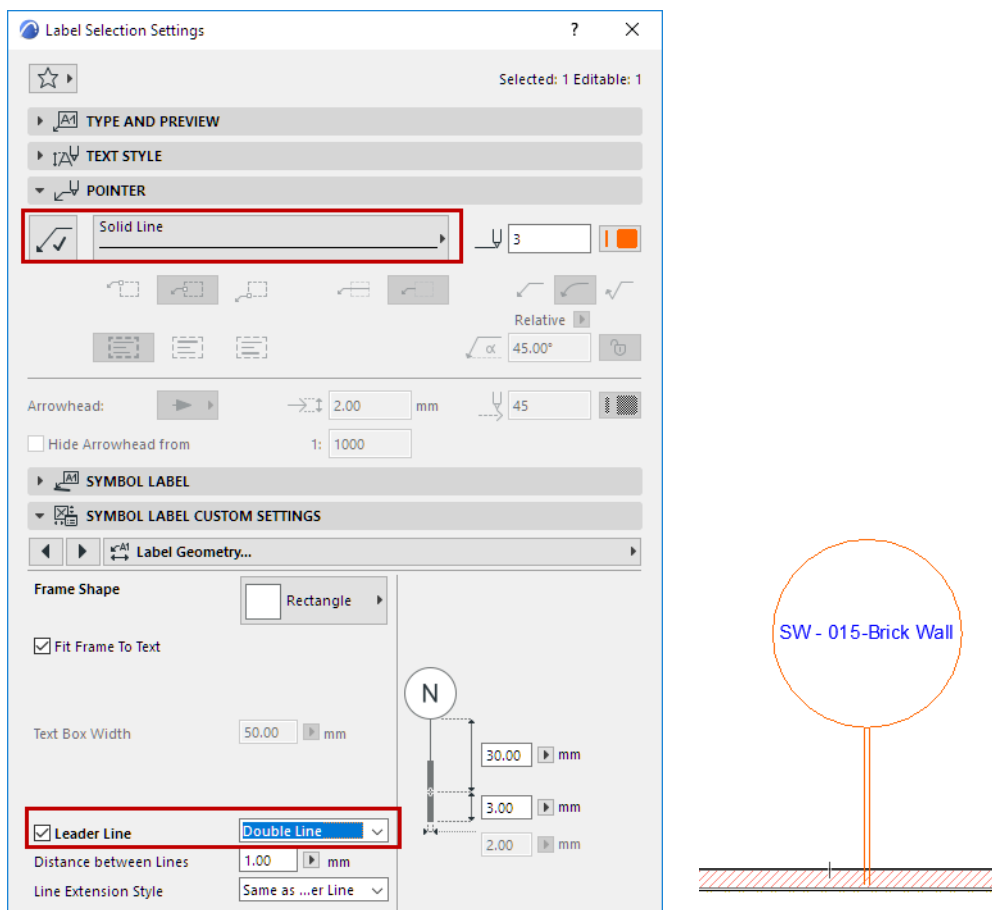
[Watch the Video](#)

Pointer Line vs. Leader Line

If you have activated a Label's Pointer Line, in the Pointer Panel, then these Pointer Line settings will prevail (the Label will *not* use the Leader Line defined on the Custom Settings panel.)



If you are *not* using the Pointer Line (the Pointer button is off), then the Leader Line (defined in the Custom Settings panel) will be used. The line and pen settings of the Leader Line are also set on the Pointer Panel.

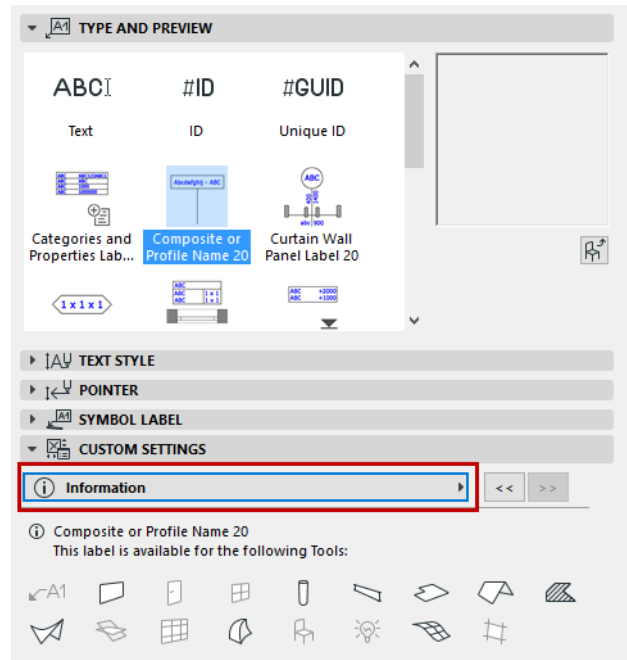
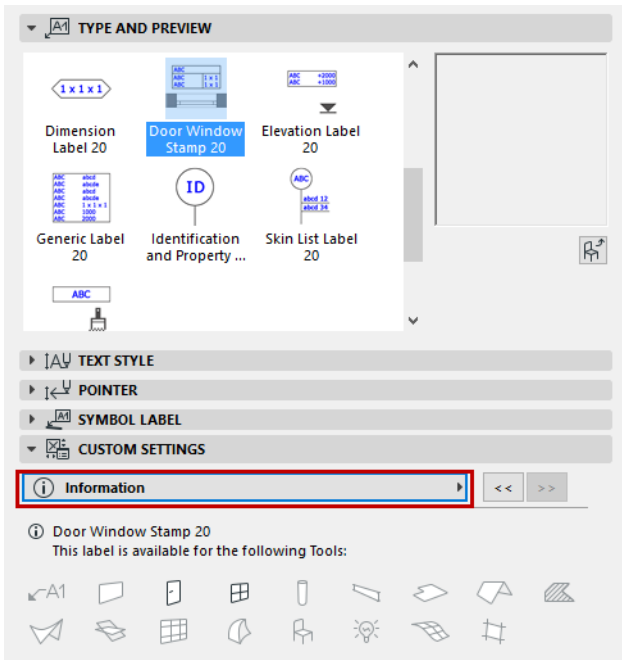


Related Topic:

[Pointer Line](#)

Information Tab Page

This is an information-only page. It shows which element types the selected Label is available for.

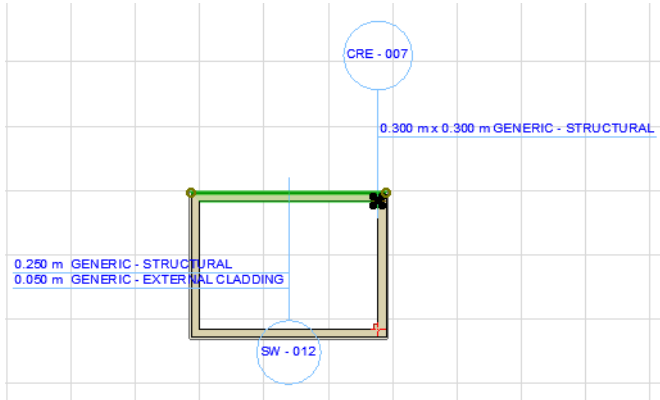


Label Multiple Selected Elements

To place multiple labels with one click:

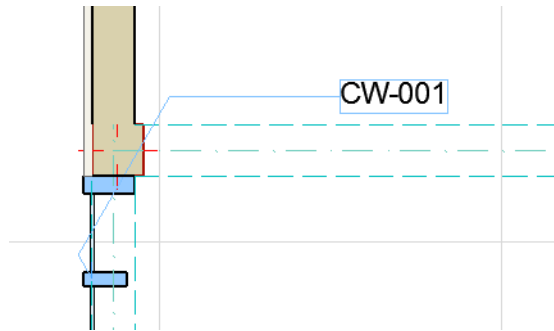
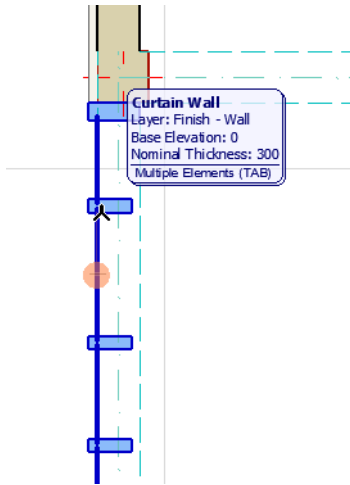
1. Select the elements you wish to label.
2. Choose **Document > Annotation > Label Selected Elements**.

The current default label is attached to all selected elements.

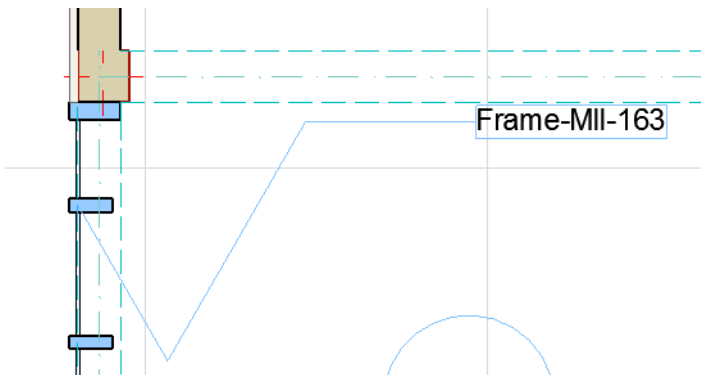
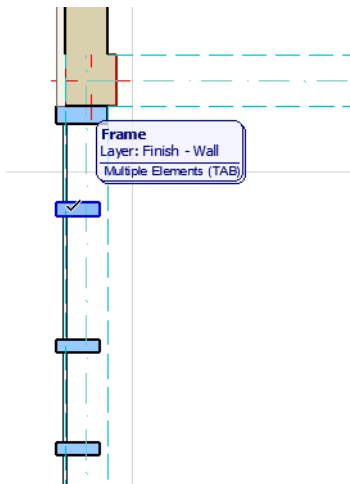


Labeling Curtain Walls

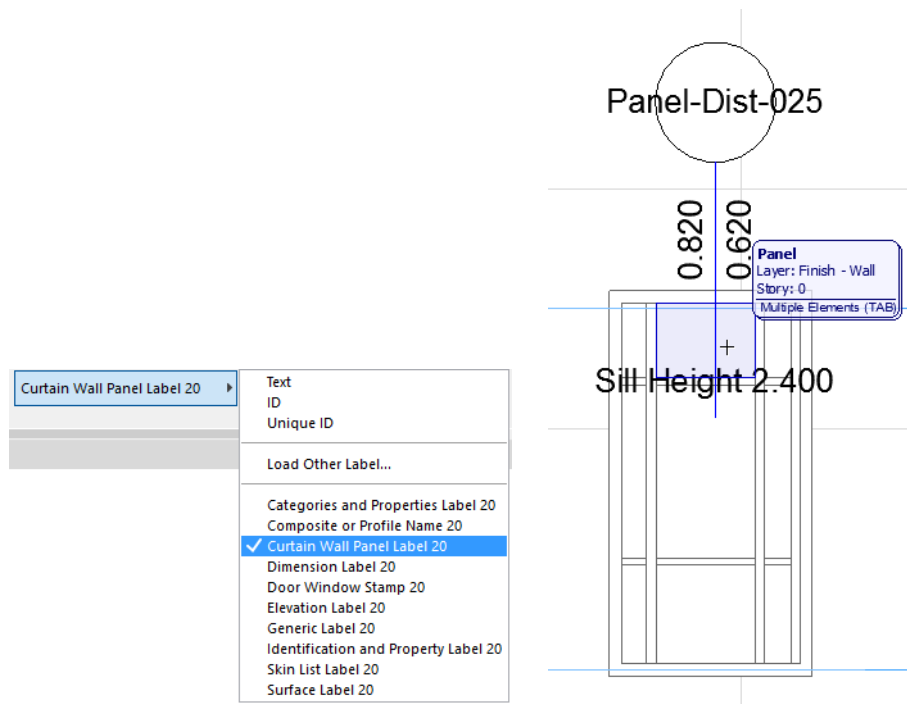
Like any other element, you can label a Curtain Wall or any of its visible members individually, on the Floor Plan as well as in Section view.



Curtain Wall with ID Label



Curtain Wall Frame with ID Label



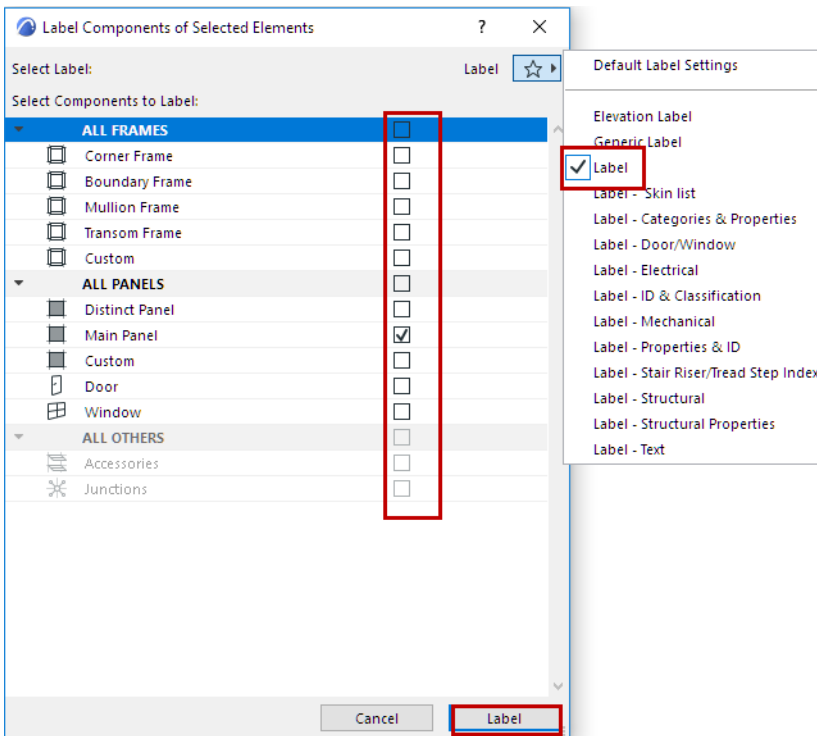
Curtain Wall Panel Label

Label All Curtain Wall Members

If you need labels on all Members, or all of a certain class of Members:

1. Select the Curtain Wall.
2. Click **Document > Annotation > Label Components of Selected Elements** to bring up the dialog box.
3. From the pop-up, choose a Label Type: the current default Label, or any of the Label Favorites.

4. Check the boxes of all the component types to which you would like to attach a Label. The Labels will be associative.



5. Click **Label** to place.



Labeling Stairs

Stair Description and Stair Dimension Labels

These symbol-type Labels are available from Label Tool Settings.

Related Topic:

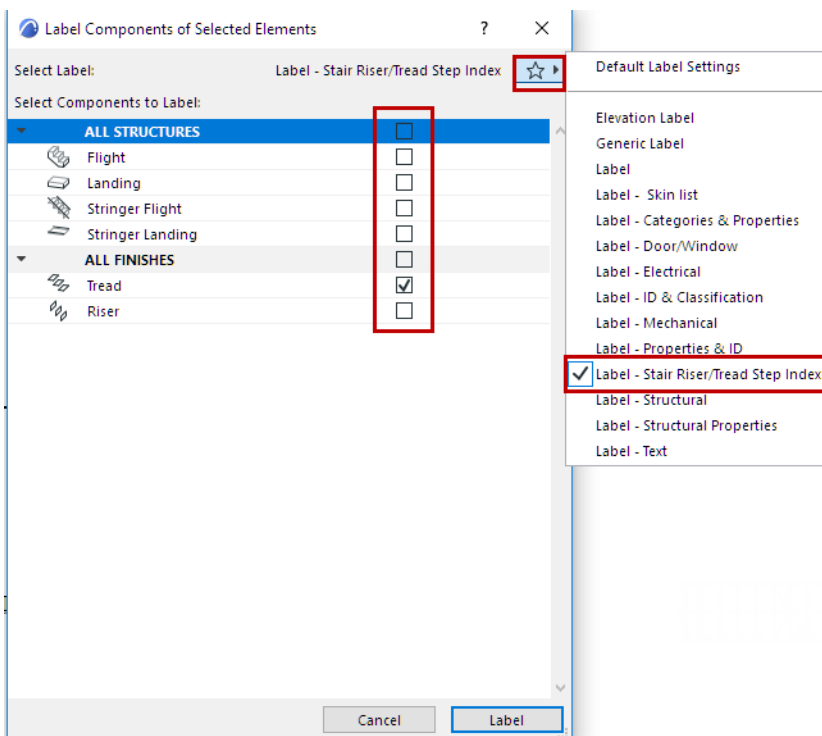
[Symbol Labels with Element Information](#)

Label Stair Components

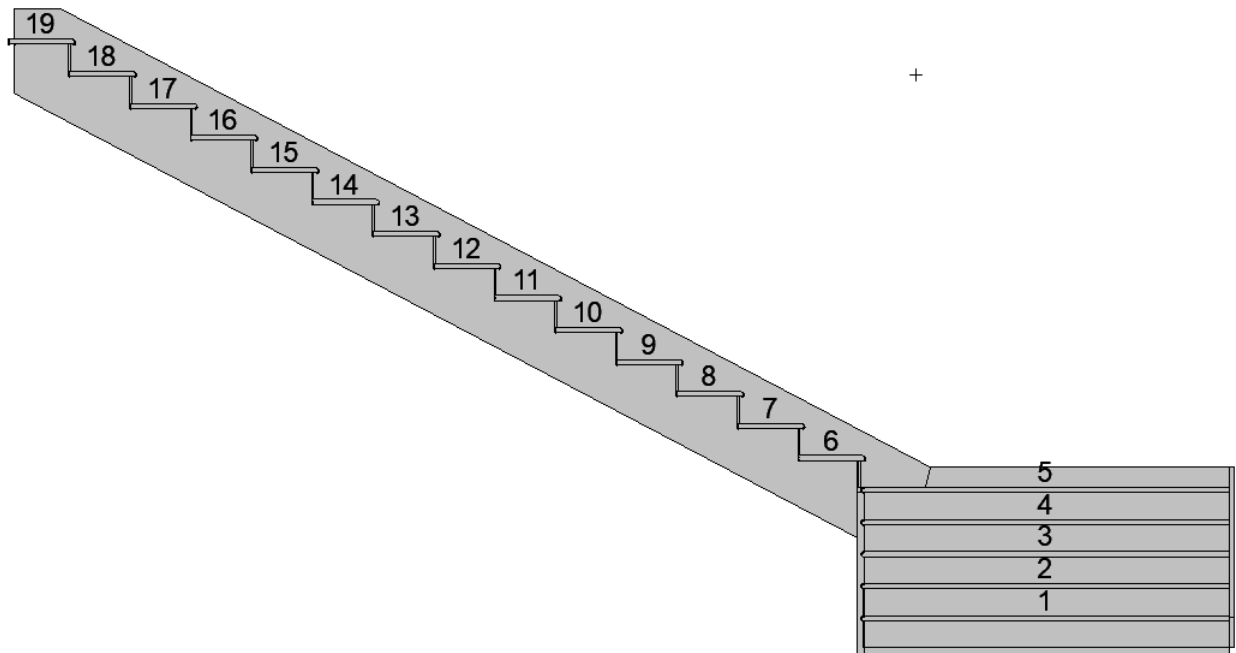
1. Select the Stair.
2. Click **Document > Annotation > Label Components of Selected Elements** to bring up the dialog box.
3. From the pop-up, choose a Label Type: the current default Label, or any of the Label Favorites.
4. Check the boxes of all the component types to which you would like to attach a Label. The Labels will be associative.

Related Topic:

[Level Dimensions](#)



5. Click **Label** to place.

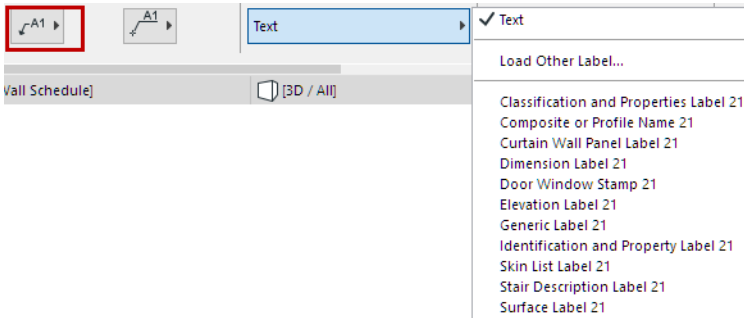


Place an Independent Label

1. Choose Independent Label from the pop-up options in the Label Info Box:



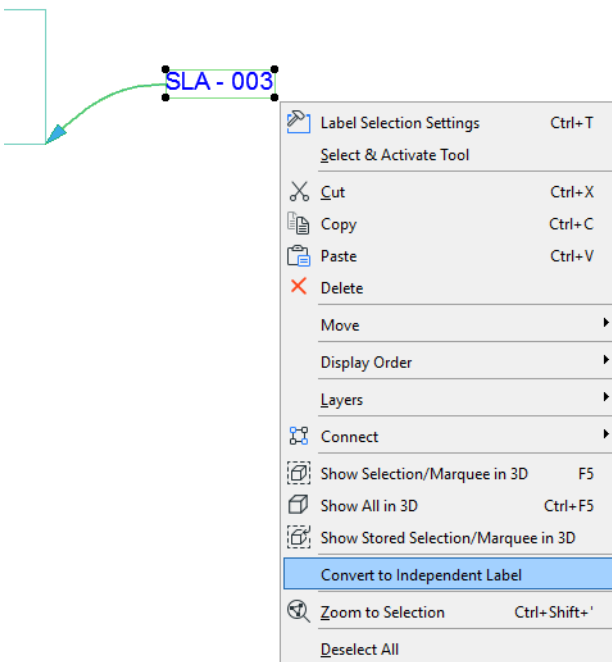
2. In Label Settings or the Info Box, choose the label type you wish to use. (See also [Text/Autotext Label Content.](#))



3. Choose a preferred Label Placement Method. (See [Label Placement Methods: Simple and Detailed.](#))
4. With the Label Tool active, click anywhere in the workspace to place the independent label.

Convert Associative Label to Independent Label

1. Select the associative label.
2. From the context menu, choose **Convert to Independent Label**.

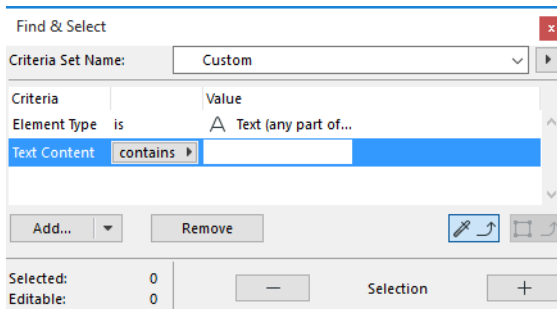


Find and Select Text Blocks/Labels

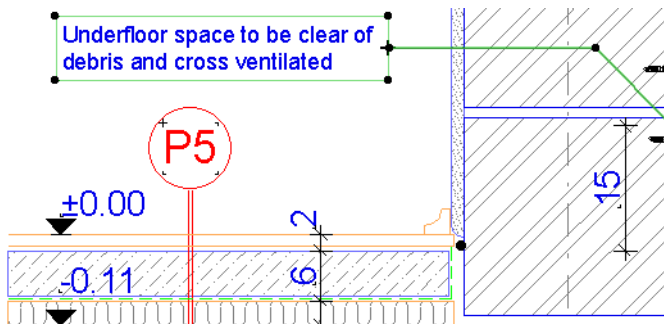
Use the Find & Select function to locate text blocks or label texts whose content matches or contains any given text string.

See [Find and Select Elements](#).

1. In the Find & Select dialog box, choose either Text or Label as the Element criterion.
2. As the next criterion, choose "Text Content."
3. As the value, enter any desired text string.
 - to find and select text fragments, choose "contains."
 - to find exact text matches, choose "is."



4. Click the **Plus** button to execute the Find & Select function. In this example, all Text Blocks or Labels in the active window which contain the entered text will be selected.



Text Editing in Archicad

Search and Replace Text

Spell Checker

Editing Commands in Text-Type Windows

For information on placing and editing Text Blocks using Archicad's dedicated Text tool, see [Text Blocks](#).

Search and Replace Text

The Search and Replace Text function is available for the following text-type elements in Archicad:

- Text blocks placed with the **Text** tool
- Labels
- Dimensions
- Zone Stamps
- Parameters of **GDL Objects**

Go to **Edit > Search and Replace Text**. Use this function to search texts in all eligible element types. It works on the Floor Plan and in Section/Elevation/IE, and Detail and Worksheet windows. It does NOT work for Autotext items.

[See *Search and Replace Text Dialog Box*.](#)

Note: The Search and Replace text function is not available for text-type windows. (Text-type windows include all Listing windows created with the Calculation function, GDL script windows, Project Notes, and Report windows.) In text type Windows, the **Find & Replace** command offers similar functions.

[For more information, see *Editing Commands in Text-Type Windows*.](#)

Spell Checker

The **Document > Annotation > Spell Checker** command allows you to check the spelling of your Archicad project.

Note: Spell Checker is not available for text-type windows.

See [Editing Commands in Text-Type Windows](#).

On Windows, Spell Checker uses Microsoft Word's Spelling Check, which means that you must have Microsoft Word installed on your computer together with the dictionary of the language you are using. In Microsoft Word, use **File > Options > Language** to set your desired preferences.

On Mac, Spell Checker uses MacOS system preferences. Set these at **System Preferences > Language & Region**.

Archicad's Spell Checker will use these preferences.

Warning: Spell Checker relies on Microsoft technology. Archicad can only locate Word's Spell Checker if its pointer is correctly written in the Windows Registry. If you have properly installed Microsoft Office, Spell Checker will work without any problem. If you have manually modified the location of Microsoft Word, Archicad will not be able to use its spell checking features. Moreover, the **Spelling** commands must be installed with Word.

Editing Commands in Text-Type Windows

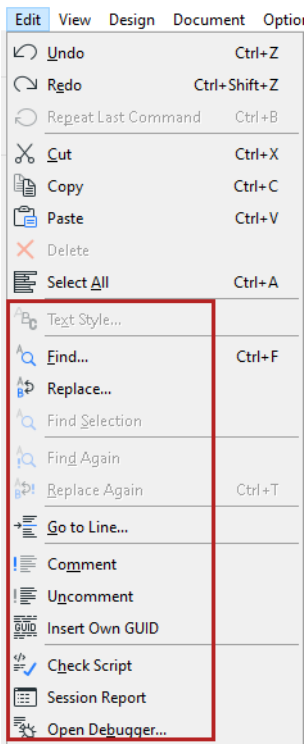
Text-type windows include all Listing windows created with the Calculation function, GDL script windows, Project Notes, and Report windows.

Only a single editing step can be undone/redone in a text-type window.

When a text-type Window is active, most of the **Edit** menu commands change in order to provide you with basic word processing features.

Note: Search/Replace Text and Spell-check do *not* work for text-type windows.

The **Text Style** command opens a dialog box determining how your text will look both on screen and for printing.



The commands in the bottom section apply to GDL scripts, and are active only if a GDL script window is active.

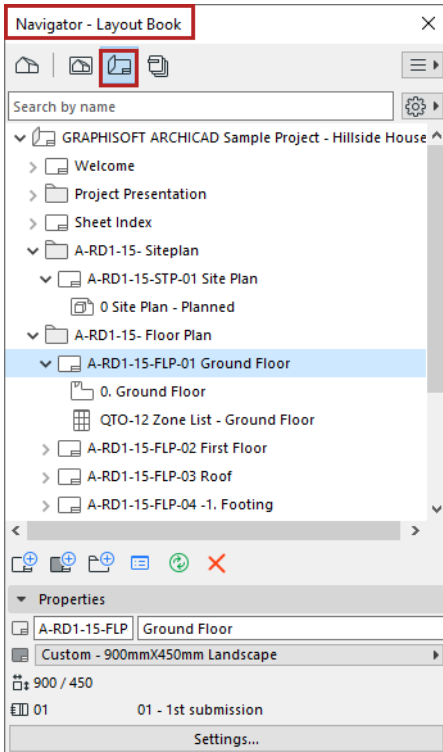
For more information, see [Script Windows](#).

Navigator - Layout Book

Contains all the project's Layouts with their Drawings, optionally organized into subsets, and formatted based on Master Layouts.

It is displayed in a tree structure in the Layout Book map of the Navigator or Organizer.

Use the search field to locate items by name.



Topics in this section:

[Layout Book List Display](#)

[Moving Layout Book Items](#)

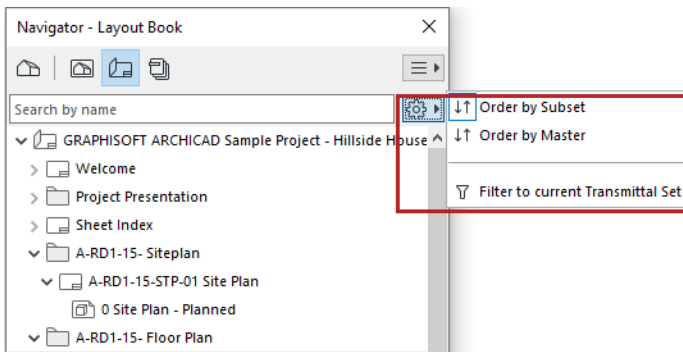
[Settings and Commands of Layout Book Items](#)

[Properties of Layout Book items](#)

[Subsets](#)

Layout Book List Display

Use the Settings pop-up to how to list the Layout book items:



- **Order by Subset** lists layouts according to Subsets - a freely defined logic. (The primary function of Subsets is to enable you to assign a customized numbering system.) The Tree also shows the names of the drawings that belong to each Layout.

[See Subsets.](#)

- **Order by Master** lists the Layouts according to their [Master Layouts](#). Drawings are not listed.
- **Filter to current Transmittal Set:** Activate this filter to show only the Layouts that have been added to the latest Transmittal Set.

[See Revision Management.](#)

Moving Layout Book Items

Use drag and drop to move and rearrange items in the Layout Book.

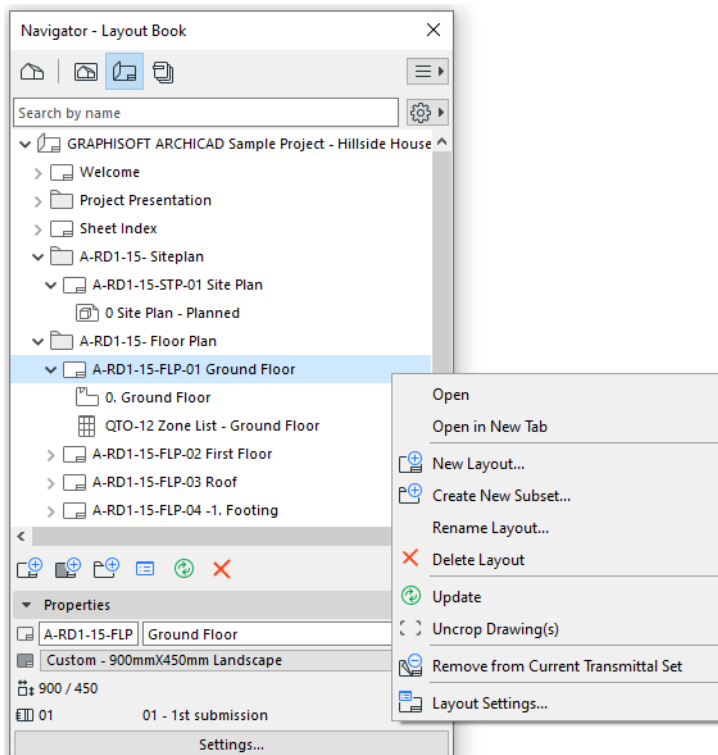
- Create hierarchical Subsets
- Move a Layout from one Subset to another
- Apply a Master Layout to a Layout
- Move a Drawing from one Layout to another Layout

Note: In this case, the proportionate distance of the Drawing from the Layout origin remains unchanged.

Important: Operations involving Navigator/Organizer items (such as drag and drop between Navigator maps, deleting items from a map, or adding items to the Publisher set) are not added to the undo queue, and are *not* undoable.

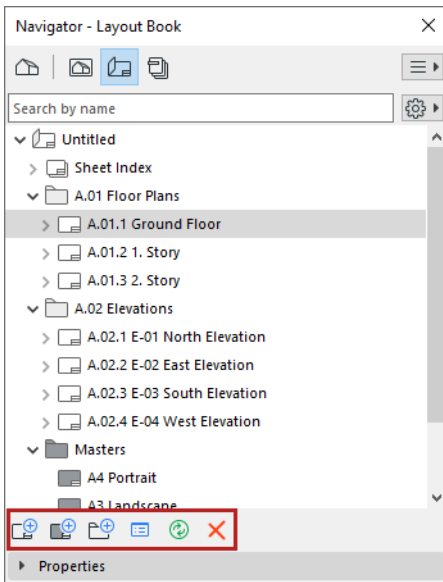
Settings and Commands of Layout Book Items

- Use context menu of the selected Layout Book item



- For any selected item, click Settings at the bottom of the Layout Book

- Use the command buttons. These are active depending on the item selected in the Layout Book (Layout Book, Subset, Layout, Drawing, or Master Layout):



- New Layout
- New Master Layout
- New Subset
- Settings
- **Update:** Updates the selected Drawing, or the Drawings on the selected Layout.
- **Delete** selected item
- Use commands of the **Document > Layout Book** menu
 - Note:** Another way to gain an overview of Layouts or Drawings in your project, or to sort them by criteria, is the Project Index function.

[See Project Indexes.](#)

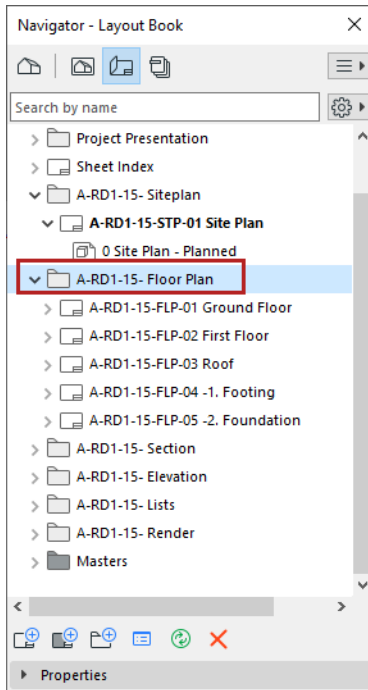
Properties of Layout Book items

Select an item (Layout Book, Subset, Layout, Master Layout or Drawing) in the Layout Book to view its Properties below.

Subsets

A subset is a folder inside the Layout Book. Subsets allow you to easily organize your layouts and to define numbering rules for them.

Place a Layout into a Subset by dragging and dropping it into place.



For more information on IDs, see [Layout and Drawing IDs](#).

When setting up the contents of a Publisher Set, you can add a shortcut to any Layout Book Subset: this means Publisher Set items are created which are linked to the Subset folder and to its contents. Any changes to the contents of the Subset in the Layout Book are also reflected in the contents of the Publisher Set.

See [Add items to Publisher Set](#).

Working with Layouts

Topics in this section:

[Open a Layout](#)

[Navigate Among Layouts](#)

[Create New Layout](#)

[Delete Layouts](#)

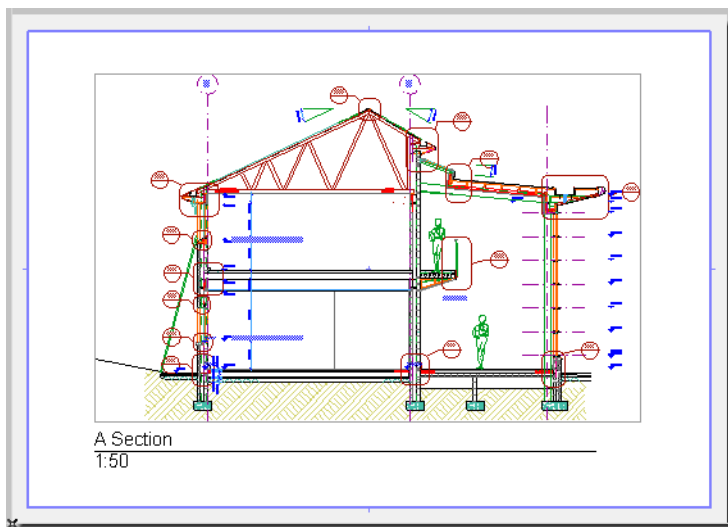
Open a Layout

Do one of the following:

- Double-click its name Navigator Layout Book or the Organizer palette's Layout Editor.
- Click the Layout's name from the **Window** menu
- Click the **Open Layout** command from the Layout drop-down in the Mini-Navigator Toolbar. Activate the desired Layout from the appearing Open Layout list.

The white area represents the usable print area of the paper delimited by a blue frame.

The gray area represents the non-printing area (margins) of the paper.



When you open a new layout or Master Layout, it can either replace the opened one, or be opened in a separate new tab. Change this tab-opening preference in **Options > Work Environment > More Options**.

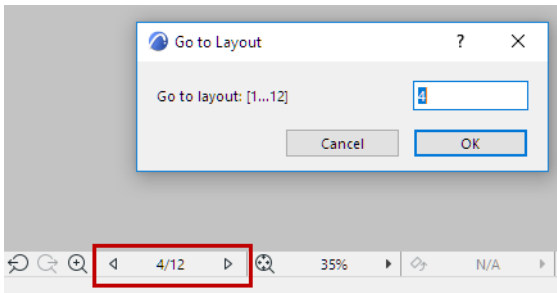
Navigate Among Layouts

To navigate sequentially among Layouts in the Layout Book:

- Click the Layout navigation arrows in the Quick Options Bar at the bottom of the Layout window.

To access the **Go to Layout** command:

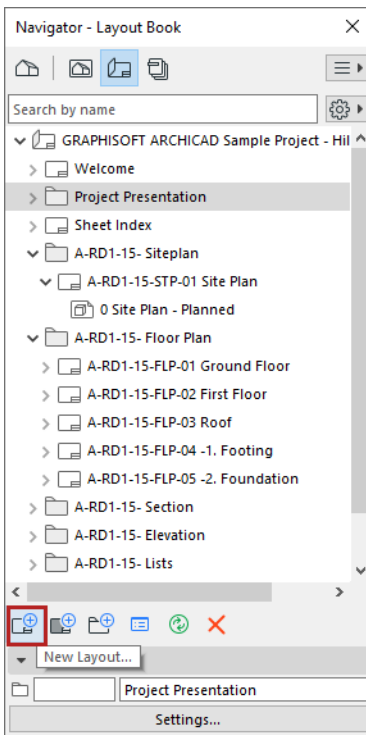
- Click the button showing the current Layout number.



Create New Layout

Do one of the following:

- From any Layout Book item in Navigator: use **New Layout** context menu command
- **Document > Layout Book > New Layout**
- **New Layout** icon in the Navigator Palette or the Organizer



Delete Layouts

You can delete any Layout in the Layout Book except the last one: the Layout Book must always contain at least one Layout.

Related Topics

[Master Layouts](#)

[Print Layout](#)

[Drawings in the Layout Book](#)

Master Layouts

Each Layout is assigned a Master Layout.

- The Master Layout defines the size of the Layout page
- The graphic and text items that you place on a Master Layout (“Master items”) appear on every Layout that uses it

Master Layouts are found in the Masters folder of the Layout Book.

Archicad includes several predefined Master Layouts. You can use one of these predefined Master Layouts, or customize your own.

Most of these options are set in Master Layout Settings.

See [Master Layout Settings](#).

Topics in this section:

[Open Master Layout](#)

[Default Master Layout](#)

[Assign Master Layout to a Layout](#)

[Create New Master Layout](#)

[Master Layout Title Block](#)

[Display of Master Layout Items](#)

Open Master Layout

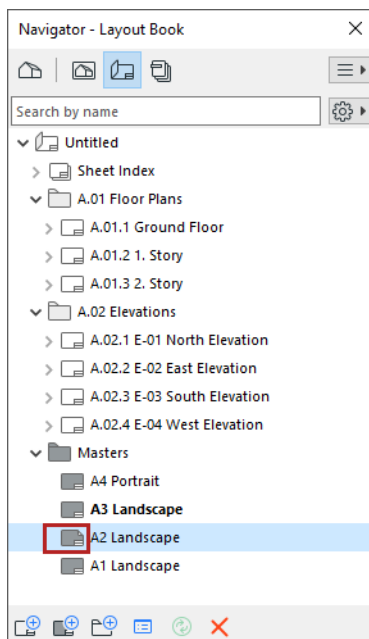
See [Open a Layout](#).

To open the current Layout’s Master, do one of the following:

- right-click the Layout window, and use **Open assigned Master Layout**
- use **Document > Layout Book > Open assigned Master Layout**

Default Master Layout

The default Master Layout is shown as dog-eared. It is assigned to every newly created Layout.



Set a Master Layout as Default

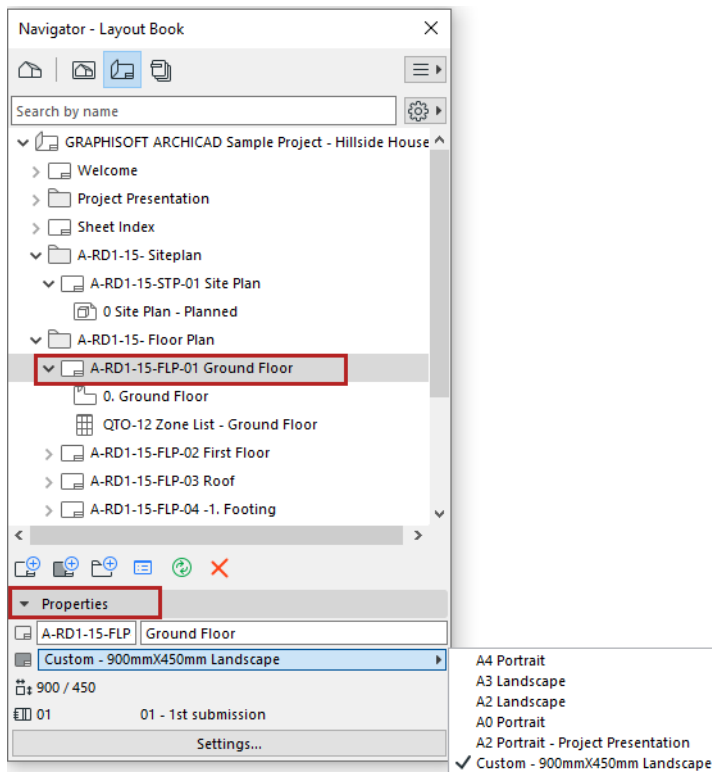
Select the Master Layout in the Navigator or Organizer, then do one of the following:

- From the context menu, choose **Set as Default**
- Open Master Layout Settings and check the **Set as Default for New Layouts** box.

Assign Master Layout to a Layout

Do one of the following in the Navigator Layout Book:

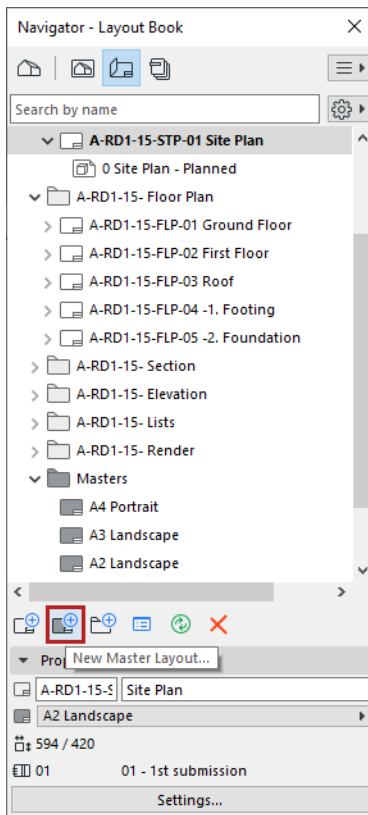
- Drag and drop the Master Layout onto a Layout
- Drag and drop a Layout onto the Master Layout
- Select a Layout, then choose another Master Layout from the pop-up menu at the bottom of the Navigator Palette or in the Settings dialog box of the selected Layout.



Create New Master Layout

To create a new Master Layout, do one of the following:

- Use the **New Master Layout** command from the context menu of any item of the Navigator's Masters folder.
- Choose the **Document > Layout Book > New Master Layout** command.
- Click the New Master Layout icon in the Navigator or the Organizer.



Master Layout Title Block


The default Master Layouts include commonly used title blocks which are composed of 2D drawing elements and text items, including Autotext.

To edit or create a title block: Open the Master Layout and use the 2D editing and text tools to add or modify the content.


For example, the Title Blocks of the Master Layout pictured below includes

- Text blocks with company and project data,
- a logo image

- text blocks with Autotext items: Drawing data, Layout and Master Layout data

 <p>GRAPHISOFT ARCHICAD Sample Project - Hillside House Rose Str. 12. Springfield City Hungary 0000</p>	
Drawing Name #Layout Name	
Drawing Status #Status	
Modified by #Modified by	Date
Checked by #Checked by	Date
Drawing Scale 1:###	
Layout ID #LayID	Revision #RevID
Master Layout Name A2 Landscape	Master Size 594 x 420
Layout Number (in this Subset) #Layout Number in Current	
Subset of #Number of Layouts in Current Subset	

Title block on Master Layout

 <p>GRAPHISOFT ARCHICAD Sample Project - Hillside House Rose Str. 12. Springfield City Hungary 0000</p>	
Drawing Name East Elevation	
Drawing Status Published	
Modified by GRAPHISOFT SE	Date
Checked by GRAPHISOFT SE - Audit	Date
Drawing Scale 1:50	
Layout ID A-RD1-15-ELE-3	Revision 01
Master Layout Name A2 Landscape	Master Size 594 x 420
Layout Number (in this Subset) 3 of 4	

Title block on actual Layout

See also [Autotext](#) and [Revision History Data on Layout and in Layout Settings](#)

Display of Master Layout Items

Color

By default, Master items are displayed in their original colors.

Assign a uniform color at **Options > Work Environment > On-Screen Options**.

Above Layout/Below Layout

Display Master items below or above the Layout: set this in Master Layout Settings.

[See Master Layout Settings.](#)

Hide Master Items

To hide the on-screen display of Master elements on all Layouts: turn off **View > On-Screen View Options > Master Items on Layout**.

Drawings in the Layout Book

Drawings are Archicad views and external files that have been placed onto a Layout.

Drawings can be created out of views from the current project file, or imported from external Archicad project files. External drawing and picture files as well as PDF files can also be placed on the Layout.

Note: Drawings can also be placed into Model Views. ([See *Drawings in Model Views*.](#))

You can customize the size, scale, title and framing options for your drawings in the Drawing Settings dialog box.

[See *Drawing Settings*.](#)

Topics in this section:

[Place Drawing on Layout](#)

[Arranging Multiple Drawings on the Layout](#)

[Drawing Frame](#)

[Drawing Titles](#)

[Managing and Updating Placed Drawings](#)

Place Drawing on Layout

The following can be placed as Drawings on a Layout

- view from current Archicad project
- view from external Archicad project
- external Drawing-type file, such as PDF or other drawing format

A view placed onto a Layout automatically becomes a Drawing.

Topics in this section:

[Place view from current project \(drag and drop\)](#)

[Place view from current project \(Drawing tool\)](#)

[Place view from external Archicad project \(menu command\)](#)

[Place view from external Archicad project \(Navigator Project Chooser\)](#)

[Place external drawing file \(multiple methods\)](#)

[Place selected pages from a PDF](#)

[Save View and Place on Layout \(command\)](#)

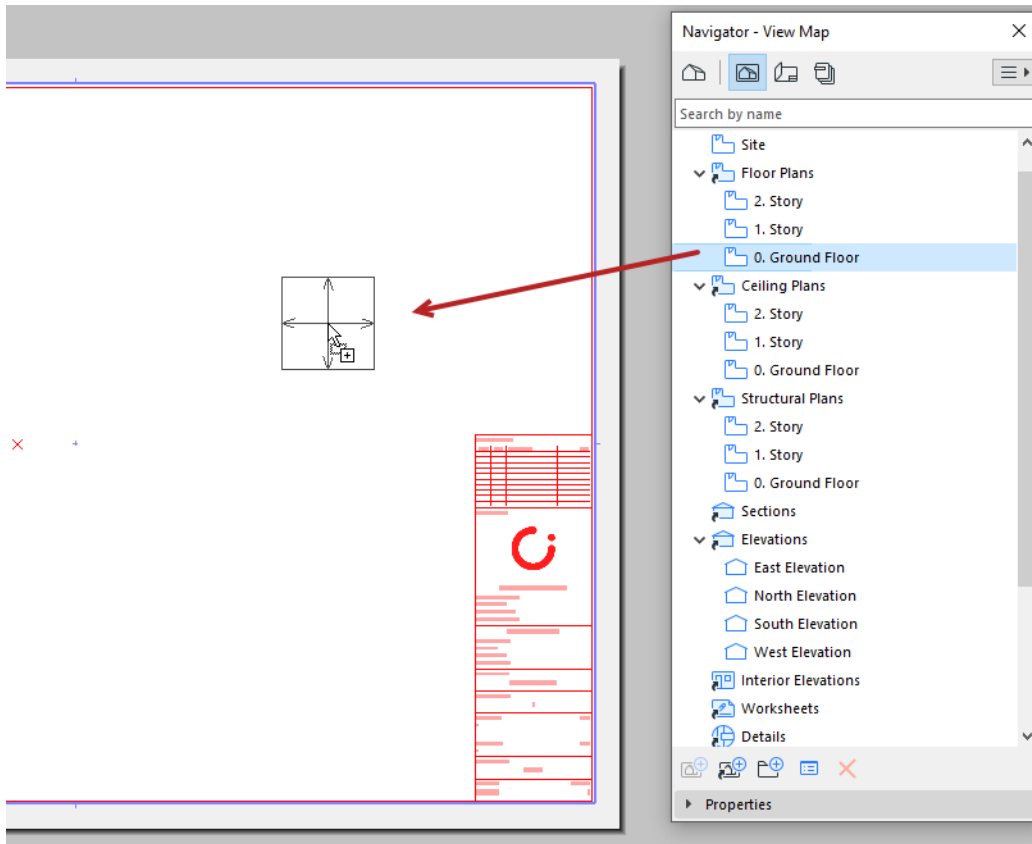
[Place Drawing using Organizer \(Layout Editor mode\)](#)

Place view from current project (drag and drop)

1. Open the Layout.
2. From the Navigator, drag and drop items (viewpoints or views) directly onto the Layout. They are now Drawings.

Notes:

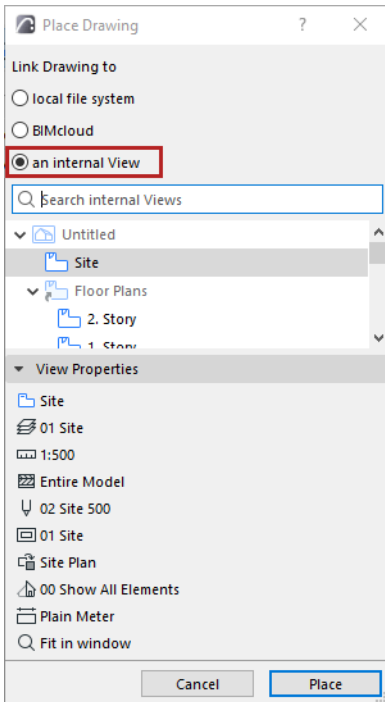
- A viewpoint added to a Layout is also converted to a view and added to the View Map.
- If you drag and drop several items to the Layout, they are arranged based on the Drawing Placement settings in Master Layout Settings.



Drag view onto Layout; it becomes a Drawing

Place view from current project (Drawing tool)

1. Open the Layout.
2. Activate the Drawing tool, then click on the Layout.
3. From the appearing dialog, choose "Internal view", then select a view.
4. Click **Place**.



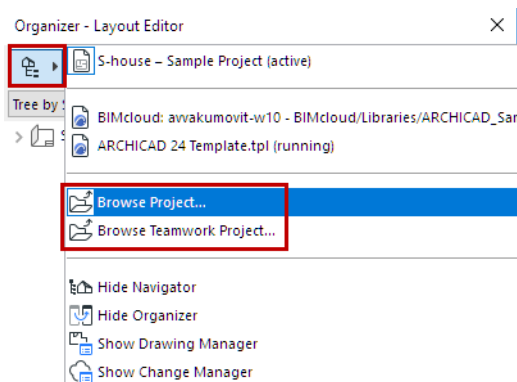
Place view from external Archicad project (menu command)

1. Use **File > External Content > Place External Drawing**
2. From the appearing dialog: choose an Archicad project (PLA or PLN).
3. From the next dialog: choose the view(s) to import and a method for adding them to the Layout Book.
4. Click **Place**.

Note: Placing a view from another Archicad project file will open another instance of Archicad. To avoid this: go to **Options > Work Environment > More Options** and check “Do not launch new instance of Archicad.”

Place view from external Archicad project (Navigator Project Chooser)

1. Open the Navigator or Organizer of the active project.
2. From the Project Chooser button: click **Browse Project** or **Browse Teamwork Project**.



3. Choose an external Archicad project (version 17 and up).

If browsing a Teamwork project: Choose the BIMcloud, then the project you need. The source project and host project must be the same Archicad version.

If needed, log in to BIMcloud. [See *Log in to BIMcloud*](#).

4. Click Open.

The project tree structure is now shown in the Navigator.

5. Drag and drop views to onto the Layout in the active window.

Note: Placing a view from another Archicad project file will open another instance of Archicad. To avoid this: go to **Options > Work Environment > More Options** and check “Do not launch new instance of Archicad.”

Place external drawing file (multiple methods)

1. Open the Layout.

2. Drag and drop a drawing file (one at a time) from your file system

Or browse the file system or BIMcloud using any of these methods:

- Activate the **Drawing tool**, then click on Layout.
Choose “local file system”, then browse.
- **File > External Content > Place External Drawing**
- **File > External Content > Place External Drawing from BIMcloud**

Choose the BIMcloud, then the project you need.

If needed, log in to BIMcloud. [See *Log in to BIMcloud*](#).

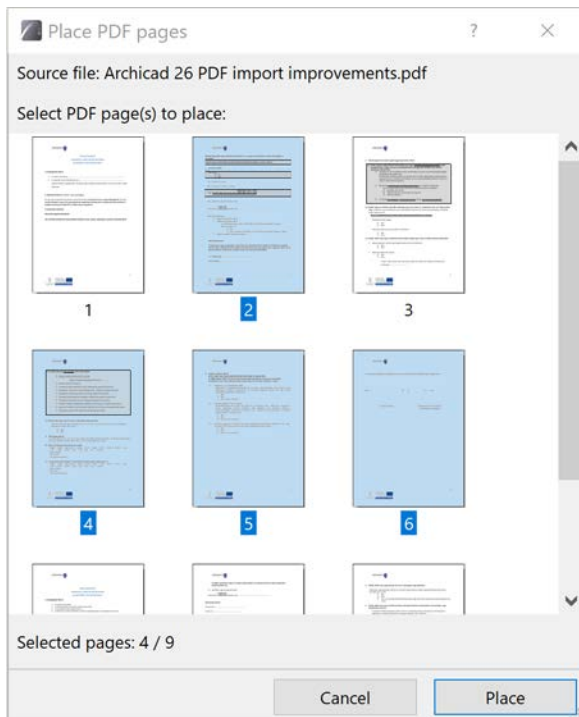
Note: To link a drawing from a Teamwork project, the source Teamwork project must have the same version number as the host project.

Place selected pages from a PDF

This works if the PDF is placed using drag-and-drop or a File menu command.

If the imported drawing is a multi-page PDF:

- 1.** A chooser dialog appears. Select one or more or all pages.
- 2.** Click **Place**.



Save View and Place on Layout (command)

Note: This command not available in every localized Archicad version.

This command is available:

- from the active window's context menu
- from the view/viewpoint's context menu in the Pop-up Navigator (in case of a view, the command is called simply **Place on Layout**)
- from the **Document** menu

This command opens the layout window (the most recently active Layout, or the first Layout in the Layout Book).

Click to place the drawing.

Place Drawing using Organizer (Layout Editor mode)

In the Organizer's Layout Editor, drag and drop viewpoints, views and external files from the hierarchy list on the left into the Layout Book on the right. (Or use the **Place Drawing** command on the left side of the Organizer.)

[See Organizer.](#)

The placement of the drawings obeys the following rules:

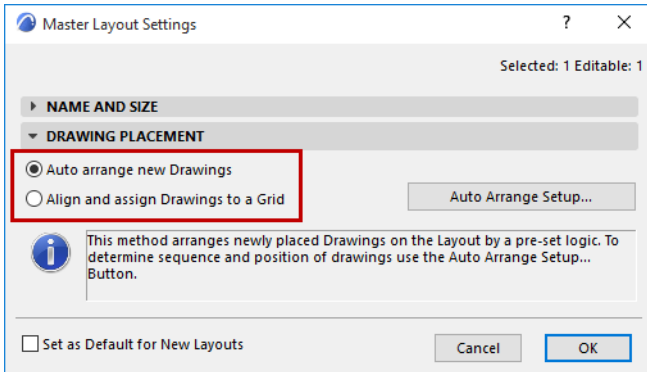
- If you drop an item on the name of an existing layout, the drawing will be added to that layout.
- If you drop an item on the name of a subset or the Layout Book itself, the drawing will be placed on a new layout created automatically inside the subset (or at the topmost level of the Layout Book).

Arranging Multiple Drawings on the Layout

You can arrange and manipulate placed Drawings on the Layout as needed. (Although you can place any number of Archicad views on a Layout, you cannot embed views within each other.)

When placing multiple Drawings onto a layout, the Drawings will arrange themselves according to the Drawing Placement logic defined in Master Layout Settings.

Note: The default logic is “Auto Arrange.” If you are placing only a single drawing on the Layout, it will be placed in the center of the Layout.



If the Layout uses a Grid for Drawing IDs, the drawings will be placed in the appropriate cells.

See [Drawing Placement](#).

Drawing Frame

Each placed Drawing has a Frame. Use it in two ways:

- **Fit Frame to Drawing** to include the full content of its source view
- **Manually Resize** the Frame to limit the visible content (and optionally crop it out)

Why Crop the Drawing Content?

If you resize a Drawing Frame, you can optionally crop out the unseen Drawing content and delete it from the Drawing cache. Cropped content no longer has to be checked or refreshed (until it is restored). Thus, **cropping Drawing content reduces file size and can speed up Drawing update.**

Frame editing and Drawing crop features work the same for all types of Drawings, including those with an external source.

Cropped content can be restored at any time, as long as its source remains available.

Topics in this section:

[Fit Frame to Drawing](#)

[Resize Frame and Crop Content](#)

[Restore Cropped Content](#)

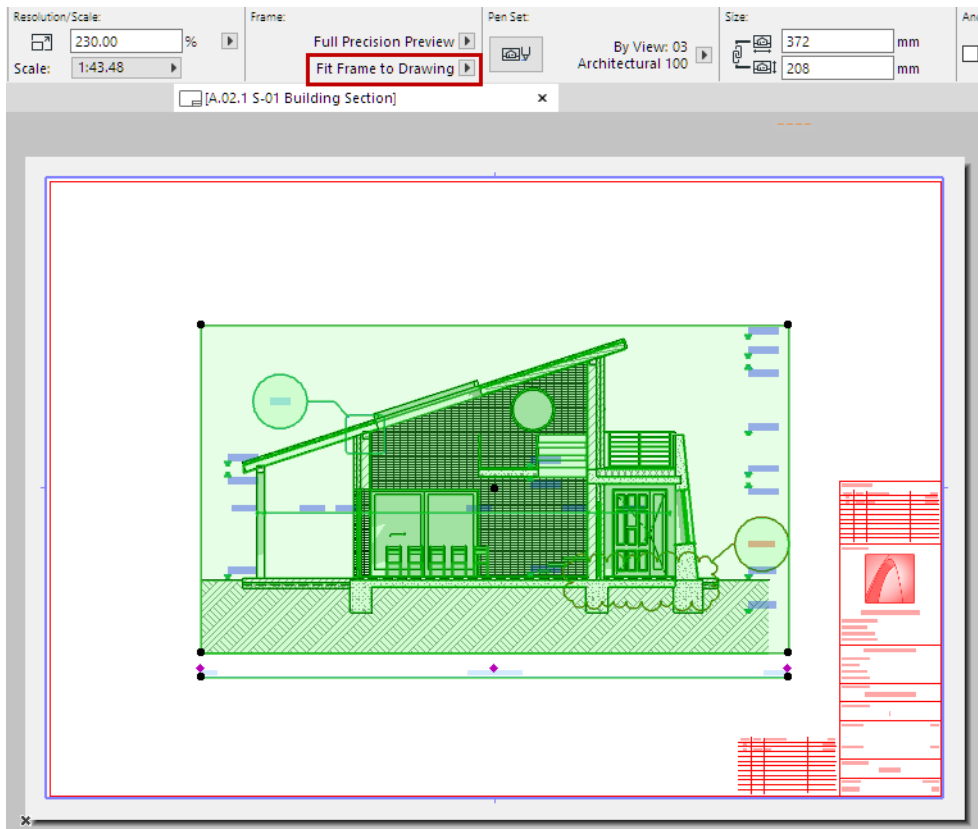
[Crop Drawing\(s\) to Frame Command](#)

[Uncrop Drawing\(s\) Command](#)

Fit Frame to Drawing

By default, the Drawing Frame fits to the full Drawing content: the **Fit Frame to Drawing** control is active, in Drawing Settings or Info Box.

This Drawing Frame is shown with a solid line: it is *not* cropped.



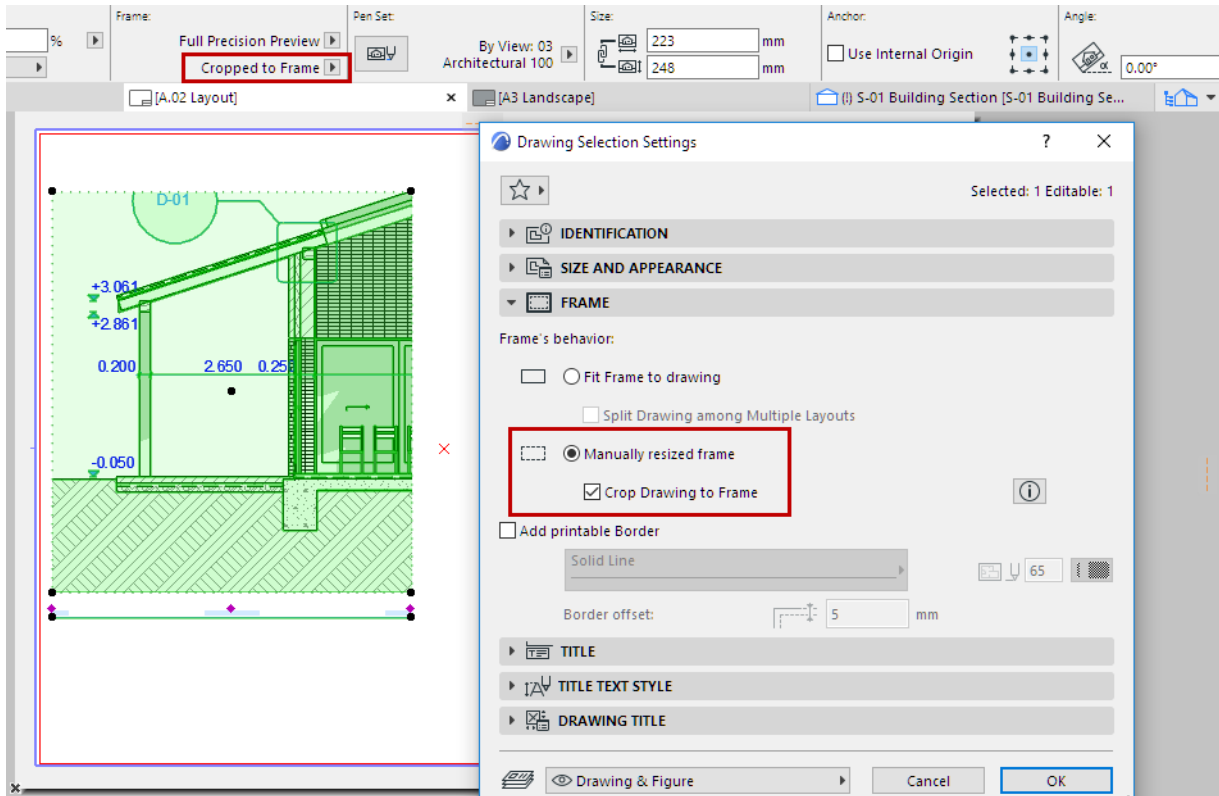
Fit Frame to Drawing (Solid Line)

Resize Frame and Crop Content

1. Select any edge of the Frame and use pet palette editing commands to modify it.
2. The **Manually resized frame** button is now activated (in Drawing Settings or the Info Box).
3. By default, **Crop Drawing to Frame** is also checked: as you resize the Frame, the Drawing content is cropped accordingly.

The Drawing Frame of a cropped Drawing is shown with a dotted line.

Cropped content can be restored at any time, as long as its source remains available. (see below)

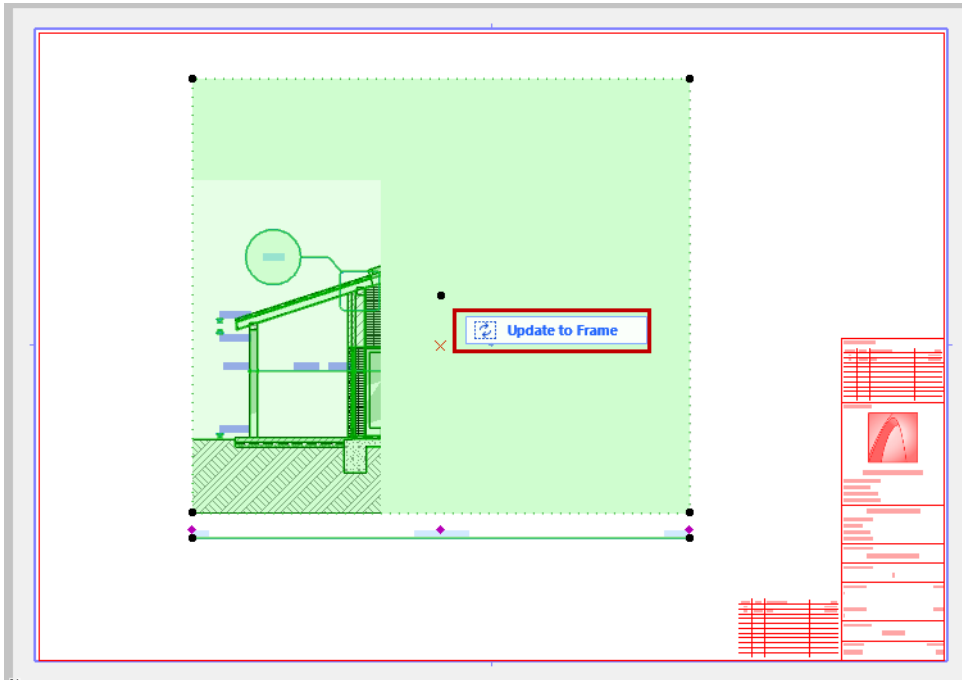


Cropped to Frame (Dotted Line)

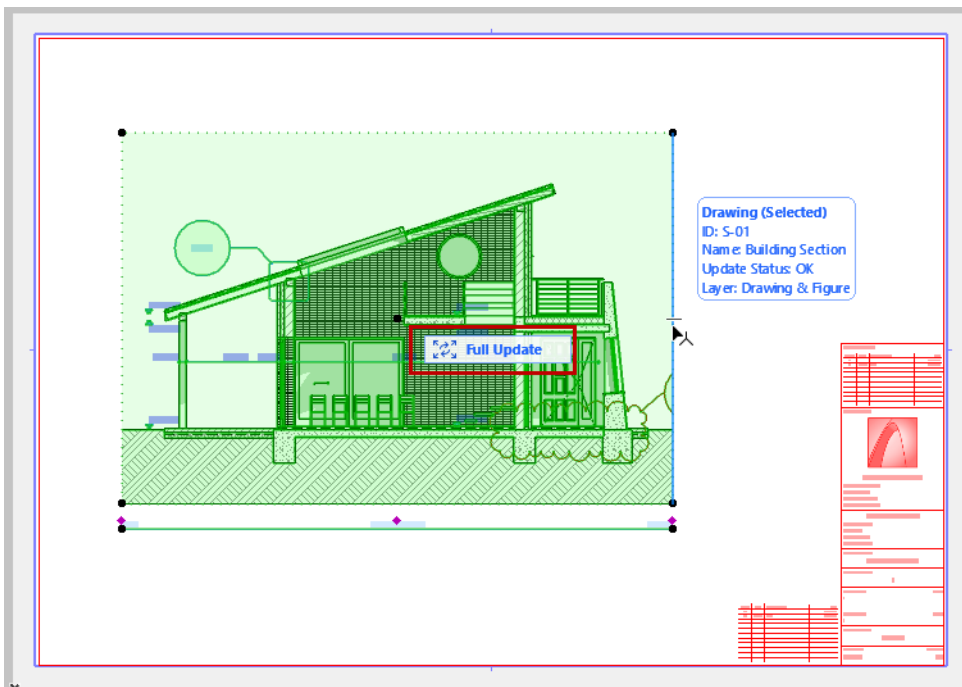
Restore Cropped Content

On a selected cropped Drawing:

1. Drag the Frame to the desired position.
2. The cropped content is indicated by the shaded area. The **Update to Frame** command appears.



3. Click **Update to Frame**. The masked (cropped) drawing content is now restored.
4. If you continue editing the Frame, the command changes to **Full Update**.
 - Click **Full Update** to update the full Drawing content in the background. All of the masked-out content is now displayed, so you do not have to keep updating while you edit the shape of the Frame.



Crop Drawing(s) to Frame Command

For a selected Drawing with a Manually resized Frame, which is not cropped:

Use the **Crop Drawing(s) to Frame** command from:

- Context menu of the Drawing or the Drawing's Navigator item
- Context menu of the Layout tab
- Info Box
- **Document > Drawings** menu
- Drawing Settings (Frame panel)

The Drawing status is now **Cropped** (as shown in the Info Box and Drawing Manager.) Any Drawing content that falls outside the current extent of the Frame is deleted from the Drawing cache.

The Drawing Frame of a cropped Drawing is shown with a dotted line.

Note: Cropped status can be set as the Drawing Tool default, in Drawing Settings or the Info Box.

Uncrop Drawing(s) Command

For any selected, cropped Drawing(s):

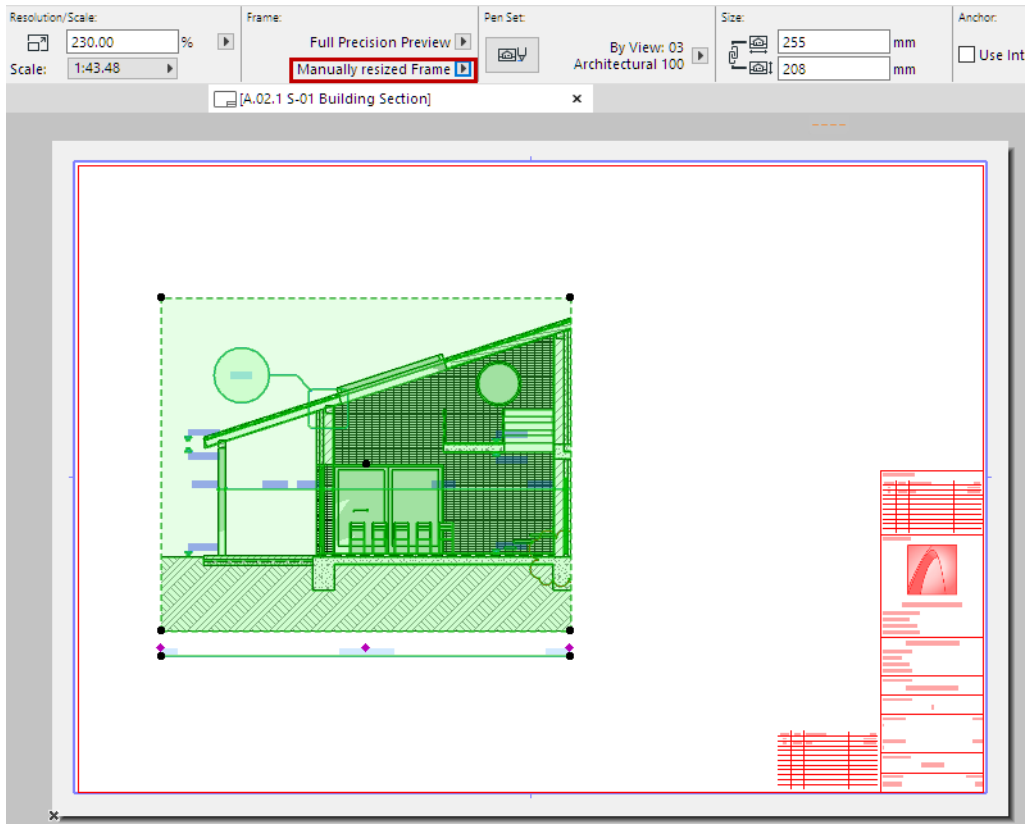
Use the **Uncrop Drawing(s)** command from:

- Context menu of the Drawing, or the Layout tab
- Info Box
- **Document > Drawings** menu
- Drawing Settings (Frame panel): Uncheck **Crop Drawing to Frame**

The Drawing status is now Uncropped (as shown in the Info Box and Drawing Manager.) Its Frame is now Manually Resized.

- For an **Auto-update Drawing**: The **Uncrop Drawing(s)** command restores previously cropped content to the Drawing
- For a **Manual-update Drawing**: To restore cropped content, use the **Uncrop Drawing(s)** command, then update the drawing.

A Manually Resized Drawing Frame is shown with a dashed line:

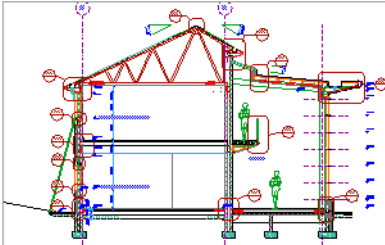


Manually Resized Frame (Dashed Line) - Drawing in Uncropped Status

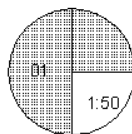
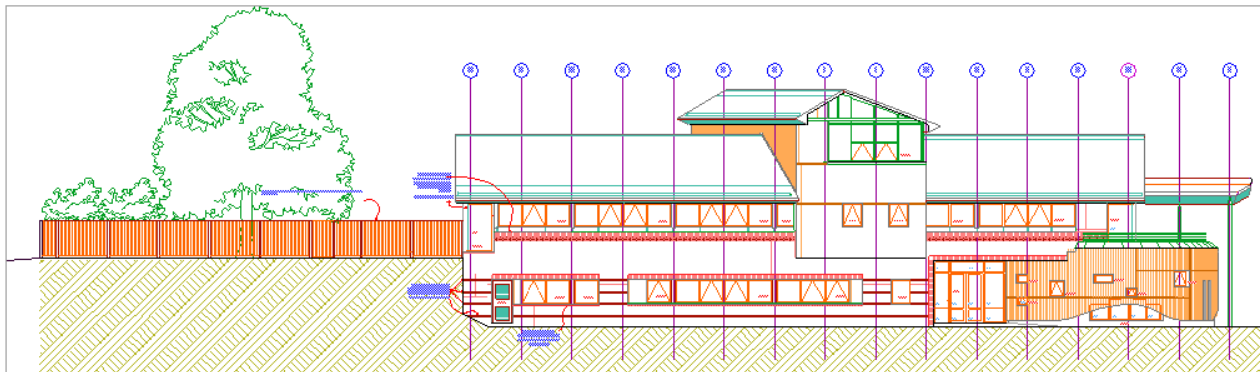
Drawing Titles

Each Drawing placed on a Layout can have a title containing graphic elements and information referring to the drawing data, such as Drawing ID, Name, and Scale.

The Drawing title is based on a Title Type. Title Types are GDL Objects that can be assigned to Drawings by default (automatically) or assigned manually to selected Drawings.



A Section
1:50



Topics in this section:

[Assign Title to Drawing](#)

[Default Drawing Title](#)

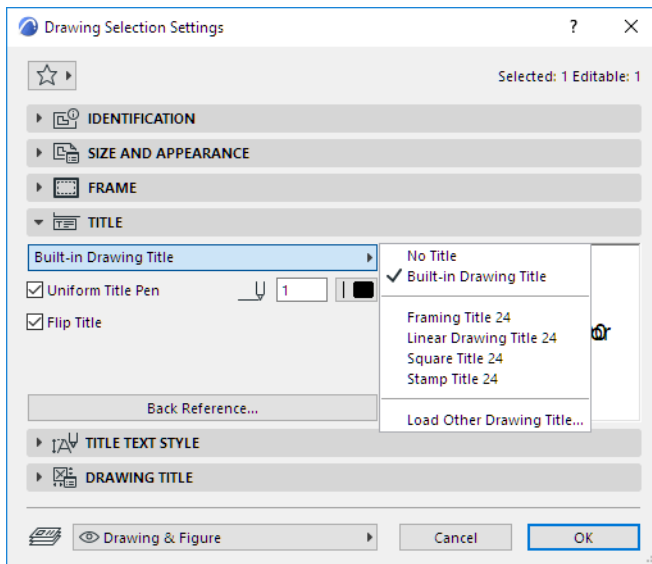
[Create New Title Type](#)

[Edit Title Type Object](#)

[Back Reference to Layout Containing Source Marker](#)

Assign Title to Drawing

1. Select the Drawing and open the Drawing Settings dialog box.
2. In the Title panel, select a Title type from the pop-up menu.



Default Drawing Title

To assign a title by default to each newly placed Drawing:

1. Open Drawing Default Settings
2. In the Title panel, select a Title type from the pop-up menu.

Drawing Titles are laid out automatically with the associated drawing.

Create New Title Type

1. In the Layout window, use the available 2D tools (e.g. lines, text blocks) to design a Title Type.
2. Select Title Type components you have created.
3. Use **File > Libraries and Objects > Save Selection as > Drawing Title** to save this object.

Once saved, your Title Type is available from the pop-up in the Title panel of Drawing Settings.

Edit Title Type Object

You can edit a custom-created Title Type graphically:

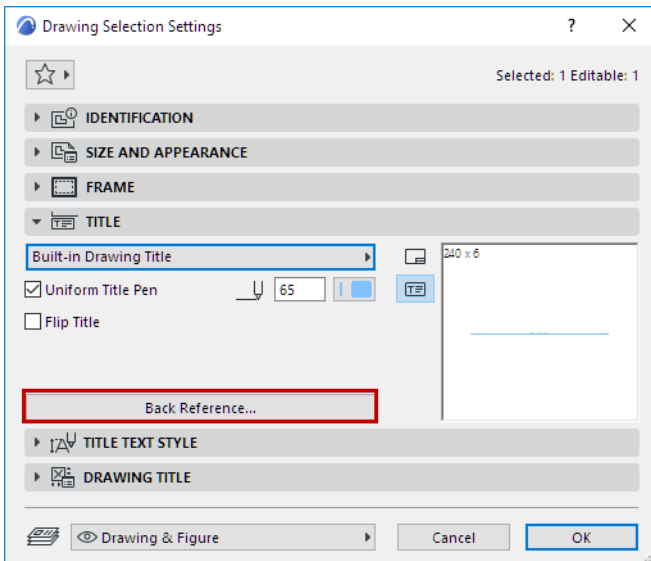
1. Use **File > Libraries and Objects > Open Object**.
2. Select the Title Type you want to edit.
3. Click “2D Symbol” to access the graphical editing window for this object.
4. Make your changes using 2D tools.

Note: Make sure the scale of the GDL 2D symbol window is set to 1:1, to make editing easier.

Back Reference to Layout Containing Source Marker

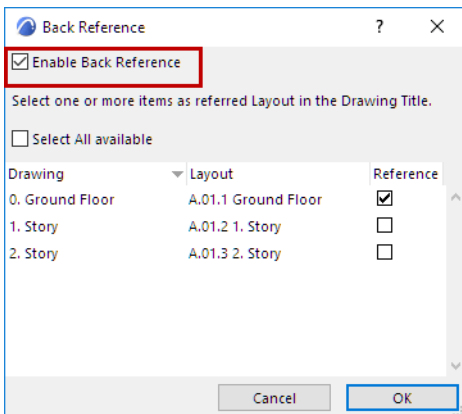
The Title Panel of Drawing Settings gives you access to the Back Reference function.

Back Reference allows you to add Layout ID information to the Drawing Title: specifically, the ID of one or more Layouts that contain a Drawing that shows the *source marker* of the current Drawing. Back Reference is useful if the current Drawing is based on a Section, Elevation, Interior Elevation, Detail or Worksheet.



Click **Back Reference** to bring up the dialog box of the same name.

This dialog box lists every Layout that displays the source marker of the current drawing.



First check the **Enable Back Reference** box, then select one or more of the Layouts whose ID you wish to display. (Check **Select All available** if you want the Drawing Title to display the ID's of *all* Layouts listed here.)

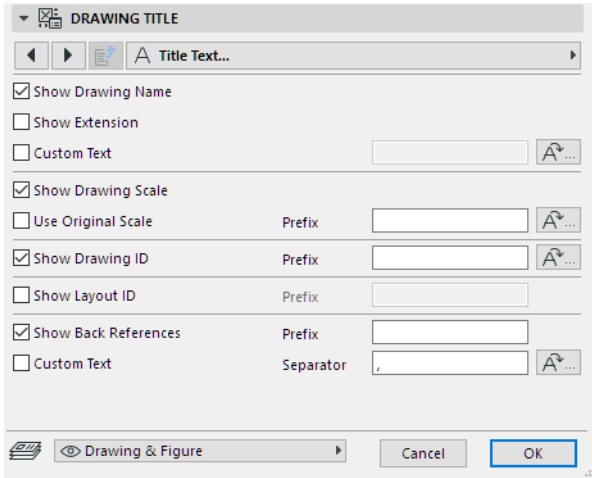
Click OK to close the dialog box and return to Drawing Settings.

The Drawing Title Preview shows an Autotext in the place where the Drawing Title will contain the Layout ID information.

In this example, if you have checked “Select All available,” the final Drawing title looks like this:



Note: Make sure that the parameters of the Drawing Title object you are using have enabled the Back Reference function:



Managing and Updating Placed Drawings

All drawings placed in the Archicad project belong to one of the two following types:

- **Automatic update.** The drawing placed on the layout will be updated automatically when the layout is activated (brought to front or selected for output). Automatic updates also occur before publishing. If an auto-update drawing comes from a different source than the currently opened project file, then Archicad will check whether it requires updating or not, and perform an update if necessary.

Note on Auto-Update Drawings in Teamwork: In a Teamwork project, an auto-update Drawing with an external source (e.g. external project, PDF, DWG, PNG) will be updated *only if you have reserved that Drawing*.

- **Manual update.** The drawing is frozen in the state its source view or file was in when placed on the layout or updated. This can be useful if you wish to keep the drawings untouched until they are approved or you are ready to move on to the next stage in your work. Manual-update Drawings are updated only if you issue an update command.

Note on Manual-Update Drawings in Teamwork: If you have reserved a manual-update Drawing, then you alone have the right to update that Drawing.

For any placed Drawing, set its default update type in Drawing Settings.

Use the **Drawing Manager** to check the status of all drawings in the project, modify their update type, reestablish, repair or change links.

You can also check Drawings and access Drawing Manager through the Action Center. (**File > Info > Action Center**)

[See *Drawing Manager*](#).

Topics in this section:

[Update Linked Drawings](#)

[Re-Link PMK File](#)

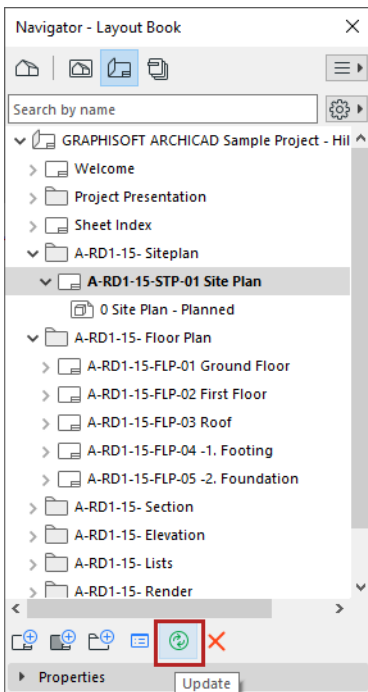
[Note on External Drawing Update \(Drawing placed from BIMcloud\)](#)

Update Linked Drawings

When opening a project with drawings whose sources have been modified, the Drawings page of the Action Center will alert you.

To update selected Drawings, do one of the following:

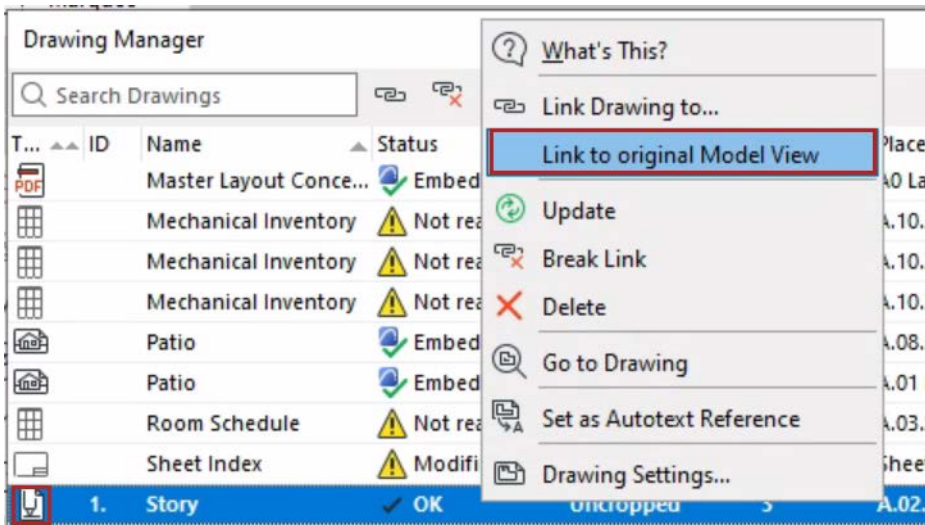
- In the Navigator, Organizer or Drawing Manager:
 - Select the drawing(s), then use **Update** from the context menu, or the palette's **Update** button.



- In the Layout window: Select the drawing(s), then use **Update** from the context menu

Re-Link PMK File

If the Layout Book contains Drawings whose source is a linked PMK file (created in the Publisher out of an Archicad model view), you can re-link all these PMK-based drawings back to their original model view. Select the PMK files in the Drawing Manager (multi-selection is available) and use the following command from the context menu: **Link to original Model View**.



See the workflow description at [Layout Workflow: Larger Projects](#).

Note on External Drawing Update (Drawing placed from BIMcloud)

Once you place a Drawing whose source is an external Teamwork project, a copy of the Drawing is created on your machine. When you subsequently update the Drawing, Archicad will automatically synchronize your local copy with the project on the server to ensure that the Drawing in your host project is fully updated.

There is one case in which Archicad will not update the Drawing placed from an external Teamwork project. This occurs if you are logged into the source Teamwork project, from your current computer, and if you have any unsent changes in that Teamwork project. In this case, Archicad detects that the Drawing may not be in fully updated condition on the server, and will not update it. You will be warned if this occurs. In this case, send in any unsent changes from the source Teamwork Project, then update the placed Drawing again from the current project.

Layout and Drawing IDs

Both Layouts and Drawings have IDs assigned to them, as an aid in organizing, navigating and outputting your project. The primary use of assigning Layout IDs is to achieve the automatic numbering of the Layout Book. While an automatic logic can be applied to the whole Layout Book, Archicad provides maximum flexibility in customizing the assignment of IDs to meet your preferences.

Topics in this section:

[Simple Layout Numbering](#)

[Automatic Layout ID Assignment](#)

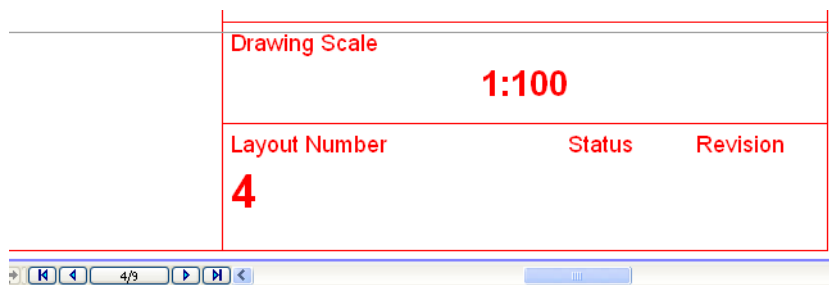
[Book Settings Dialog Box](#)

[Custom Layout/Subset IDs](#)

[Drawing IDs](#)

Simple Layout Numbering

The simplest way to number your Layouts is to number each layout sequentially, like the pages in a book. To do this, you do not need to consider Layout IDs; just place an [Autotext](#) for “Layout Number” in the Master Layout. The Layout Number is a number assigned to each layout in the order it is created; it is shown at the bottom of your Layout Window. Through the Autotext, this number will appear on each Layout.



However, if you want to use a “smart” numbering system that accounts for adding and removing Layouts in the Layout Book hierarchy, you should use the [Automatic Layout ID Assignment](#) logic.

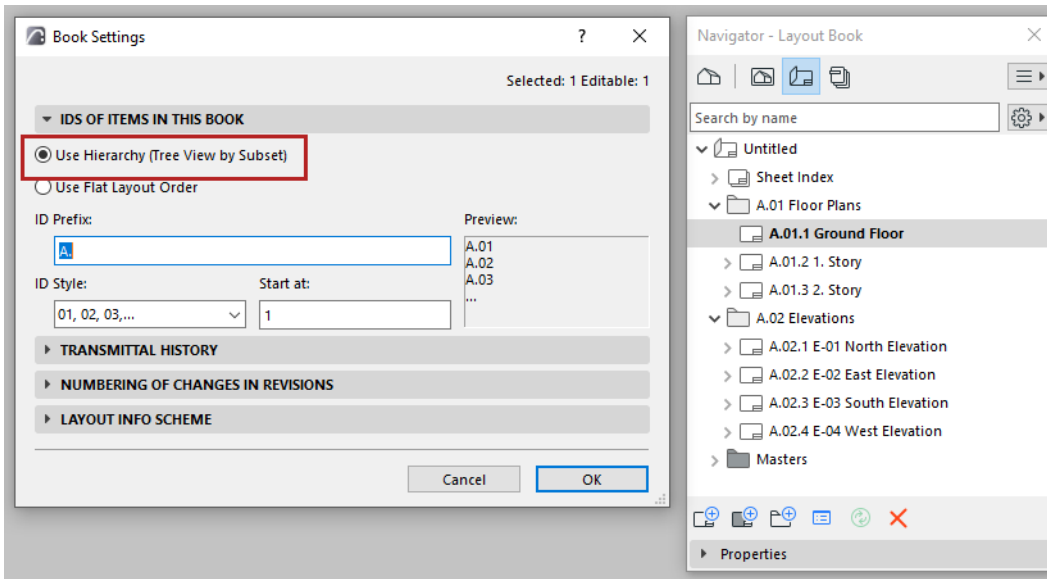
Automatic Layout ID Assignment

Due to the complexity of most architectural projects, and to enable you to retain the desired numbering sequence even after Layouts are added and removed from the Layout Book, most users will prefer to use an automated method which assigns an ID to each layout. The smart ID assignment system automatically adjusts itself whenever the Layout Book is changed (for example by the addition of new Layouts, changes in the Layout order or in the document structure).

“Layout ID” is available as an [Autotext](#) entry, so if you add the Layout ID Autotext to the Master Layout’s Title block, then each Layout will be numbered with this ID.

Go to **Document > Layout Book > Book Settings**. Choose your preferred method for assigning Layout IDs:

- **Use Hierarchy** (Tree view by subset): Use this method if you want your Layout IDs to reflect the Layout’s location in a particular subset. IDs will reflect the tree structure hierarchy: Layout IDs will “inherit” the IDs of the subsets in which they are located - as in the illustration below.

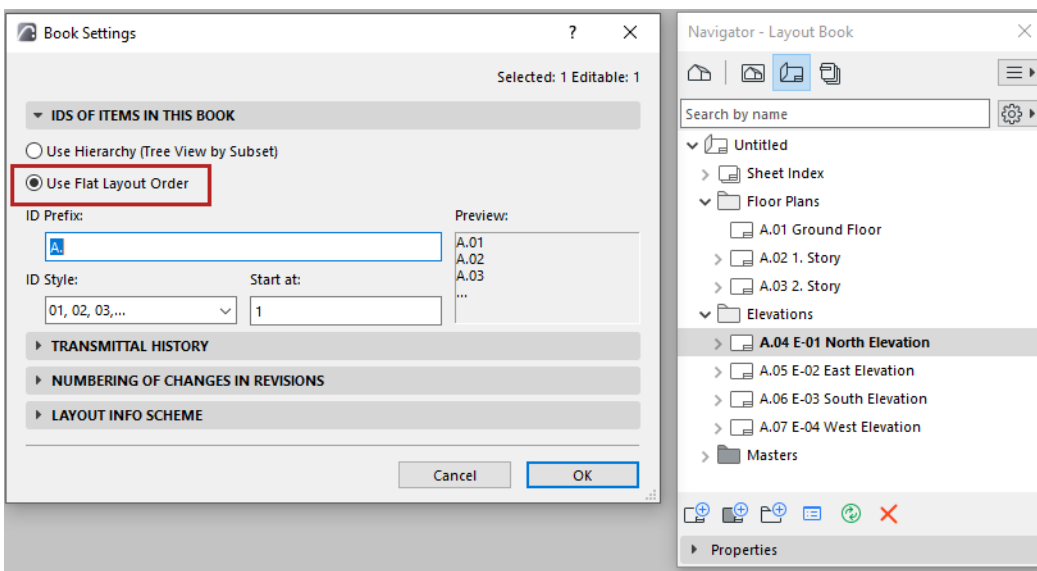


Both the prefix and the style (1-2-3, A-B-C, etc.) of the layout/subset ID can be customized. The resulting layout ID is usually a letter-number combination, such as A-03/a.

See [Book Settings Dialog Box](#).

Note: At any level of ID assignment (Book Settings, Layout Settings or Subset Settings), you can choose a style of “No ID”. As part of the automated ID system, every sub-item in that group (e.g. every layout in that subset) will be assigned an “empty” ID, which is like no ID at all.

- **Use Flat Layout Order:** This method will assign an ID to Layouts only (not taking subsets into account), from top to bottom in the Navigator Layout Book. Layouts will receive sequential IDs regardless of their hierarchical position - as in the illustration below. If you change the order of Navigator items, their IDs will change accordingly.



Custom Layout/Subset IDs

Even if you use the [Automatic Layout ID Assignment](#) logic defined in Book Settings, you can still assign a custom ID to any Layout or Subset.

In Layout or Subset Settings, assign a custom ID to the selected layout or subset. The Layout/Subset will still “count” as part of the ID sequence (e.g., if the previous Layout is A-02, the custom Layout might be Suppl.-02, and the following Layout will be A-03.)

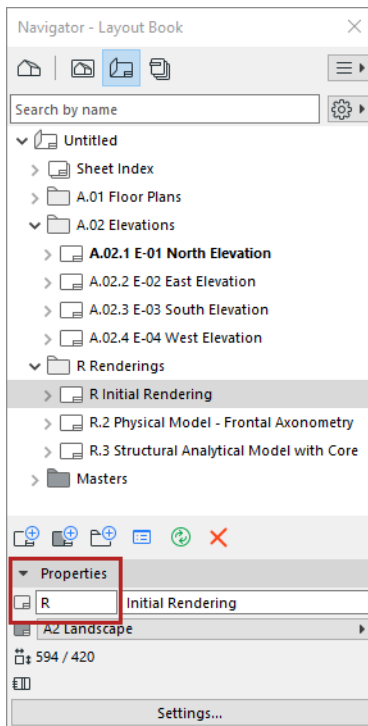
- If you do not want the custom ID to be counted in the automatic ID sequence, check the “Do not include this Layout (Subset) in ID sequence” box.

The image shows a 'Subset Settings' dialog box with the following configuration:

- SUBSET IDENTIFICATION:**
 - ID: Do not include this Subset in ID sequence
 - Assign ID automatically
 - Custom ID (highlighted with a red box)
 - Name: Renderings
 - ID: Render
- IDS OF ITEMS IN THIS SUBSET:**
 - Continue using ID assignment of Upper Levels
 - Customize ID assignment
 - Use Prefix & IDs of Upper Levels
 - Add Prefix for this Subset
 - ID Style: 1, 2, 3, ...
 - Start at: 1
 - Preview: Render.1, Render.2, Render.3, ...

Once you have customized a Subset ID, you then have the further choice: in the **ID of Items in this Subset** panel, you can make the IDs of Layouts in this subset either follow the new logic (incorporating the subset’s custom ID), or else retaining the ID logic of the overall Layout Book structure irrespective of the customized subset ID.

Another way to assign a custom Layout or Subset ID is by typing it directly into the Properties panel of the selected Layout (Subset) in the Navigator Layout Book.



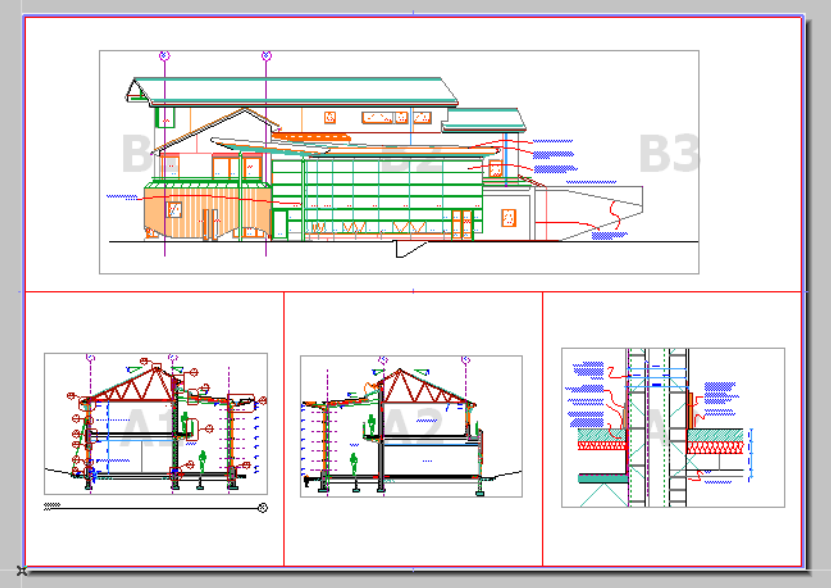
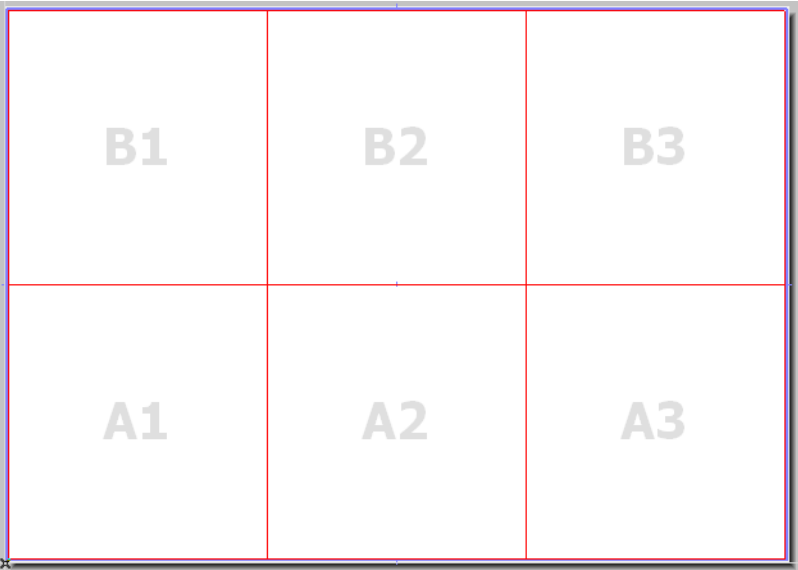
Drawing IDs

Each drawing placed onto a layout is automatically assigned a **Drawing ID**. The Drawing ID appears in front of the Drawing Name in the Navigator and the Organizer palette. The method of ID assignment is defined in Drawing Settings. Note that the Drawing Name and the Drawing ID are defined separately.

In the Drawing Identification panel of Drawing Settings, choose one of these options for defining the Drawing's ID:

- **By View ID:** The Drawing will have the same ID as its source view. This option is often the most useful if your Layouts contain one Drawing each. (This option is only available if the Drawing's source is an Archicad view.)
- **By Layout:** The Drawing will get an ID based on its "Parent" Layout: the Drawing ID will have the Prefix/Style you define in Layout Settings, in the "IDs of Drawings on this Layout" panel. This option is most useful if your Layouts contain multiple Drawings.
- **Custom:** Assign any custom ID to this Drawing.

You can also use the Grid for Drawing IDs feature of Master Layout Settings to automatically place drawings in cells and assign IDs in a matrix or a flat structure.



Layout Workflow Schemes

Archicad's Integrated Workflow Control ensures optimal workflow for projects of any size, complexity and type (single-user or shared project). The following outline introduces some workflow models you might use in various situations.

Topics in this section:

[Layout Workflow: Smaller Projects](#)

[Layout Workflow: Larger Projects](#)

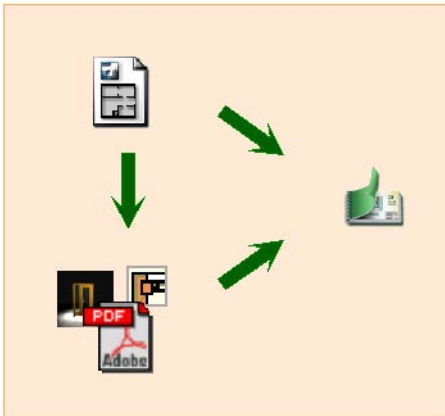
Layout Workflow: Smaller Projects

Smaller projects typically mean smaller design teams (or solo practices) and smaller drawing sets. Smaller practices tend to be less structured, and more often than not the same architect takes care of the entire design process, including the layout phase.

In such cases, the project usually consists of a single file which contains all the drawings that make up the documentation, with the possible exception of a number of external files such as pictures, PDF format tables and descriptions, and standard details.

When using the single-file model, you can build up the structure of your layout book by defining the necessary number of subset folders and adding to them the viewpoints and views set up previously in the project file.

If the placed drawings are defined as automatically updated, the layout book will be continuously in synch with the design work. The activated layout will reflect the latest changes made on the plans, sections and other drawings.



Layout Workflow: Larger Projects

In larger architectural practices which handle large, multi-building projects, it may be more practical to break down the large project into smaller entities. Each of these subordinate entities (for example, buildings located on the same site) will then constitute a project file, which can still refer to a single set of templates, modules and libraries.

However, the documentation will probably extend to the entire project, taking drawings from all the project files.

In this case, it makes sense to define a single Layout Book that will accommodate views and files originating from all of the project files that make up the project.

There are basically three solutions:

Single File Containing Model and Layout Book

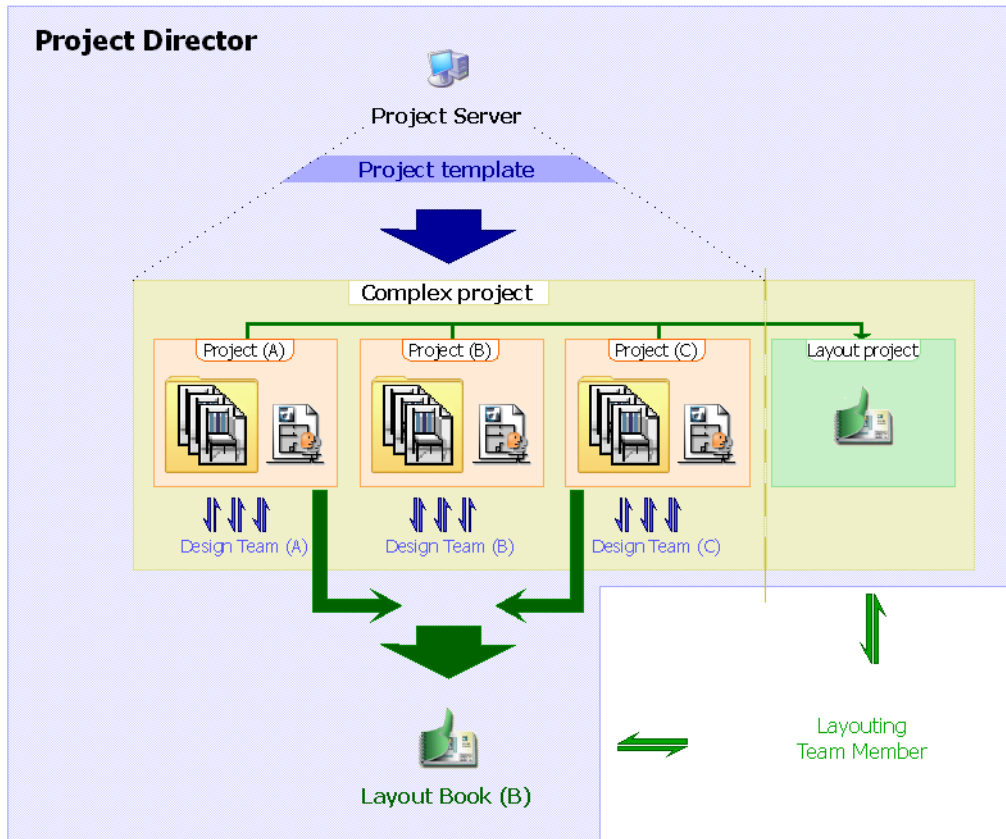
If there is a large main file in the project with one or two smaller additional files, you can opt to build up the structure of the layout book in the larger file as you would in the single file model above, and then also open the smaller files, locate their views in the Organizer, and add them to the Layout Book.

[See *Place view from external Archicad project \(Navigator Project Chooser\)*.](#)

Multiple Project Files, with Separate File for Layout Book Only

If there are many project files of about the same size and complexity, it may make sense to create an additional project file that does not contain any construction elements, just the Layout Book. You can then open the other files one by one and add their views to the Layout Map, possibly by creating a subset folder for each of them or by defining another custom hierarchical structure.

When a layout is activated in the multiple file model, Archicad checks whether it contains drawings that originate from project files other than the active one. If there are, it will then start updating those drawings that are set to be updated automatically. If the file containing the given drawing is open in another Archicad session, the update occurs instantly. If the file containing the given drawing is not currently open, then another Archicad session will be launched to open it.



Multiple Project Files, Separate Layout Book, Separate Drawings (PMK Files)

This workflow can be appropriate if several individuals are working concurrently on the model and documentation through a network.

In this workflow, as in the previous one, the model is located in one or more project files, and the Layout Book in a separate project file. The Layout procedure, however, is different: Layouts will contain Drawings whose source is in an external file, rather than directly linked to project views.

The model designer will create views as usual in the Archicad file. Instead of placing these views directly onto the Layout, s/he will use the Publisher function to save the views in PMK format, a native Archicad drawing format which takes up relatively less hard drive space than many other formats.

[See *Publisher File Formats*.](#)

The architect responsible for the Layout Book will then place these PMK drawings on the Layout, using the Drawing Tool or the **File > External Content > Place External Drawing** command.

[See *Place external drawing file \(multiple methods\)*.](#)

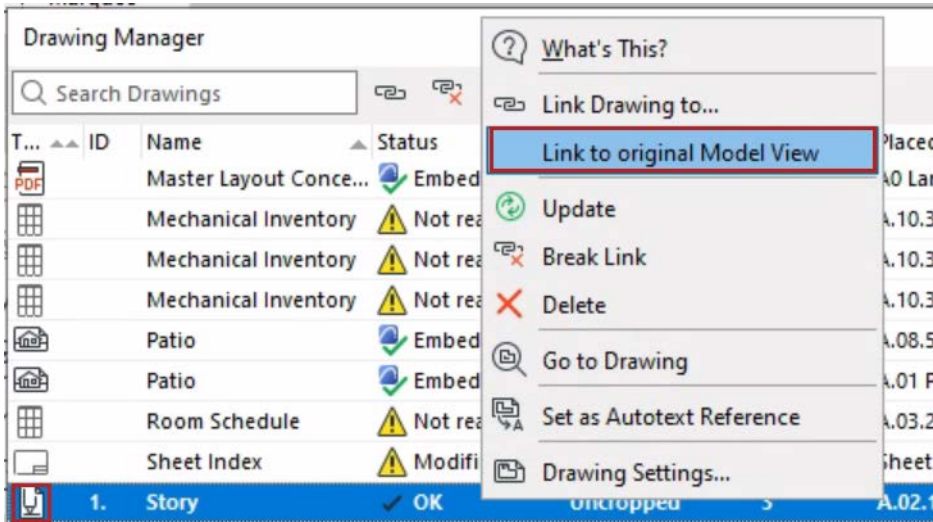
These placed Drawings, like any other Drawing, are either Auto-update or Manual-update. Their source is the PMK file. Consequently, it is the model designer's responsibility to ensure that every time an Archicad view is modified, the PMK file created from that view is also replaced, so that the Drawings linked to these PMK files can be updated.

Advantages of this workflow:

- Updating documentation may be quicker, because it involves updating a single, small-sized drawing file at a time.

- Opening the Layout Book might take less time, since the PMK files serving as the Drawing sources are located outside the project file.
- If the PMK-based Drawings are set to Auto-update, then they will be automatically updated when the Layout is published.

If necessary, you can break the Drawing's link to the PMK file and re-link the Drawing to its original model view. To do this, select the PMK file(s) in the Drawing Manager (multi-selection is available) and use the following command from the context menu: **Link to original Model View**.



Revision Management

During the course of an Archicad project, you will probably issue several sets of Layouts at different times. Between each such Transmittal Set, the model changes, and so do the Layouts. Revision Management helps you keep track of changes, Layout revisions, and Transmittal Sets.

[Revision Management Workflow](#)

[Revision Management: Default Schemes in the Template](#)

[Working with Changes](#)

[Feedback on Layout Revisions](#)

[Revision IDs](#)

[Managing Transmittal Sets](#)

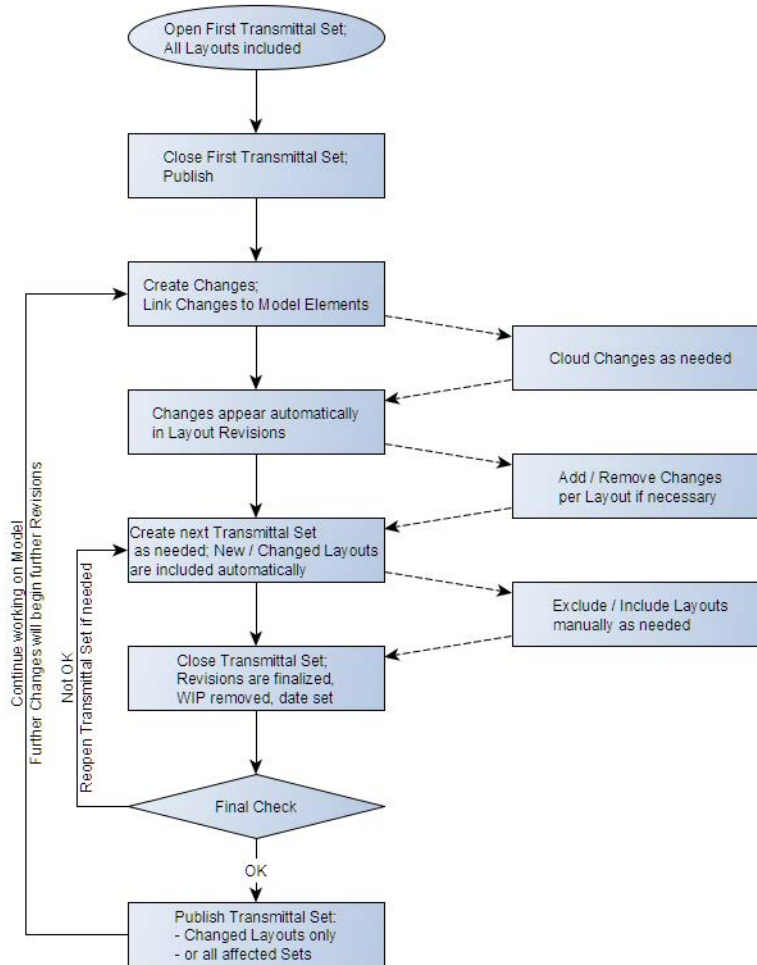
[Change List and Transmittal History](#)

[Revisions in the Publisher](#)

[Changes from External Files](#)

Revision Management Workflow

This chart, and the summary that follows, describe a Revision management workflow which takes advantage of Archicad features that automate your Layout Revisions. For this automation to work, you must place Changes that are connected to model elements. This way, all Layouts affected by these Changes are automatically given a new Revision number.



Phases of the Workflow:

First Transmittal Set

- Create the First Transmittal Set. This typically includes all the Layouts in your project.
- Close the Transmittal Set.

Changes and Revisions

Archicad automatically tracks your changes on Layout Revisions, provided that your Changes are linked to elements, and provided that Drawings on the Layouts are up to date.

- As the project develops, create Change entries and link them to elements in the model (use Markers and/or Clouds if desired).

- These Changes are automatically added to the Layout Revision History of every Layout affected by the Change: that is, every Layout containing an element that is linked to a Change.
- A new Layout Revision is created automatically as soon as a new Change appears on a Layout since its last Transmittal Set.
- A Change will appear on every Layout that is affected, but on only one single Revision of a given Layout.
- Changes of any nature can also be added manually to a Layout Revision.
- Only the latest Revision of any given Layout is editable. The Revision data (and changes) of any earlier, already transmitted Layout cannot be modified.

Subsequent Transmittal Sets

- Do a Drawing Update for the entire Layout Book.
- Create the next Transmittal Set. All Layouts having a new Revision since the previous Transmittal Set, plus any new Layouts, are automatically included in the Second Transmittal Set.
- Remove any unneeded Layouts from the second Transmittal Set.

Close and Publish Transmittal Set

- Close the Transmittal Set. Layout Revisions are now finalized.
- Review the contents of the Transmittal Set. If you see problems, you can reopen the Transmittal Set, fix it, then close it again.
- Publish Transmittal Set contents from the Publisher (either every affected Publisher Set, or just the changed Layouts in the current Publisher Set.)
- Continue working on the model. Any further change on a Layout will automatically generate a new Layout Revision.

Revision Management: Default Schemes in the Template

To help you get the full benefit of the Revision feature, the Archicad template for each language version includes predefined defaults for the Revision-related interface. Among these:

- Master Layout with **Revision History** object
- Default data for the Change Tool, **Change Scheme** and Transmittal Set Scheme
- New default Project Indexes: **Change List, Transmittal History**

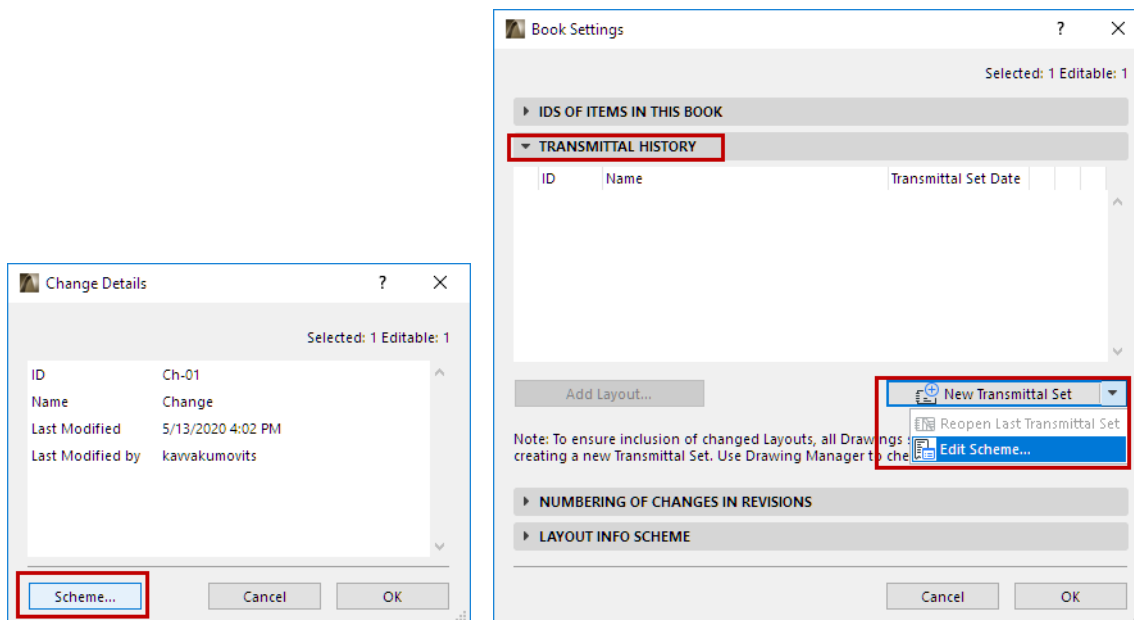
If you are using your own company template instead of the Archicad template, you will not see these items in your project by default. You can do the following:

To access the Revision History (a GDL object)

- Make sure you migrate your library, or that you can otherwise access the Revision History object (from Object Tool Settings), and place it on a Master Layout as needed.

To access default Change Scheme or Transmittal Set Scheme data fields:

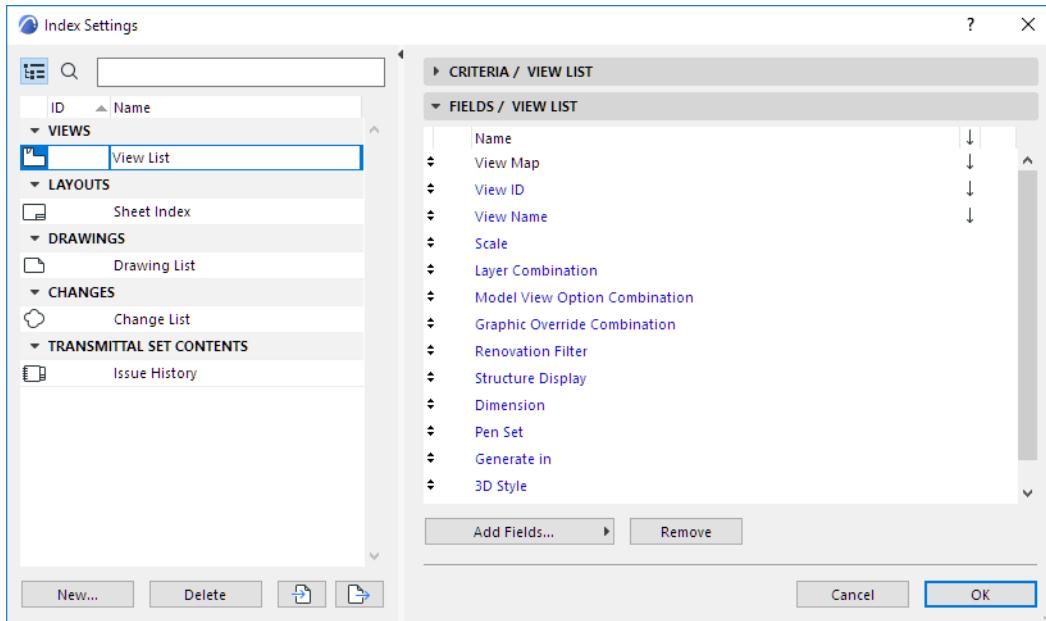
1. Start Archicad and choose the default (factory shipped) Archicad 27 template file to create a new project.
2. Open the **Edit Scheme** dialog:
 - for Change Scheme: from Change Details
 - for Transmittal Set Scheme: from Book Settings



3. Click Export to save the Change/Transmittal Set Scheme to a location on your computer.
4. Open your own project/template.
5. From Edit Change Scheme (or Edit Transmittal Set Scheme), click Import to bring in the saved Scheme.

To access the Change List and Transmittal History (Project Indexes):

1. Open the Archicad 27 template file.
2. Open Document > Project Indexes > Index Settings to access the predefined Project Indexes.



3. For each desired Index file (e.g. Change List, Transmittal History), click Export to save it to a location on your computer.
4. Open your own project/template.
5. From Index Settings, click Import to bring in the Indexes.

Working with Changes

Create a Change to mark and/or describe a change in the project.

A Change can be placed into the project either with the **Change Tool**; or you can create and list a change using the **Change Manager**.

If you want Archicad to detect Changes automatically on every affected Layout, then you must link your Changes to one or more elements or Change Markers (placed with the Change Tool).

These functions are described below.

[Change Manager Palette](#)

[Create Change](#)

[Link/Unlink Change Elements](#)

[Add/Remove Changes on Layout Revisions](#)

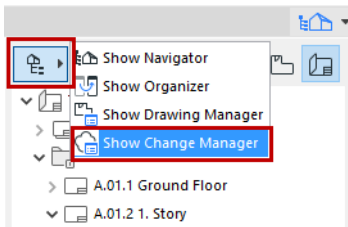
[Change Details](#)

[Archive Changes in Change Manager](#)

Change Manager Palette

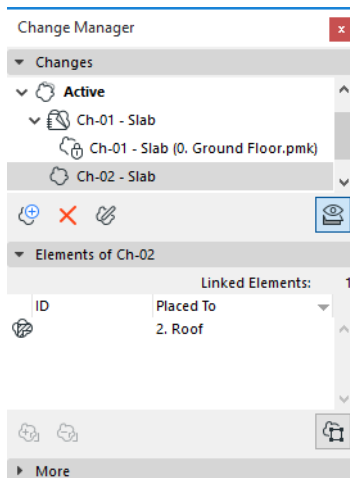
Keep track of your changes in the Change Manager Palette, available here:

- Window > Palettes > Change Manager
- **Show Change Manager** from the pop-up menu at the upper left corner of the Navigator/Organizer



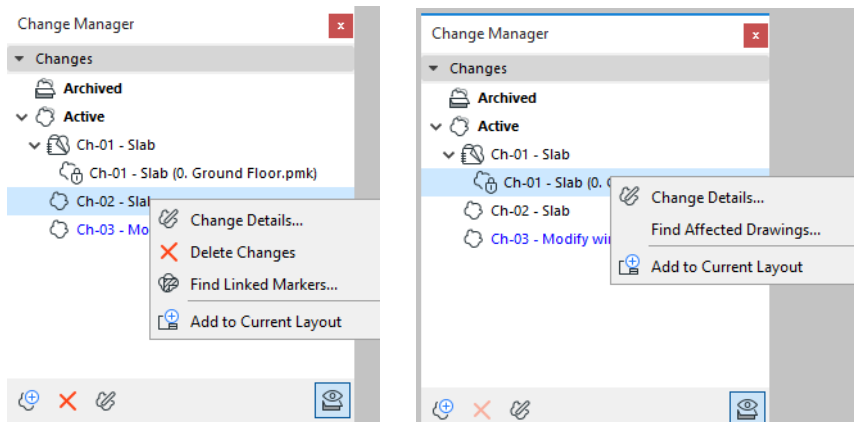
The top of the Change Manager lists all Changes in the project.

Note: Change Markers placed as “unlinked” are not listed in Change Manager. Unlinked Change Markers are graphical elements that are not linked to anything, but you can link them at any time to existing Changes.



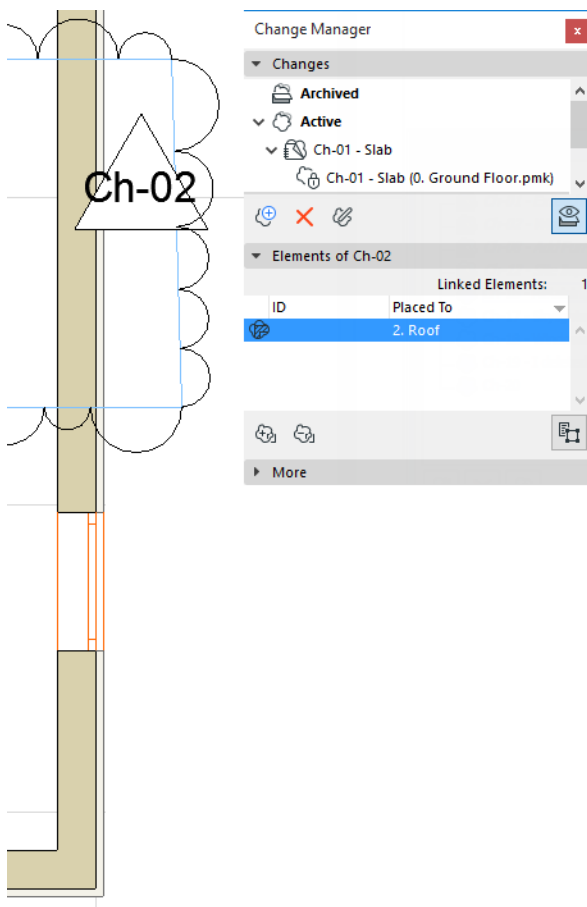
- **Untransmitted Changes:** A Change on the currently active Layout that has not yet been transmitted is also listed in blue type in the Change Manager. (The Layout window must be open for this feedback to be visible.)
- **Transmitted Changes:** Changes that have been transmitted on at least one Layout revision have a green Transmittal Set icon on it. (These changes can still be added to other Layout revisions.)
- **Archived Changes:** Optionally, you can archive some or all of your transmitted changes and list them separately in the Change Manager. ([See Archive Changes in Change Manager.](#))
- **Changes in external files:** Changes (except Archived Changes) that arrive in the host file via a Hotlink or external drawing are shown with a lock icon. ([See Changes from External Files.](#))

Use the context menu of any selected Change to access relevant commands. Note that these differ for transmitted and for untransmitted Changes. (Transmitted Changes cannot be deleted from the project, but they can be archived.)



Zoom to Linked Element

From the linked element(s) at the bottom of the Change Manager, select any element and double-click to zoom to that element in the open window (if it is visible) or in the source view (if it is not currently visible).



[Watch the Video](#)

Create Change

[Create Change from Change Manager Palette - With Elements Selected](#)

[Create Change from Change Manager Palette - With No Elements Selected](#)

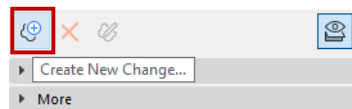
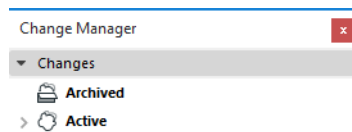
[Create New Change Using Change Tool](#)

[Place New Change Marker Linked to Existing Change](#)

[Create New Change on a Layout Revision](#)

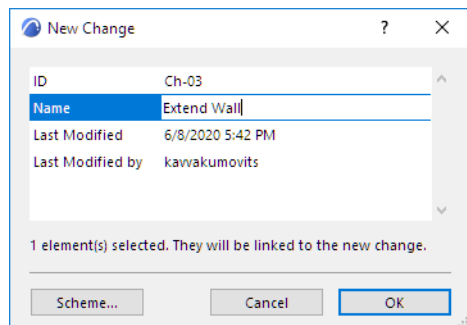
Create Change from Change Manager Palette - With Elements Selected

1. In any model window, select element(s) involved in the Change that you wish to document.
2. In Change Manager, click **Create New Change**.



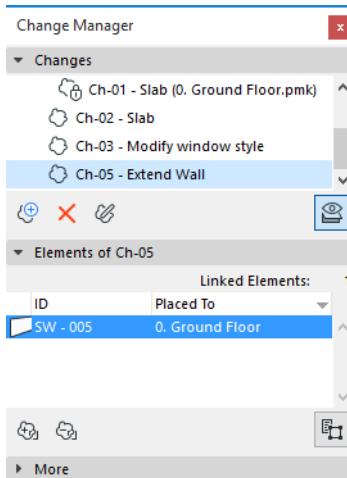
3. The **New Change** dialog box appears. Enter a Change ID (required), name, and any custom data as needed.

See [Edit Change Scheme to Define Custom Data](#).



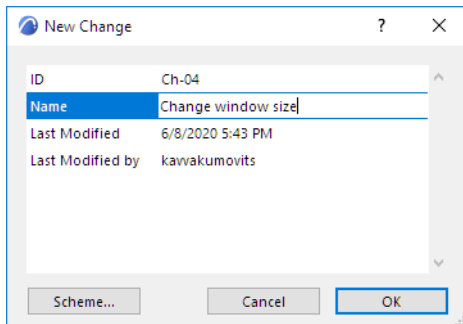
4. Click OK to create the Change.

The new Change is listed in the Change Manager. The selected elements are linked with the Change and listed in the panel below.



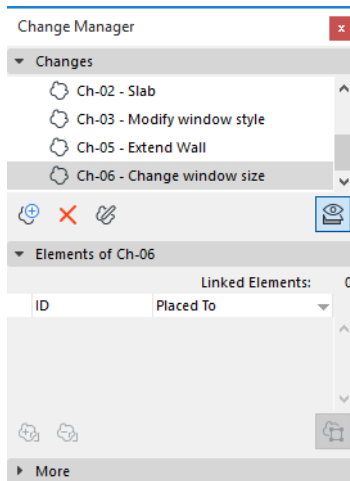
Create Change from Change Manager Palette - With No Elements Selected

1. On the Change Manager, click **Create New Change**.
2. The **New Change** dialog box appears. Enter an ID, name, and other information as needed.



3. Click OK to create the Change.

The new Change is listed in the Change Manager. Note that it is not linked to any elements in the project; no Linked Elements are listed at the bottom of the Change Manager. Consequently, this Change will not show up automatically on any Layout Revision.



If you want Archicad to detect Changes automatically on every affected Layout, then you must link these Changes to one or more elements.

See [Link Element to Change](#).

Create New Change Using Change Tool

Use the Change tool to place a Change Marker (cloud and/or marker) in any of the following windows: Floor Plan; Section; Elevation; Interior Elevation; 3D Document; Worksheet; Detail; or Layout window.

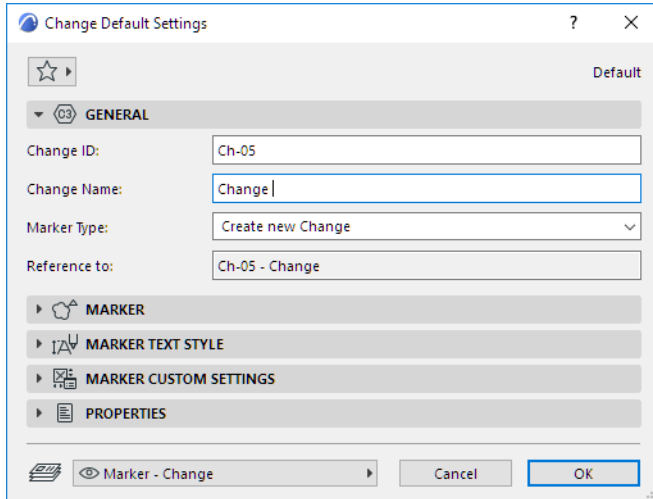
- The Change element (cloud and/or marker) can either be linked to a new or existing Change, or is an unlinked graphical element. If it is linked, then the Change Marker is listed in Change Manager, and is added automatically to every affected Layout Revision.
- Visibility of Change Markers is defined globally for each Transmittal Set, in the Transmittal Set Details dialog box.

See [Show or Hide Change Markers in This Transmittal Set](#)

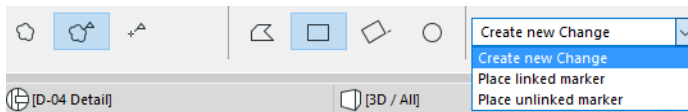
1. Activate the Change Tool
 - from **Document > Documenting Tools**, or
 - from the Archicad toolbox



2. In Change Tool Settings, enter the Change ID and Change name.



Marker type: By default, the Change Tool is set to create a new Change (The other options - also available in the Info Box - are to place a linked or unlinked marker).



3. From Change Tool Settings or the Info Box, choose the component(s) you want to place: a Cloud plus a Marker Head, or just one or the other.



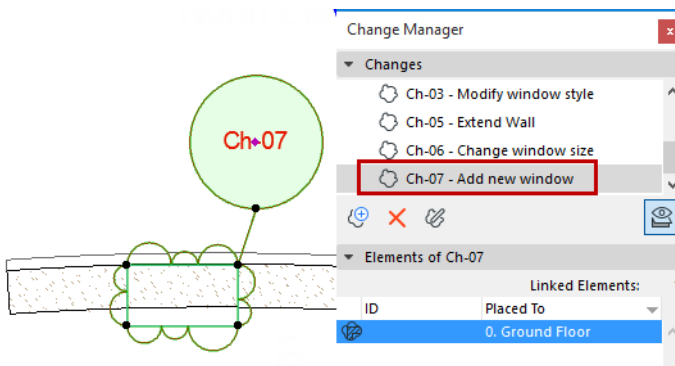
4. From the Info Box, choose a geometry method for the Cloud part of the Marker (if you are using a Cloud):



- Polygonal
- Rectangular
- Rotated Rectangular
- Circular

5. Place the Change Marker in the project.

6. A new Change appears immediately in the Change Manager. The Change Marker is linked to the Change you just placed.



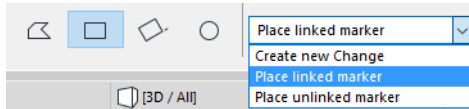
Note: The cloud component includes an on-screen only polygon. If needed, you can hide this polygon with **View > On-Screen View Options > Marker Range**.

The new Change is listed in the Change Manager.

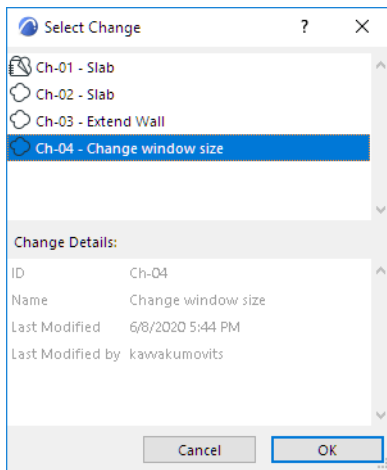
[Watch the Video](#)

Place New Change Marker Linked to Existing Change

1. Activate the Change Tool and adjust its settings as described above.
2. From the Info Box, choose **Place linked marker**.



3. In the Info Box pop-up, choose the Change that you wish to link the new Change marker to; or click **Select Change**.
4. In the appearing Select Change dialog box, choose the Change and click OK.



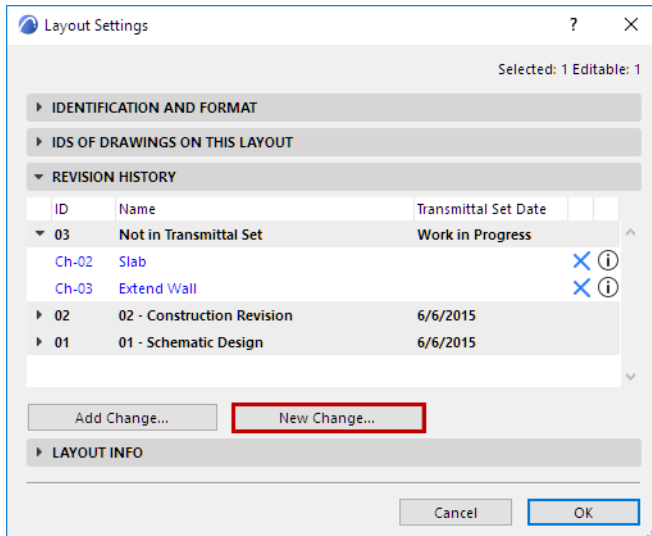
5. Place the Change Marker into the project. It is linked to the Change you selected.

[Watch the Video](#)

Create New Change on a Layout Revision

You may wish to note a Change on a Layout Revision manually. For example, if a new Drawing was added to the Layout; or the Drawing order on the Layout has been changed; or if the Master Layout has changed.

1. In the Revision History panel of Layout Settings, click **New Change**.



2. In the New Change dialog box, enter data for the change as needed. ([See New Change Details.](#))
3. Click OK to create the Change and add it to the Layout Revision.

The new Change is listed in the Change Manager. Note that it is not linked to any elements in the project; no Linked Elements are listed at the bottom of the Change Palette.

Another way to do the same thing:

1. Open the Layout.
2. Use the Change Tool to create a new Change, with a marker, directly on the Layout. Again, it is listed in the Change Manager and added to the open Layout Revision.

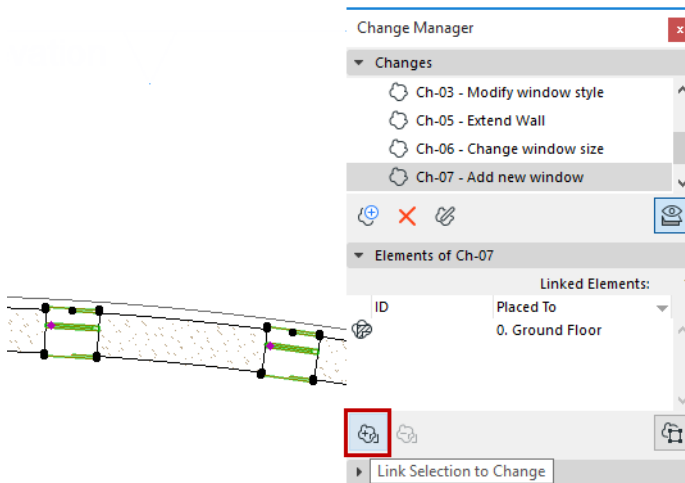
Link/Unlink Change Elements

Link Element to Change

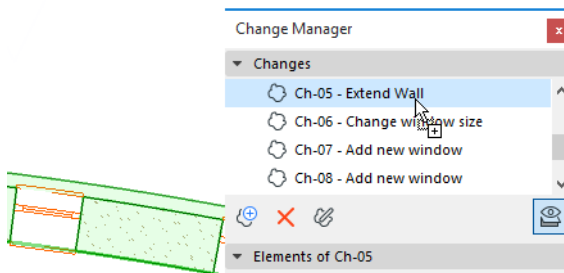
Unlink Element from Change

Link Element to Change

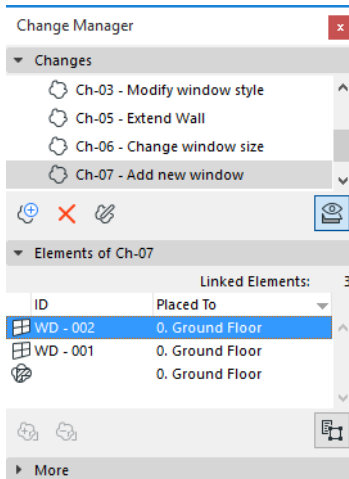
1. Select one or more elements.
2. In the Change Manager, select the Change to which you want to link the selected element(s).
3. Do one of the following:
 - Click the **Link Selection to Change** button in Change Manager



- Select a Change in the Change Manager. From the Floor Plan, drag and drop the selected construction element into the Change Manager to link it. (Make sure you keep the mouse button pressed throughout the drag-and-drop.)

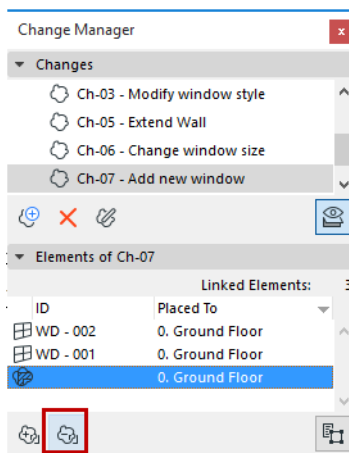


4. The linked element(s) appear at the bottom of the Change Palette.



Unlink Element from Change

1. Select that item from the Linked Elements list in the Change Manager Palette.
2. Click **Unlink Selection from Change**.



[Watch the Video](#)

Add/Remove Changes on Layout Revisions

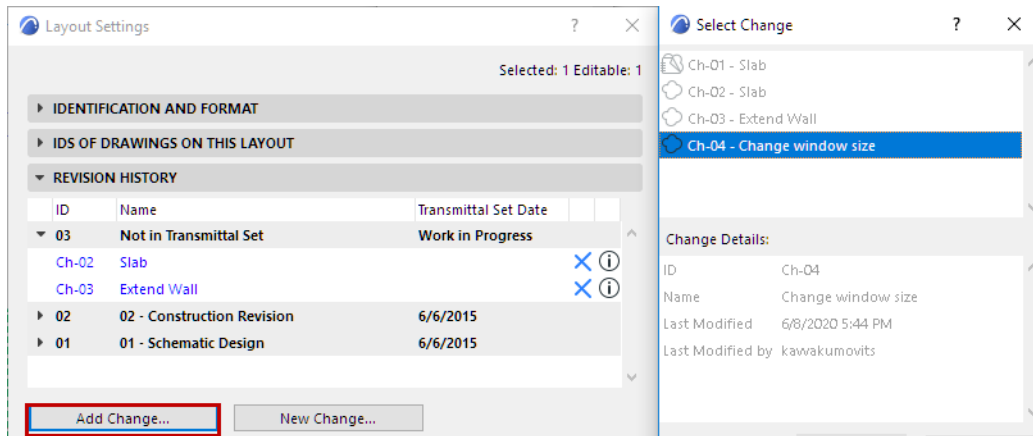
Add an Existing Change to a Layout

Remove Change from Layout Revision

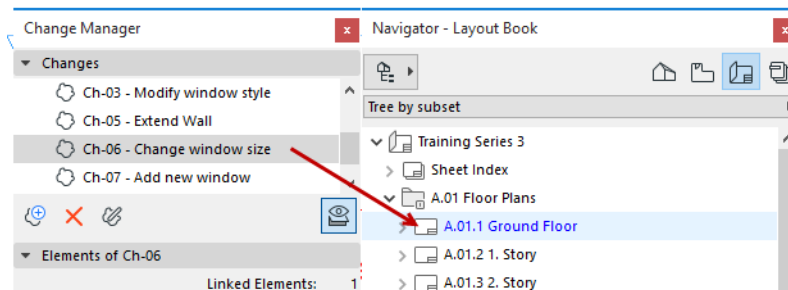
Add an Existing Change to a Layout

Do one of the following:

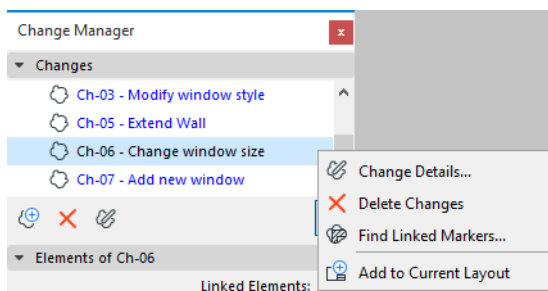
- In the Revision History panel of Layout Settings, click **Add Change**. From the appearing dialog box, select the Change you want to add to this Layout's Revision History and click OK.



- From the Change Manager, drag and drop the change onto the Layout (either its item in the Navigator, or onto the Layout itself).

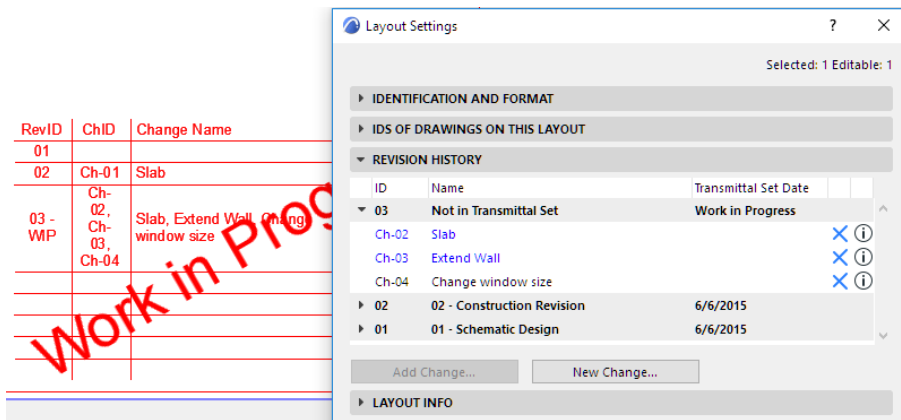


- With the Layout window active, right-click a Change in the Change Manager to add it to the currently active Layout.



The new Change is now shown in the Revision History, both in Layout Settings and on the Layout itself.

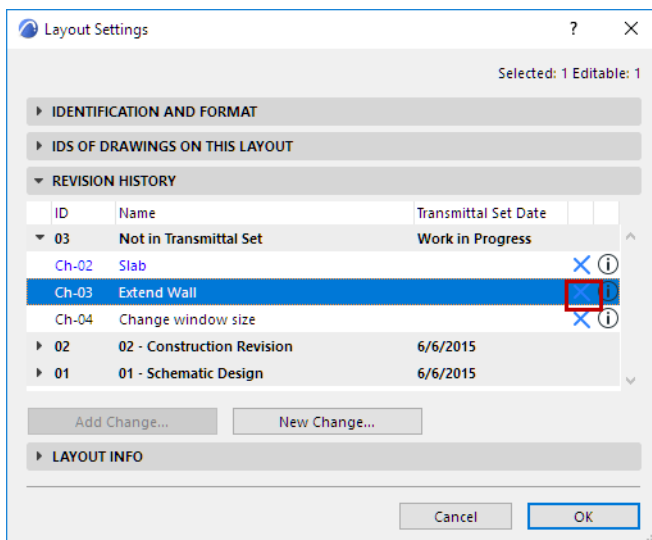
[See also Revision History Data on Layout and in Layout Settings.](#)



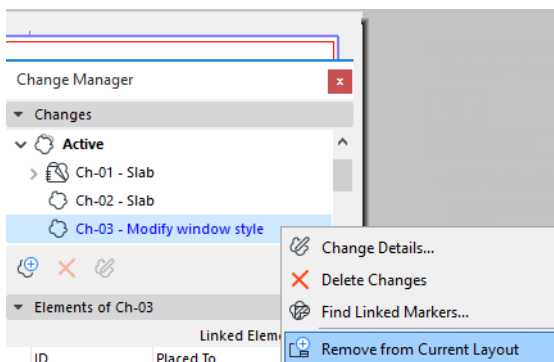
Remove Change from Layout Revision

At any time, you can remove a Change from an open Layout Revision. Do one of the following:

- In Layout Revision History (Layout Settings), just select the unneeded Change and click the blue X to remove it from the Layout Revision. The Change will not appear again on this Layout unless you add it back manually (using Add Change).



- With the Layout window active, right-click a Change in the Change Manager to remove it from the currently active Layout.



You cannot remove Changes from the Revisions of closed Transmittal Sets.

Change Details

New Change Details

Edit Change Scheme to Define Custom Data

Import/Export Change Scheme

Modify Change Details

Modify Details of Transmitted Change

New Change Details

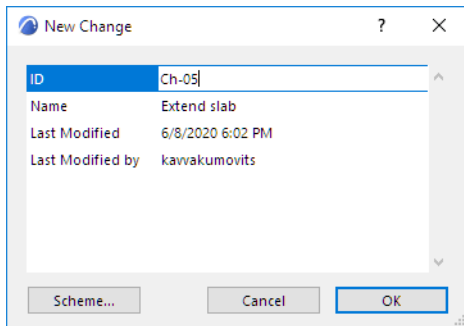
The following data are defined in the **New Change** dialog box.

Change ID: This value is mandatory and must be unique for each Change. With each new Change, the last number is automatically increased by one.

Name: This optional value will serve as the name of the Change. By default, the previous Change Name is shown here, but you can modify it as you like.

Last Modified: Time/date of last modification of the Change Details. Linking or unlinking elements or markers does *not* count as a Change modification. (Once a Change is transmitted, you can no longer edit this field.)

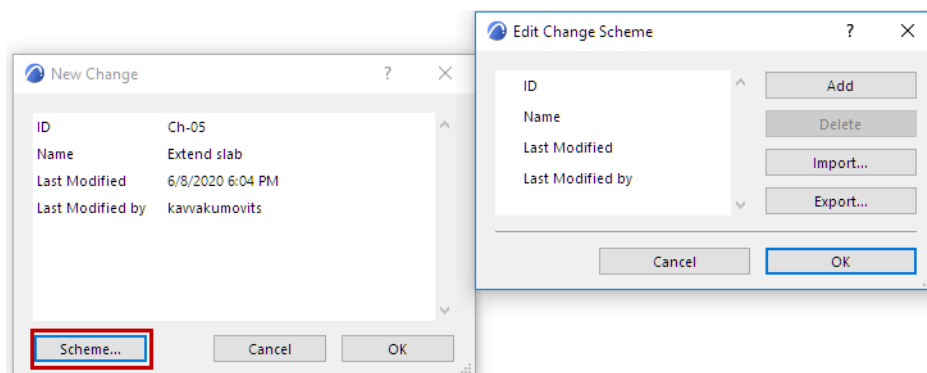
Last Modified by: User name who carried out the last modification (relevant for Teamwork projects). (Once a Change is transmitted, you can no longer edit this field.)



Edit Change Scheme to Define Custom Data

Use this Scheme to define any Custom data types for all Changes in this project.

Click **Scheme** from the New Change or the Change Details dialog box to open the **Edit Change Scheme** dialog box.



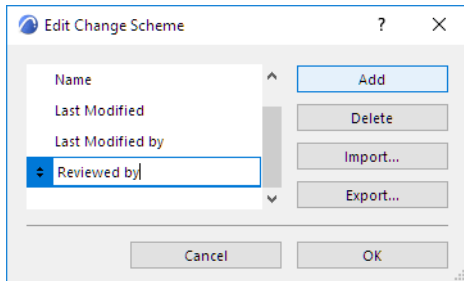
The Scheme contains the set of four fixed Change data items for the project; you cannot redefine these:

- ID

- Name
- Last Modified
- Last Modified By

Below these fields, you can define any number of additional fields.

1. Click **Add** to add another field.
2. Type in any custom field name (e.g. Reviewed by).



3. Click **Add** to add another custom field, or **Delete** to delete it.
4. Use the arrows at the left to change the order of the custom fields (the four fixed fields are always at the top of the list).

Import/Export Change Scheme

Click **Export** to save your Change Scheme as an xml file.

Then you can **Import** it for use in other project files.

See also Revision Management: Default Schemes in the Template.

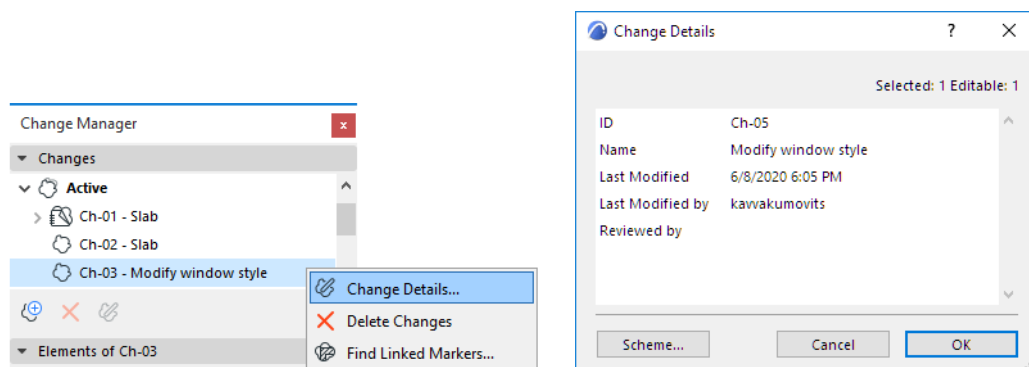
Modify Change Details

To modify these data, select one or more Changes in the Change Manager.

1. Access **Change Details**: right-click and select the **Change Details** command, or double-click on the Change, or click the **Change Details** button.
2. Edit the fields as needed.

Note: Details of Changes from external projects cannot be edited.

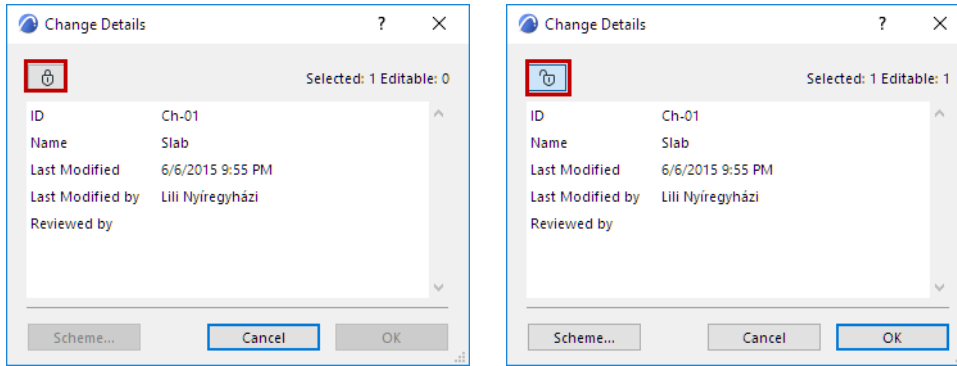
Note: In Teamwork, you must reserve the Change Manager before modifying Change Details. Without reserving, you can only modify the details of Changes which you created and which have not yet been sent to the server.



Modify Details of Transmitted Change

If the selected Change is transmitted (or Archived), the Change Details dialog box has a lock on it. It is not advisable to edit such changes. If you must modify a transmitted or an archived Change anyway, click the icon to unlock it.

You can edit the ID, Description, and any custom fields, but you cannot edit the Last Modified date or user name.

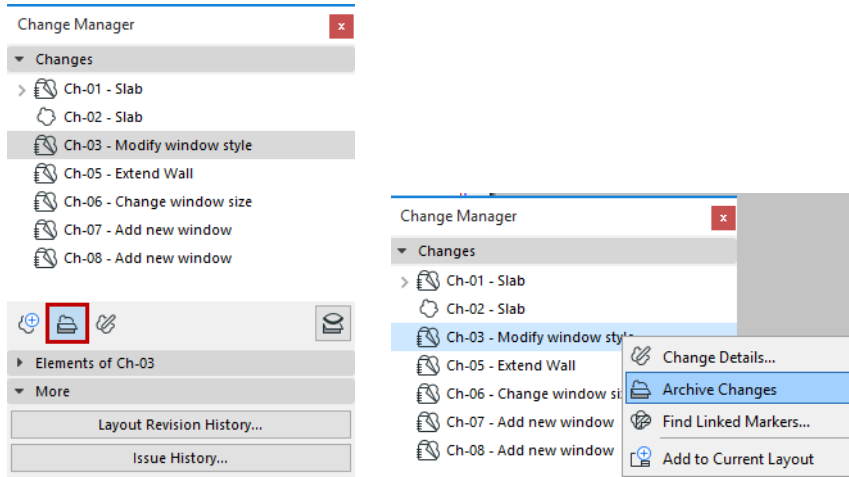


Once you leave the dialog, the Change is locked once more.

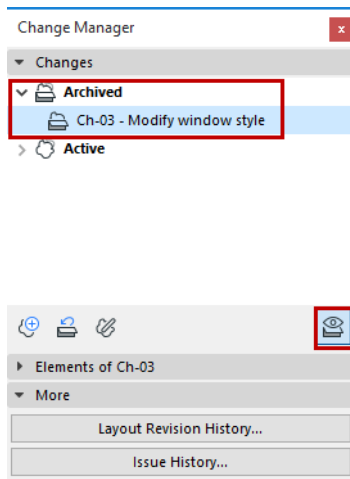
Archive Changes in Change Manager

Any transmitted Change can be optionally archived (for example, if your list of Changes is very long, and you have many transmitted Changes that are no longer relevant for the current Transmittal Set).

In Change Manager, select any transmitted Change(s) from the list and click the **Archive Change** button, or the context menu command.



Archived Changes, identified with this icon, are placed in their own folder in the Change Manager, which you can hide with the **Show/Hide Archived Changes** button.



The data of transmitted or Archived Changes can only be modified to a limited extent.

Feedback on Layout Revisions

Important: This feedback is up to date only if the Drawing(s) on the Layout are up to date!

Blue Layouts

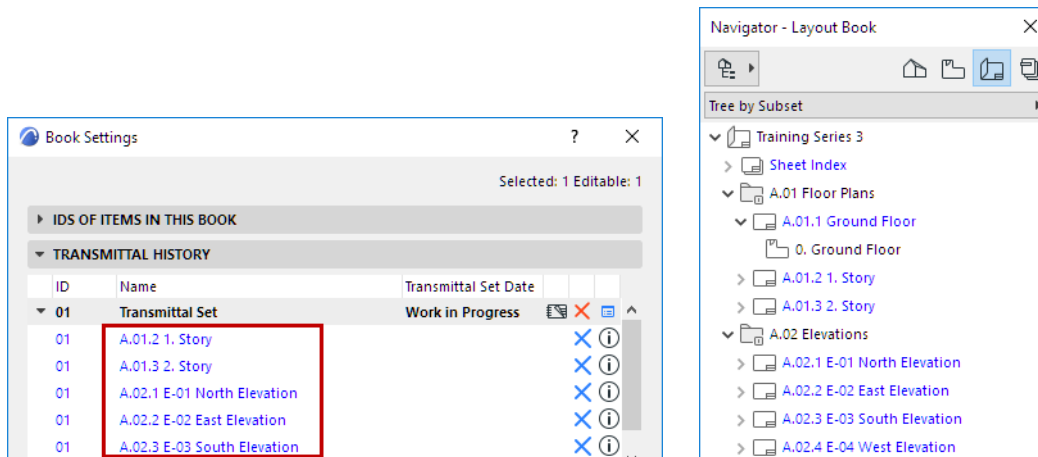
A **Layout name** is shown in blue if its current Revision has not been transmitted yet.

That is:

- the Layout has never been transmitted before (it's a new Layout), or
- the Layout contains new Changes since its last Transmittal Set

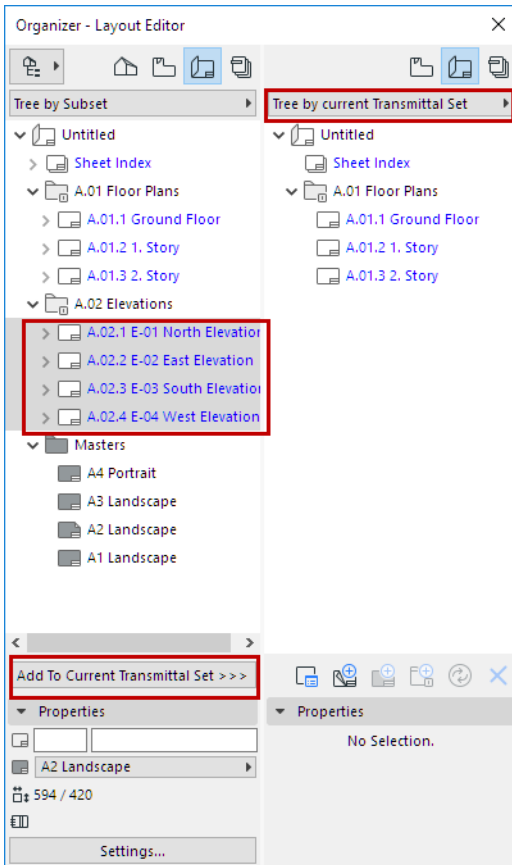
Layout names can be listed in blue in the following locations:

- the Navigator
- the Transmittal History
- the Add Layout dialog box



The blue Layout feedback can be particularly useful when you are adding Layouts to the current Transmittal Set.

See [Add Layout to Current Transmittal Set](#).



Note: In the Publisher Set, Layouts that are part of the current Transmittal Set are listed in green.

[See Revisions in the Publisher.](#)

Blue Changes

A Change is listed in blue type in the Change Manager if this Change has not yet been transmitted on the currently active Layout. (The Layout window must be open for this feedback to be visible.)

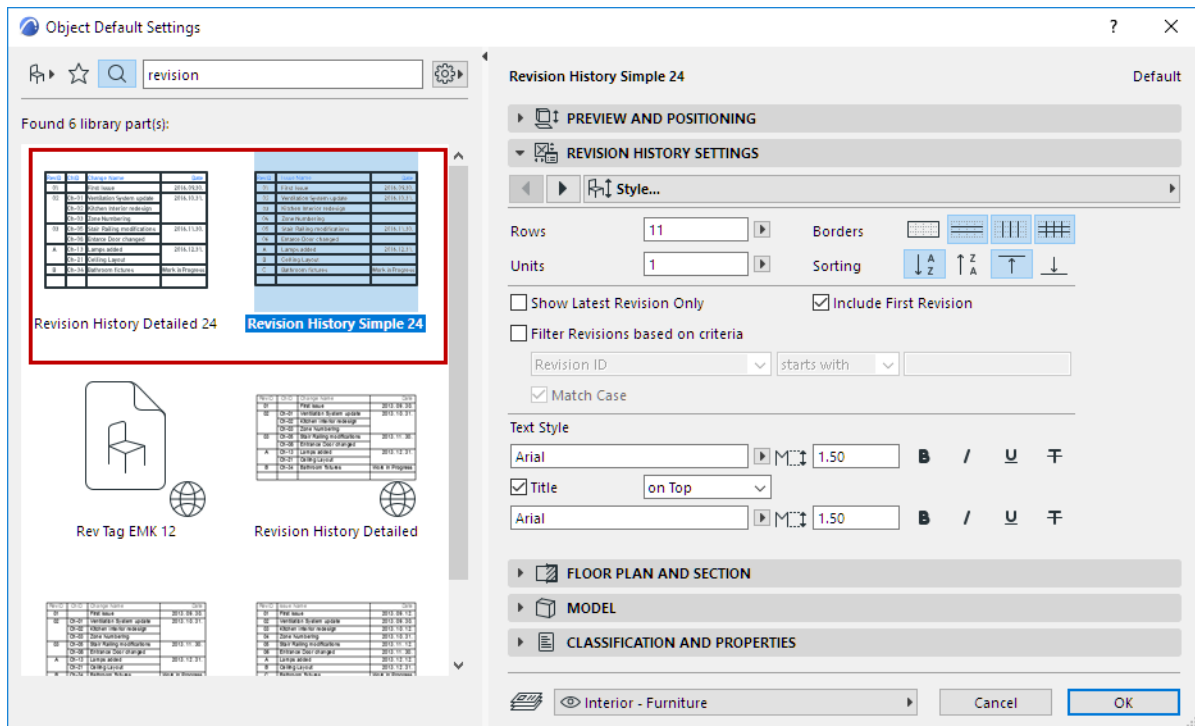
RevID	ChID	Change Name	Date
01 - WIP	Ch-06	Redefine Slab structure	Work in Progress

[Watch the Video](#)

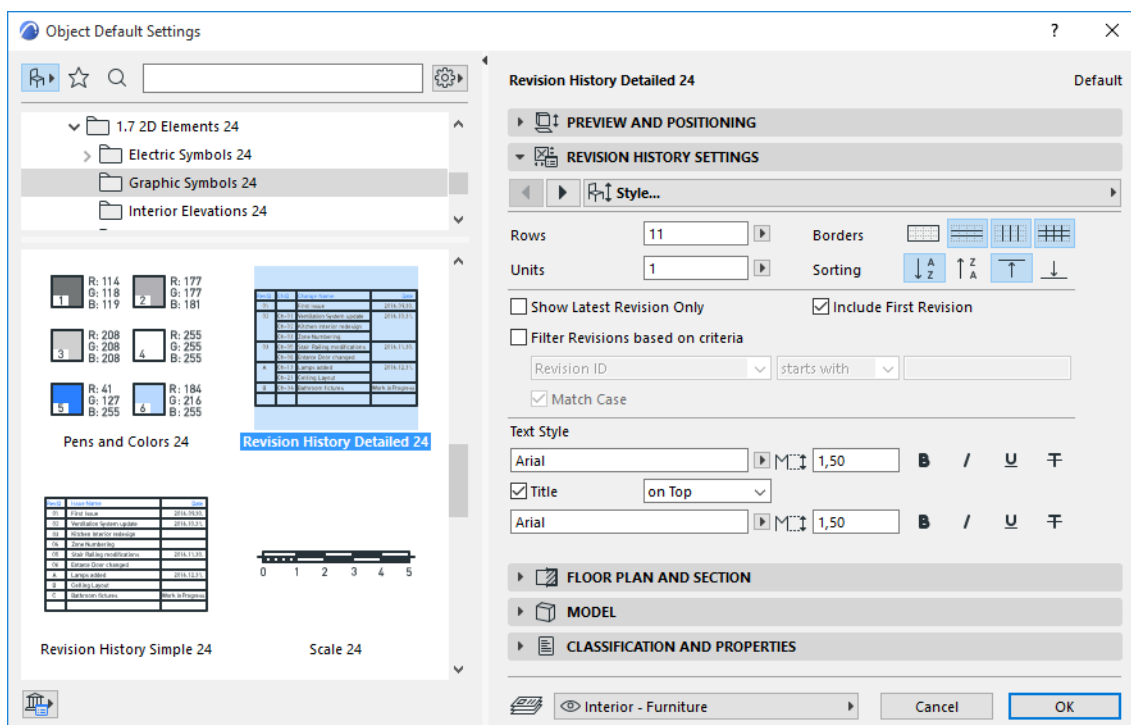
Revision History Data on Layout and in Layout Settings

Each Layout can display a Revision History.

This is a GDL object which must be placed on a Master Layout or directly onto a Layout. (It is available in your Archicad Library, in the Graphic Symbols folder.) Use the Object Settings to customize its content and style.




The Revision History object has two predefined styles: Simple and Detailed.



Even if the Revision History object is placed on the Master Layout, its content is specific to each Layout. In other words, data in the Revision History are updated based on the current status of the particular Layout.

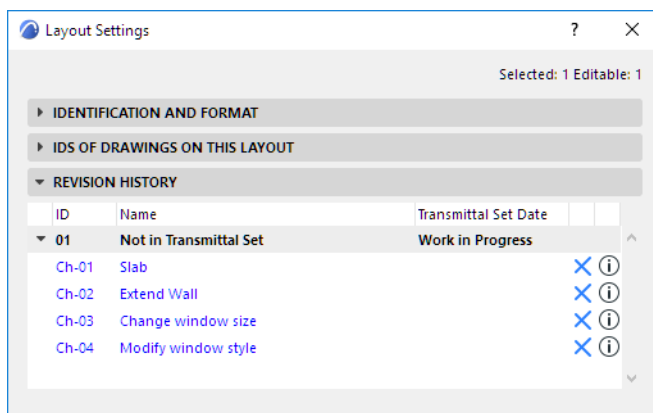
Note: If you place the Revision History object onto a Layout in the form of a Drawing, the Revision History data will not be “smart”. Place the object onto a Master Layout or directly onto a Drawing.

Revision History			
RevID	ChID	Change Name	Date
01			12/13/2013
02	Ch-01	Extend Northeast Corner	1/10/2014
	Ch-02	Window Deleted!	
	Ch-08	Add Furnishings	
03	Ch-10	Add partition Ground Floor	1/10/2014
	Ch-11	Partition Bedroom	
04	Ch-12	Site Improvements	Work in Progress
	Ch-16	Retaining walls	
	Ch-19	I deleted a window!	

Company Title	
	
Company name	
Street	
City	
State/Country	
Postal Code	

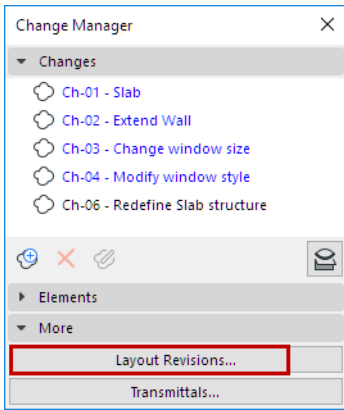
The Revision History (its detailed view) lists the Layout's Revisions, plus all Changes associated with each Revision.

The same data is available in the Revision History panel of Layout Settings.



[Watch the Video](#)

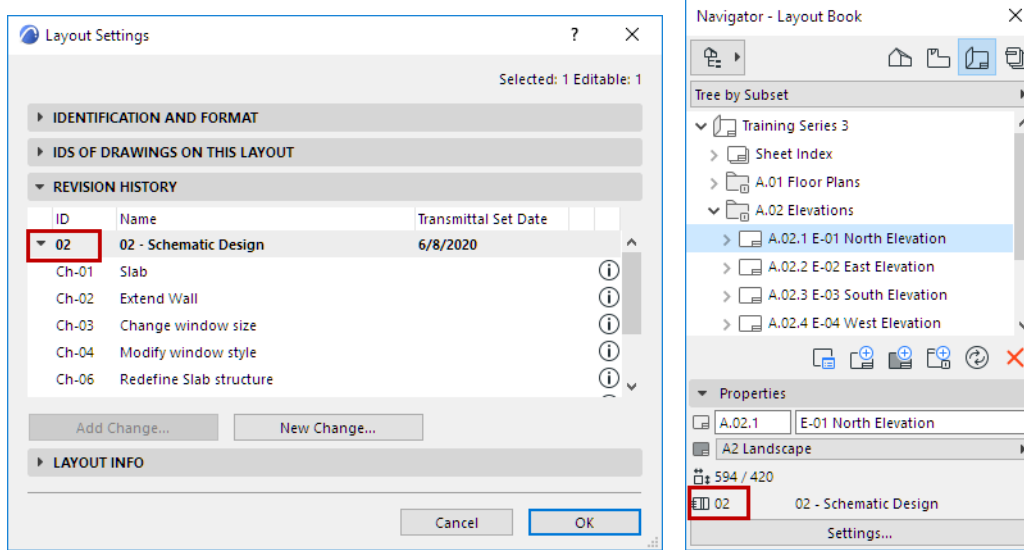
With the Layout window active, you can access the current Layout's Revision History panel from the Change Manager, too:



Revision IDs

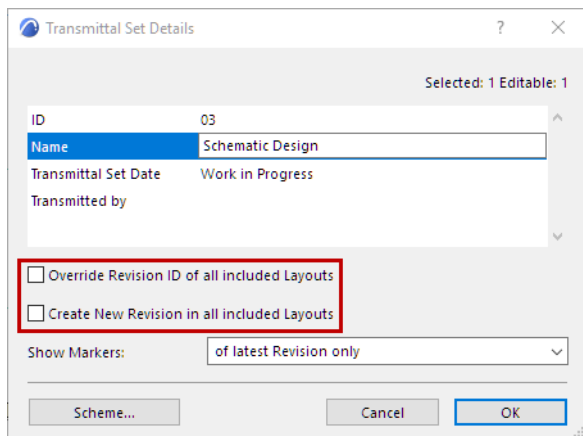
By default, any Layout affected by a new Change (since the last Transmittal Set was closed) automatically gets a new **Revision ID**. This is shown in the Revision History panel of Layout Settings, in the Navigator, and in the Revision History stamp of the Layout.

See also [Revision History Data on Layout and in Layout Settings](#).



RevID	ChID	Change Name
01		
	Ch-01	Slab
	Ch-02	Extend Wall
	Ch-03	Change window size
02	Ch-04	Modify window style
	Ch-06	Redefine Slab structure
	Ch-07	Adjust slab geometry
	Ch-08	Change partition width

Use the **New Transmittal Set** or the **Transmittal Set Details** dialog box to define the logic for numbering the Layout Revisions for the current Transmittal Set. Two checkbox control this function.



Revision IDs for the Project's First Transmittal Set

For the project's very first Transmittal Set, the second checkbox is always activated. All Layouts of the project are "new" (all are listed in blue), and all are added to the Transmittal Set. This means that all Layouts will be assigned a Revision ID.

Default Revision Numbering for All Subsequent Transmittal Sets

For subsequent Transmittal Sets, by default, neither box is checked. This means that a new Layout Revision (with a new Revision ID) is created only if a Change has been added to the Layout (either by hand, or automatically when a change is linked to an element shown on this Layout).

Override Layout Revision Numbering Logic

The two checkboxes provide different methods of Layout Revisioning.

- **Create New Revision in all included Layouts:** Every Layout that is added to this Transmittal Set is assigned a new Revision ID (even if it doesn't contain any Changes).

In other words, if you add a Layout to the Transmittal Set, it gets a new Revision number. This is a simplified Revision numbering logic that is independent of any Changes.

- **Override Revision ID of all included Layouts:** Use the Transmittal Set ID as the Revision ID for all Layouts in this Transmittal Set.

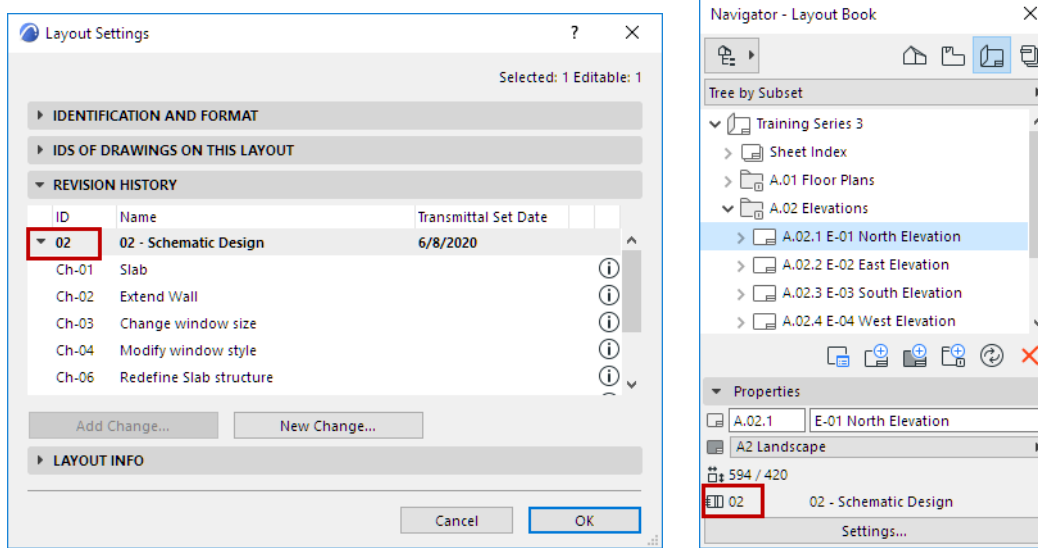
In other words, all Layouts in this Transmittal Set get a single, uniform ID, which is the same as the Transmittal Set ID. This can be useful at a stage of the project when you want a different, simplified Revision numbering logic.

[Watch the Video](#)

Modify Revision ID

Revision IDs can be modified in the Revision History and the Navigator only if

- the Layout Revision has not been transmitted yet
- the Revision ID has not been overridden by the Transmittal Set ID (using the first checkbox in **Transmittal Set Details**)



Managing Transmittal Sets

Create New Transmittal Set

Transmittal Set Details

Show or Hide Change Markers in This Transmittal Set

Add Layout to Current Transmittal Set

Remove Layout from Current Transmittal Set

Close Transmittal Set

What Happens When A Transmittal Set Is Closed?

Reopen Last Transmittal Set

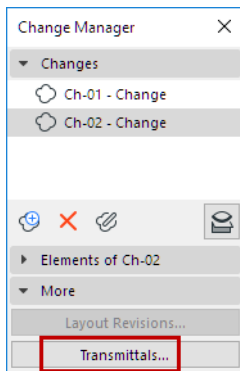
Delete Transmittal Set

A Transmittal Set is a set of Layouts that is transmitted at a given time.

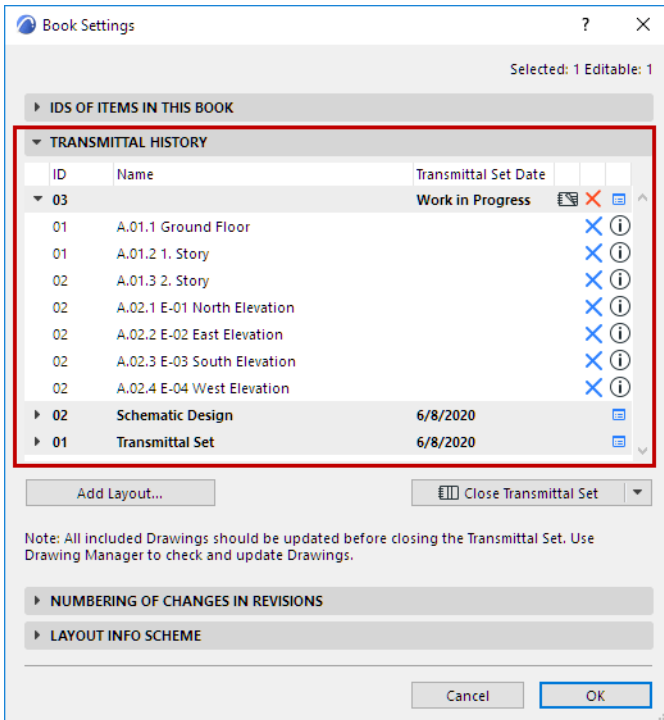
Typically, you will create a Transmittal Set when the required Layout set is considered complete.

To create and manage Transmittal Sets, use the **Transmittal History** panel of Layout Book Settings.

You can also access Transmittal History via the button at the bottom of Change Manager.



Here, all project Transmittal Sets (both the closed Transmittal Set and the currently open Transmittal Set) are shown with the list of Layout Revisions they contain.



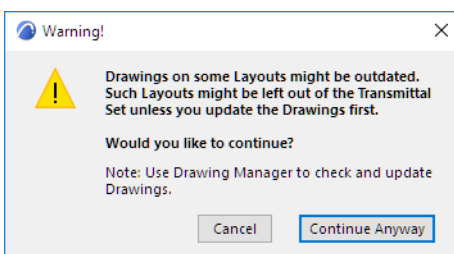
Create New Transmittal Set

This command is only available if no other Transmittal Set is currently open. Only one Transmittal Set can be open at a time.

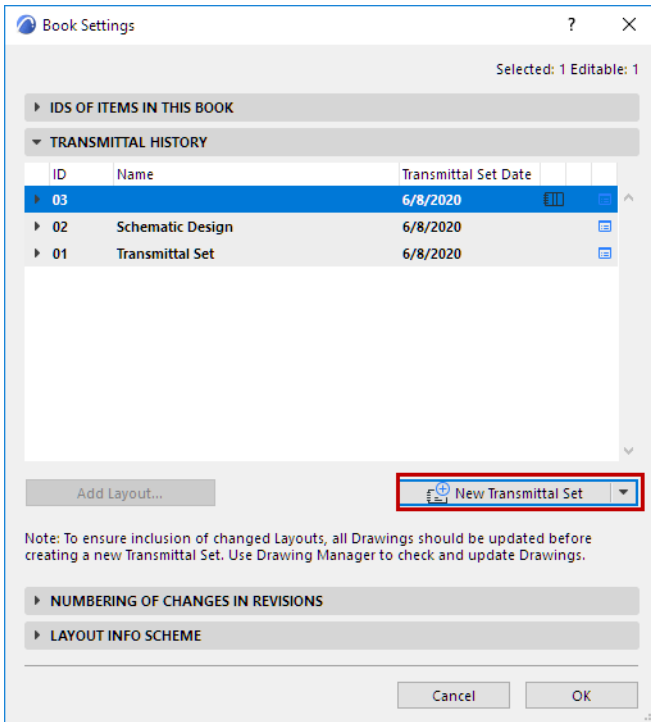
[Watch the Video](#)

Every time you create a New Transmittal Set, all open Layout Revisions - that have not yet been transmitted (and are listed in blue) are automatically added to the new Transmittal Set.

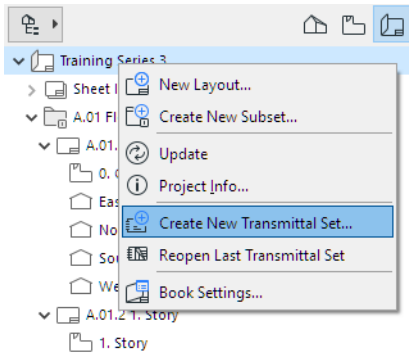
- 1. Update Drawings** as needed. You should do a Drawing Update in the Drawing Manager before the New Transmittal Set is created. (Otherwise, some Layouts with new Changes may not be added to the Transmittal Set.)



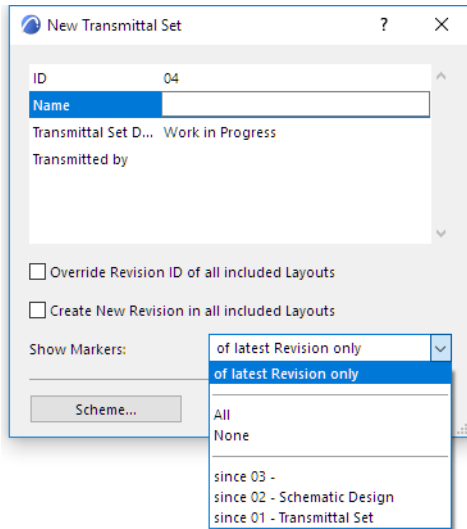
- 2. Use the New Transmittal Set** command.
 - from Layout Book Settings:



from the Navigator, using the context menu of the Layout Book



3. In the **New Transmittal Set** dialog box, enter the needed data to identify the new Transmittal Set.

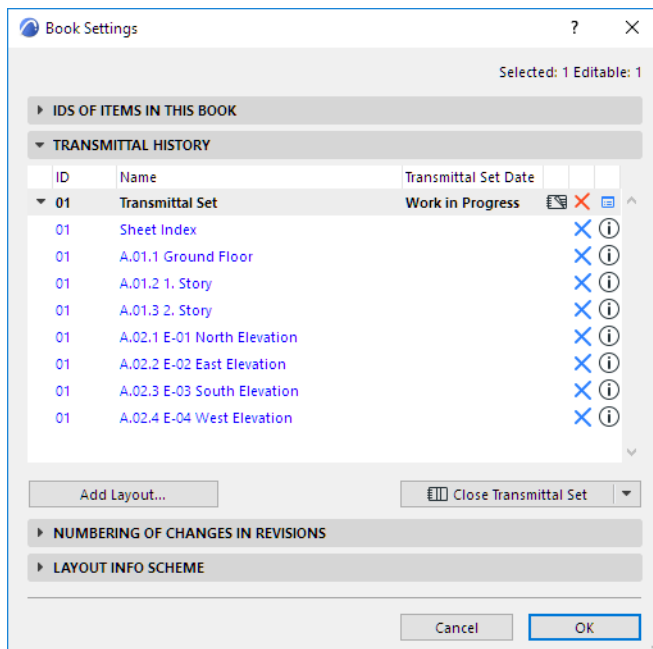


See also [Revision IDs and Show or Hide Change Markers in This Transmittal Set.](#)

4. Click OK to create the new Transmittal Set.

- **The Project's First Transmittal Set:** For the project's very first Transmittal Set, all Layouts of the project are “new”, and so all the Layouts are added to the Transmittal Set.

Note: A Layout listed in blue means that it has not been transmitted yet in its current form.

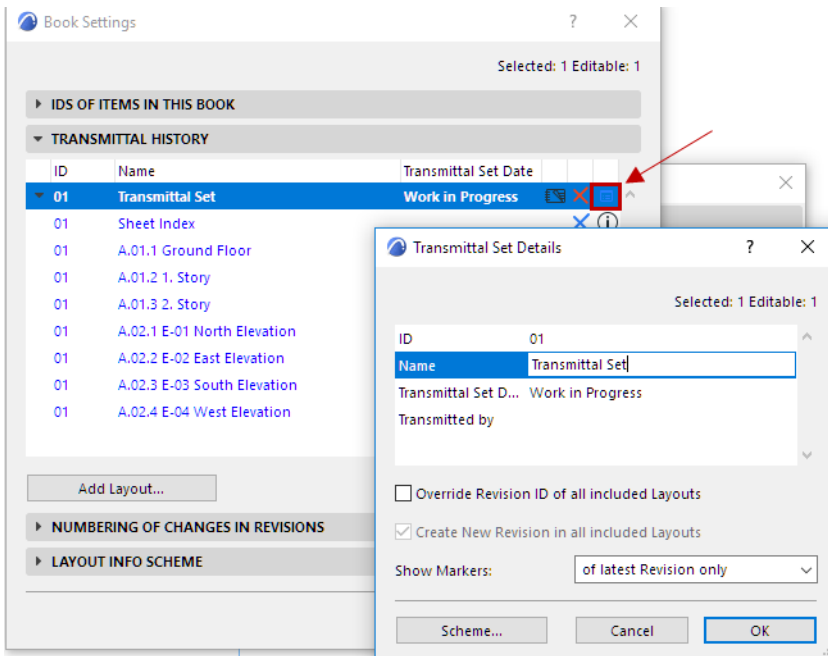


- **All Subsequent New Transmittal Sets:** Every time you create a New Transmittal Set after the first one, all open Layout Revisions - that have not yet been transmitted (and are listed in blue) are automatically added to the new Transmittal Set. (You are free to remove any Layouts that you don't need from the Transmittal Set. [See Remove Layout from Current Transmittal Set.](#))

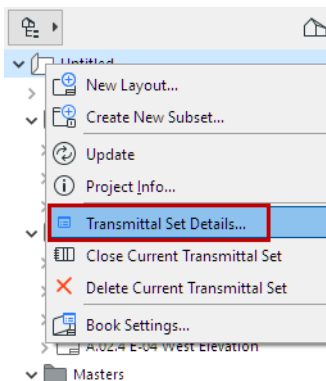
Transmittal Set Details

Transmittal Set Details are defined in the New Transmittal Set dialog box when you first create it, but you can modify these details at any time for the currently open Transmittal Set.

Access Transmittal Set Details by clicking the icon from Transmittal History (in Layout Book Settings):



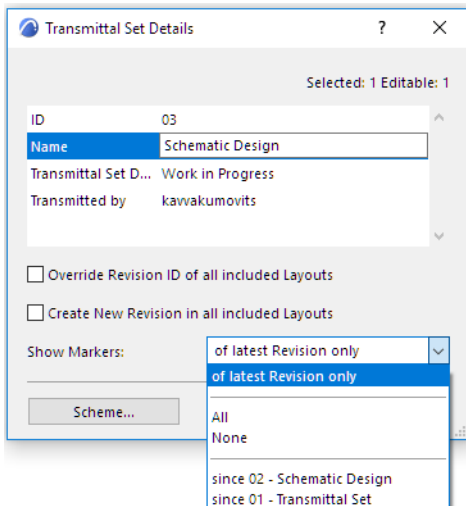
When the project has a Transmittal Set open, Transmittal Set Details is also accessible from the Navigator Layout Book context menu.



For information on the Revision ID-related checkboxes in Transmittal Set Details, see: [Revision IDs](#).

Show or Hide Change Markers in This Transmittal Set

Use the Show Markers pop-up of the New Transmittal Set or Transmittal Set Details dialog box to define how to show or hide the project's Change Markers on the Layouts included in this Transmittal Set.



- **Of latest Revision only:** Only the Markers that are part of the latest Revision will appear on the Layouts of this Transmittal Set. (Markers associated with previous Revisions will not be shown on the Layout.)
- **All:** All Change Markers are shown.
- **None:** No Change Markers are shown on any Layout.
- **Since a particular Transmittal Set:** Change Markers are shown from all the Revisions starting with the Transmittal Set chosen here.

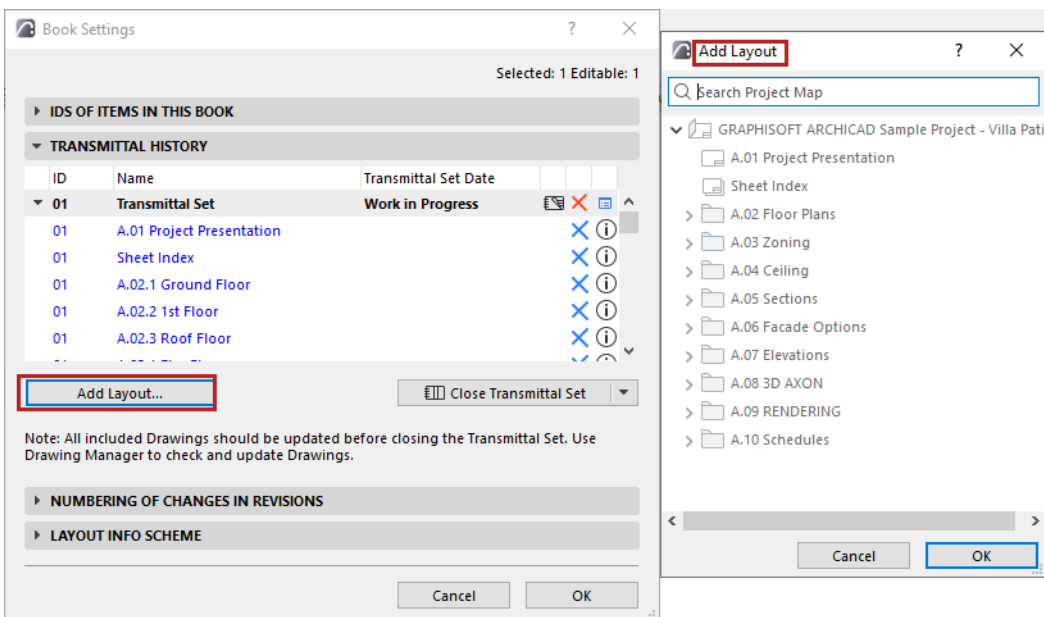
[Watch the Video](#)

Add Layout to Current Transmittal Set

Layouts that have gotten a new Revision since the last Transmittal Set are automatically added to the current Transmittal Set (provided that the Layout Drawings are up to date).

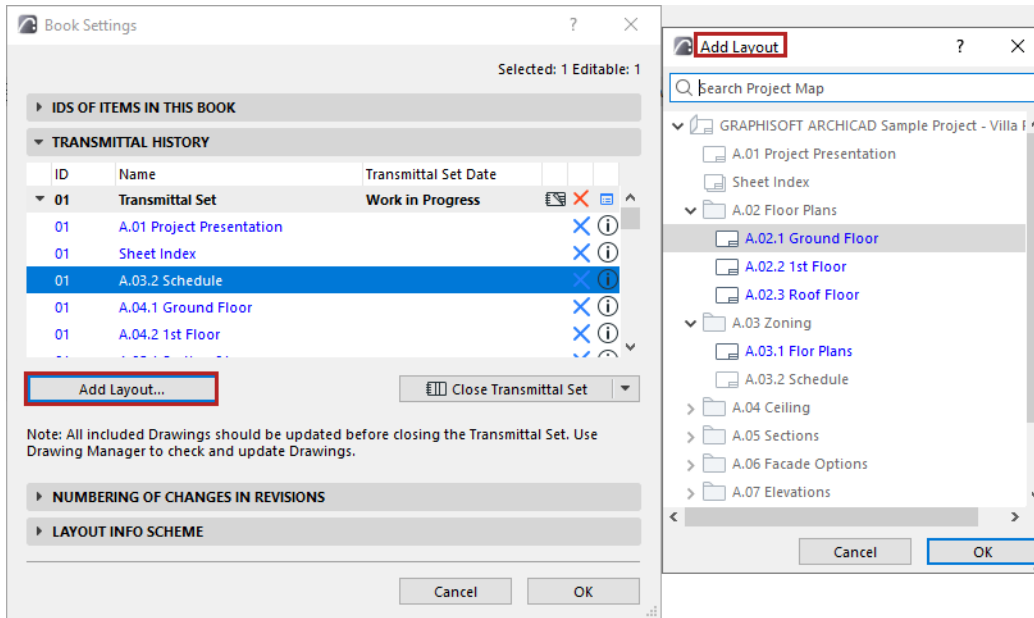
To add any other Layout:

Click **Add Layout** from the Transmittal History panel:

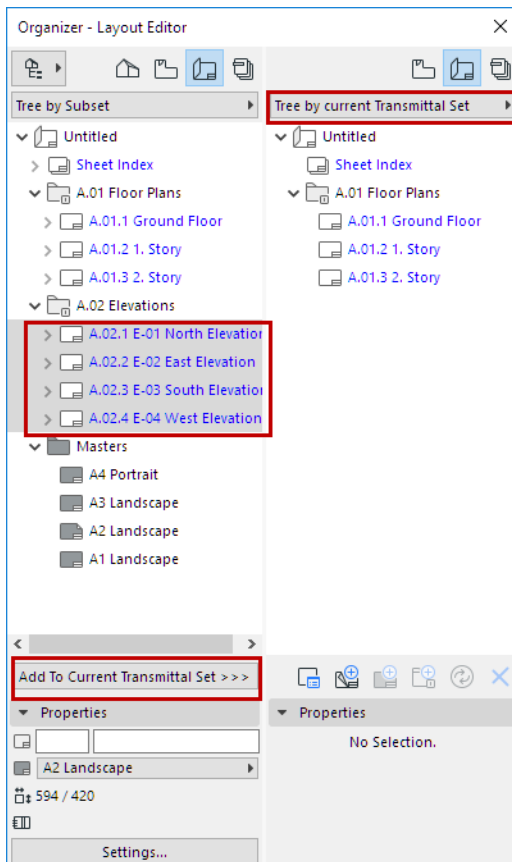


Note: Layouts listed in gray text in the Add Layout dialog box have already been added to the Transmittal Set; you cannot add these. You can add the ones in **black** type (not yet in the Transmittal Set) and/or those in **blue** (untransmitted Revisions that are not yet in this Transmittal Set).

or from the Navigator Layout Book in “Tree by current Transmittal Set” mode:

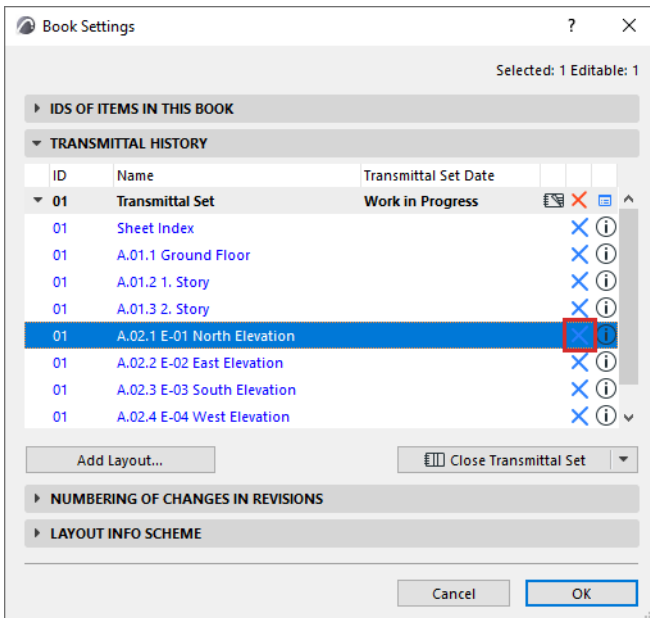


In the Organizer’s Layout Editor, with the right side in “Tree by current Transmittal Set” mode, drag and drop Layouts into the Layout Book to add them to the current Transmittal Set:

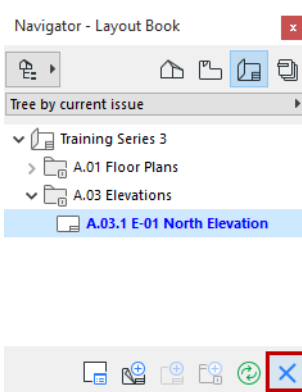


Remove Layout from Current Transmittal Set

1. Open Layout Book Settings.
2. From the Transmittal History panel, select the Layout to be removed.
3. Click the blue delete icon to remove it from the Transmittal Set.



or from the Navigator Layout Book in “Tree by current Transmittal Set” mode:

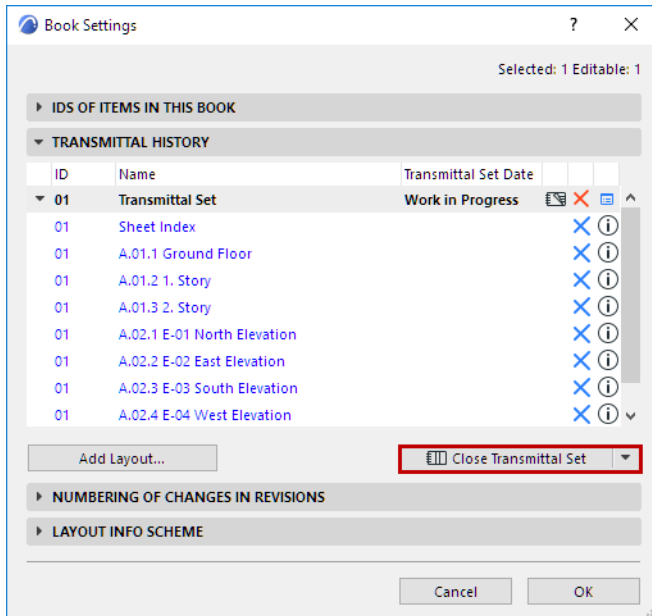


You cannot remove Layouts from closed Transmittal Sets.

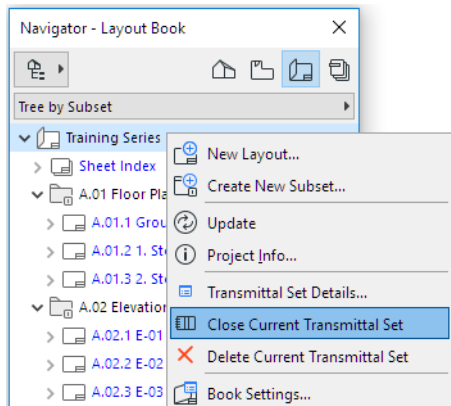
Close Transmittal Set

Use the Close Transmittal Set command:

- from the Transmittal History panel of Layout Book Settings

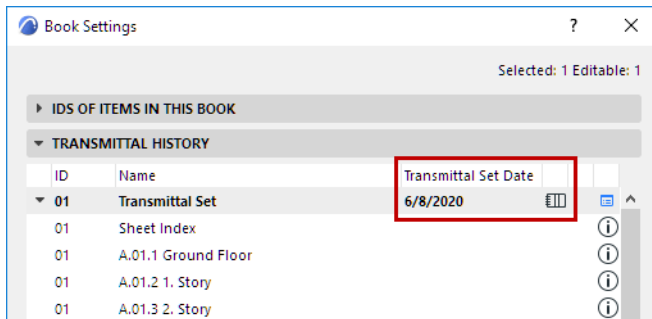


- from the context menu of the Layout Book in the Navigator

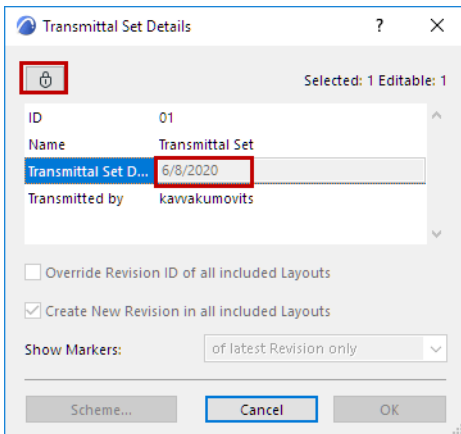


What Happens When A Transmittal Set Is Closed?

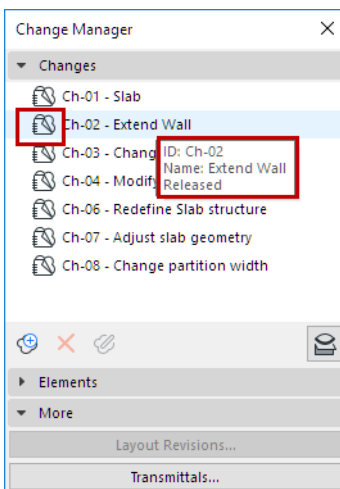
- The icon in the Transmittal History panel of Layout Book Settings changes to “closed”.
- The Transmittal Set Date is finalized (no longer indicates “Work in Progress”)



- Data in Transmittal Set Details are read-only and can no longer be modified. The Transmittal Set Details dialog box displays a lock icon.



- The current **Revision** of each Layout included in the Transmittal Set is closed
 - The Revision ID can no longer be modified
 - The WIP (Work in Progress) suffix is removed
 - No Changes can be added or removed from this Layout Revision
 - Any new Change to the Layout results in a new Revision being created
- All **Changes** on transmitted Layouts get the status of “Released”.
 - They are listed with the transmitted icon in the Change Manager



- Transmitted Changes can no longer be deleted; they can only be archived

See [Archive Changes in Change Manager](#).

- The data of transmitted Changes can only be modified to a limited extent.

See [Modify Change Details](#).

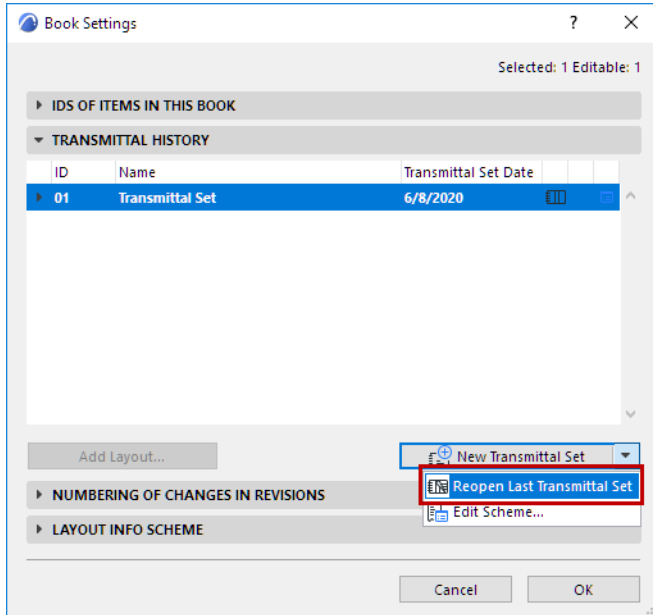
- Transmitted Changes can still be linked to model elements. This way, an transmitted Change can still take part in a later Transmittal Set, on a different Layout.
- Transmitted Changes can be archived in the Change Manager (see below)

Reopen Last Transmittal Set

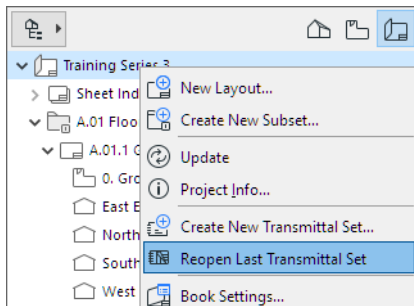
Note: In Teamwork, the Transmittal Sets - Modify History access right is required to reopen a Transmittal Set. (By default, this right is assigned to the Lead Architect role.)

A closed Transmittal Set can be reopened using the **Reopen Last Transmittal Set** command

- from Transmittal History



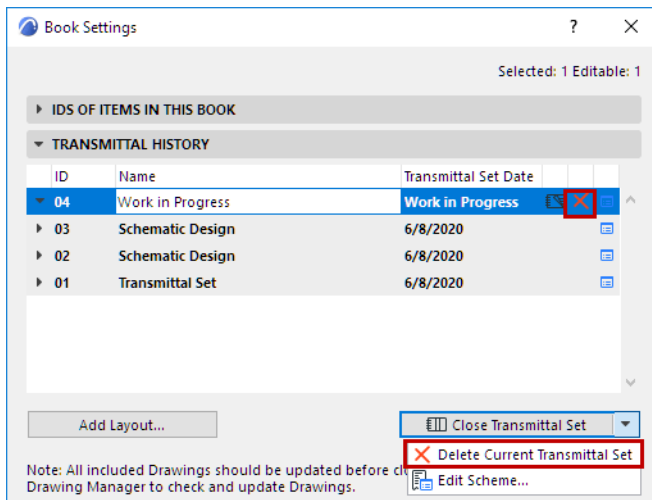
- from the Layout Book's context menu in the Navigator



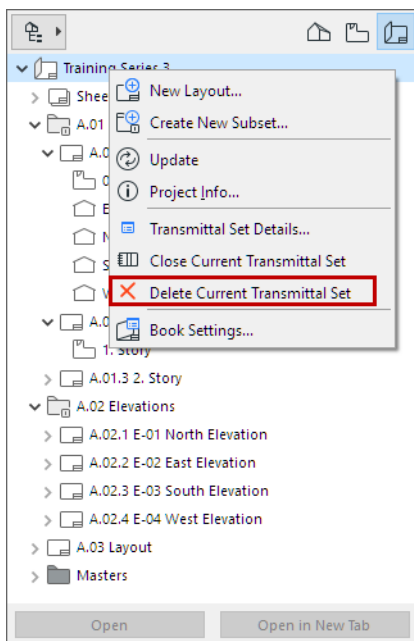
Delete Transmittal Set

You can delete the current open Transmittal Set using the Delete Transmittal Set command:

- in the Transmittal History panel of Layout Book Settings, in two different places as shown here:



- from the Layout Book context menu in the Navigator or Organizer:

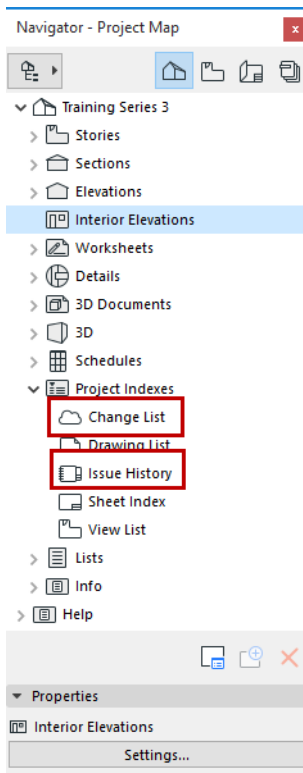


Deleting Transmittal Sets is not undoable.

[Watch the Video](#)

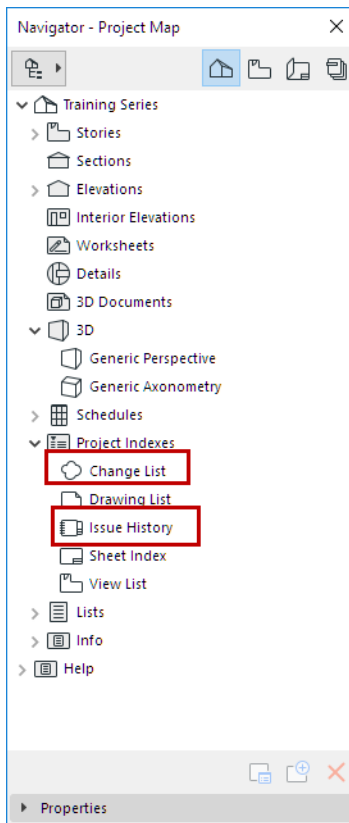
Change List and Transmittal History

Two preconfigured Project Indexes are available for easy listing of Revision-related information:



Change List

A Project Index that lists all the changes in a particular project. Use the Criteria and Fields parameters of Index Settings to list exactly the data you require.



Transmittal History

A Project Index that lists Layouts by Transmittal Set and by Revision, and can also include Changes for each. Note that the Transmittal History list will include all transmitted Layouts - even those which you may have deleted from the project since the Transmittal Set.

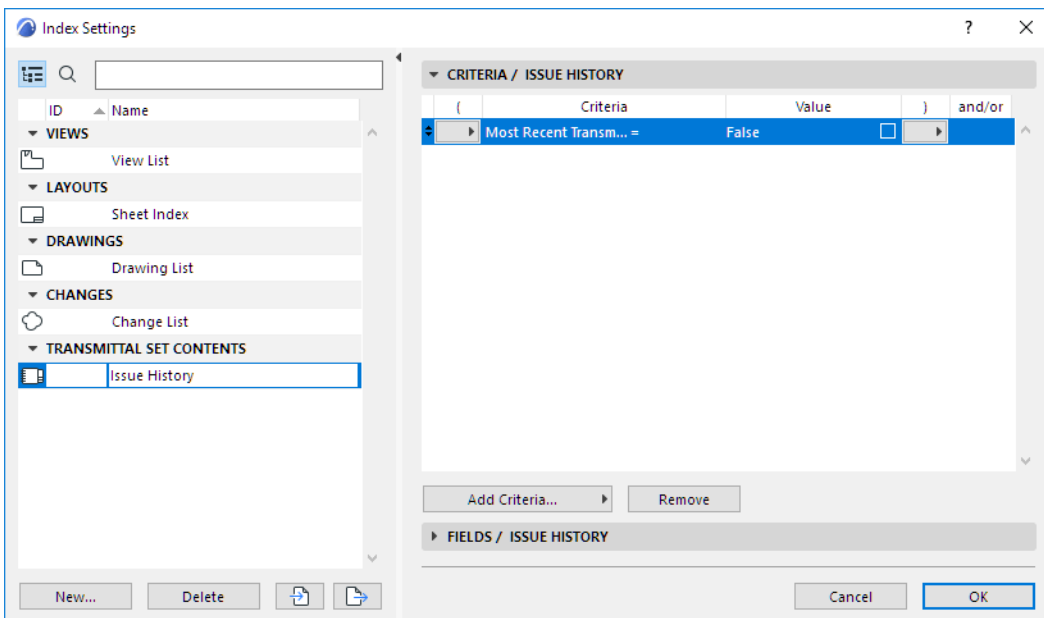
Use the Criteria and Fields parameters of Index Settings to list exactly the data you require.

Issue History				
Issue	ID	Name	Revision	Changes
01 , First Issue , 12/13/2013 4:49 PM				
		Sheet Index	01	
	A.01.1	Ground Floor	01	
	A.01.2	1 Story	01	
	A.01.3	2 Story	01	
	A.02.1	E-01 North Elevation	01	
	A.02.2	E-02 East Elevation	01	
	A.02.3	E-03 South Elevation	01	
	A.02.4	E-04 West Elevation	01	
02 , Second Issue , 1/10/2014 2:41 PM				
	A.01.1	Ground Floor	02	Ch-01, Ch-02, Ch-08
	A.01.4	Layout_for_PDF	01	
	A.02.1	E-01 North Elevation	02	Ch-02, Ch-08
	A.02.2	E-02 East Elevation	02	Ch-01, Ch-08
	A.02.3	E-03 South Elevation	02	Ch-01, Ch-08
	A.02.4	E-04 West Elevation	02	Ch-08
03 , Third Issue , 1/10/2014 2:50 PM				
	A.01.1	Ground Floor	03	Ch-10, Ch-11
	A.01.4	Layout_for_PDF	01	
	A.02.1	E-01 North Elevation	03	Ch-10
	A.02.2	E-02 East Elevation	03	Ch-10
	A.02.3	E-03 South Elevation	03	Ch-10
	A.02.4	E-04 West Elevation	03	Ch-10
04 , Fourth Issue , 1/10/2014 3:05 PM				
	A.02.1	E-01 North Elevation	03	Ch-10
	A.02.2	E-02 East Elevation	04	Ch-11, Ch-12
	A.02.3	E-03 South Elevation	04	Ch-11, Ch-12
	A.02.4	E-04 West Elevation	03	Ch-10
05 , Fifth Issue , 1/10/2014 3:17 PM				
	A.01.1	Ground Floor	04 - WIP	Ch-16, Ch-12, Ch-19
	A.01.3	2 Story	02 - WIP	Ch-17
	A.02.3	E-03 South Elevation	05 - WIP	Ch-17, Ch-16

Transmittal Record

To create a **Transmittal Record** - that is, to list the contents of any given Transmittal Set - just adjust the Index criteria.

To narrow the index to list only the current Transmittal Set contents, use the Most Recent criterion for the Transmittal Set Date.

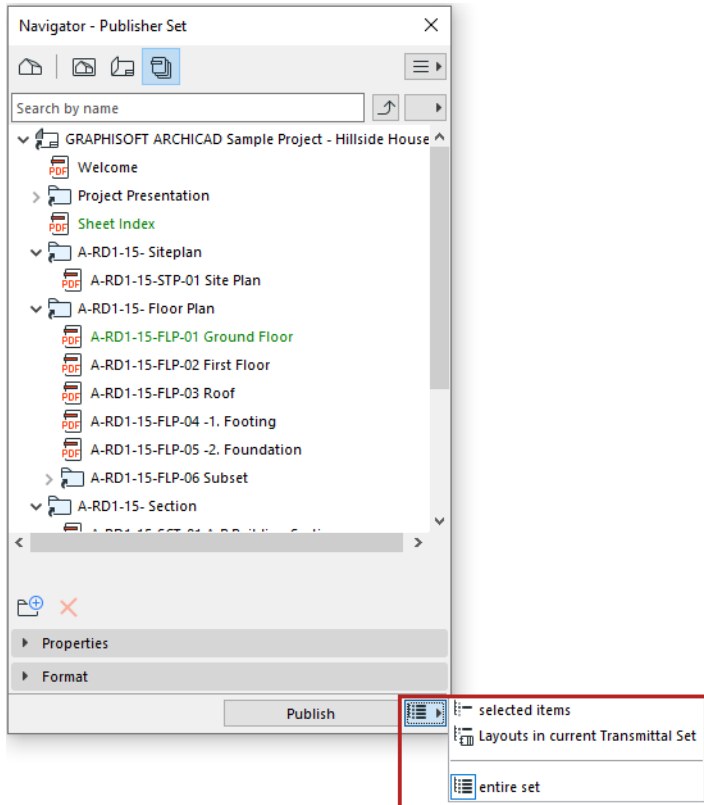


Issue	ID	Issue History		Revision	Changes
			Name		
05 , Fifth Issue , 2/17/2014 11:32 AM					
	A.01.1	Ground Floor		04	Ch-16, Ch-12, Ch-19
	A.01.3	2 Story		02	Ch-17
	A.02.3	E-03 South Elevation		05	Ch-16, Ch-17

Revisions in the Publisher

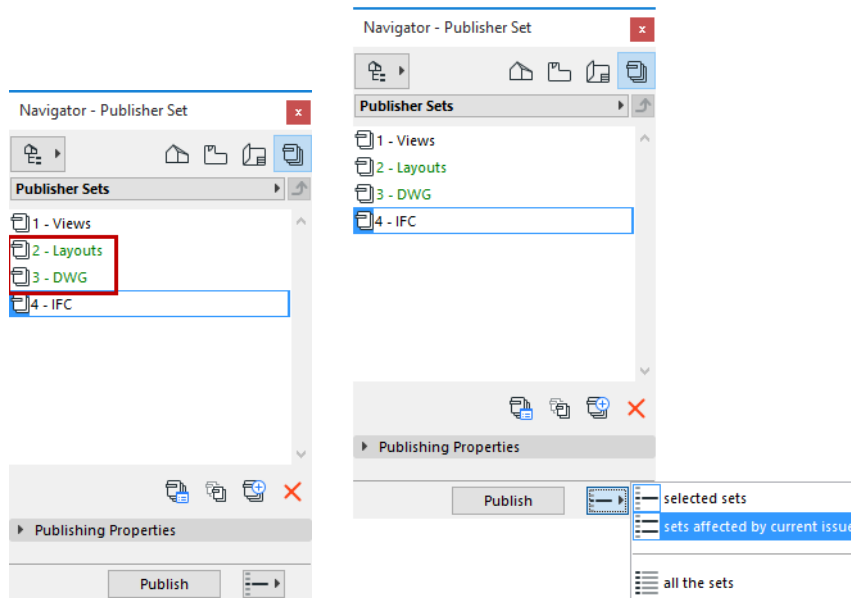
Once you have created a Transmittal Set and are using Layout Revisions, the following automatic feedback in the Navigator's Publisher mode can help you in defining your Publisher Sets:

Layouts in Current Transmittal Set



- Within a Publisher Set: all Layouts that are *in the current Transmittal Set* are shown in green
- To publish just the green Layouts, use the **Layouts in current Transmittal Set** Publish option. This will publish all layouts of the active publisher set that are part of the current Transmittal Set.

Publisher Sets Affected by Current Transmittal Set



- In the list of Publisher Sets: the name of any Publisher Set that includes a green Layout (that is, it includes a Layout that is part of the current Transmittal Set) is also shown in green.
- To publish all of these green Publisher Sets, you can use the Publish option: **Sets affected by current Transmittal Set**.

[Watch the Video](#)

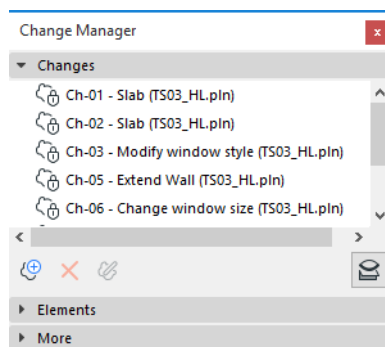
Changes from External Files

When placing an external Drawing (via an Archicad view/Layout) or a Hotlinked Module in your model, the arriving files may include Change data.

Changes in the Hotlinked file or external view are placed into the Host file and added to the host file's Change Manager. However, note the following:

- Such external Changes are shown with a lock icon and their details cannot be edited
- Archived Changes in a Hotlinked file are not added to the host project
- Changes in external Drawings are added to the host project only if they are visible on the Layout

If the Change in the imported item has the same Change Number as a Change in the host file, they are combined as entries in the Change Manager.



When a Transmittal Set is closed, local copies of the imported Changes are created. From this point on, they are preserved in the project history.

Publishing

Outputs are the end result of the architectural design workflow; Archicad gives you a high degree of flexibility in printing, plotting, and electronic publishing.

For quick outputs of the current on-screen view, the **Print and Plot** commands are available directly from Archicad's file menu; the Print and Plot dialog boxes contain familiar options.

You will normally use printing for creating fast outputs on which you can check various aspects of your design and plotting for creating the final documentation that will be submitted and sent to the client, the building authorities and the subcontractors.

For large-scale, planned outputs of one or more publisher sets, the Publisher function is a handy concept dedicated to the architectural design workflow. Using the Publisher palette, you set up and save preferences for any number of publisher sets: you define whether to print, plot or save them to disk. Once a publisher set is defined, you can publish or republish it at any time, using the same properties, at the push of a button.

Topics in this section:

[Print](#)

[Plot](#)

[Publisher](#)

[Create PDF Output Using the Publisher](#)

Print

The **File > Print** command will bring up a Print dialog box which varies slightly depending on the view you are printing: Layout, 2D or 3D.

[Print 2D Document](#)

[Print 3D Document](#)

[Print Picture](#)

[Print Layout](#)

Note: The following descriptions are based on the Windows interface.

Items to be Printed

The **Print** command enables you to print the entire contents of the current window, or just a particular Print Area, as set in the [Print 2D Document](#) dialog box. From the Layout Window, you can print just the Layouts which are selected in the Navigator.

Items that cannot be printed include **Hotspots**, selection dots, **Section** depth lines, **Roof** pivot lines, **Cameras** and their paths.

The lines of the Construction Grid can be optionally printed from the 2D windows by checking the **Print Grid** checkbox. Note that you can only print the grid if Grid Display is ON.

See [Show/Hide Construction Grid](#).

Items on a **Trace view** can also be printed.

Note: Elements that are located on remote stories but individually set to display on the current story will always appear on the output.

Use the **Document > Model View > [Model View Options](#)** dialog box to specify the display of model items on the Floor Plan, and thereby the printed output.

Header/Footer

When printing from Archicad's 2D or 3D window or Layout window, you have the option to place a **Header** or **Footer** on the printed page. You set the content of the Header/Footer in [Header/Footer Settings](#).

Output Color

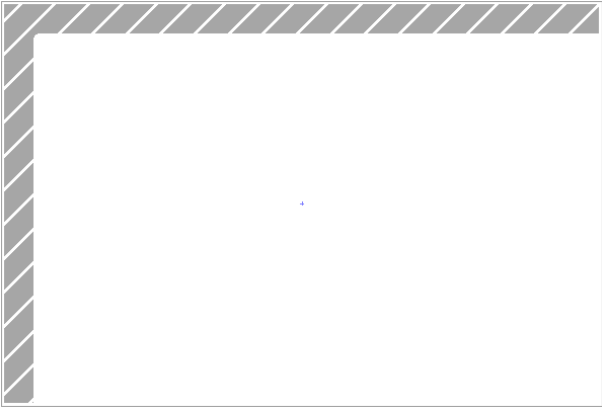
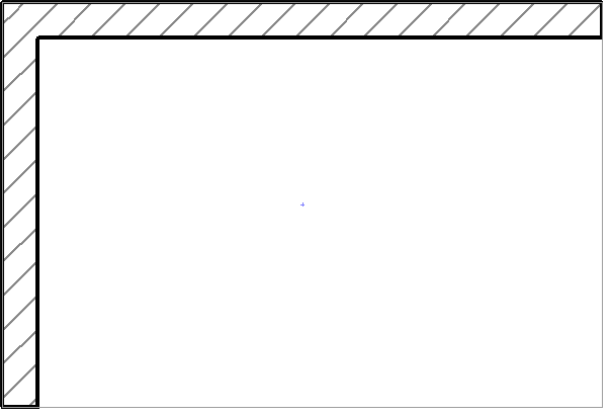
If you are using a printer that supports color or grayscale output, the color of the printed elements is defined by:

1. The setting made in the Print dialog box. Checking the Black and White box will print all colors in black independently of any other setting.

For more information, see [Print 2D Document](#).

2. The setting made on the Properties panel of the [Drawing Settings](#) dialog box for each individual drawing placed on the layout.

Note: In the Drawing Settings Properties panel, the **Pen Set** pop-up allows you to choose a pen set for the drawing; the **Colors** pop-up allows you to choose the colors that will be used for printing it. Since the color and grayscale pensets are not connected, it makes a difference whether you first choose the Color pen set and then choose to display the drawing in grayscale or the other way round. The two results can be entirely different as shown below.



Plot

The **File > Plot** command brings up the [Plot 2D Document](#) dialog box.

Note: The following descriptions are based on the Windows interface.

Plotter drivers shipped with Archicad all use the HPGL vector-graphic language. We refer to any printing device that supports HPGL as a 'plotter'. Graphisoft provides dedicated plotter drivers for most plotters; for printing, you must use printer drivers provided by the vendor.

Note: Many current large-format output devices come with printer drivers which make the device function as a system printer. If the device is also HPGL compatible, you can use the dedicated plotter drivers provided by Graphisoft.

Plotter Setup

When outputting documents to a plotter, you first need to set up the plotter for Archicad. The communication with the plotter device is ensured by specific plotter drivers stored in the **PlotWare** folder located by default in the same directory as Archicad.

Choose **File > Plot Setup** to open the [Plot Setup](#) dialog box.

Note: The ReadMe file also contains extensive information on issues related to plotting, such as recommendations on the choice of drivers and cables, and network plotting.

When first using a plotter with Archicad, you need to connect to it by clicking the **Setup** button in the Connection area.

You can only plot graphic data from Archicad. If a text window is in front, both the **Plot Setup** and **Plot** commands appear in gray in the File menu.

Plot versus Print

Some of the basic differences with printing:

- You cannot define a plot range; the entire contents of the active window will be plotted. From a Layout Window, you can choose to plot the Layouts selected in the Navigator.
- You can choose a color for the lines of the window grid, if you wish to plot it from a 2D window.
- Elements in a Trace view cannot be plotted.

You may prefer Plotting to Printing in these cases:

- **Spool Folder:** You can output a print/plot job into a file in both Printing and Plotting. However, if you need to collect your output jobs into a spool folder, use the Plotting function. Set the location of the Spool Folder in the Plot Setup dialog box.
- **Grayscale Output Color:** When Plotting, you have the option to select the color choice of the output: Color, Grayscale and Black and White are the available options. The Print dialog box lets you mark the Black and White checkbox if you prefer Black/White to color. (You can, however, also achieve grayscale output when printing: use grayscale Pen Sets for Model Views, or set grayscale display for Drawings placed on Layouts.)
- **Speed:** Plotting is generally faster than Printing, so if you have a high-volume output job, you should consider Plotting.

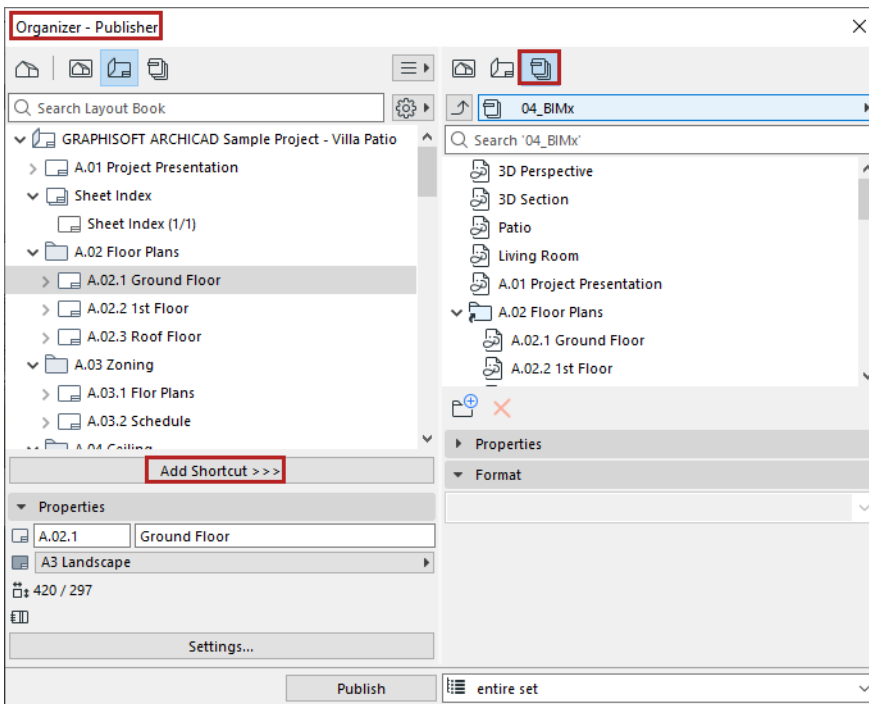
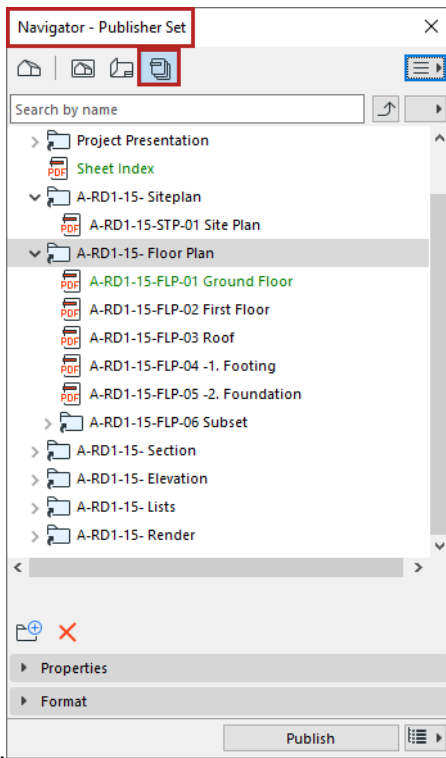
Publisher

The Publisher function is the fourth map in the Navigator or Organizer.

Publisher map items are views and Layouts, collected in Publisher Sets. Use the Publisher to set up their output in a variety of formats.

Access Publisher Map

- Select the Publisher map from the top of the Navigator/Organizer; or
- Use **Document > Publish**. (This opens the Organizer in Publisher mode)



The Publisher Map contains the project’s Publisher Sets, which are collections of Publisher items.

Each Publisher item is a shortcut which refers directly to a View or to a Layout.

Publisher Sets are stored with the project, so you can access them any time, adjust settings if needed, and re-publish them.

Topics in this Section

List of Publisher Sets

Create new Publisher Set

Add items to Publisher Set

Properties of Publisher Set items

Rename Publisher Items

Define Publishing Method: Print, Save, Plot, BIMx

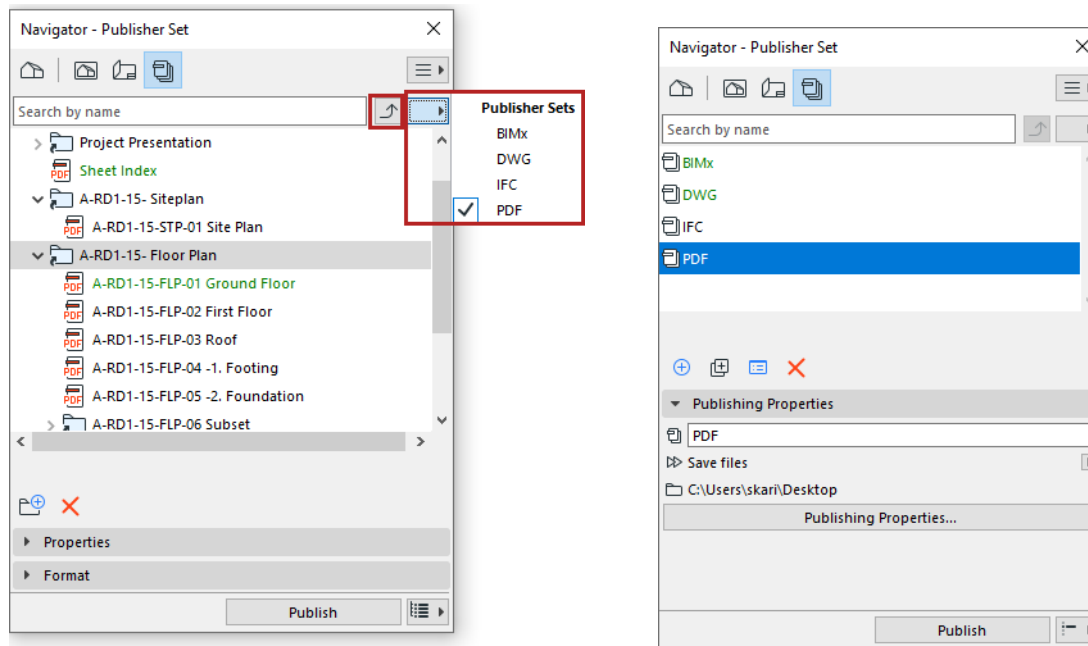
Related Topics

Publisher File Formats

Publishing Process

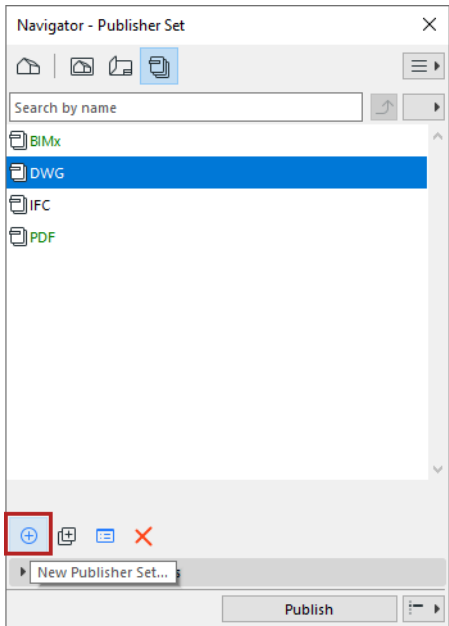
List of Publisher Sets

To see the list of all Publisher Sets defined in this project, click the “Level up” button or the pop-up button.



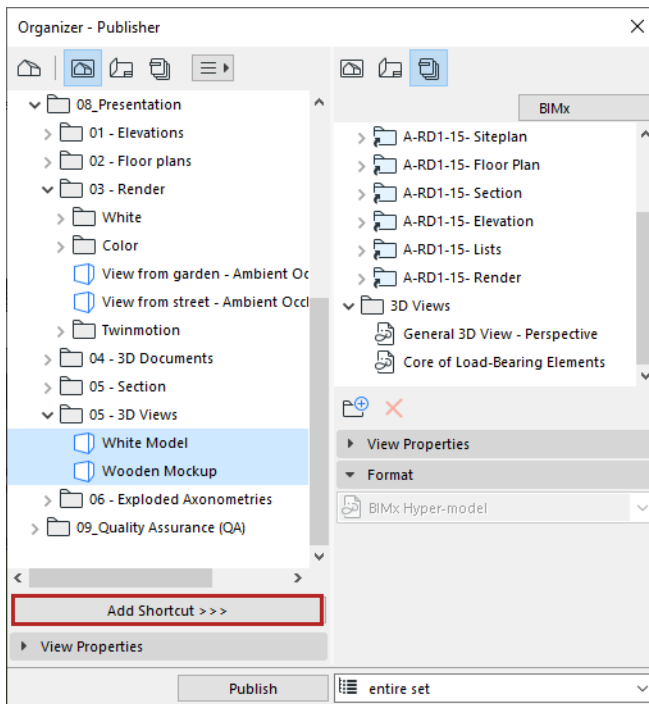
Create new Publisher Set

From the Publisher set list: Click the **New Publisher Set** button.



Add items to Publisher Set

1. Open the Publisher mode of the Organizer:
 - Document > Publisher
 - Open the Organizer, and make sure the Publisher set is on the right
2. From the left side of the Organizer: Select a view (from the View map) or Layout item (from the Layout Book),
3. Drag and drop the item into the Publisher Set, or click **Add Shortcut**.



Notes:

- If you add a Layout Subset to the Publisher set using the **Add Shortcut** button: this creates a corresponding “clone” folder in the Publisher Set, and any changes to the original subset are automatically updated in the corresponding Publisher Set folder.
- If you add a Layout Subset to the Publisher set using **drag-and-drop**: the folder is added, but this folder is not a clone and is not updated automatically.

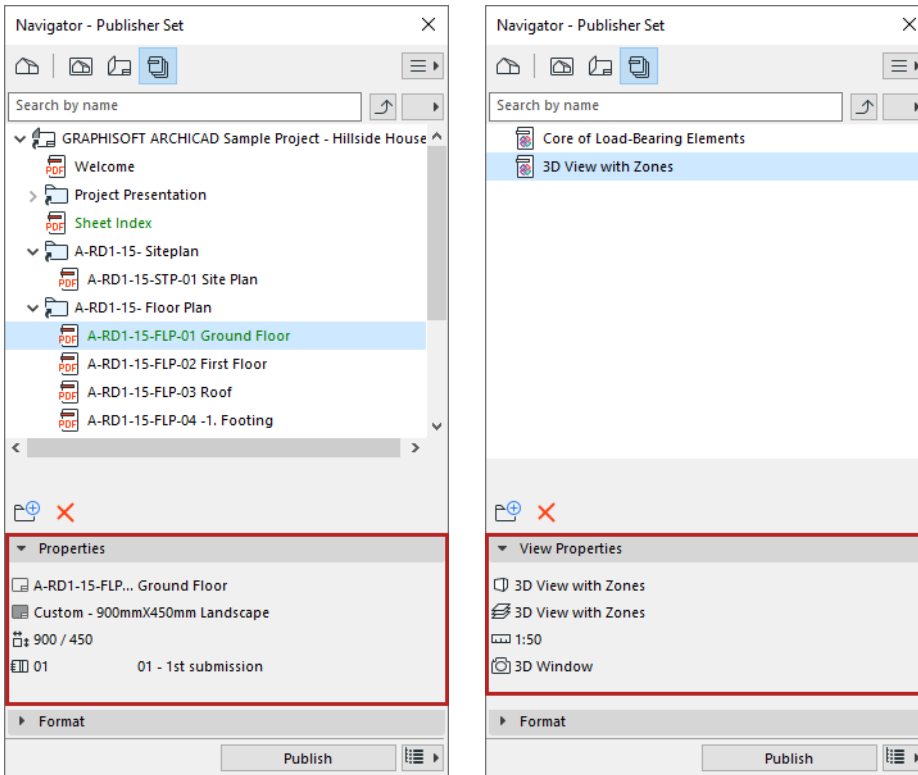
To define the output format for each Publisher item:

[See *Publisher File Formats*.](#)

Properties of Publisher Set items

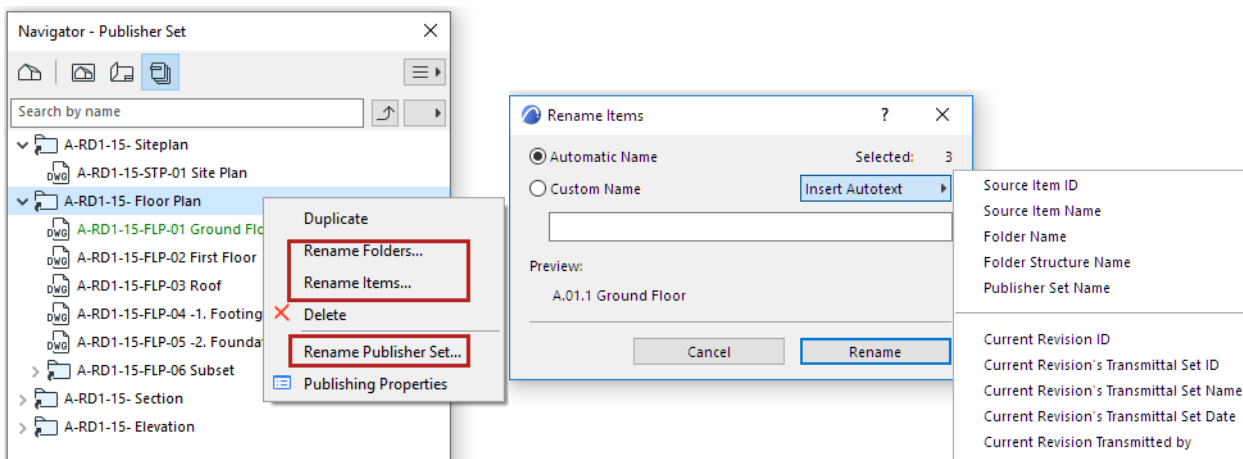
Select an item from the Publisher Set to view its Properties below.

These fields display the properties of the view or layout linked to the selected publisher item. They cannot be edited here.



Rename Publisher Items

You can rename a Publisher Set, as well as Publisher items, folders and their naming logic. Select one or more items or folders and use the context menu commands.



- A new item placed into a Publisher folder (or sub-folder) will take on the naming logic of that folder (or sub-folder)
- Use the **Custom Name** option to define any name for the item(s).
- The **Autotext** options can help you conform to local file naming conventions:
 - **Source ID:** Renames the item using View ID or Layout ID, depending on the source of the Publisher item
 - **Source Item Name:** Renames the item using View Name or Layout Name, depending on the source of the Publisher item

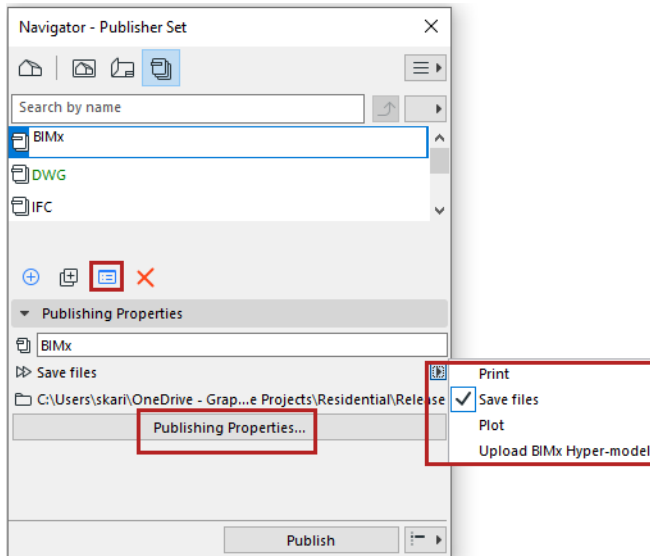
- **Folder Name:** Renames the item using the folder name in which it is located
- **Folder Structure Name:** Renames the item using the entire folder path of the item's location in the Publisher Set
- **Publisher Set Name:** Renames the item using the Publisher Set name
- **Automatic Name** means that the Publisher item will have the same name as its source (e.g. Layout name or Drawing name).

Note: A *folder* in the Publisher which was created as a shortcut from the View Map or Layout Book can also have an Automatic name: this is the same name as the name of the original View Map folder/Layout Book subset.

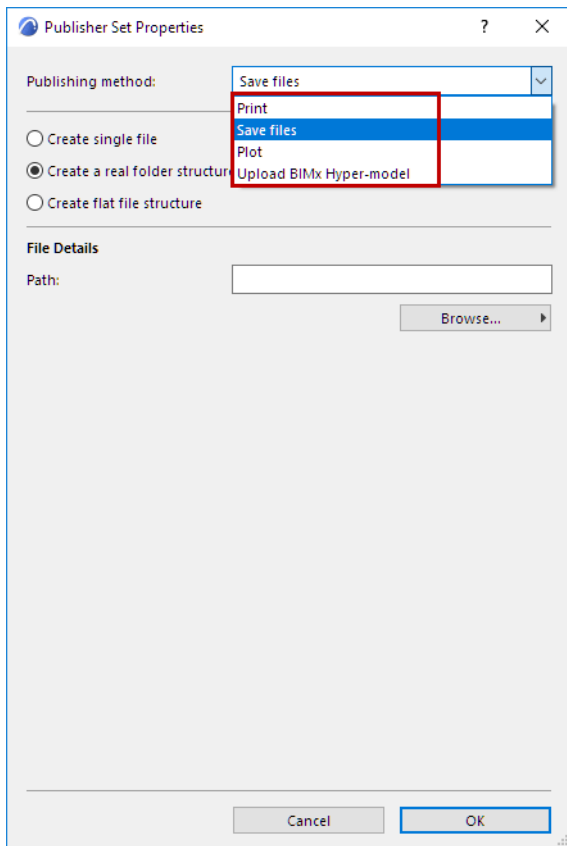
Define Publishing Method: Print, Save, Plot, BIMx

Choose a Publishing method for each Publisher Set.

1. In the Publisher map, make sure you are viewing the list of Publisher Sets. ([See List of Publisher Sets.](#))
2. Do one of the following:
 - Click the **Publishing Properties** button
 - Click the **Publishing Properties** command among the command icons
 - Click the pop-up button to choose a publishing method.



3. From the Publishing Properties dialog (or the pop-up), choose a publishing method:



Print/Plot

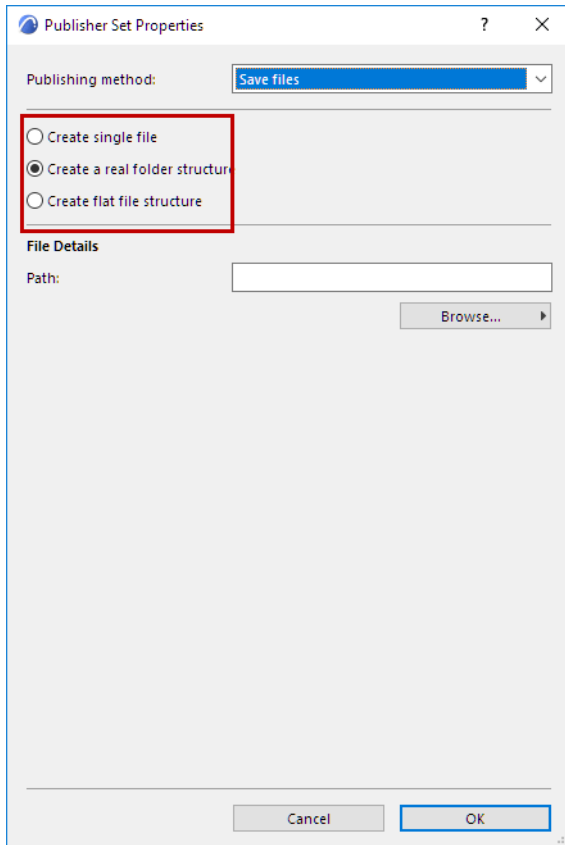
- use the buttons to open the corresponding dialog boxes (e.g. Page Setup, Print Settings).

For more information, see [Print 2D Document](#) and [Plot Setup](#).

Upload BIMx Hyper-model

Creates a Hyper-model for viewing and collaborating in BIMx - available as mobile app, desktop viewer, and web viewer. [See *Publish BIMx Hyper-Model from Archicad*](#).

Save files (to local folder or to BIMcloud)



- **Create single file:** Save the entire Publisher Set to a single file. The available formats are PDF and BIMx Hyper-model.

See [Publish BIMx Hyper-Model from Archicad](#).

- **Create a real folder structure:** Save your files in a hierarchical structure.
- **Create flat file structure:** Save your files in a flat structure.

Path: Enter the path of the folder you want to save your files in, or click **Browse** to locate a folder or BIMcloud.

For info on how to define the output format for each Publisher item, see [Publisher File Formats](#).

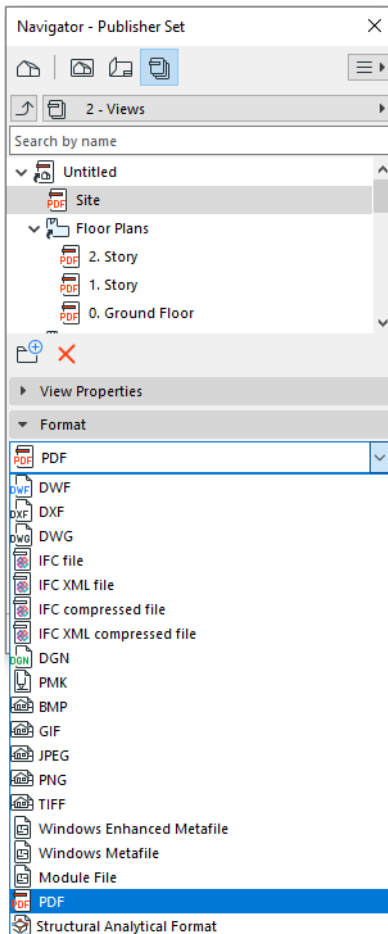
If you have finished setting up the publication but do not want to actually start publishing, you can simply close the palette without clicking the Publish button. All Publisher settings are preserved with the project.

Publisher File Formats

If you chose **Save** in the Publisher Set Properties dialog box:

Use the **Format tab page** to choose an image format for saving the item, either individually or by folder.

- Available formats depend on the view you are publishing (e.g. 3D model, or 2D list, etc.)
- Depending on format: choose a Translator and/or click **Options** for additional settings



PDF Format

Two options pages are accessible: **Document Options** and **Page Options**.

[See Create PDF Output Using the Publisher.](#)

PMK Format

The **PMK** format is a native Archicad drawing format which takes up relatively less hard drive space than many other formats. Saving views in PMK format can be a good workflow if your project is divided among multiple files (e.g. Model.pln and Documentation.pln).

[See Layout Workflow: Larger Projects.](#)

DWG or DXF Format

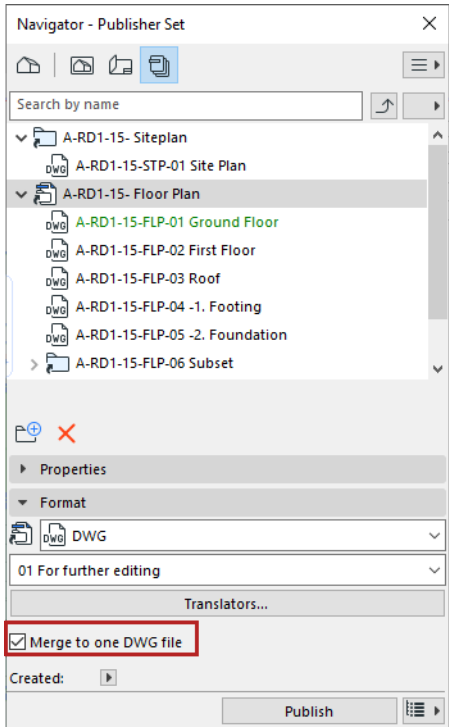
Choose a DXF-DWG Translator to control the content of the published output.

[See DXF/DWG Translation Setup.](#)

DXF Format: Merge to one DXF file

If all the items in a folder are *Layouts*, and if the entire folder is set to DWG format, an additional checkbox appears: **Merge to one DWG file**. If you use this option:

- The resulting DWG file will contain as many Paper Spaces as the number of Layouts that have been used as Publisher Set items. Thus, an entire Layout Book's contents can be saved in a single DWG file.



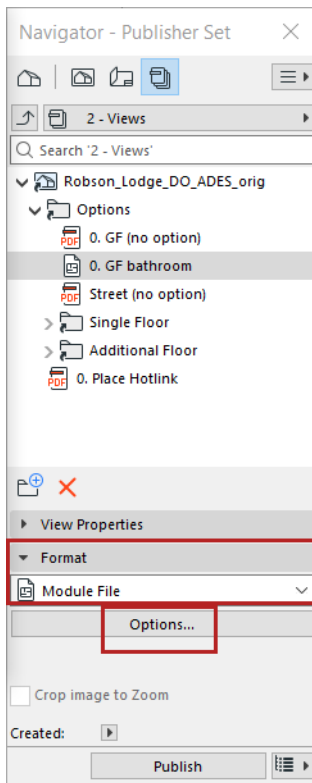
DWF Format: Crop Image to Zoom

If your publishing format is DWF: you can check **Crop image to Zoom** to crop the image to the current zoom on the Floor Plan.

When exporting an Archicad view for use by AutoCAD, you can ensure that only a selected part of the view is exported: First, publish the view in DWF format to a separate folder, using the **Crop image to zoom** option. Open the DWFs in Archicad and save them again as DWG files. Only the cropped view will be visible in the DWG file.

Module (.mod)

Available for views saved from the Floor Plan.



Click **Options** to set which stories and elements should be part of the module. (See [Module Save Options](#).)

IFC Format

For any of the IFC formats, use the pop-up to choose an IFC Translator for Export to control the content of the published output.

[See IFC Translators: Overview](#).

Click the **Translators** button for information on Translator content.

[Watch the Video](#)

Structural Analytical Format (SAF)

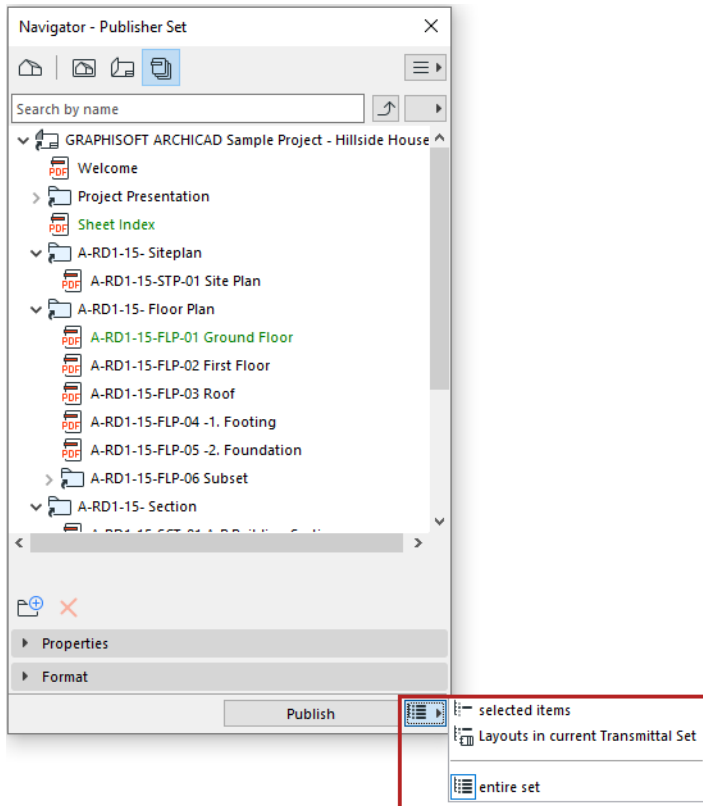
Choose this format for export Structural Analytical Model views from Archicad to external applications that use the finite element method (FEM).

[See Structural Analytical Model](#).

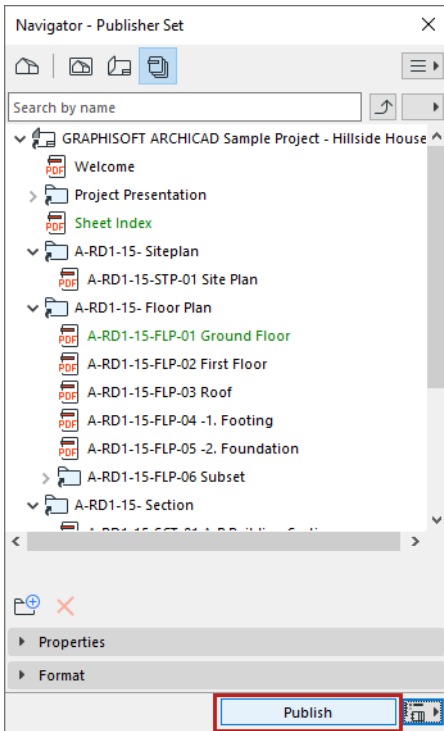
Publishing Process

Publish a Single Publisher Set

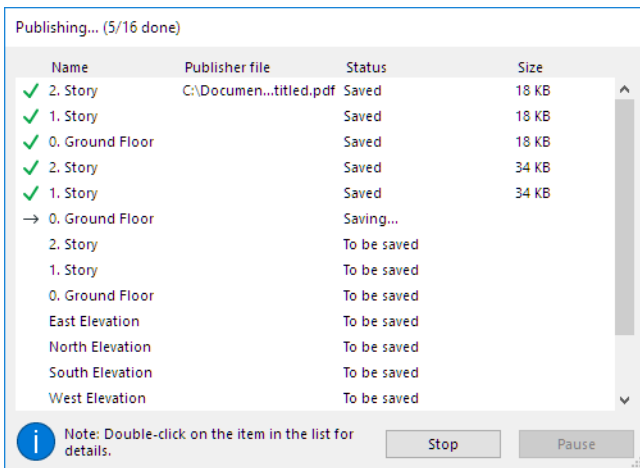
1. Go to the Publisher map in the Navigator or Organizer.
2. Select a Publisher set.
3. Choose which item(s) to publish.



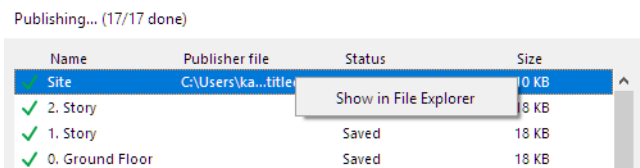
- **Entire set:** the entire active publisher set
 - **Selected items:** Only items currently selected in the active publisher set
 - **Layouts in current Transmittal Set:** All layouts of the active publisher set that are part of the current Transmittal Set. (Such Layouts are listed in green.)
4. Click **Publish**.



5. The Publishing progress dialog alerts you to the status of each item.

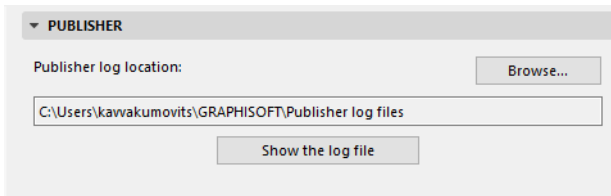


- Double-click any listed item for more info
- Right-click to navigate to its published location



Publisher Log File

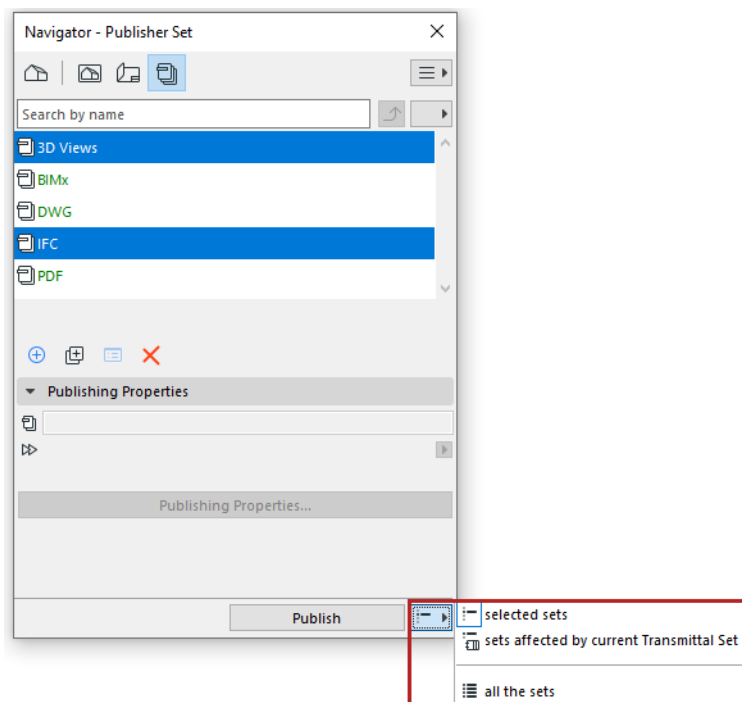
A log file containing information on each publishing process is saved to the location you specify in **Options > Work Environment > Publisher** (“Publisher log location”).



To view the log, click the **Show the log file** button.

Publish Multiple Publisher Sets

1. Go to the Publisher map in the Navigator or Organizer.
2. Go to the list of Publisher Sets. ([See *List of Publisher Sets.*](#))
3. If needed: select the Publisher sets to output.
4. Choose which set(s) to publish:



- **selected sets**
- **sets affected by current Transmittal Set:** All publisher sets that contain at least one layout that is part of the current Transmittal Set. (Such publisher sets are listed in green.)

[See also *Revisions in the Publisher.*](#)

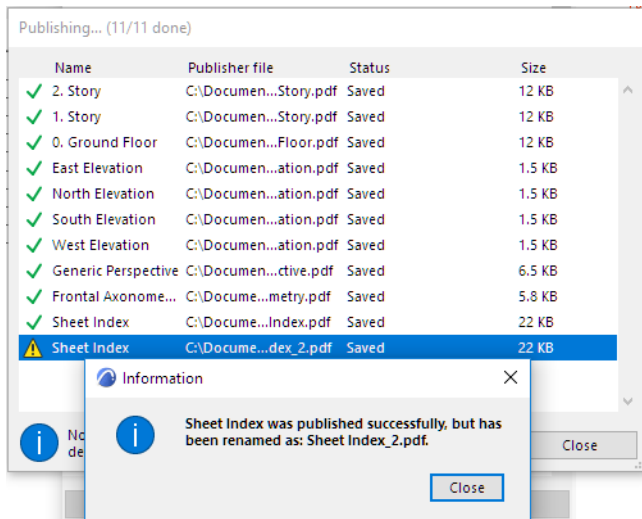
- **all the sets**

Revisions of Publisher items

You may need to output a Publisher item multiple times to keep pace with revisions. You may wish to keep the older published files as a record.

The file name of each output Publisher item is identical to the name of the Publisher item itself. This means that if you output a Publisher item, and later rename it to reflect a new revision, you can re-publish it under the new name, without overwriting the previously output file.

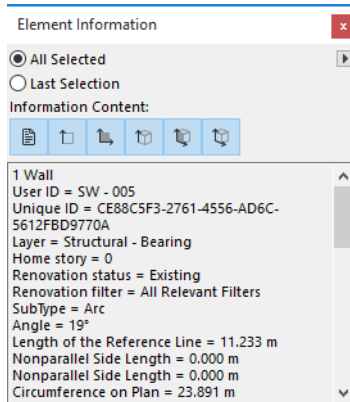
If you publish multiple items of the same name simultaneously, a yellow warning triangle appears in the Publishing process window; double-click to read the message, as shown below. Archicad will automatically rename such files.



See also [Republish Hyper-model](#).

Element Information

The **Element Information** Palette displays information (e.g. properties, dimensions) of element(s) selected on the **Floor Plan**. (Use **Window > Palettes > Element Information** to display it.)

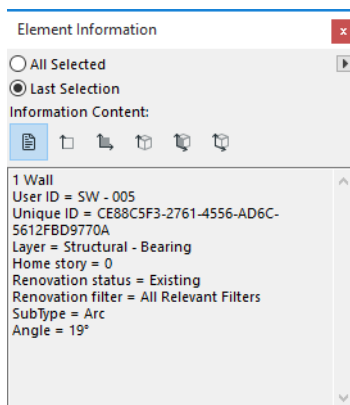


Choose either **All Selected** or **Last Selection** to define which selected elements should be considered.

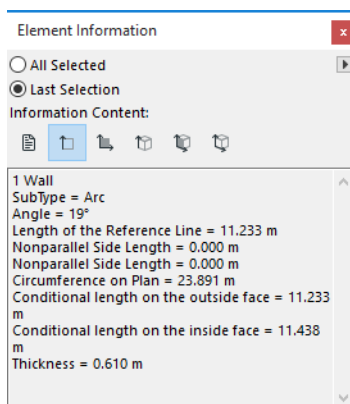
The **Information Content** buttons in the top section of the palette define what information will be shown. If none of the buttons is active, only the number and type of the selected elements is displayed.

The six buttons display, from left to right:

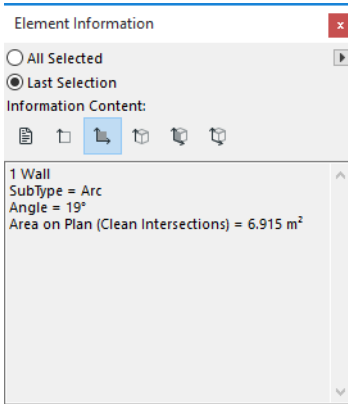
- **Classifications and Properties**



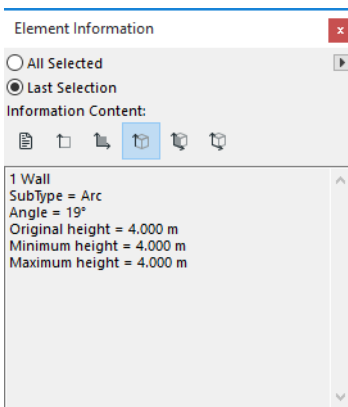
- **Length, circumference and thickness values**



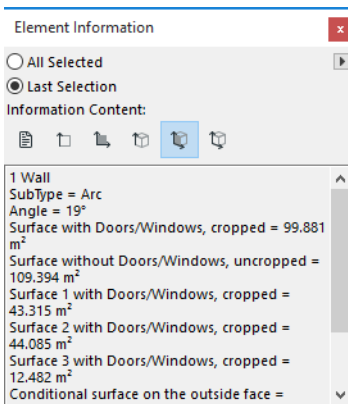
- Area values



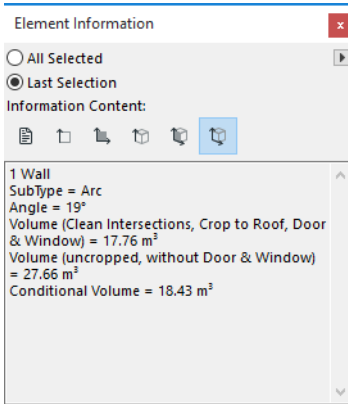
- Height values



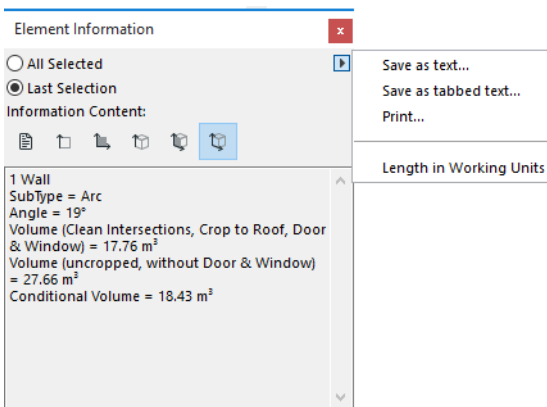
- Surface area values



- Volume values

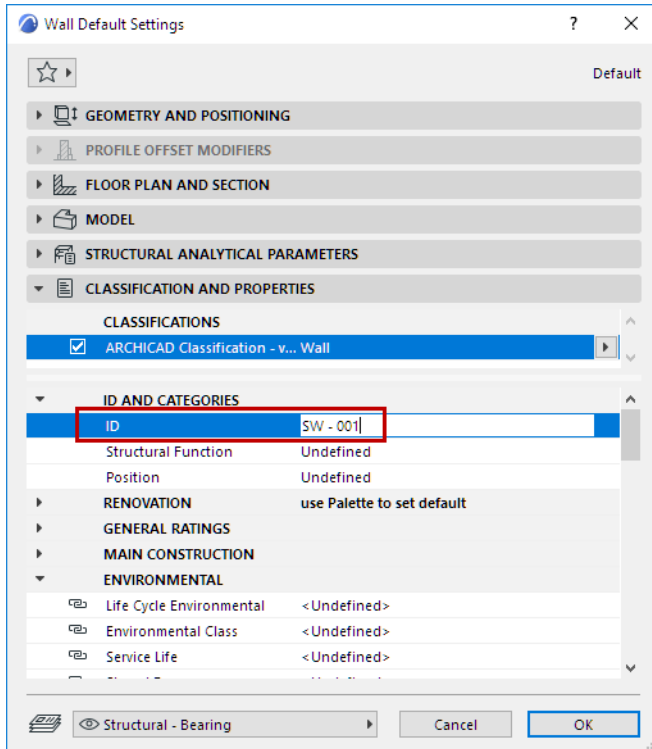


Use the pop-up commands in the top right corner of the palette to save or print the data shown.



ID Management

The **ID** field is located in the **Classification and Properties** settings panel of all construction element tools, as well as the **Fill** and **Zone** tools.



The ID identifies and groups elements in list views. You can also use the ID in the Label associated to elements. Any character can be used.

Element ID is also important if you wish to save the Archicad Model in BIMx Hyper-model format: BIMx provides feedback on selected elements based on their Element ID. (Make sure you include Classifications and Properties, including element ID data, when saving the Archicad model as a BIMx Hyper-model - otherwise, BIMx cannot provide ID-based feedback on the model's selected elements. See [Step 3: Info Set](#))

Note: All construction elements also have an automatically generated unique ID (GUID) which is conserved throughout the Project's life. You can also use this identifier for labeling or in lists.

Many List Schemes filter or group elements by their IDs. Totals obtained in calculations can refer to elements that have the same ID. Therefore, the content of the ID may be significant.

Important points

- If a number is included anywhere in the ID field, drawing successive elements adds one to this number for each new element, provided that the **Assign New Element ID to Each New Element** checkbox is enabled the **Options > Work Environment > More Options** dialog box. If there is no number in the ID field, each new element of the given type will have the same ID.
- If elements are duplicated or multiplied, the IDs of the replicas will remain the same as those of the originals.
- If you paste elements into a Project, you may have elements with conflicting IDs.

See also [Element ID Manager](#).

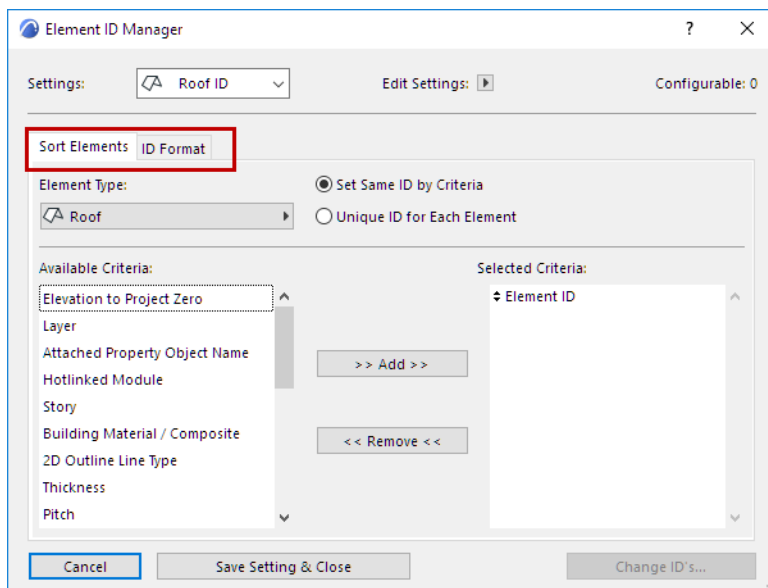
Element ID Manager

Use this dialog box at **Document > Listing Extras > Element ID Manager** to set the ID numbers of the project's elements based on their criteria. You can also use it to modify automatically generated ID numbers. Settings can be saved and stored for future use.

The configurations you create and save with the **Element ID Manager** help you prepare Project elements for clearer reading of quantity takeoffs.

Note: This command is only available from the Floor Plan.

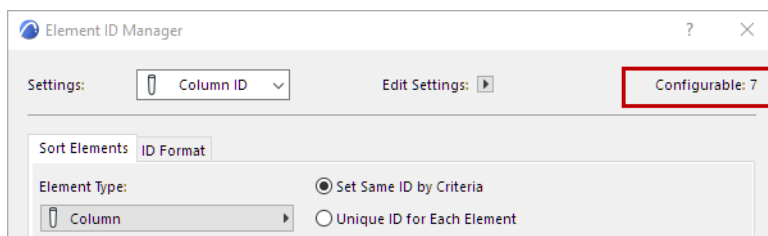
The dialog box includes two tab pages: **Sort Elements** for defining attribute choices and **ID Format** for defining custom ID sets.



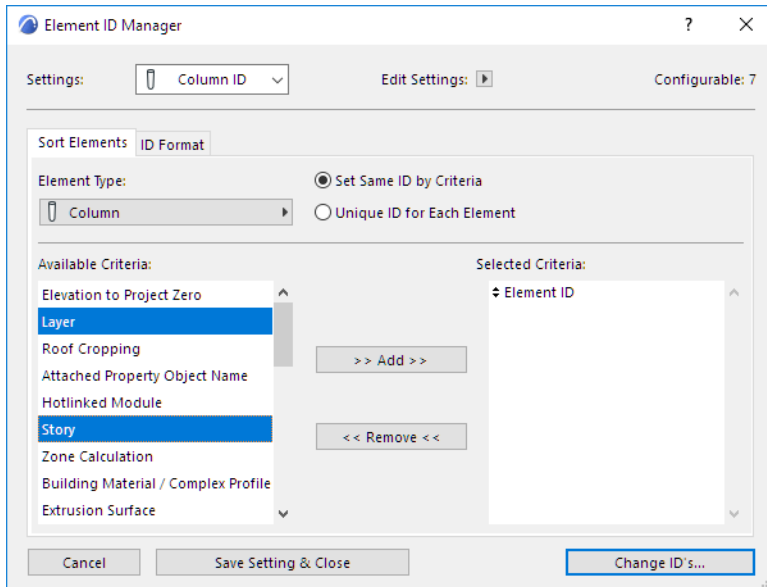
Sort Elements

When the **Sort Elements** tab is active: Choose an Element Type.

- If any elements are selected on the Floor Plan, Element ID Manager works on the selected elements only.
- If you select elements for assigning ID's, and have no Selected Criteria, the ID numbers will be assigned in the order the elements were selected.
- If nothing is selected on the Floor Plan, the command works on all placed and visible (not on hidden layers) elements.

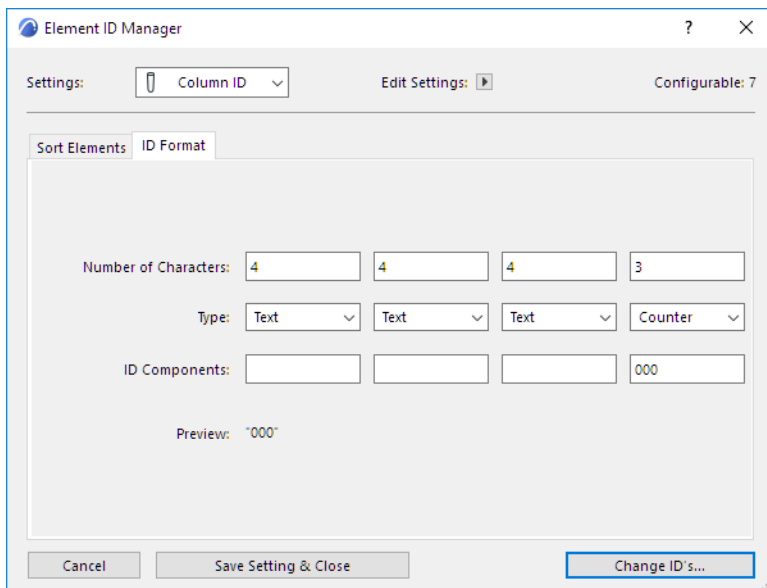


Select and add criteria to the list on the right.



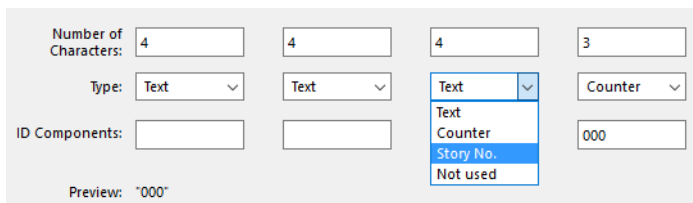
ID Format

Click the **ID Format** tab page. This allows you to define custom ID Formats:



Element ID Manager allows you to subdivide the ID into four parts.

For any of the parts, you can type plain text, a counter or a story number into the ID Components field.

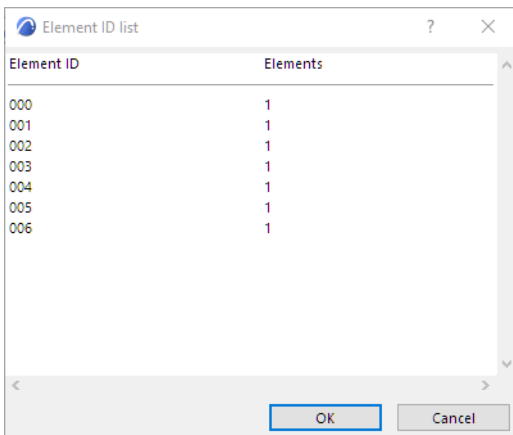
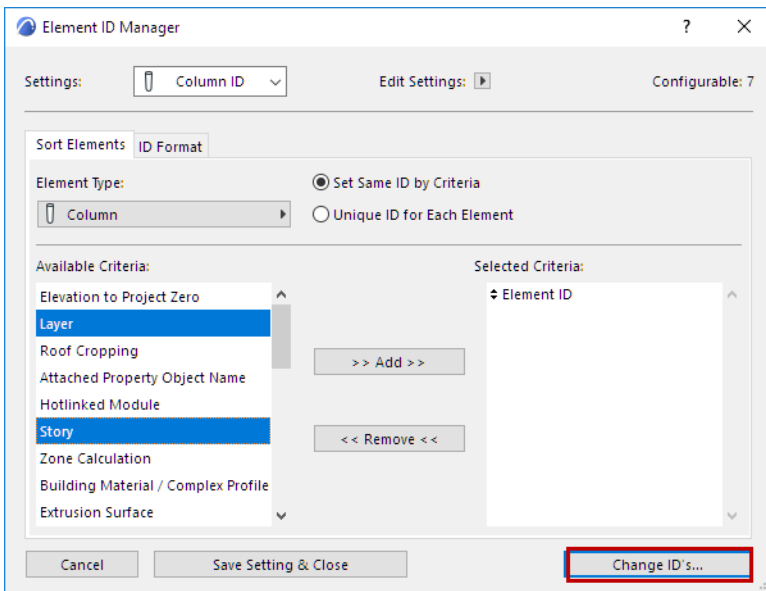


- Use **Text** Type to enter custom text.
- Use **Counter** Type to enter either numerals or letters such as “aa” that will change incrementally in the ID.
- Use **Story No. Type** to insert the ordinal number of the element story.

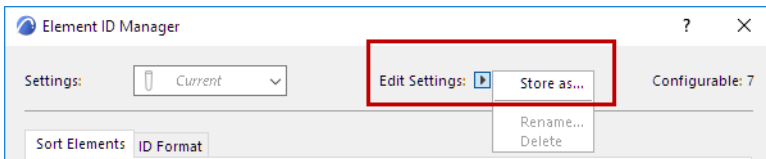
Note: Story No. is only available as a type if you've selected "Story" as a Grouping criterion.

Generate Element IDs

Click **Change ID's** to generate the **Element ID List** dialog.



You can save your settings using the **Edit Settings** pop-up menu on top of the dialog box.



Calculation (Legacy)

The Calculation function - Archicad's integrated database of construction elements - can be used to display, in list format, the number, quantities and components of elements in a project and the elements' spatial disposition (zones).

The report-generation process described in this section is a simple output function; use predefined list templates to generate simple lists.

Advanced users may wish to use customized formatting and define customized filtering properties.

The entire process is described in detail in the "Calculation Guide.pdf", available from Archicad's Help menu.

Calculation (Legacy) vs. Interactive Schedule (Supported)

The Calculation functionality is considered legacy, and is unlikely to support new developments in Archicad.

- The Stair and Railing Tools (as of version 21) are not supported.
- Segmented and tapered Beams and Columns (available as of version 23) are not supported. Single-segment Beams and Columns are supported as in previous versions.

Most users will prefer Archicad's Interactive Scheduler. Interactive Schedules are part of the Project Map, and the schedules produced can be saved as views and output on layouts. An Interactive Schedule is a two-way editable item which is linked to the project elements that it contains.

Topics in this section:

[Calculation Setup](#)

[List Types](#)

[Displaying Lists](#)

Related Topics:

[Interactive Schedule](#)

[Project Indexes](#)

[Calculation Units Preferences](#)

Calculation Setup

Some of the commands referenced in this section are not visible in Archicad's default work environment profiles. If you plan to configure lists (recommended for advanced users only), first use **Options > Work Environment > Menus** to add the following commands to an existing menu (i.e. **Document > Listing Extras**):

- Set up List Schemes (*this dialog box can be reserved in a Teamwork project*)
Note: List Schemes will function in Teamwork projects only if you save them to the project's Embedded Library.
- Edit Database (*this dialog box can be reserved in a Teamwork project*)
- New Properties
- Edit Properties
- Last Selection's Properties
- Link Properties to Criteria (*this dialog box can be reserved in a Teamwork project*)

List Types


Archicad can generate three types of calculation reports: Element Lists, Component Lists and Zone Lists.

Element Lists are best used when creating schedules and inventories, and to display the parameters of construction elements in a project. To generate Element Lists, Archicad filters the project or the selection set for construction elements according to the configuration of the selected List Scheme. Elements matching the filter settings will be listed together with element Parameters, Components and Descriptors, if so defined in the List Scheme.



General element list								2/7/01
Element	Story	Layer Name	User ID	Library part	Width / Thickness	Height	Surface	Volumes
WALL	0. Story	Exterior walls	Wall-013		0.20 m	2.44 m	19.07 m ²	3.87 m ³
	0. Story	Exterior walls	Wall-014		0.20 m	2.44 m	9.81 m ²	1.99 m ³
	0. Story	Exterior walls	Wall-015		0.20 m	2.44 m	16.66 m ²	3.39 m ³
	0. Story	Exterior walls	Wall-016		0.20 m	2.44 m	18.37 m ²	3.73 m ³
	0. Story	Exterior walls	Wall-017		0.20 m	2.44 m	11.34 m ²	2.30 m ³
	0. Story	Exterior walls	Wall-018		0.20 m	2.44 m	12.21 m ²	2.48 m ³
	0. Story	Exterior walls	Wall-019		0.20 m	2.44 m	1.94 m ²	0.33 m ³
	0. Story	Exterior walls	Wall-020		0.20 m	2.44 m	10.28 m ²	1.99 m ³
	0. Story	Exterior walls	Wall-021		0.20 m	2.44 m	3.37 m ²	0.58 m ³
	0. Story	Exterior walls	Wall-022		0.20 m	2.44 m	8.86 m ²	1.73 m ³
WALL	0. Story total						111.91 m²	22.40 m³
WALL	total for all stories						111.91 m²	22.40 m³
COLUMN	0. Story	Columns	COL-001		0.41 m	2.44 m	2.97 m ²	0.23 m ³

Component Lists are generated when bills of materials, quantity takeoffs or price lists are required. These reports typically sum up and display component type properties; however, certain element parameters can also be listed.

	**** TEXTFIELD 1 ****	*****
	**** TEXTFIELD 2 ****	*****
	**** TEXTFIELD 3 ****	*****
	**** TEXTFIELD 4 ****	*****

Component List by keys 2/7/01							
Key Code/Name	Code	Name	Database Qty.	Unit	Ref. Qty.	Ref. Unit	Quantity
004 Masonry							
	100	common brick 24*11.5*7.1 cm	383.200	pieces	3.386	m ²	1,297.510
	100	common brick 24*11.5*7.1 cm	383.200	pieces	5.045	m ²	1,933.271
	100	common brick 24*11.5*7.1 cm	383.200	pieces	1.992	m ²	763.507
	100	common brick 24*11.5*7.1 cm	383.200	pieces	3.732	m ²	1,430.132
Total	common brick 24*11.5*7.1 cm						5,425.00 pieces
	101	mortar	0.169	m3	1.992	m ²	0.337
	101	mortar	0.169	m3	3.386	m ²	0.572
	101	mortar	0.169	m3	5.045	m ²	0.853
	101	mortar	0.169	m3	3.732	m ²	0.631
Total	mortar						3.00 m3
	102	plaster	0.030	m3	19.611	m ²	0.588
	102	plaster	0.030	m3	33.327	m ²	1.000
	102	plaster	0.030	m3	36.733	m ²	1.102
	102	plaster	0.030	m3	49.656	m ²	1.490
Page 1							

Zone Lists are generally used to create room schedules and finish schedules. Zone Lists can include parameters of Zones and related construction elements. When related construction items are listed, the report is in fact an Element List where the range of calculated elements is limited by the Zones they belong to. Depending on what you want to appear listed in your list, you may choose the type of List you need to use. Some common examples are:

You want construction elements listed with basic data and parameters about them listed	Element List
You want parameters specific to construction elements (such as: Door Sill Height, Number of holes in Slab) listed	Element List
You want to create Door/Window schedules	Element List
You want quantities of components associated with construction elements listed	Component List
You want descriptors associated with construction elements listed	Component List
You want to create room schedules	Zone List
You want Zones or detailed Zone parameters (such as: Zone Extracted Area, Zone Area Reduction) to be listed	Zone List
You want main Zone parameters (such as: Zone Name, Zone Number) listed for Zones where listed construction element is located	Element List

Displaying Lists

The range of elements that will be used in calculation depends on the current selection made on the Floor Plan.

- If there is no selection made, all elements will be used and filtered by the listing command's List Scheme by type or other criteria;

- If there is a selection made, only selected elements will be used.

For more specific element ranges, use the Setup List Schemes command and define filters.

Design Options

Design Options feature helps you manage constantly changing designs and switch among alternatives.

Create design variations and easily switch among them – whether they include the whole building, just parts of it, or multiple parts at the same.

- Choose the best and most efficient design alternatives, in 3D or any 2D view – without having the different options intersecting each other
- Study designs in the context of their surroundings and evaluate them for aesthetics, performance, efficiency
- Design Options are a view-based setting: document your options via any 2D or 3D output
- Finalize Options by integrating a selected Option into the main model, or adapt Options along the lifecycle of a Project by merging, duplicating, or rearranging Options, or moving Options between Option sets.

Filter and extract BIM data according to different Options: create Option-based Graphic Overrides, and extract data via Schedules and Labels

Options are widely integrated into all 2D and 3D elements, including Annotation and Markers such as Sections, Elevations, Details and Worksheets.

[Suggested workflow for Design Options](#)

[Design Options interface](#)

[Create and manage Design Options and Sets](#)

[Set a default Option](#)

[Activate an Option](#)

[Focused editing](#)

[Relink elements to a different Option](#)

[Working with Design Option Combinations](#)

[Design Options in Teamwork](#)

[Hotlinks and Design Options](#)

[Comparing Design Options](#)

[Element properties for Design Options](#)

[Merge Design Option to Main Model](#)

Suggested workflow for Design Options

1. Do the **Main Model** first (all the parts that are already decided)
2. For the parts that are still undecided: create all the possible **Options** (for example: one “Suite bathroom” option and one “Family bathroom” option).
3. While creating an Option:
 - Set that Option as the **default**: this way, your new elements automatically go into that Option. (Or else you can place elements into the Main Model, then later select and relink them to a Design Option.)
 - Use the “**focused editing**” on-screen controls to make sure you are editing the right elements.
4. Group your options in logical Design **Option Sets** (for example, “Bathroom layouts”)
5. If needed, use **Hotlinks** to bring in Options to the model.
6. Create Design Option **Combinations**. In each Combination, you can **activate** (display) one option from each set. Mix and match.
7. Assign the Design Option Combinations to **views**, so you can see different options in different views.
8. **Explore**, compare, and publish the Design Options for review by stakeholders.
9. Merge the chosen, final Options into the Main Model.

What's what in Design Options	
Term	Definition
Option Set	A logical group of Options - for example, “Furniture layouts” or “Curtain Wall facades”
Option	A Design Option - one of several possibilities. Each Option belongs to a logical Option Set. For example, the Furniture layouts Option Set contains an “Open plan” Option and a “Closed plan” Option Elements in a Design Option do not intersect with elements in other options, or with the Main Model.
Main Model	Elements in the Main Model are already decided, so they don't belong to any option. The Main Model is your entire building model, except the elements that belong to an Option.
Active Option	From each Option Set, at most one Option is active (visible) at a time.
Inactive Option	Elements that belong to a currently inactive Option are not visible in the model until you activate the Option do not intersect with visible elements are listed in Element Schedules, but cannot be selected
Default Option	As you create new elements, they are automatically added to the Option defined as the default. (The Main Model can also be the default.) You can switch the default Option at any time.
Option Status	This is an element property. It tells which Design Option (or Main Model) the element belongs to.

Option Combination	Activate up to one Option from each Option Set, and make them into a Combination. (For example, "Budget Combination" vs. "Luxury Combination"). You can then assign Option Combinations to views.
Accept & Merge Option	Use this command to accept one Option from an Option Set as the final one, and merge it into the Main Model. When you do Accept & Merge, the rest of the Options from that Option Set, and the Set itself, will all be deleted.
Design Option Manager	Use this dialog to create and manage all Options, Sets, and Combinations. You can also activate Options and save them into combinations.

Related topics[Design Options interface](#)[Create and manage Design Options and Sets](#)[Set a default Option](#)[Activate an Option](#)[Focused editing](#)[Relink elements to a different Option](#)[Working with Design Option Combinations](#)[Hotlinks and Design Options](#)[Comparing Design Options](#)[Element properties for Design Options](#)[Merge Design Option to Main Model](#)

Design Options interface

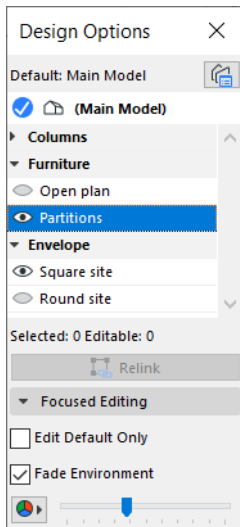
Open Design Options palette

Open Design Option Manager

Open Design Options palette

Use one of the following:

- Document > Design Options > Design Options
- Window > Palettes > Design Options



Use the palette while you work with Options in the model: to activate or set a default Option, and to edit a particular Option. You can also relink elements from one Option to another.

Related topics

[Set a default Option](#)

[Activate an Option](#)

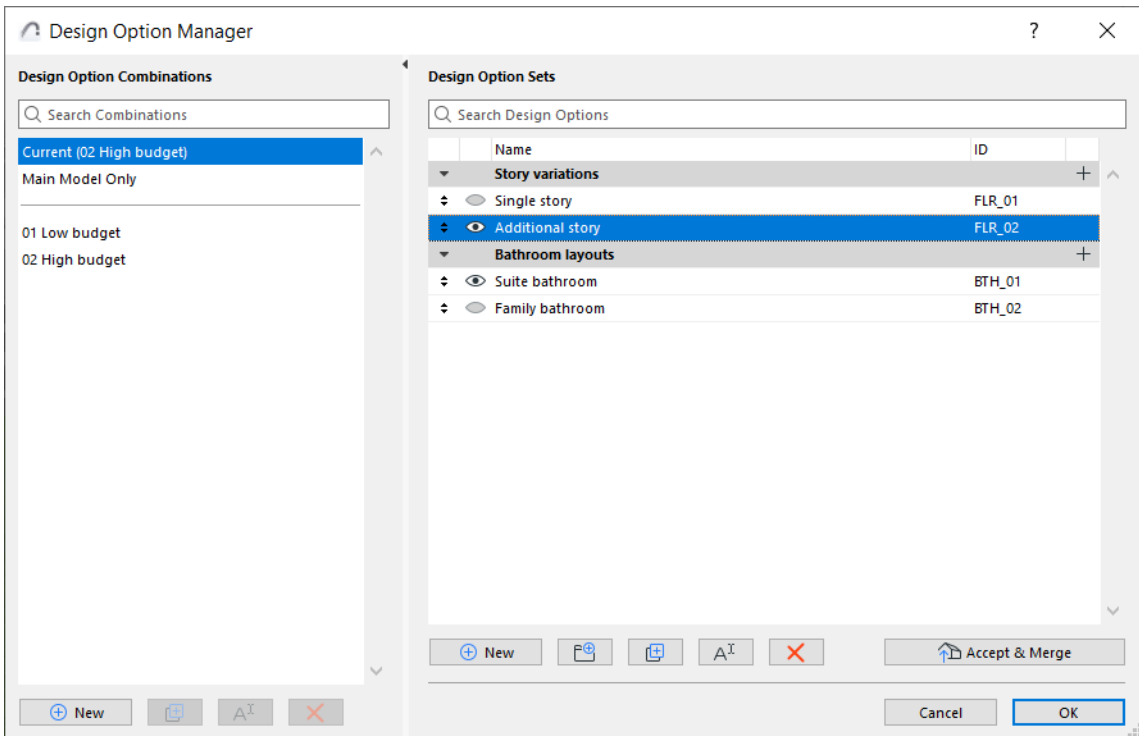
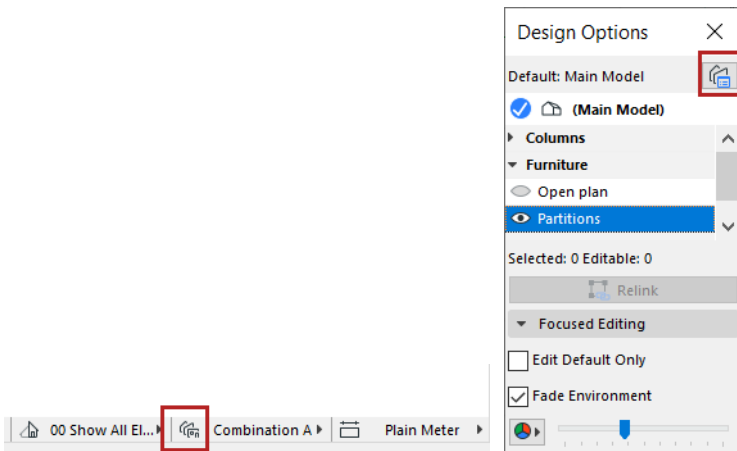
[Focused editing](#)

[Relink elements to a different Option](#)

Open Design Option Manager

Use one of the following:

- **Document > Design Options > Design Option Manager**
- Click the Design Option Manager button from
 - Quick Options Bar at the bottom of the window
 - Design Options palette



The left side lists Combinations, and the right side lists Design Option Sets and their Options.

Use this dialog to create and manage Options, Sets, and Combinations.

You can also activate Options and save them into Combinations.

Related topics

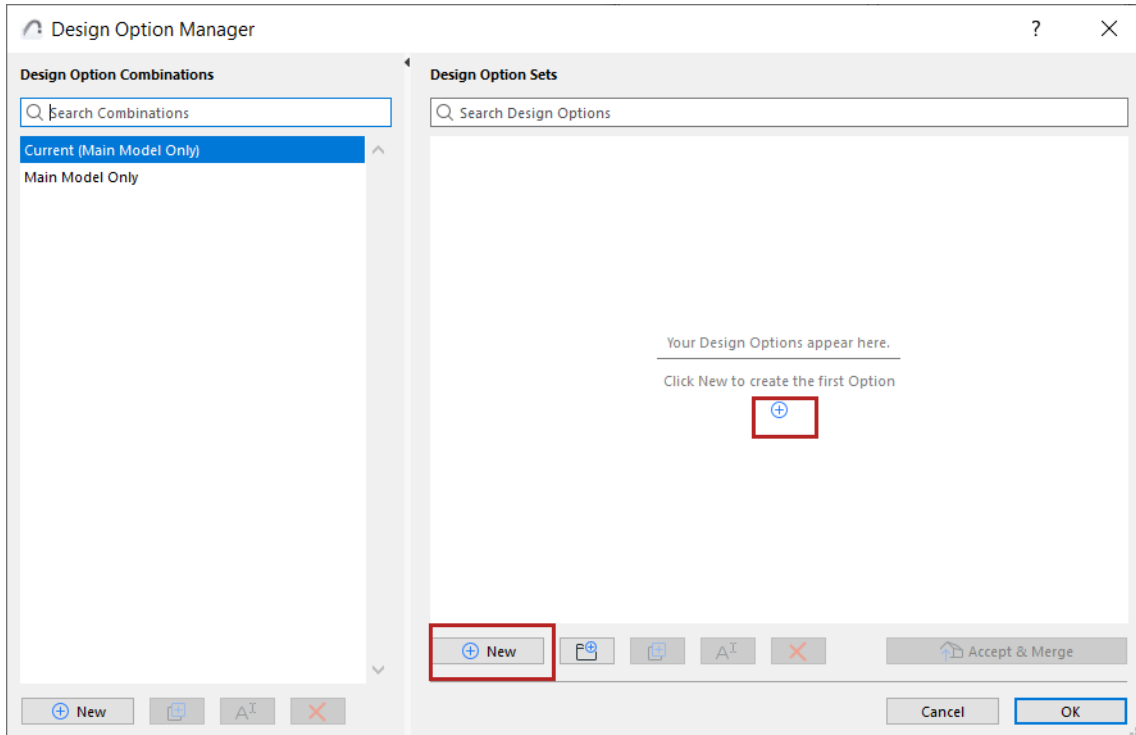
[Create and manage Design Options and Sets](#)

[Working with Design Option Combinations](#)

Create and manage Design Options and Sets

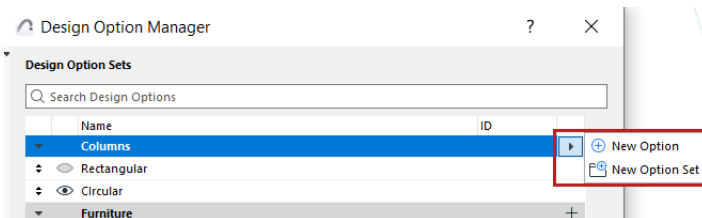
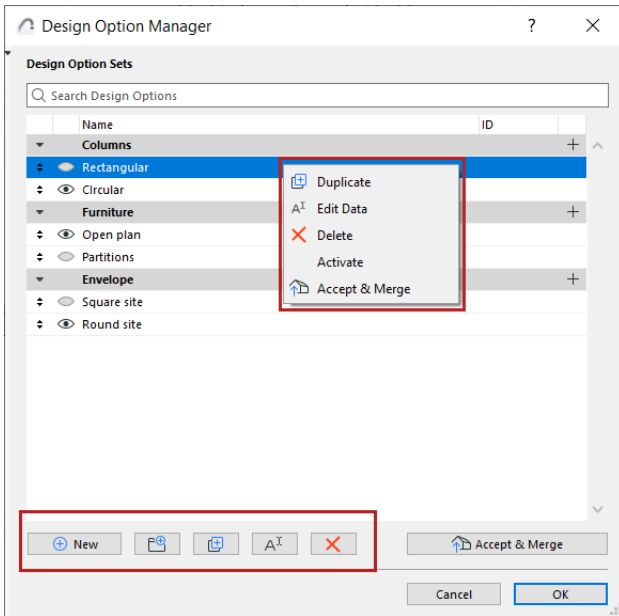
Open the Design Option Manager.

Click the **New** button to create your first Design Option.



Add, rename, duplicate, or delete Options

Use the command buttons as well as context menus in the Manager:



Drag and drop Options among Sets

In Design Options Manager: Drag and drop one or more Options among different Sets, or change their order.

Related topics

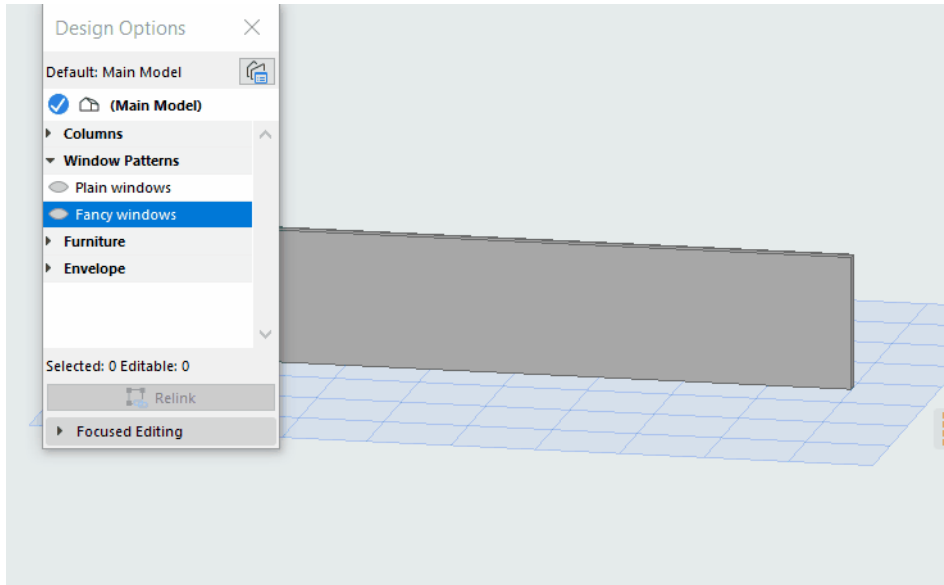
- As you create new elements, they are automatically added to the “default” Design Option. See [Set a default Option](#).
- Don’t forget that you can add 2D elements to your Design Option. See [Add 2D elements to Design Options](#)
- To work with Doors, Windows, and Skylights: see [Doors, Windows, and Skylights in Design Options](#).

Doors, Windows, and Skylights in Design Options

If a Wall is part of a Design Option, its doors and window automatically belong to that same Option. For example, you can't put a Wall into one Option and its windows into a different Option. (Similarly: In Teamwork, when reserving Walls, you also reserve their Windows.)

How to create variable Door/Window options

However, as long as the host Wall is part of the Main Model (instead of belonging to an Option), you can create several Options with different window patterns.



Skylights in Roofs work the same way as Doors and Windows in Walls.

Add 2D elements to Design Options

2D elements, such as Section markers and Lines, can also be added to Design Options - so you can include specific documentation as part of the Option.

This helps you ensure that 2D elements appear in the context that is relevant, and are not visible in other options on which they are not relevant.

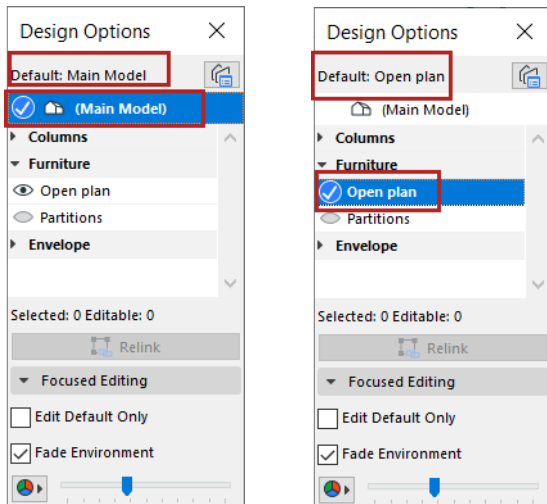
Note: Hotspots and Figure elements can't be assigned to a Design Option.

Set a default Option

As you create new elements, they are automatically added to the “default” Design Option. (The Main Model can also be the default.)

To set the default Option:

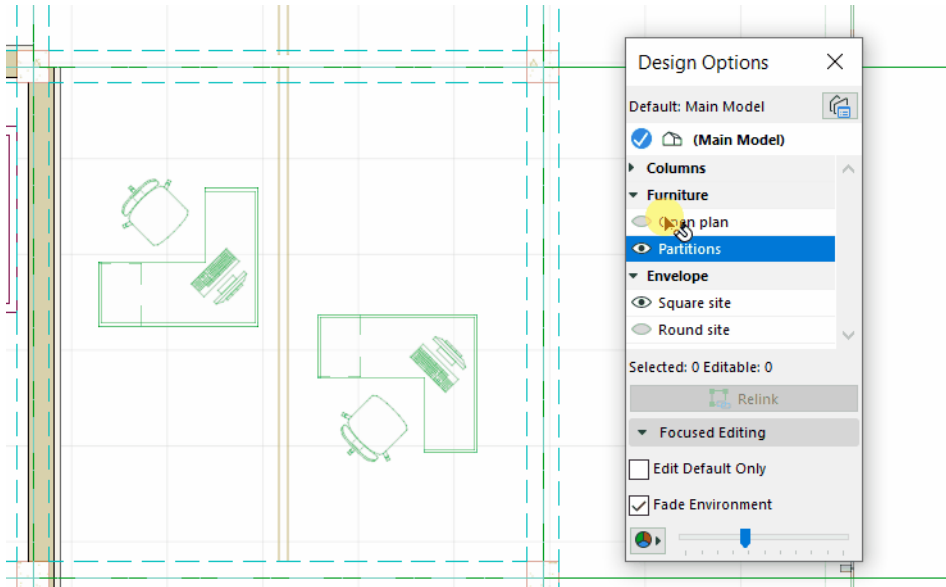
1. Open the Design Options palette.
2. Double-click an Option to make it the default. It now has a blue checkmark. (It is also activated, if it wasn't active before.)



Activate an Option

In each Option Set, one Option can be active (visible) at a time.

- Click an Option to activate it (eye icon), in the Design Options palette or the Manager
- Or use the Activate command from the context menu, in the Manager

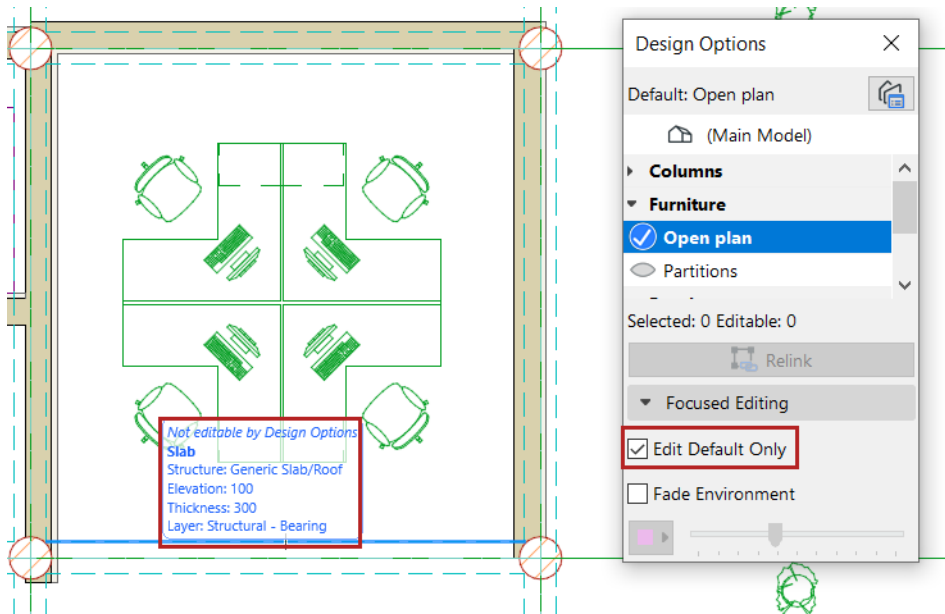


Focused editing

These functions, on the Design Options palette, help you edit exactly the right thing while you are creating your Options.

Edit Default Only

Check this to make sure you only modify elements that are part of the default Option.



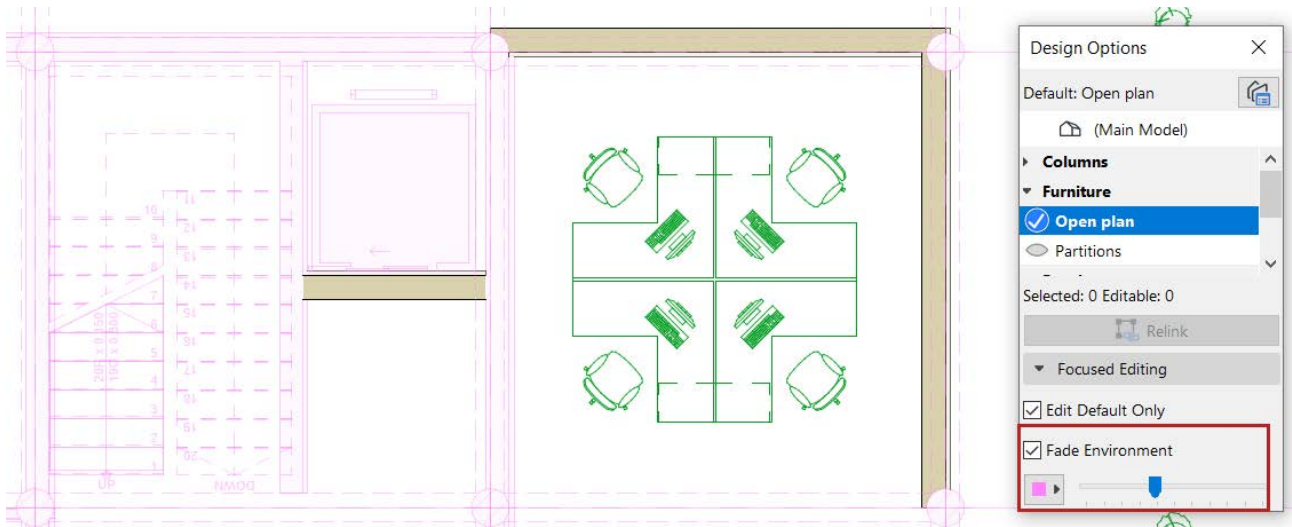
For example, if you are working on your “Open plan” furniture Option, you set that Option as the default, so all new elements automatically belong to the Open Plan.

If you also check Edit Default Only, then you won’t accidentally modify other parts of the model.

Fade Environment

Fades all model elements that don’t belong to the current default Option. This helps you focus on the Option you are working on.

- Use the slider to set the degree of fading
- Optionally, choose a different color to display the environment (all the non-default elements)



Note: The color is a screen-only highlight that doesn't affect layouts or final exported documents. To display color highlights on layouts, use Graphic Overrides.

Relink elements to a different Option

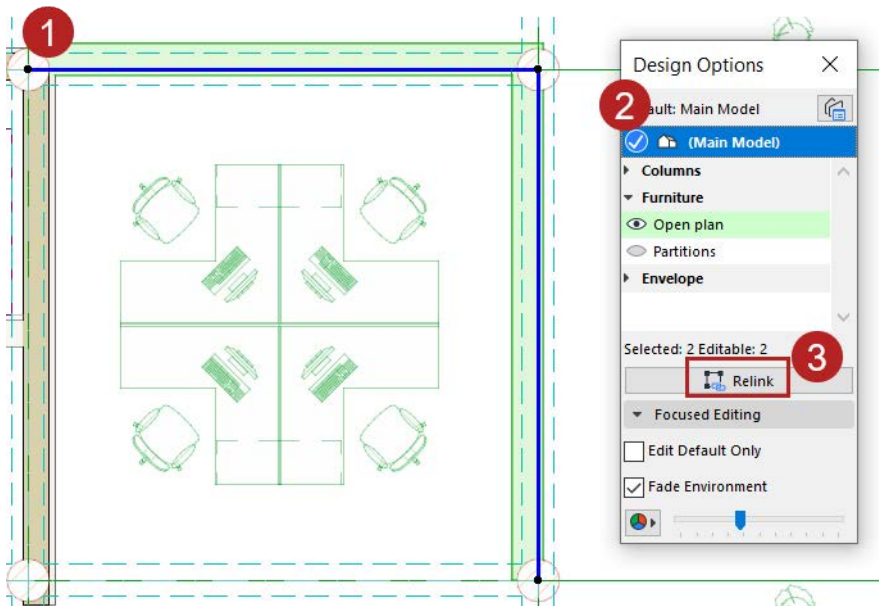
Suppose you have added some Walls to an Option, and you decide that they should be part of the Main Model instead. You can relink these elements to the Main Model.

You can do this using either the Design Options palette or element settings.

Note: You can only relink elements that are editable. Non-editable (locked) elements won't be relinked. Check the palette or dialog feedback to make sure your selected elements are editable.

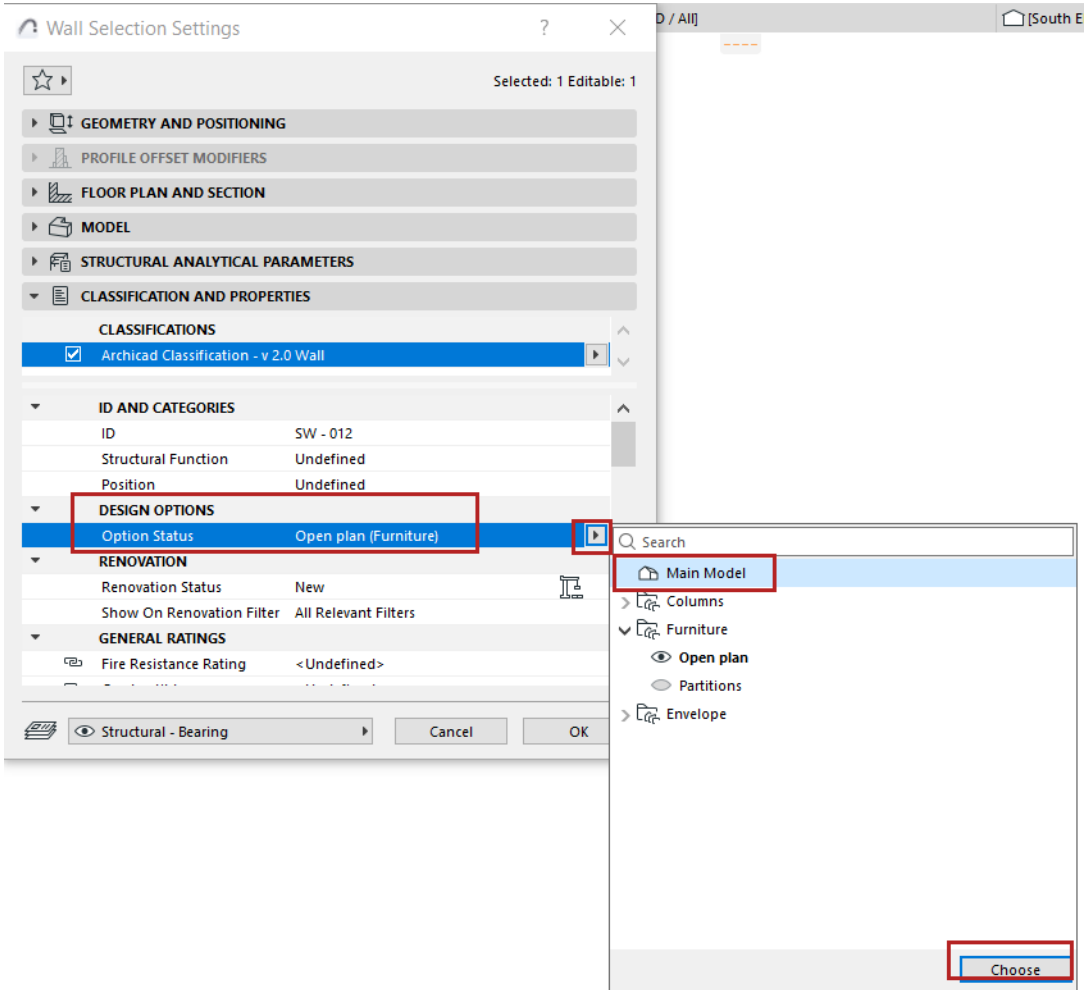
Using Design Options palette

1. Select the elements to relink.
2. In the Design Options palette, click on "Main Model", which is where you want to link the elements to. The green highlight shows that the selected elements currently belong to the "Open" Option.
3. Click **Relink**. The elements now belong to the Main Model.



Using element settings

1. Select the elements to relink.
2. Open the element Selection Settings dialog.
3. In the Classification and Properties panel (at Design Options - Option Status), choose the desired Option, or Main Model.
4. Click **Choose** to relink the selected elements.

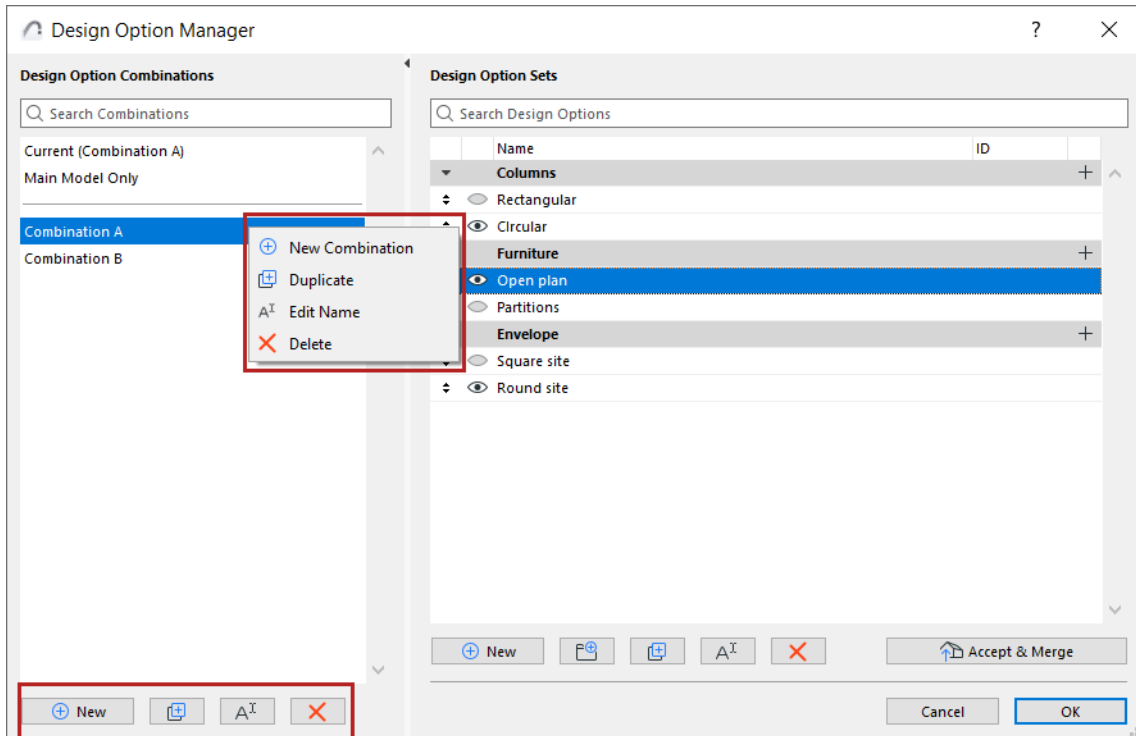


Working with Design Option Combinations

A Combination saves a set of active Options (maximum one active Option from each Option Set). Create and manage Combinations in the Design Option Manager.

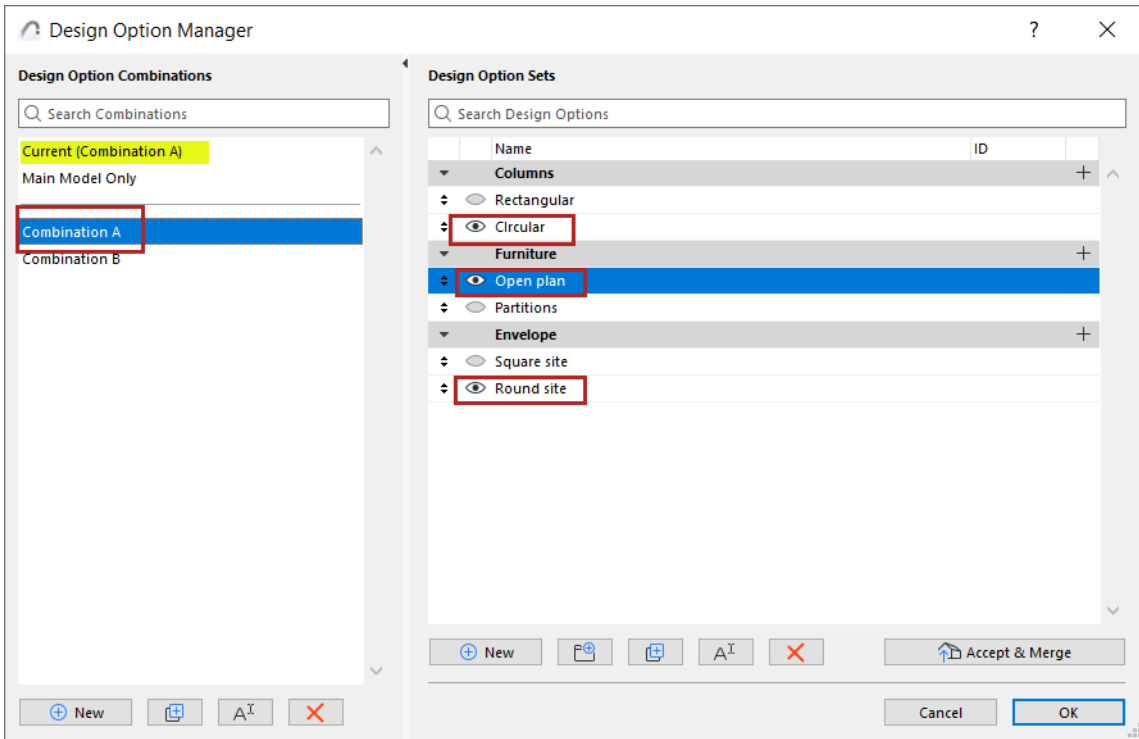
Add, rename, duplicate, or delete Combinations

Use the command buttons as well as context menus in the Manager:

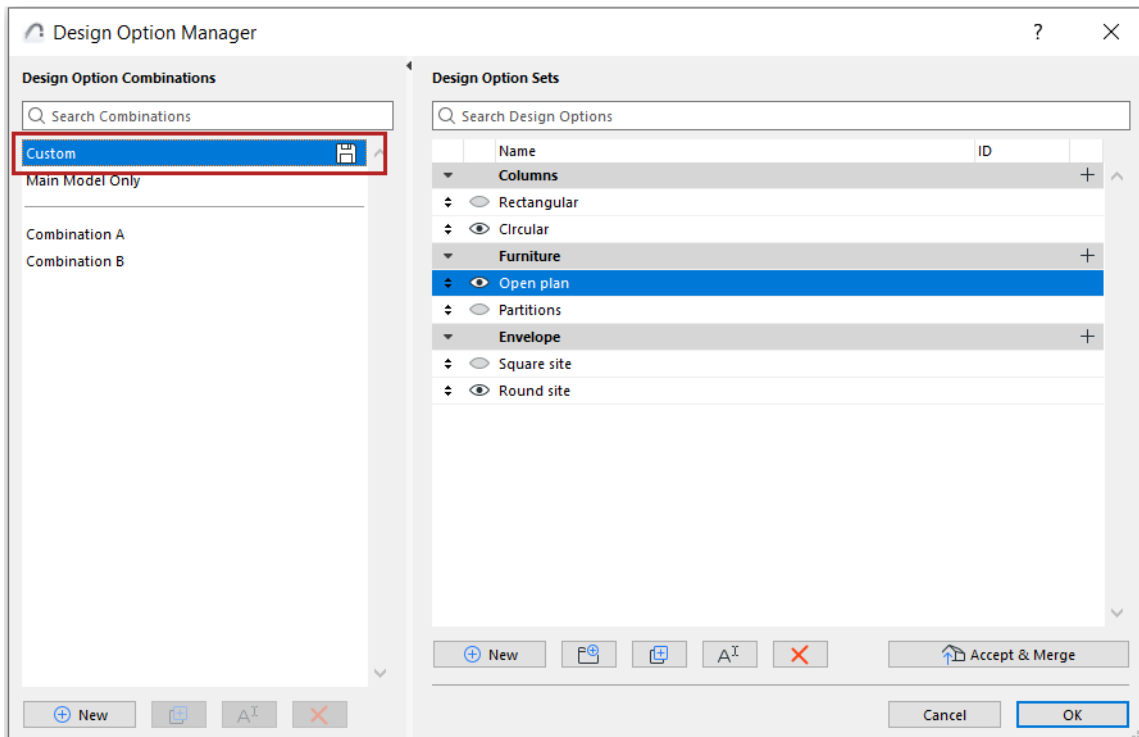


Current and Custom

When you open the Manager, the "Current" Combination is selected - the model is displayed according to this current Combination.



Whenever you change which Options are active in the “Current”, the name of the edited Combination changes to “Custom”. You can store it under a new name if needed.



Related topics

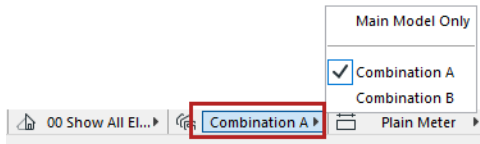
[Switch among Design Option Combinations](#)

[Save Design Option Combination in a view](#)

Switch among Design Option Combinations

Do one of the following:

- Use the Design Option Combinations pop-up from the Quick Options Bar at the bottom of the window

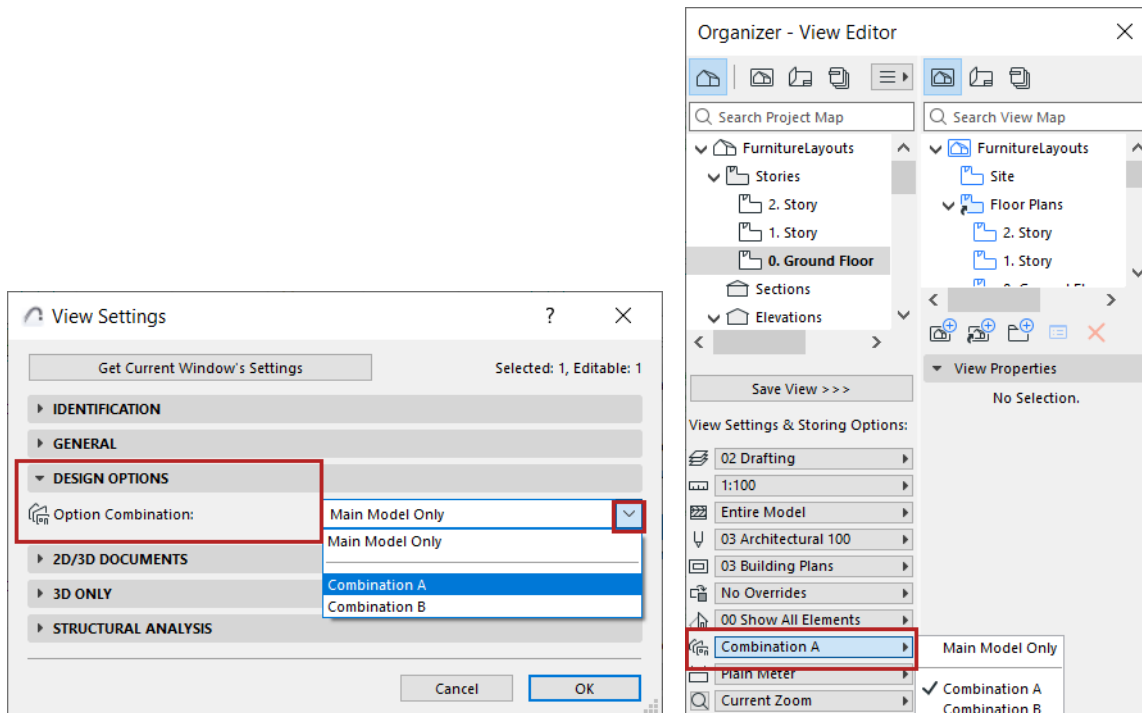


- In Design Option Manager, select a Combo and click OK to close the dialog
- Use the Document > Design Options menu

Save Design Option Combination in a view

Do one of the following:

- Use the View Settings dialog box (choose a Combo from the Design Options panel)
- In the Organizer View Map: use the Combo selector from View Settings and Storing Options



Design Options in Teamwork

Teamwork permissions for Design Options:

- Design Options - Create
- Design Options - Delete/Modify

Missing Design Option Status

It's possible for an element's Design Option to be "missing" in a Teamwork situation.

If you create some elements in a Design Option (without sending them to the server), and another user meanwhile deletes the Design Option, your elements' Design Option status will be "Missing."

You can use the "Design Option Missing" property to locate such elements (using Find & Select), or list them in a Schedule and relink them to a different Design Option.

See also [Element properties for Design Options](#).

Hotlinks and Design Options

Use the regular Hotlink workflow to place a module. [See Place Hotlink](#).

When you are placing the module, you can choose specific Design Options to place from the source file.

You can also place the same module several times, using a different Design Option in each.

Elements of the placed Options must be assigned to an existing Option in the host file. But as with any hotlink, modules based on Options can be updated to follow changes in the source file. If Options are modified (e.g. deleted, renamed) in the source file, these changes will also take place in the host file after an update.

Choose which Options to place (from source file)

Choose an Option for the placed Hotlink (in host file)

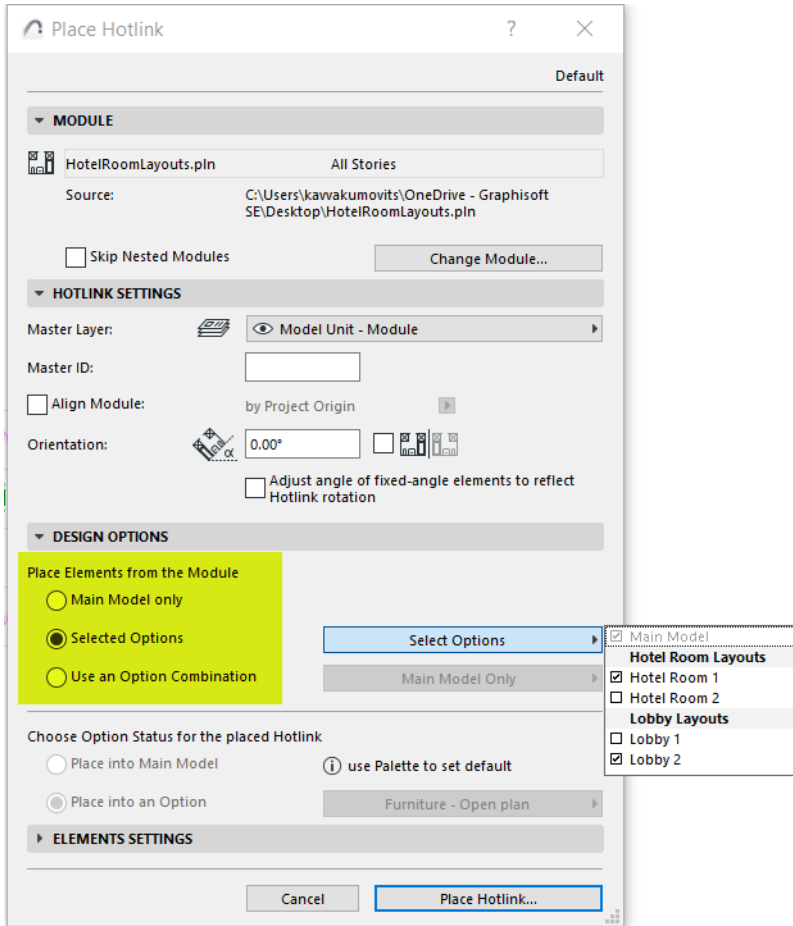
Place different Options from the same module

Which Design Options are part of the Hotlink source file?

Choose which Options to place (from source file)

When you place a hotlink, choose which Options or Combinations from the source file should be placed. Only elements that are linked to these Options will be placed into the host file.

If you are selecting particular Options, you can logically place only one Option per Option Set.



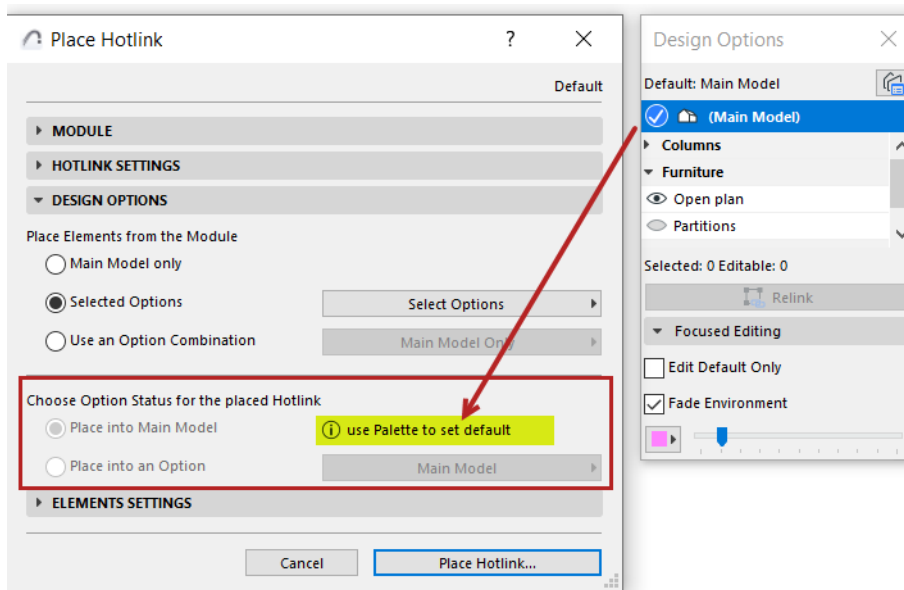
- If the source file does not contain any Options or Combinations, the relevant controls are disabled.
- If the source file is .mod format, not all Options in the original model may be available here - depending on how you saved the .mod file: all elements, or just visible elements. (See [Module Save Options](#))

Choose an Option for the placed Hotlink (in host file)

All elements of the placed Hotlink are assigned to an existing Design Option (or to the Main Model) in the host project.

Default host file Option for placed Hotlink

At placement, you cannot choose the placed hotlink's Design Option: all the hotlink elements are automatically assigned to the Design Option (or Main Model) that is currently the **default** in the host file. Check the host file's Design Options palette to see which is the current default.

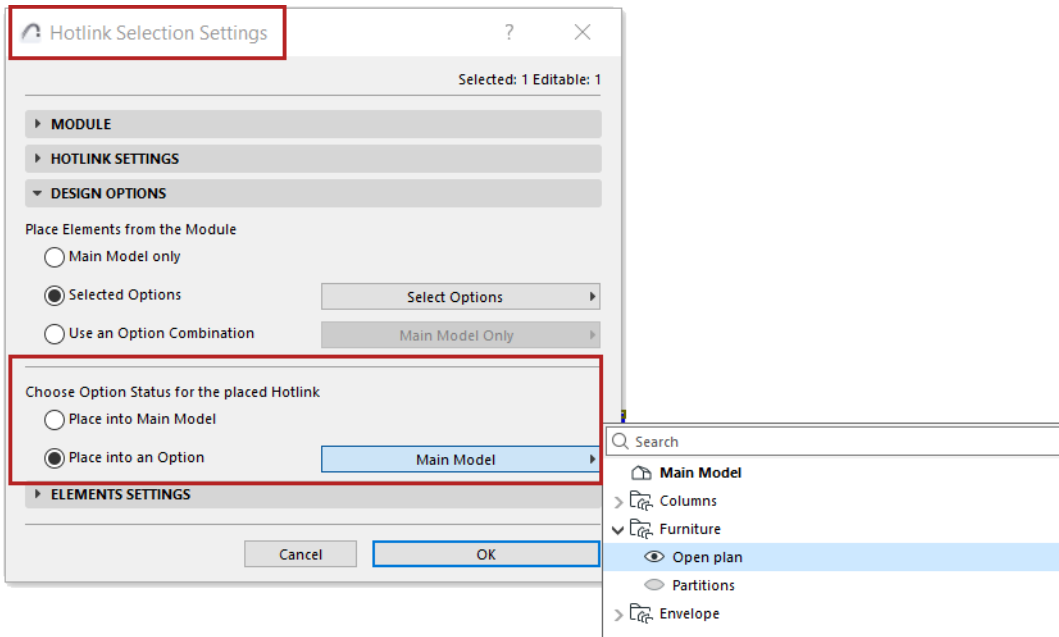


Change host file Option for placed Hotlink

After placement, you can reassign the Hotlink module to a different Option (or Main Model).

1. Select the Hotlink.
2. In Hotlink Selection Settings, choose a different Option or Main Model:

Note: If the new Option is not currently active in the model, you won't see any of its elements until you activate the Option.



Place different Options from the same module

Once the hotlink is placed in the host file, you still have access to all the options in the source module. You can change the module's Option content at any time.

For example:

1. Place a module in the host file, using Combination A from the source file.
2. In the host file, make a copy of the placed module.
3. Select the second copy of the module.
4. In Hotlink Selection Settings, change the module's Option Combination.

This way, you have two instances of the same module side by side, each displaying a different Option Combination.

Which Design Options are part of the Hotlink source file?

If the hotlink source is a .pln file

All Design Options and Option Combinations, plus the Main Model, are part of the source file. When you place the Hotlink, you choose which Options or Combinations to place. See [Choose which Options to place \(from source file\)](#).

If the hotlink source is a .mod file

When you save or publish a model or view in module (.mod) format, use the **Save Options** dialog (“Filter Elements” controls) to filter which Design Options should be part of the file.

[See Module Save Options.](#)

If the hotlink source is a .mod file saved from selected elements

If you used “Save Selection as Module” to create a .mod file:

- Only the selected elements are saved into the module, with their Option Status (the Option they belong to)
- Exception: Associative elements (Door, Window, Skylight, Wall End) will always be saved into the module with their host Wall or Roof, regardless of their Option Status.

Comparing Design Options

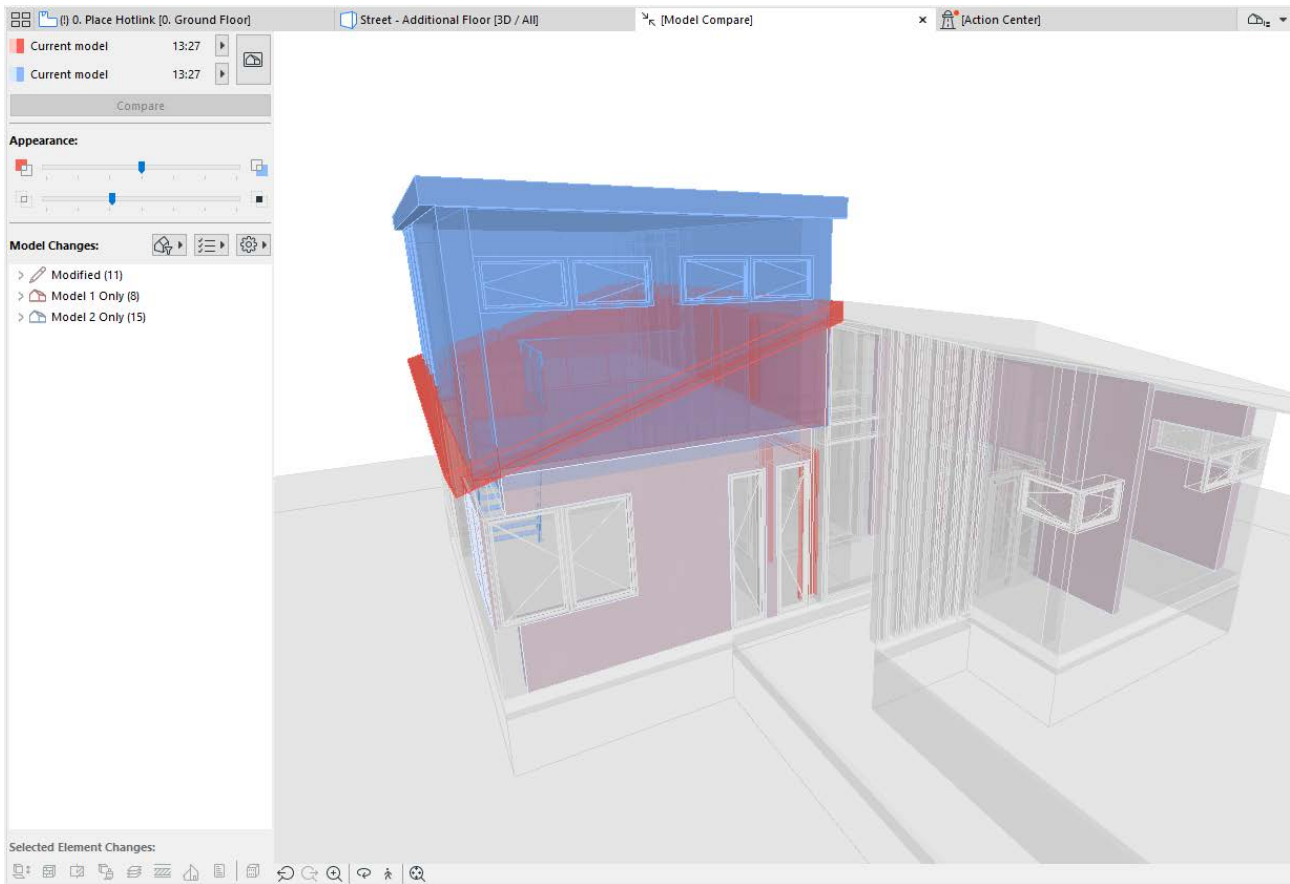
Option Combinations are a view-based setting. Each view has an associated Design Option Combination. (See [Save Design Option Combination in a view.](#))

To compare different Design Options, use the Archicad functions that let you compare views. For example:

- In the model itself: Use the Quick Options bar to switch among saved Option Combinations



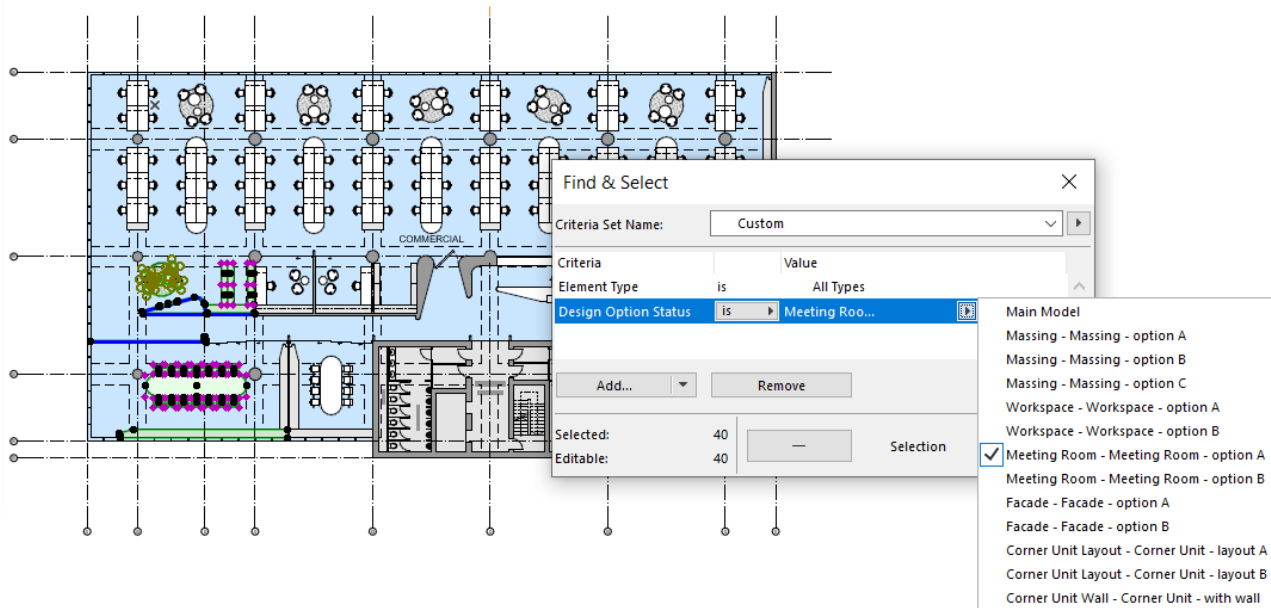
- Place saved views having different Option Combinations side by side on Layouts
- Compare two views using Trace and Reference
- Use Model Compare to see the difference between any two 3D views



Element properties for Design Options

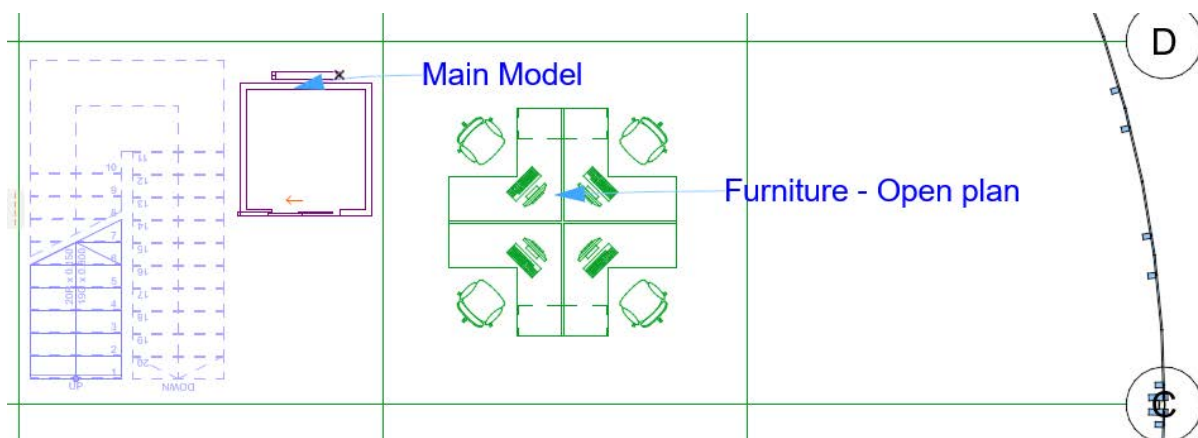
Use the following properties to find and manage element data by Design Option:

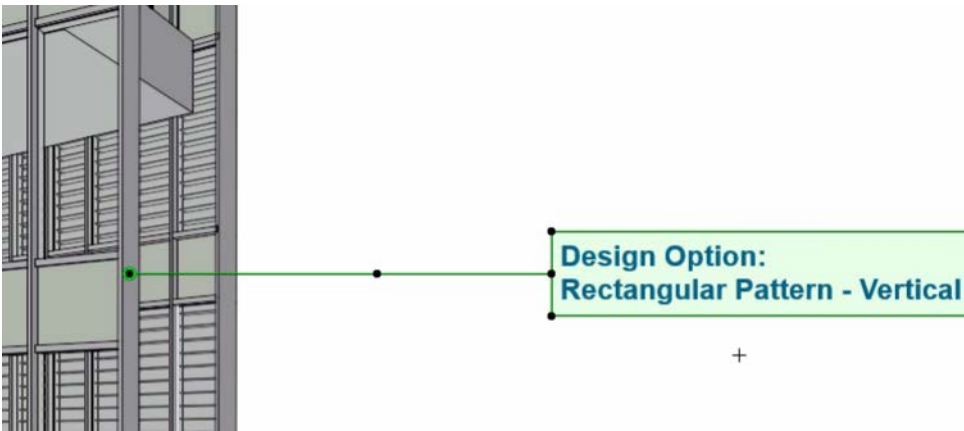
- Design Option Status
- Design Option Set
- Design Option Missing (see [Design Options in Teamwork](#))



The following additional parameters are available in Labels and Schedules:

- Design Option ID
- Design Option Name
- Design Option Set Name





Design Options in Element Schedules

Element Schedules can list elements by Design Option. For example, it's easy to create Option-based quantities lists (such as surface areas or number of bricks.)

To compare the results, create a schedule for each Design Option, then save each schedule as a view.

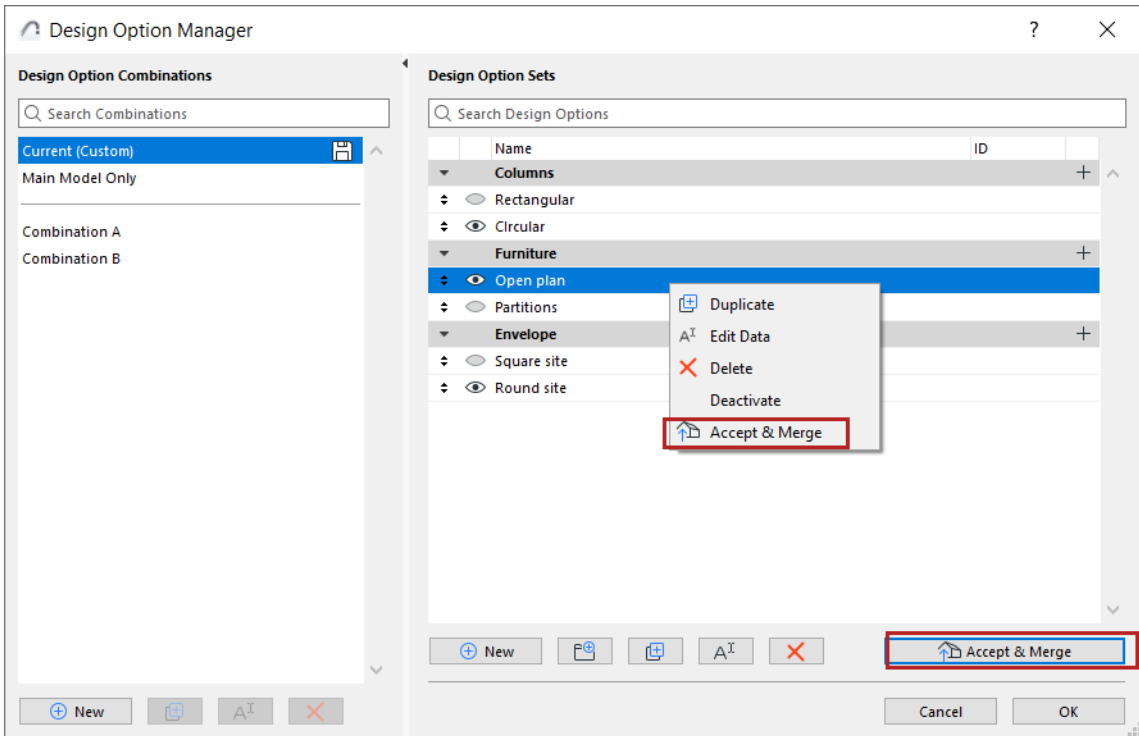
Curtain Wall Surface Areas by Design Option		
Design Option Name	Home Story	Surface Area
Rectangular Pattern - Vertical		
	L2	4,596.6
	L8	2,334.0
	L12	1,228.1
	L15	1,088.0
	L19	106.7
		9,353.4 m²

Merge Design Option to Main Model

When a final Design Option is chosen and good to go, select the Option in Design Option Manager, and click Accept & Merge. (Use the command button or the context menu.)

You can only select and merge one Option at a time.

- Elements of the selected Option are now merged into the Main Model.
- The entire Option Set is now deleted, together with all its Options.



Collaboration

The ability for several individuals and teams to collaborate effectively on a single project is a basic necessity of architectural work. Teamwork for Archicad is optimized to meet these needs, and specifically designed to meet the demands of a modern, multinational team.

Architects often need to work together with each other and with professionals of other disciplines who use other software. Moreover, interactive communication with clients is also a growing concern. Archicad's sophisticated collaboration tools are designed to address these challenges.

Related Topics:

[Teamwork](#)

[Hotlink Modules](#)

[Model Compare](#)

[Collision Detection](#)

[Archicad Design Checker](#)

[Issue Handling](#)

[Working with XREFs](#)

[Interoperability](#)

[Structural Analytical Model](#)

[MEP Modeler](#)

Teamwork

Teamwork is based on a client-server architecture and is designed to ensure maximum flexibility, speed and data safety to enable teams - even those spread out around the world - to collaborate on large projects.

The following sections describe Archicad's Teamwork related features and functions.

Topics in this section:

[Teamwork Functions at a Glance](#)

[Graphisoft BIMcloud: Features and Benefits](#)

[Teamwork Setup/Preparation](#)

[Share a Teamwork Project in Archicad](#)

[Network Diagnostic Information \(BIMcloud\)](#)

[Join or Leave a Teamwork Project](#)

[Open or Join Teamwork Project](#)

[Close Teamwork Project](#)

[Save Local Copy of Teamwork Project](#)

[Send/Receive Changes in Teamwork](#)

[Manage Local Teamwork Data](#)

[Working Online/Offline in Teamwork](#)

[Remote Access to Teamwork Projects](#)

[Teamwork Interface](#)

[Reserving Elements in Teamwork](#)

[Reserving Teamwork Project Data \(Non-Elements\)](#)

[Reserve All Elements and Data \(Teamwork\)](#)

[Review Your Workspace \(Teamwork\)](#)

[Creating and Sending Project Data \(Teamwork\)](#)

[Change Tracking in Teamwork](#)

[Assign Elements \(Teamwork\)](#)

[Request Elements or Project Data \(Teamwork\)](#)

[Release Elements or Project Data \(Teamwork\)](#)

[Messaging \(Teamwork\)](#)

[Add Comment to Activities \(Teamwork\)](#)

[Libraries in Teamwork](#)

[Hotlink Modules and XREFs in Teamwork](#)

[Create Support Package \(Troubleshooting\)](#)

Teamwork Functions at a Glance

Function	Teamwork in Archicad
Reserving Elements	Reserve any existing elements individually, by selection or criteria No reservation needed to create new element/data Assign elements to any user Whenever possible, reservation occurs automatically without user action (not available in BIMcloud Basic)
Reserving Other Project Data	Reserve as needed, at any time, by any authorized user Whenever possible, reservation occurs automatically without user action (not available in BIMcloud Basic)
Releasing Elements	Released by the user at any time, as needed
Feedback on Reservation Status	Always up to date (if user is online) Color-coding for at-a-glance feedback
Ownership Conflicts	Use "Request" function to signal need for data
Send & Receive	Download/upload involves only data that have been modified since last Send or Receive.
Team Communications	Use simple built-in messaging system
Project Management	Central registry of users defined for all projects on BIMcloud Each user's role can vary by project
Working Via Internet	Users access the BIMcloud from any remote computer via Internet
Working Offline	Local copy created automatically. Local copy can be saved as "Travel pack" for transfer among several computers.

Graphisoft BIMcloud: Features and Benefits

Graphisoft BIMcloud responds to an industry need - allowing remote teams to work together and handle large files. It also allows working offline, and enables working from just about any location, so that team members can log in and work anywhere. Any project data that can be imported into Archicad can be communicated via the Graphisoft BIMcloud.

- When implementing BIM on a larger scale, architects often run into bottlenecks in model accessibility and workflow management. Graphisoft's BIMcloud is the industry's pioneering solution for model-based team collaboration for companies who use Teamwork in Archicad.
- The BIMcloud's **Delta Server™ technology** reduces network traffic to a minimum for instant and reliable data-exchange, both within the office and over the Internet. In the BIMcloud environment, long waiting times for server synchronization are a thing of the past: team members can collaborate in real-time on BIM models through standard Internet connections from virtually anywhere in the world.
- The BIMcloud's **robust system architecture** ensures the integrity of the BIM project at all times. Any data errors from malfunctioning components are blocked, preserving the project's integrity on the network.
- The BIMcloud was **designed for Architects**, not for IT professionals - so it works in any standard office setup through any standard computer network.
- BIMcloud supports both **online and offline work**: if Archicad loses the connection to the BIMcloud, users can continue to work with their already reserved elements. Once the connection is re-established, they will be able to send their changes to the BIMcloud and Receive changes submitted by other users.
- Built-in **Teamwork Messaging** boosts the communication between team members by putting the messages into context: simply select any element(s), write a comment and send it to a team member user for a review. A pop-up notification will inform the other user about the incoming message. After opening the message, he will be able to zoom to the element and read the message in context.
- **BIMcloud Roles** allow the browser-based BIMcloud Manager to define different permission sets for the BIMcloud users and assign them based on their responsibilities. It is very easy to limit the functions available for a specific user to prevent accidental damage or unwanted modifications to the project. Each user can have different Roles for different projects - to fit their responsibilities in that particular project. Even a view-only role can be created, allowing people to Join the project without being able to modify anything.

For more information on features, see the [BIMcloud User Guide](#).

Teamwork Setup/Preparation

Teamwork for Archicad is based on a client-server technology, in which one or more server applications are connected through the network to client applications. Therefore, using Teamwork in Archicad requires

1. the Archicad client application, and
2. the Graphisoft BIMcloud.

Each of these two components must be installed separately. The BIMcloud should be installed on a computer dedicated to server functionality.

[See the *BIMcloud User Guide* for details.](#)

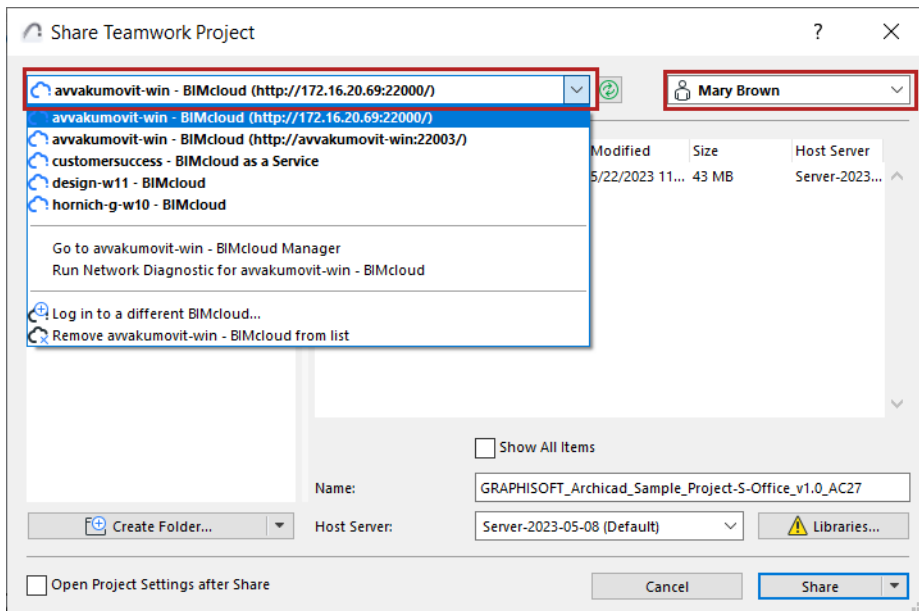
After you install and configure Archicad and the BIMcloud, you are ready to start using Teamwork within the office.

Share a Teamwork Project in Archicad

To create a new Teamwork project, open a solo Archicad project file (.pln), then use the **Share** command to add it to the BIMcloud. All users who have access rights can then work on this shared project in Teamwork.

Follow these steps to share a Teamwork Project:

1. Open a solo Archicad Project file (.pln).
2. Use one of these commands to open the **Share Project** dialog:
 - Teamwork > Project > Share
 - File > New > Share



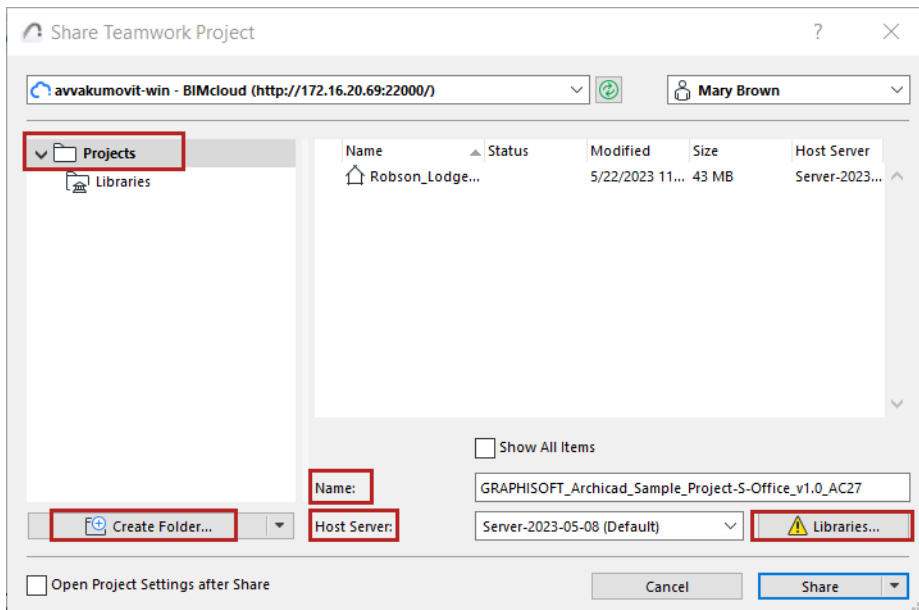
3. Choose a BIMcloud server from the list, which shows the addresses of all BIMclouds to which you successfully logged in at least once.
 - If you don't see the BIMcloud you need, you can add it with **Log in to a different BIMcloud.** (see [Log in to BIMcloud](#))
4. Choose a user.
 - If you are not logged in yet, choose **Log in** from the Select User field.
 - If you don't see the user you need, choose **Log in as a different user.**

See [Log in to BIMcloud](#).

If you no longer need to see an item, select it and use the **Remove** command from the drop-down lists

The Share Project dialog lists the folders and projects/libraries to which you have at least view permissions on the selected BIMcloud.

- If needed, click **Refresh** to refresh the BIMcloud status and content



5. Choose a folder and a name for the shared project

➤ Required permission:

- Simplified Management: Project or Server Administrator
- Detailed Management: Modify details and Modify folder content for the destination folder

- If needed, use **Create Folder** to create a new one

Note: To delete or otherwise manage folders, you must use the browser-based BIMcloud Manager. (Click the **Manage Folders** command from the Create Folder button to access the Projects page of the BIMcloud Manager.)

6. Enter a Name for the shared project.

Important: Use a different name for each project. Do not use the same name as an existing project, even if the letter cases are different (e.g. ParkingGarage and PARKINGGARAGE). If you do, users may not be able to Open or Join the project they need.

7. Check the displayed **Host Server**, which is the default Host Server of the selected project folder. If needed, choose a different Host Server.

Note: Not available in BIMcloud Basic.

➤ Required permission to access the pop-up:

- Simplified Management: Server Administrator
- Detailed Management: Modify details for the selected folder

➤ Available BIMcloud Servers in the pop-up:

- Simplified Management: All BIMcloud Servers for a Server Administrator
- Detailed Management: Those for which you have **Modify Hosted Data on BIMcloud Server** permission.

8. Click **Libraries** to open the dialog and decide how to handle the project libraries.

- When you share a solo project, use the Libraries dialog to optionally upload the solo project's linked libraries to the BIMcloud.

See [Libraries Setup for Teamwork Project Sharing](#).

Share

Click **Share**. The newly shared project opens.

If you do not want to open the shared project right away, use the **Share without Join** option from the Share button's pop-up.

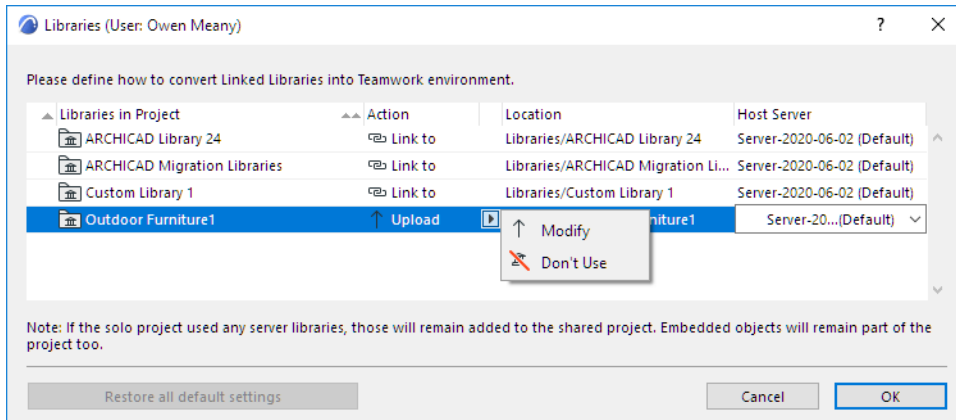
Open Project Settings after Share: Check this box to navigate to the Projects page of the BIMcloud Manager after the Share is complete.

Libraries Setup for Teamwork Project Sharing

Libraries can be used in a shared Teamwork project only if they are hosted on the BIMcloud. When you share a solo project, use the Libraries dialog to optionally upload the solo project's linked libraries to the BIMcloud.

The Libraries dialog is available from the Share Teamwork Project dialog. (See [Share a Teamwork Project in Archicad](#)).

The Libraries dialog lists the solo project's Linked Libraries in the left column.



For each selected linked library, choose an Action from the **Action** pop-up. The available Actions vary, depending on the specific situation.

Upload

Uploads the linked library to the selected Host Server shown here. This is the default option if:

- the linked library is available
- the target folder does not already contain a library of the same name

Upload Later

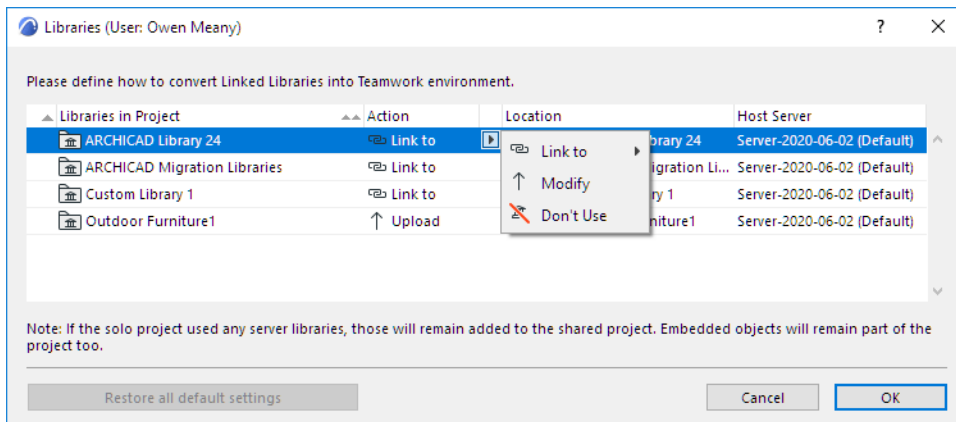
This option appears if

- the linked library you want to upload is missing (e.g. it has been deleted)
- You do not have necessary permissions to access the Host Server and/or to upload libraries

If you choose **Upload Later**, you can still share the project, but the linked library is not uploaded now. If someone else (e.g. a Server Administrator) uploads the Library to the specified location, the project will automatically find it.

Link to

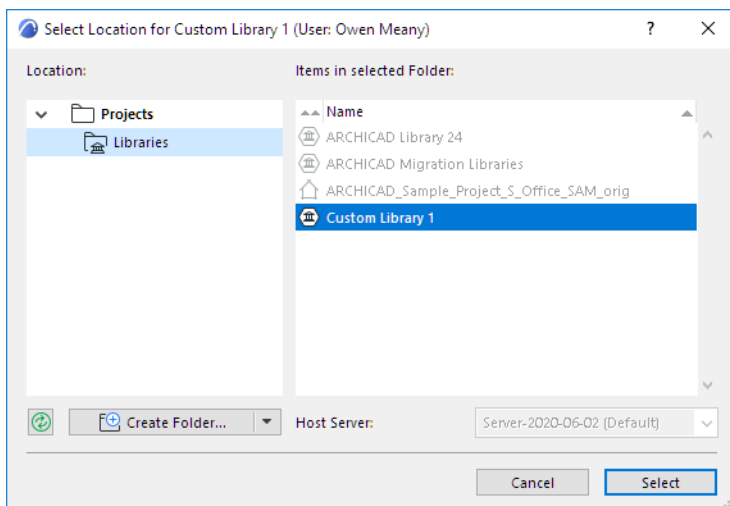
“Link to” is the default option if a library of the same name is available in the target folder of BIMcloud. This option links the shared project to this library. Or click the pop-up to link to a library of the same name at a different BIMcloud location.



Modify Library Folder

Use this option to upload the library to a different folder (other than the default) on the BIMcloud.

Note: Not available in BIMcloud Basic.



Required permission:

- Simplified Management: Project Administrator
- Detailed Management: Modify Content, Modify Details, View Item permissions for the Libraries folder

Modify Host Server

Note: Not available in BIMcloud Basic.

To upload a library to a different server (other than the default) on the BIMcloud, click the Host Server column in the Libraries share folder and choose from the pop-up.

You can also switch Host Servers from the **Select Location** dialog box (provided that the BIMcloud has multiple Servers to which you have access).

Required permission:

- Simplified Management: Server Administrator
- Detailed Management: Modify Hosted Data on BIMcloud Server

Select

This action appears if the Projects folder on BIMcloud has no defined target location.

(On the Projects page of BIMcloud Manager, the Library Folder setting is “Defined Manually from Archicad.”)

In this case, a red warning triangle appears in the Libraries dialog, and you cannot Share the project at all until you click Select. Then use the **Modify** command to define the target folder.

Don't Use

Removes the selected linked library from the project you are sharing. This option may result in a Teamwork project with missing library parts. In this case, a warning shows you the number of missing library parts that will result. Consult Library Manager for a list of these library parts.

Apply changes

From the Libraries dialog box, click **OK** to return to the **Share Teamwork Project** dialog box.

Your library conversion choices will take effect after you click **Share**.

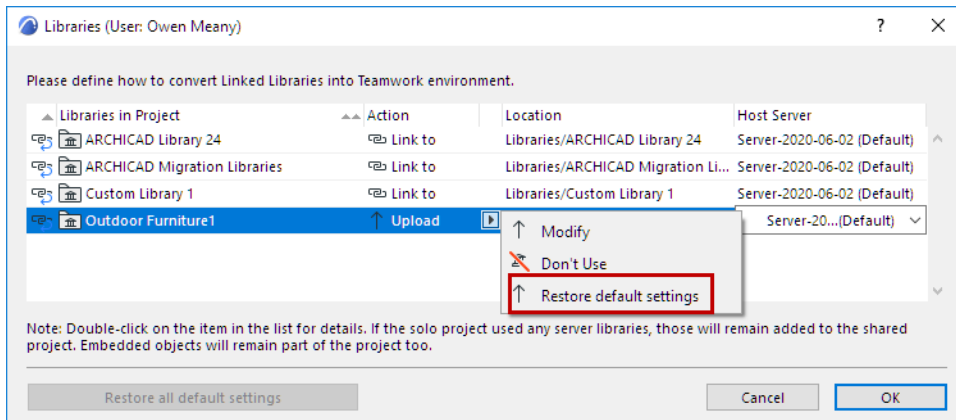
Default Location vs. Customized Location

During the Share process, BIMcloud defines a default location for the project’s linked libraries.

You can modify these defaults in the Libraries dialog, by placing a linked library in a different folder or on a different Host Server - as described above. Its settings are now customized.

To return the library to its default location, click the Libraries in Project item and use **Restore default settings** from the pop-up.

To restore all listed libraries to their default locations, use the **Restore all default settings** button at the bottom of the dialog.

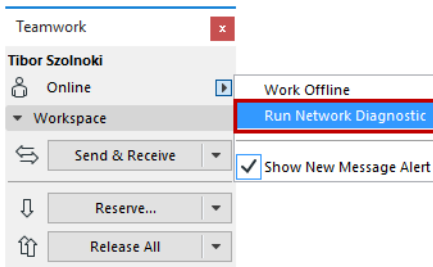


Network Diagnostic Information (BIMcloud)

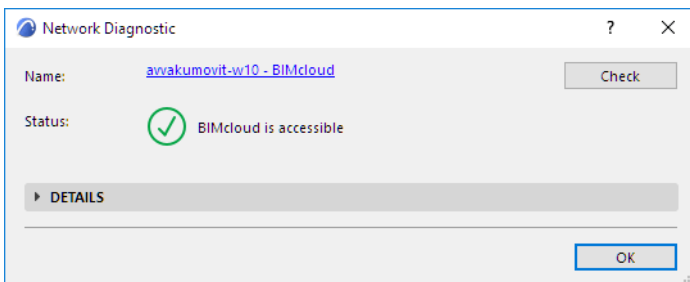
This dialog box, accessible to BIMcloud users, provides current information on your network connections and how to solve any problems.

Usually, your CAD Manager will set up the server addresses so that you will be able to access server(s) from Archicad without any problem, and you will not have to use Network Diagnostic.

If you do need it, this dialog box is accessible from all parts of the Archicad interface at which you select and log in to a server, plus the Teamwork Palette, and the Work Environment Settings Network and Update page (if you have installed a Delta Cache Server with BIMcloud).



If the currently connected server is working, the Network Diagnostic dialog box opens in compact form:

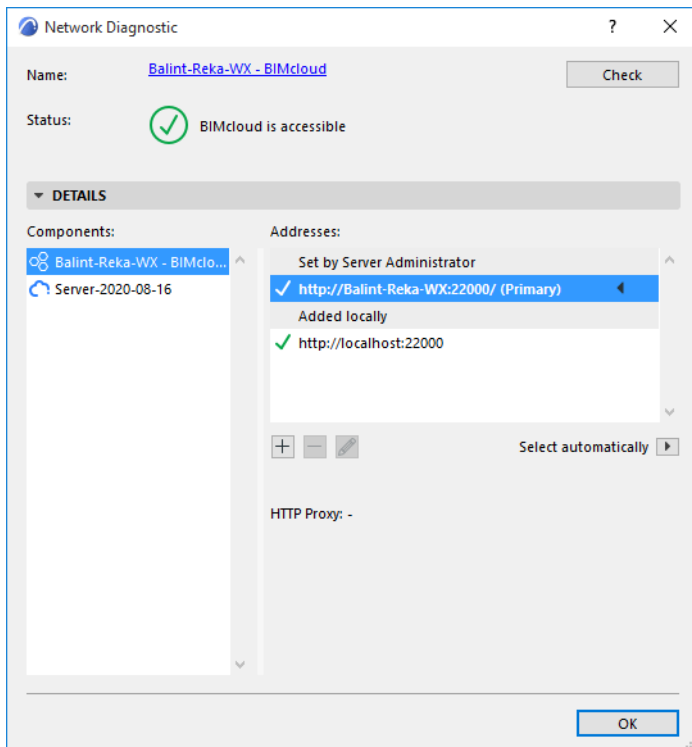


Name: This is name of the server to which Archicad is currently connected. If nothing is connected, this field shows an editable URL.

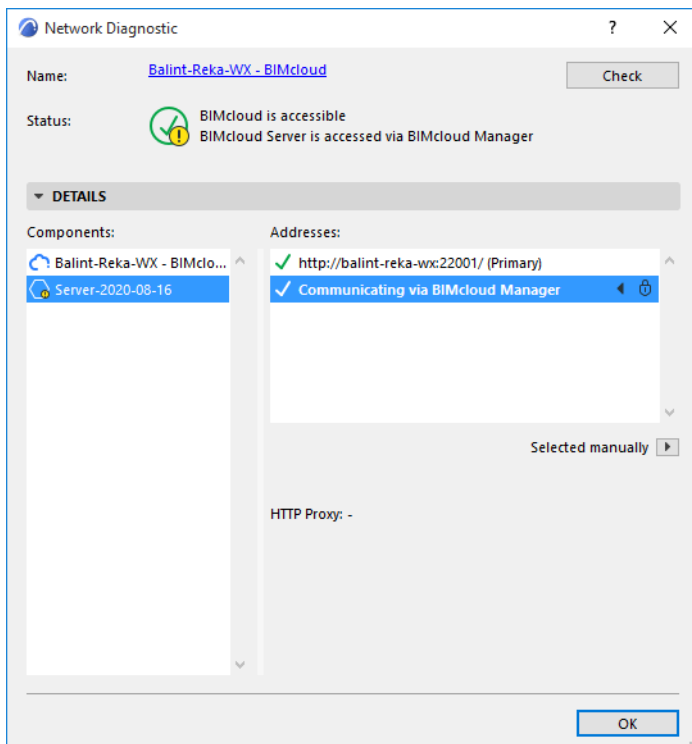
At right, click **Check** to refresh the server's status information.

Status: Basic feedback on the server's connection.

- The green checkmark means everything is accessible and working.



- The checkmark with exclamation point means that you can work with this server connection, but the connection might not be optimal. For example, if the server is communicating with Archicad via the BIMcloud Manager (and not directly); or if the server connection is not a secure one.



- The red X means that you cannot work with this server connection. Open the **Details** panel for more information.

Components

The Components list at the left side of the Details panel lists all the server components of the installed BIMcloud.

For any component selected here, its addresses are listed on the right (see below).

Addresses

This is the list of server addresses at which your Archicad can access the selected server component.

Your Server Administrator, using the BIMcloud Manager, sets up this list of addresses and their priorities.

Archicad will receive these addresses upon the first successful connection, and automatically pick the address most suitable for the current situation. Alternatively, you can select an address manually here in the Network Diagnostic dialog box ([see *Select Manually/Select Automatically, below*](#)).

If a Manager component is selected, you have the option of adding additional local server addresses (see below).

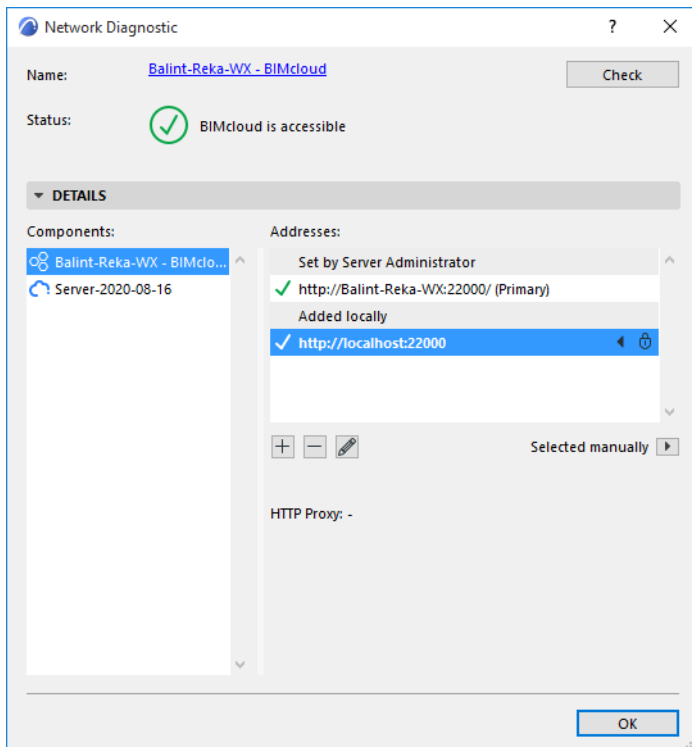
The address marked with an arrow is the address that is currently in use for the selected server component.

Advantages:

- The reason for multiple addresses is to optimize data transfer to fit various network configurations.
- If a user wishes to access the server from outside the network, via the Internet, he/she can do so using a public IP address, and does not have to provide the full domain (although this is still recommended).
- Enables the efficient use of various IT infrastructure configurations
- Supports single-port communications, simplifying IT tasks

The address marked with an arrow is the address that is currently in use for the selected server component.

- **Set by Server Administrator:** These addresses are defined by the Server Administrator and cannot be modified in Archicad.
- **Added Locally:** These server addresses are optionally added by you, to allow for alternative server access paths. The option to add local server addresses is available if the selected component is BIMcloud Manager.



Use the plus, minus and pencil icons to add, remove, or edit a selected address. Only locally added addresses can be managed here.

Select Manually/Select Automatically

By default, Archicad will go through the list of server addresses and use the first successful connection. In this case, the Network Diagnostic will show, for this selected address, that it was “Selected Automatically.”

If you prefer to use any other server address, and not use any of the others, click the Selected pop-up and choose “Always use selected address.”

Now this address will show a lock icon (since it won't change automatically), and it will be shown as “Selected Manually.”

Open or Join Teamwork Project

To open or join a Teamwork project in Archicad:

1. The project must already have been shared to the BIMcloud.

See [Share a Teamwork Project in Archicad](#).

2. You must be a user on the BIMcloud.

See the [BIMcloud User Guide for details](#).

Even if you are in offline mode, you can **open** a project which you joined earlier. However, since you cannot synchronize with the server when you are offline, you cannot be sure that the local copy you open is up to date.

Open Recent Teamwork Project

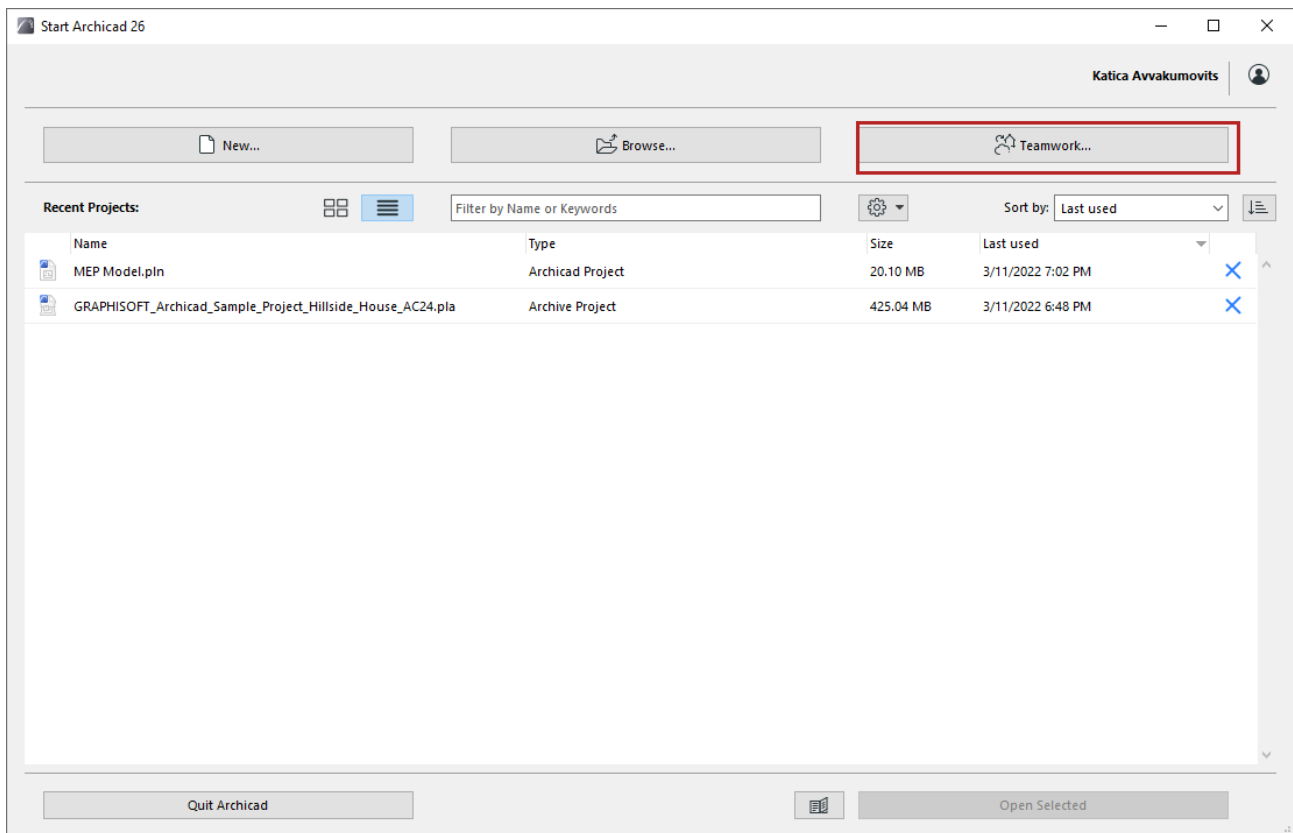
If you have recently opened the project on this computer with the current user name, do one of the following:

- From the **Start Archicad** dialog box, select it from the listed Recent Projects
- Use **File > Open** and select it from the listed Recent Projects

Choose Project from a BIMcloud

Do one of the following:

- From the **Start Archicad** dialog box, click **Teamwork**

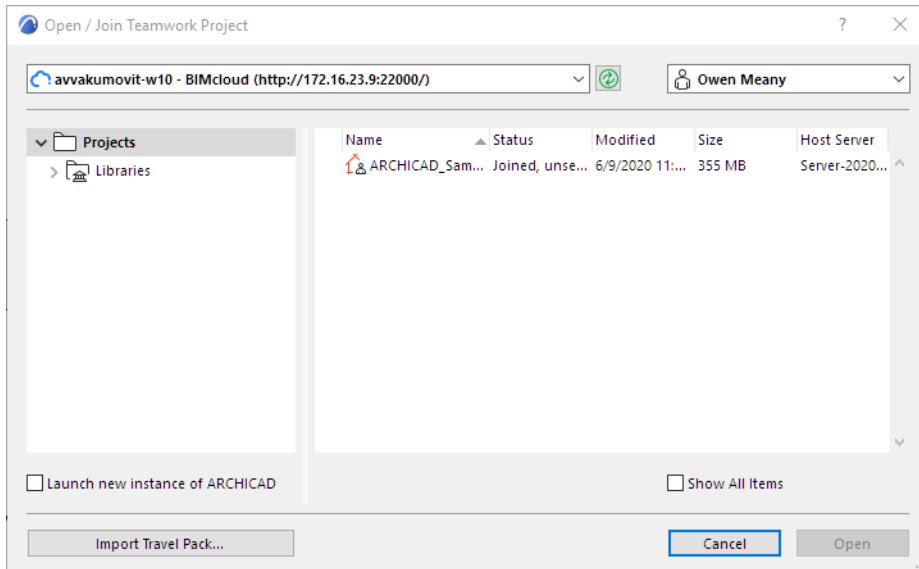


- **File > Open > Open/Join Teamwork Project**
- **Teamwork > Project > Open/Join Teamwork Project.**

Select a BIMcloud and user. If needed, log in to BIMcloud. See [Log in to BIMcloud](#).

Select Project

In the **Open/Join Teamwork Project** dialog box:



- For the selected server, the tree structure at left lists the project folders and projects on the BIMcloud to which you have access.
- Click Refresh to refresh the contents of the tree and the folders.
 - **Permissions required to view items:**
 - **Simplified Management:** You are a member of a folder's child project
 - **Detailed Management:** View item permission for the folder or any child folder/project
- For each project, the dialog box shows its status (whether you have joined it already), the last time it was modified, its size and the BIMcloud Server on which it is housed.
- If the selected BIMcloud is "Offline", the tree structure shows only those projects that you already joined at an earlier time, and whose local data is available on your machine. You can open such a project and continue working on it in offline mode.
- Locked projects are shown with a lock icon. You can open it in read-only mode.

Click **Open**.

(If you have not yet joined the project, the command changes from "**Open**" to "**Join**.")

Import Travel Pack

The **Import** button in the Open Teamwork Project dialog box enables you to import a Travel Pack file (.twtravel). (You create a Travel Pack to transport a Teamwork Project and/or project libraries to different locations).

See [Travel Pack: Move Your Teamwork Project and Libraries to a Different Computer](#).

Close Teamwork Project

Use **File > Close** to close a Teamwork Project in Archicad.

Your local copy of the project remains intact. If you have *unsaved* changes, you can opt to save them to the local copy. If you have *unsent* changes, you can opt to send them.

Close Project Online

When you close a project with unsent changes while online, the program prompts you with several options:

- **Discard changes:** Any unsaved and unsent changes are discarded. Your reservations remain intact.
- **Send & Save:** Send changes to the server and also save any unsent changes to your local copy.

Keep or Release Your Reservations with Send & Save

What happens to your reservations when you click **Send & Save**?

By default, your reservations remain unchanged. The next time you open this project, you will find that the data reserved by you before you closed the project are still yours.

To change this default, you can re-set a Work Environment preference, at **Options > Work Environment > Data Safety and Integrity**.

Here, check the **Release All when closing Teamwork Project with Send Changes** box. This means that when you close a Teamwork Project, all your reservations will be released.

Click the arrow button to the right of the Send & Save button for two more variations:

- **Save without Send:** Save your changes locally, but do not send them to the server
- Save & Send and Release All

or

Save & Send without Release

(The command shown - either “without Release” or “Release All” - will be the opposite of your general preference set in **Options > Work Environment > Data Safety & Integrity**.)

If you will not work on the project for a while after closing it, you are advised to release your reservations - before closing the project - so that other users can access the data they need.

Close a Project vs. Leave a Project

When you finish working each day, *do not leave* the project; just **Close** it.

The next day, you will not have to join the project (you are already joined in); just **Open** it.

As long as you are joined in to a Teamwork Project, you can Open or Close the project as many times as you like, without losing your reservations and without dumping your local copy.

Note: If you change the default setting “**Close Teamwork Project**” in **Options > Work Environment > Data Safety & Integrity**, then your reservations *will* be discarded if you click **Save & Send** when closing the Teamwork Project.

See [Keep or Release Your Reservations with Send & Save](#).

However, once you **Leave** the project (as opposed to closing it), you lose your reservations and dump your local copy. It is recommended that you do not leave the project unless you do not plan to work on it any more in the foreseeable future. Typically, you will leave a Teamwork project when you no longer participate in it.

Close Project Offline

If you are offline when you Close the project, you cannot send changes to the server. Your only options are to discard the changes, or else to save them locally.

If you choose “**Discard Changes**,” then your unsaved changes will be lost, and you will not synchronize your local copy with the server.

Save Local Copy of Teamwork Project

When you join a Teamwork project, local copies of the project and of its libraries are created on your hard drive.

Local data is also created when you link a Drawing from another project to your own project. The source project data is saved as local data to your machine, even though you did not join that project.

The **Save** command refreshes this local data. The **Save** command does not create a PLN file, and it does *not* send your changes to the server.

Local data are stored in the **Local Data Folder**, which by default is located in your installed Graphisoft folder on your hard drive.

All data from all Teamwork projects, regardless of the Archicad version they are in, are saved to this one location.

To manage your local data (e.g. move it to a new location or to delete it), go to the **Local Data Manager** dialog box (**Teamwork > Project > Local Data Manager**).

[See *Manage Local Teamwork Data*.](#)

Saving to a Recovery File for a Teamwork project behaves the same way as for solo PLNs: it automatically stores your Teamwork project to the Local Data Folder at predefined intervals, without your having to issue the Save command. Each user sets his/her own Recovery File preferences locally, at **Options > Work Environment > Data Safety & Integrity**.

Save as PLN

If you wish, you can save your Teamwork project as a **PLN** file. (Use **File > Save As**, and choose PLN file format); this newly saved PLN will contain all project elements. This PLN file is *solely a local file* and cannot be sent to the server, but you can re-share the PLN file as a Teamwork project.

Recommendation: If you are working in Teamwork and suspect any kind of data error, the first thing you should do is to save your project as a PLN file.

Necessary permission to save a project as a PLN: Save as PLN,PLA,MOD

Reload Project from Server

This command is located at **Teamwork > Project > Reload Project from Server**.

Ordinarily, you do not have to use this command during your normal daily workflow. But if you are working along and then realize that you want to discard your local changes and start fresh, or if you suspect any kind of data error in the project you are working on, use Reload Project from Server. This means that your unsent changes will be discarded, and you will get the current, entire state of the project as it exists on the server. (In contrast, when you Receive project changes from the server, your local data, including unsent changes, remain intact.)

Force User to Leave a Teamwork Project

If you have the necessary permissions, you can force any user to leave the project.

As a result, the user cannot send his/her changes to the project and will lose all his/her reservations.

Unless you also remove the user from the project's user list, this user will be able to re-join the project immediately.

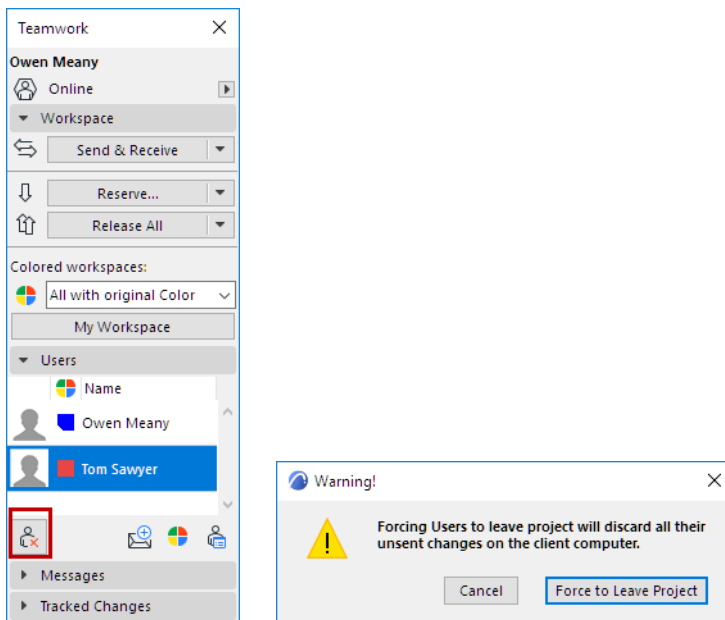
Consequently, the "Force Leave" function is a temporary situation that can serve to force the release of reserved elements or data, without waiting for the owner to release them.

To force a user to leave the project:

- Use the Teamwork Palette: select the user name from the Users panel, then click the "Force User to Leave Project" icon or the command from the context menu.
- In BIMcloud Manager (Users page): Select the user in the Navigator. From the Joined Projects list in the Summary section, click Force Leave next to the project that the user must leave. Multi-select projects as needed. Or use Force Leave All to eject user from all joined projects.
- In BIMcloud Manager (Projects page): Select the project in the Navigator. From the Joined Users list in the Summary section, click Force Leave next to the name of the user who must leave. Multi-select users as needed. Or use Force Leave All to eject all users.

A Warning tells you that the user will not be able to send his/her unsent changes to the project.

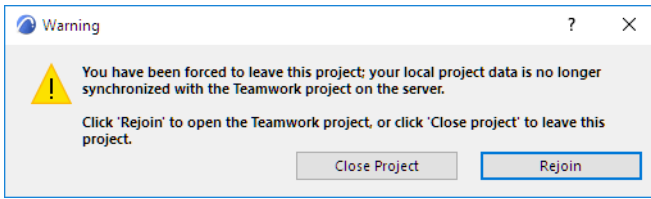
The user is notified that he/she has been forced to leave the project, and about the options for continuing.



➤ Necessary permissions to force leave:

- Simplified management: Project or Server administrator
- Detailed management: Modify access

At this point, the user is forced out, and receives the following message:



Join or Leave a Teamwork Project

To join or leave a Teamwork project, you must be online.

Join

Typically, you will join a Teamwork project and remain joined in, for the full period of time that you are a participant in the life of the project.

As long as you are joined in to a Teamwork Project, a copy of the project exists locally on your machine and is synchronized with the project on the server whenever you send and receive changes.

As long as you are joined in to a Teamwork Project, you can **Open** or **Close** the project as many times as you like, without losing your reservations or dumping your local copy.

To join a project, follow the process described here: [Open or Join Teamwork Project](#).

The only difference is that the project you choose from the Open Teamwork Project list will not have a blue head (which would indicate that you already joined it), and the dialog box command changes to **Join**.

Leave

Typically, you will leave a Teamwork project when you no longer participate in it. Once you leave a project, you relinquish your reservations and the local copy on your machine is discarded.

Note: Even if you leave (and not just close) the project and later re-join it, full library downloads will be not be repeated if the local library copy has remained on your machine. In contrast to the local copy of the project, which is discarded when you leave the project, the local library copy is not discarded when you leave the project.

Close a Project vs. Leave a Project

When you finish working each day, *do not leave* the project; just **Close** it.

The next day, you will not have to join the project (you are already joined in); just **Open** it.

As long as you are joined in to a Teamwork Project, you can Open or Close the project as many times as you like, without losing your reservations or dumping your local copy.

However, once you **Leave** the project (as opposed to closing it), you lose your reservations and dump your local copy. It is recommended that you do not leave the project unless you do not plan to work on it any more in the foreseeable future. Typically, you will leave a Teamwork project when you no longer participate in it.

[See also Close Teamwork Project](#).

Force Leave

[See Force User to Leave a Teamwork Project](#).

Send/Receive Changes in Teamwork

The **Send** and **Receive** commands send and/or receive changes - which have occurred since the last Send/Receive.

Send and Receive commands are available

- from the Teamwork Menu, and
- from the Teamwork Palette

Changes to model data must be **sent** to the server and **received** by other users, in order for all users to have an up-to-date project.

Teamwork's DELTA-Server technology ensures that only the changes since the last Send & Receive will be sent through the network and processed by the server or clients; this greatly reduces network traffic and improves performance.

Send/Receive commands involve refreshing BIMcloud data, but these commands *do not save anything locally* on your machine.

[See also *Save Local Copy of Teamwork Project*](#).

Exception: If you do a Send or Receive, then close your project, a Save will automatically occur.

When you **Receive** from the server, your unsent modifications in your local copy are not affected – they remain intact.

Send/Receive does not affect the reservation status of data; reservation status is always up-to-date as long as you are online.

When you work **offline**, your copy of the project is by definition out of sync with Teamwork server data. When you go online again, you must do a Send and Receive to ensure that your data and also the server data are up-to-date.

Archicad will prevent any user from reserving any element or data type that is not in sync with the latest server data. In such a situation, the program will enforce a Receive. Similarly, the program will prompt you to do a send or receive if that is required to carry out a command: for example, if you try to release an element without first having sent your changes to the server, you are prompted to "Send and Release."

Warning: The project's undo queue is cleared entirely whenever you issue any of the Send/Receive commands in Teamwork

Send & Receive, Then Compare

The Receive or Send & Receive commands can be used together with Model Compare. Use this Compare workflow in Teamwork after receiving changes, to see what has changed in the model.

[See *Compare Teamwork Model Versions \(Before and After Receiving Changes\)*](#).

Automatic Reminder to Send Changes

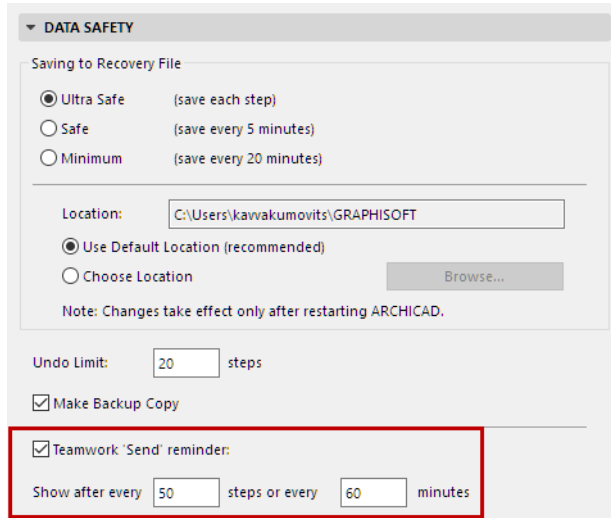
If you have been working in a Teamwork project for an extended period and/or have made numerous changes or edits to the project, Archicad will display an on-screen notice that it may be time to Send your changes.



- You can click **Send** from the notice itself to send your changes.

- Click **Snooze** to make the notice disappear immediately; it will not appear again for another 30 minutes or 100 steps.
- If you do nothing, the message will disappear on its own after 10 seconds.

This is an optional feature. Enable or disable it at **Options > Work Environment > Data Safety and Integrity**. Here, you can also set how often the notice should appear.



DATA SAFETY

Saving to Recovery File

Ultra Safe (save each step)
 Safe (save every 5 minutes)
 Minimum (save every 20 minutes)

Location: C:\Users\kavvakumovits\GRAPHISOFT

Use Default Location (recommended)
 Choose Location Browse...

Note: Changes take effect only after restarting ARCHICAD.

Undo Limit: 20 steps

Make Backup Copy

Teamwork 'Send' reminder:
Show after every 50 steps or every 60 minutes

Reload Project from Server

This command is located at **Teamwork > Project > Reload Project from Server**.

Ordinarily, you do not have to use this command during your normal daily workflow. But if you are working along and then realize that you want to discard your local changes and start fresh, or if you suspect any kind of data error in the project you are working on, use Reload Project from Server. This means that your unsent changes will be discarded, and you will get the current, entire state of the project as it exists on the server. (In contrast, when you Receive project changes from the server, your local data, including unsent changes, remain intact.)

Related Topic:

[Compare Teamwork Model Versions \(Before and After Receiving Changes\)](#)

Manage Local Teamwork Data

You may have joined Teamwork projects in the past that are no longer active; in this case, you should delete the unneeded local data that is just taking up space on your machine.

The **Local Data Manager** (**Teamwork > Project > Local Data Manager**) feature gives you feedback to help you manage the local data from Teamwork projects you have joined.

Topics in this section:

[What is Local Data?](#)

[Get Feedback on Local Teamwork Project/Library Data](#)

[Review Status and Properties of Local Data](#)

[Move Local Teamwork Data to a Different Folder](#)

What is Local Data?

When you join a Teamwork project or link a Drawing from an external project, local copies of that project and of its libraries are created on your hard drive.

Local data are stored in your **Local Data Folder**, which by default is located in the “Graphisoft” folder, under your user data, on your hard drive.

All data from all Teamwork projects, regardless of the Archicad version they are in, are saved to this one location.

[See also *Save Local Copy of Teamwork Project*.](#)

When you **Leave** a Teamwork project, the local data for that project are deleted automatically. The problem of unneeded data only arises for projects that you remain joined into.

[See *Join or Leave a Teamwork Project*.](#)

Get Feedback on Local Teamwork Project/Library Data

To see the status and amount of local Teamwork data stored on your computer, go to the **Local Data Manager** dialog box (**Teamwork > Project > Local Data Manager**).

In this dialog box:

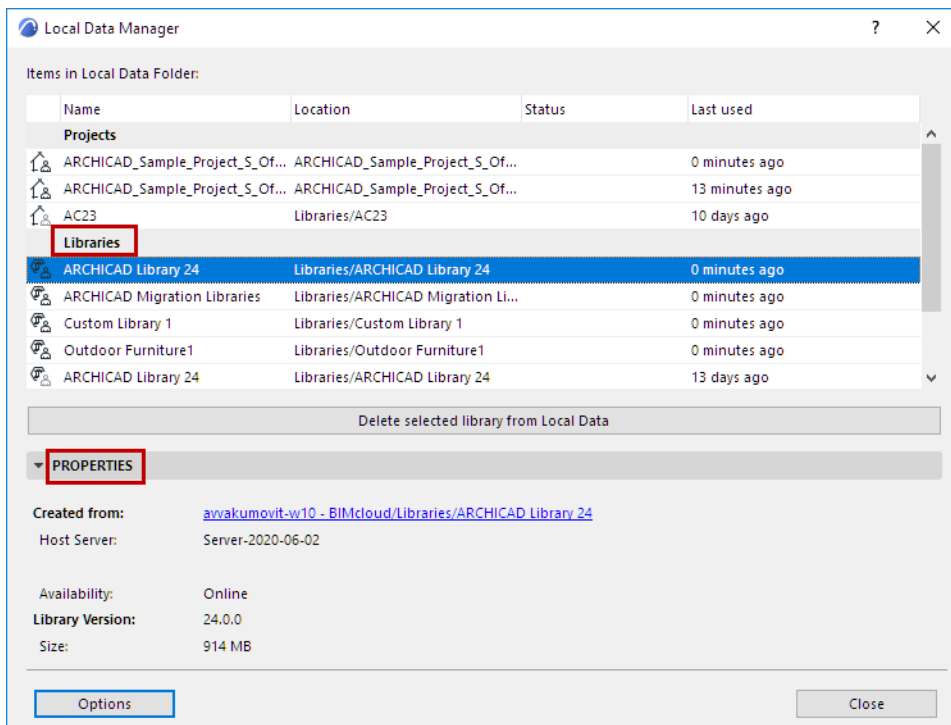
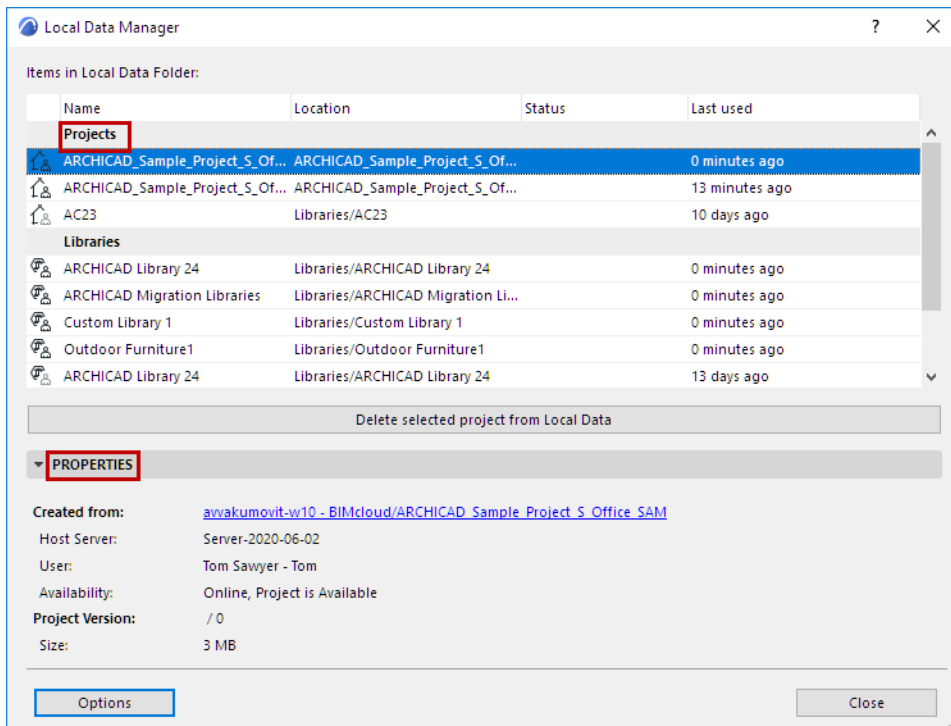
- “Projects” are the *locally saved data* from your Teamwork Projects. (Project data includes the Embedded Library, but not any BIMcloud Libraries).
- “Libraries” are *local copies* of the BIMcloud Libraries used in one or more of your Teamwork projects.

Review Status and Properties of Local Data

Each project and library is listed with:

- **Status:** The project or library data’s status with respect to the server, e.g. “Unsent Changes” or “Detached”.
- **Location:** The folder of the project/library on the BIMcloud
- **Last Used:** This date is the last time you accessed this project.

For libraries, this date is the last time the library was downloaded to your machine (usually, the last time you opened a Teamwork project using this library).

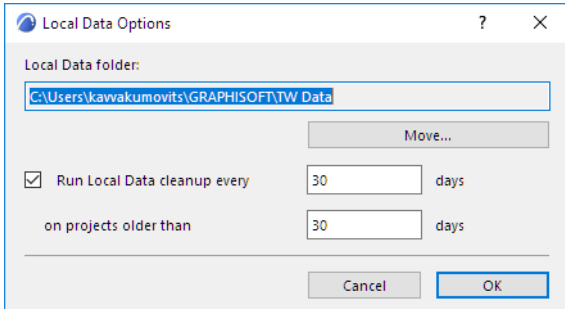


For any selected Project or Library in the list, view the **Properties** tab page for information on:

- the **Name, Address** and **Availability** of the Host Server or Module on which the project or library is located
- the **Version, Size** and **Folder** of the Project or Library

Move Local Teamwork Data to a Different Folder

1. Close all running Teamwork projects. (Save and/or Send, as necessary.)
2. Go to **Teamwork > Project > Local Data Manager**
3. Click the **Options** button to open the **Local Data Options** dialog box.



4. Click **Move...**
5. In the appearing **Browse for Folder** dialog box, choose or create a new folder location.
6. Click **OK**.

All data is moved to the new location, and all Teamwork local data - regardless of Archicad version - will henceforth be saved to this location.

Related Topics:

[Move Local Teamwork Data to a Different Computer](#)

[Delete Local Teamwork Data](#)

[Schedule Teamwork Data Cleanup Reminder](#)

Move Local Teamwork Data to a Different Computer

IMPORTANT: It is possible to move your local data to a different computer, as described below. However, if you are working on one Teamwork project at a time, *the Travel Pack function is the preferred workflow* for transporting it between computers.

See [Travel Pack: Move Your Teamwork Project and Libraries to a Different Computer](#).

The Travel Pack is optimized for transporting one project at a time, whereas moving local data involves all the data from all Teamwork projects on your machine.

It is possible to use the “Move Local Data” feature to move your local data to a different machine - for example, your home computer - via a pen drive:

1. At the office, move your local data to a pen drive.
2. Take it home. On your home computer, start Archicad and use the “Move Local Data” function again, defining the folder on the pen drive as the new location.
3. Since this folder contains data, Archicad will henceforth use this location and this data as Local Data.

Delete Local Teamwork Data

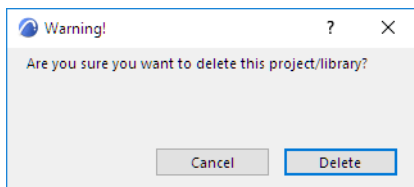
Use the Local Data Manager (**Teamwork >Project > Local Data Manager**) to delete your local copies of Teamwork projects and libraries. This means, of course, that you will lose any changes that you have not yet sent to the server.

Deleting your local data is equivalent to leaving the project, except that your reservations will remain. Should you wish to work in the project again, you must again join the project, thereby downloading a local copy of the project as well as its libraries.

1. Go to **Teamwork > Project > Local Data Manager**.
2. From the list, select the Project or Library you wish to delete.
3. Click **Delete** at the bottom of the dialog box.
4. The selected project or library copy is deleted from your Local Data Folder.

Note: If you choose to delete the local data of a project from a BIMcloud that is online, Archicad will recommend that you leave the project instead of just deleting the local copy, because leaving the project will also release your reservations.

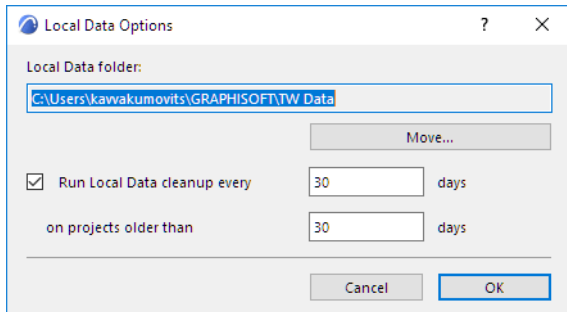
In any case, if you should prefer to delete the local copy, you should also release your reservations.



Schedule Teamwork Data Cleanup Reminder

To help with data management, you can set Archicad to issue data cleanup reminders for data of a certain age.

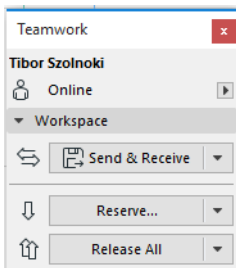
1. Go to **Teamwork > Project > Local Data Manager**.
2. Click the **Options** button to open the **Local Data Options** dialog box.



3. Check the **Run Local Data cleanup** box.
 - Enter the number of days - (for example 15). This means that Archicad will check for old and unused data every 15 days.
 - Enter the number of days (e.g. 15 days) beyond which a project is considered “old.”
4. At the scheduled intervals, Archicad will suggest that you consider a cleanup of your local data, if old data is found.

Working Online/Offline in Teamwork

At the top of the Teamwork Palette, you can view your online status:

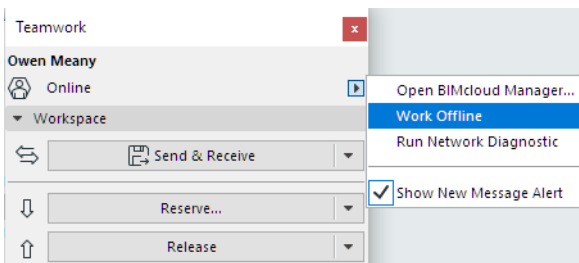


The client machine must be online to use the following Teamwork functions:

- Reserve
- Send and Receive
- Messaging

If the client is offline, interface items relating to these functions will be grayed.

To work Offline, click “Work Offline” from the pop-up arrow at the top of the Teamwork Palette.



Click this command once more to go online again.

Remote Access to Teamwork Projects

Archicad supports several ways for you to access and work on a Teamwork project from different locations. This is helpful if, for example, you are traveling and want to take the project with you; or if you regularly work on the same project from several locations (e.g. home and work).

Related Topics:

[Working from Home and the Office: Suggested Workflow for a Teamwork Project](#)

Read this section for a step-by-step workflow.

[Travel Pack: Move Your Teamwork Project and Libraries to a Different Computer](#)

Read this section to find out how to move your project and/or libraries easily from one computer to another.

[Avoid Conflicting Work Sessions in Teamwork Projects](#)

Read this section to find out how to avoid losing edits due to conflicting work sessions from multiple locations.

Working from Home and the Office: Suggested Workflow for a Teamwork Project

The following describes how to work on a single project while alternating between two *online* locations.

For alternative scenarios involving working offline and how to move project data, see: [Travel Pack: Move Your Teamwork Project and Libraries to a Different Computer](#).

This scenario works under two conditions:

- You must have online access at both locations
 - After each session, you must do a Send & Receive before closing the project
1. At the office, finish working, do a save and Send/Receive.
 2. On your home computer, start Archicad. You must be online to download the Teamwork Project and libraries from the BIMcloud.
 3. Work on it as needed, either online or offline.
 4. Once you finish, you must be online: do a Send/Receive.
 5. Save and close the project.
 6. Back at the office, open the project again.
 7. On the office computer, do a Send & Receive. The modifications you made at home are now merged into the project on the office computer, and you can continue working.

You can keep going in this manner, alternating your home and office computers, as long as you need to.

Travel Pack: Move Your Teamwork Project and Libraries to a Different Computer

The Travel Pack function creates a portable copy of your local data from the current Teamwork project. By converting your local data into a Travel Pack, you can transfer project data and/or libraries to a remote computer and continue working there.

Important: Make sure that you are using Archicad programs with identical versions and identical build numbers, on both the remote and original computers. Otherwise, you will not be able to merge changes to the project.

Topics in this section:

[Why Use a Travel Pack?](#)

[Create a Travel Pack](#)

[Import Travel Pack](#)

[What does “Traveling Mode” mean?](#)

[Travel Pack Workflow 1: Work on a Project at a Remote Location without Online Access](#)

[Travel Pack Workflow 2: Work on Single Project while Alternating Between Two Online Locations](#)

Why Use a Travel Pack?

It is possible, of course, to just go to another computer and access and download the project from the BIMcloud through a network, or through the Internet. The Travel Pack, however, gives you more flexibility in accessing project data from multiple locations. Using the Travel Pack is recommended in the following cases:

- **You cannot access the BIMcloud from the remote computer.** In this case, the only way you can access project data is to bring it with you to the remote computer.

At the original location, you will make a Travel Pack out of the project, take it home, unpack it, work on it in offline mode, pack it up again, then return to the original location (or other machine with convenient access to the BIMcloud). You can now go online and synchronize your local copy with the project data on the server.

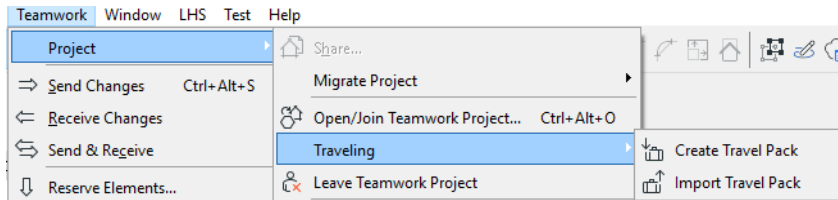
- **Your network or internet access is slow.** While you can download the project over the internet at the remote location, the sheer volume of data makes this prohibitively time-consuming. Instead, create a Travel Pack at the original location, then open it at the remote location. This way, the project's local data is saved to the machine - no downloading necessary.

When you do go online, all you need to do is send/receive changes to synchronize your local copy with the project data on the server. This requires much less data transfer capacity than downloading the entire project with all its libraries.

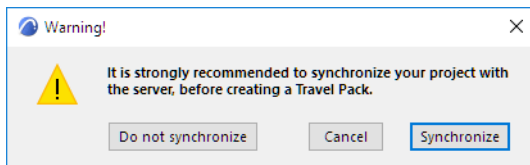
- **You can access the BIMcloud easily, yet the project's BIMcloud libraries are so large that you do not want to wait for the whole download.** On the original computer, create a *new* shared project, adding the BIMcloud Libraries you will need. This project contains no model data, just libraries. Out of this, create a Travel Pack. On the remote computer, open the Travel Pack project: this step will unpack those libraries and save them to your Local Data folder. Now, access the BIMcloud to open the project itself, the one you want to work on. The program will find that the libraries have already been downloaded locally, so the download is limited to just model data - no library downloads.

Create a Travel Pack

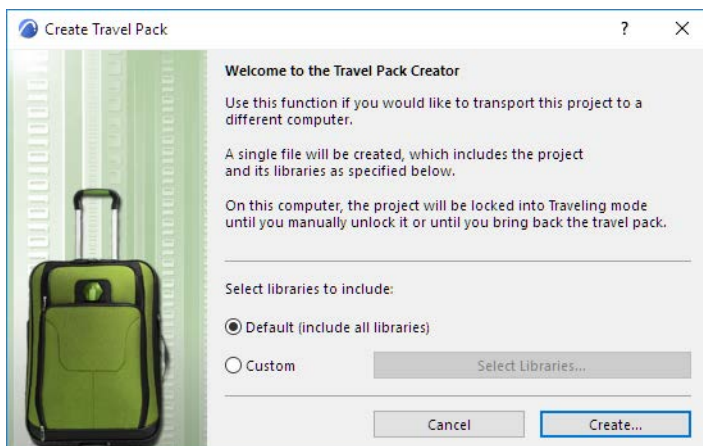
1. Open the project out of which you would like to create a Travel Pack.
2. If you plan to work on the Travel Pack project in offline mode, make sure you reserve all the data and elements you will need.
3. Save any changes to the project. If you are online, do a Send/Receive.
4. Use the **Teamwork > Project > Traveling > Create Travel Pack** command.



Note: If you are online and did not do a Send/Receive, you are prompted to do so.

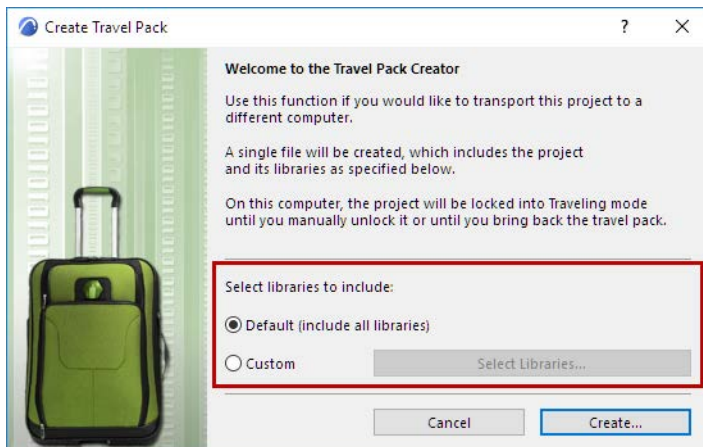


5. The **Create Travel Pack** dialog box appears.



In addition to the project data, all of the project's libraries will be included in the Travel Pack. If you prefer to include only certain libraries in the Travel Pack (or no libraries at all), click **Custom**, then **Select Libraries** to specify them. (Uncheck any libraries you do not want to include.)

Note: If you do not include libraries in the Travel Pack, your Travel Pack will be much smaller. However, the *first* time you use Travel Pack to transfer a project to a remote computer, make sure the Travel Pack does include the libraries you will need. In the next "round" of creating the Travel Pack, you can skip the libraries to save time in the packing/imported process, provided that the needed libraries still exist on the remote computer, and provided that no changes to the libraries have occurred on the server in the meantime.



6. Click **Create**. The **Pack Project** dialog box appears. Choose a location to which to save the file (for example, a pen drive).

The file name of the Travel Pack is identical to the original project name by default, but you can enter a different name if desired. The file extension is *.twtravel.

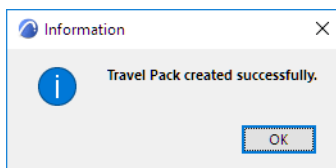
7. Click **Pack**. This action will close the current project.

Important: Even if your Work Environment is set to automatically do a Release All when you Close the project, you will *not* release all your elements at this point.

See also [Keep or Release Your Reservations with Send & Save](#).

If you have any unsent changes, you are notified that these will be saved before the project is closed.

8. The Travel Pack, containing your local copy of the project data and/or its libraries, is ready for transport.



Import Travel Pack

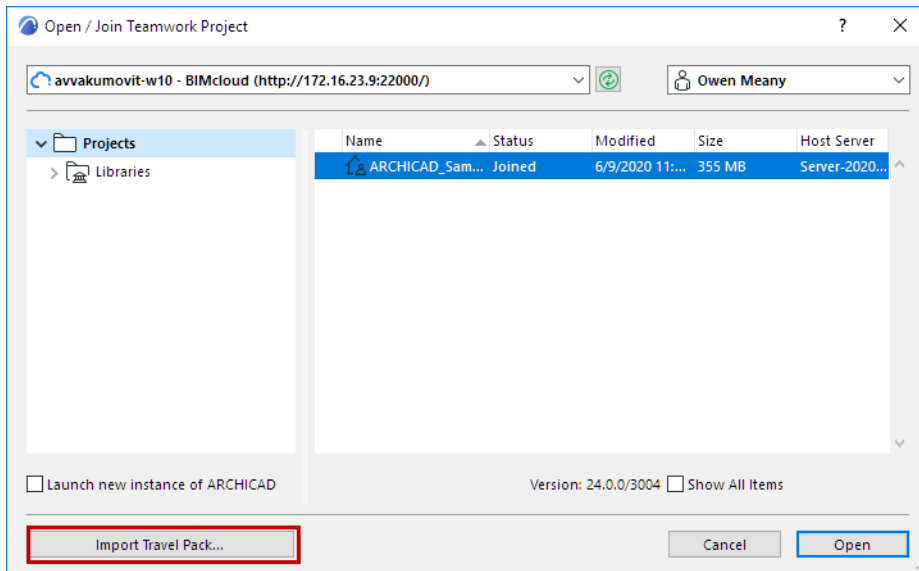
On any computer that is running a compatible version of Archicad (with the same version number and build number as on the original computer), you can open the Travel Pack and keep working.

1. Go to **Teamwork > Project > Traveling > Import Travel Pack** and browse for the travel pack file.
2. Click **Import**.

If you have worked on this project on this computer before, and if the local data is still on the machine, you receive a warning, because presumably the two versions of the local data are in conflict. If you want to continue, click **Overwrite**: the data in the Travel Pack will replace the existing local data on the hard drive.

If you have not worked on this project on this computer before, then the Travel Pack is imported.

3. Click **OK** to bring up the Open Teamwork Project dialog box. Choose a BIMcloud, then select the project and click **Open**.



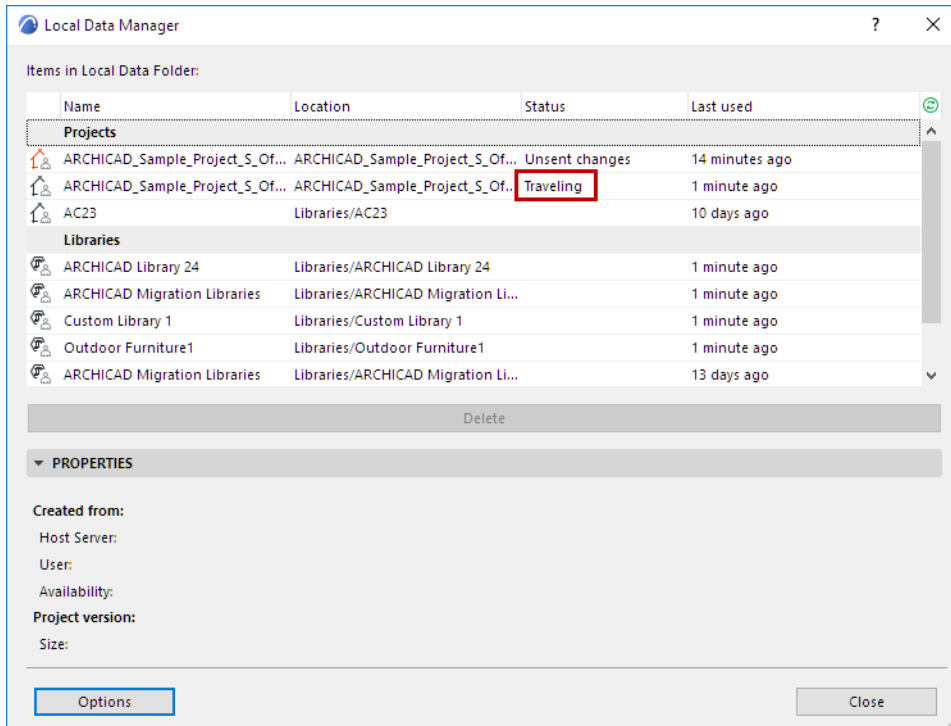
Alternatively, click **Import Travel Pack** to import a different Travel Pack project.

Whenever you close a project that you imported from a Travel Pack, Archicad will ask you whether you want to recreate the Travel Pack. You may want to do this, if you want to import the latest data back to its original location. Once you recreate the Travel Pack, the project's local data is removed from the computer you have been working on.

[See *Travel Pack Workflow 1: Work on a Project at a Remote Location without Online Access.*](#)

What does “Traveling Mode” mean?

Once you create a Travel Pack out of a project’s local data, that local data stored on the hard drive is flagged as being in Traveling mode. The Local Data Manager, as well as the Open Teamwork Project dialog box, will show this project with the status of “Traveling”.



The Traveling mode “flag” is removed from the project automatically as soon as you import a Travel Pack of that project.

If you later *open* the project again (rather than importing the Travel Pack) on the same, original computer, Archicad will warn you that you have already made a Travel Pack out of the local data (and presumably have worked on it, or will work on it, at another location).

It is recommended that you do NOT unlock and open the project here, unless you are certain that will not import the Travel Pack onto this machine at a later time.

For such a scenario, see [Travel Pack Workflow 2: Work on Single Project while Alternating Between Two Online Locations](#).

Travel Pack Workflow 1: Work on a Project at a Remote Location without Online Access

Suppose you are working on a Teamwork project from your office computer, but wish to take the project and work on it **in offline mode** on your portable notebook computer. For example, you are flying on a plane without internet access, but want to work during your flight.

1. Reserve all the data and elements you will need before creating the Travel Pack.

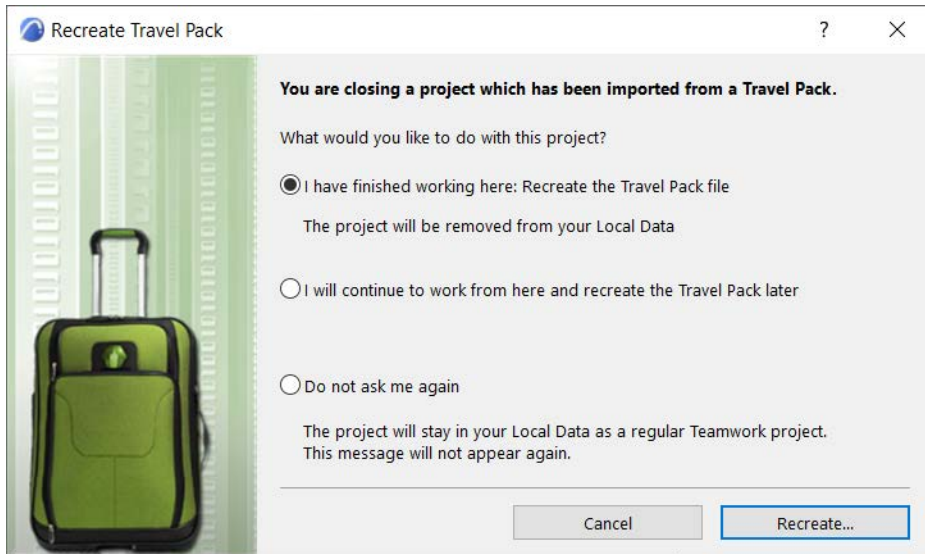
Create a Travel Pack at the office and save the resulting file (*.twtravel) to a portable device.

See [Create a Travel Pack for details](#).

Note: The project’s local data on the original computer is now flagged as being in “Traveling Mode.”

2. Once you are on the plane, start Archicad on your notebook.
3. Import the Travel Pack, then Open the project.
4. Work on the project as needed in offline mode.

5. When you are done, close the project. The following dialog box appears:



6. Choose the first option to recreate the Travel Pack from the local data on your notebook, and save the resulting file (*.twtravel) to a portable device. The local project data is deleted from your notebook.

7. Back at the office, start Archicad, import the Travel Pack, then open the project.

8. You are ready to work as usual.

Travel Pack Workflow 2: Work on Single Project while Alternating Between Two Online Locations

In this scenario, you make a Travel Pack once, to transfer the local data from Location A to Location B the first time. Once the local data exists in both places, there is no more need for a Travel Pack.

Important: You must issue the **Send** command, then the **Receive** command at the end of your work session at both locations. (Do not use Send & Receive.)

Note: To work from several online locations, you do not necessarily need a Travel Pack: if you can access the BIMcloud online, then you can download the project and libraries from any machine, work as needed, then Send & Receive before closing the project.

[See *Working from Home and the Office: Suggested Workflow for a Teamwork Project*.](#)

However, if you do not want to download the project data and/or libraries online, you can use the Travel Pack to transfer this data physically from one location to another, as described here.

This scenario works under two conditions:

- You must work online at both locations
- You must issue the **Send** command, then the **Receive** command at the end of your work session at both locations. (Do not use Send & Receive.)

1. At the office, finish working, do a save.

2. Issue the **Send** command.

3. Issue the **Receive** command.

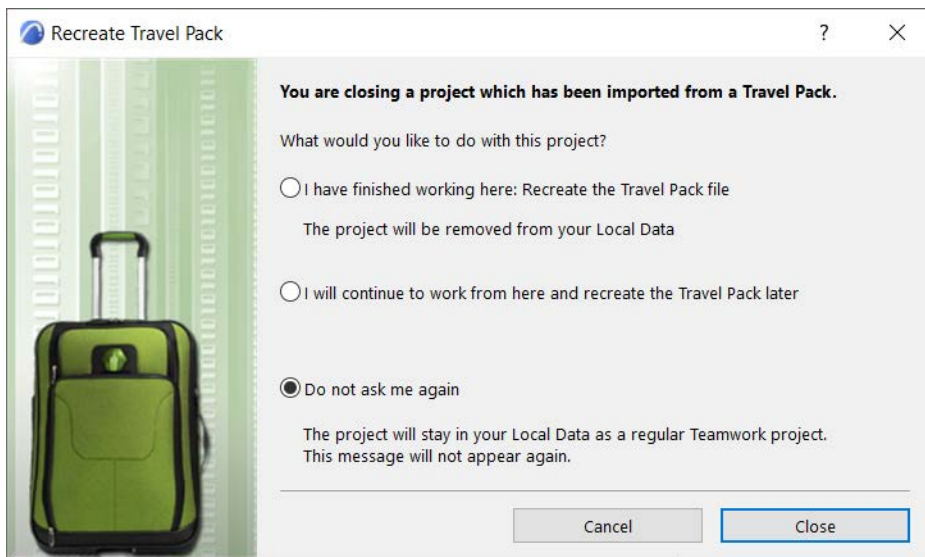
4. Create a Travel Pack.

5. Do a save, then close the project.

[See *Create a Travel Pack for details*.](#)

The local data on your office computer is now flagged as being in “Traveling” mode.

6. On your home computer, start Archicad and unpack the project. Work as usual.
7. Once you finish, you must be online so that you can do a Send/Receive.
8. Issue the **Send** command.
9. Issue the **Receive** command.
10. Save the project.
11. Close the project. The following notice appears:



Since you have done a Send/Receive, the local data is synchronized with the server, and you do not need a Travel Pack to move the local data manually back to the office. Choose the third option (Do not ask me again), and click **Close**. The local project data is no longer in traveling mode on your home computer, and is a regular Teamwork project.

12. Back at the office, open the project again. You are alerted that the local data on this computer is in Traveling mode.
13. Since the modifications you made at home have been sent to the server, you can go ahead and choose Unlock and Open. (The local data on your office computer will thus cease to be in Traveling mode.)
14. On the office computer, issue the **Receive** command. The modifications you made at home are now merged into the project on the office computer, and you can continue working.

You can keep going in this manner, alternating your home and office computers, as long as you need to:

15. On the office computer, finish working, issue the **Send** command; then issue the **Receive** command; do a save, then close the project. If you want to keep working at home, you do *not* need to make another Travel Pack, because the project’s local data is already on your hard drive at home.
16. On the home computer, start Archicad, open the project, issue the **Receive** command. The modifications you made at the office are now merged into your local data at home, and you can continue working, with no further need for a Travel Pack.
17. As needed, repeat the process as described beginning at Step 6 above.

Avoid Conflicting Work Sessions in Teamwork Projects

If you are working on a Teamwork project that you later plan to open from a different computer, make sure you send in all your changes before you close it.

If you open and work on a Teamwork project on one computer, then subsequently open it on a different computer, you may run into editing conflicts if you did not send in all your changes.

In this case, you will be warned of an Editing Conflict.

If you choose **Edit Anyway**, you will lose any unsent modifications that you made to this project from the first computer.

Therefore, when you return to this project on the first computer, you will be warned that the project version here is no longer current, and the unsent changes can no longer be sent in to the server.

You have these options:

- Discard Changes and Quit
- **Discard Changes and re-Open:** Open the project, losing the changes.
- **Save solo Project:** If you do not want to lose the changes, your only option is to save this version of the project as a solo project. Your edits will remain, but you can no longer send them in directly to the BIMcloud. You can, however, manually paste your elements into the Teamwork project.

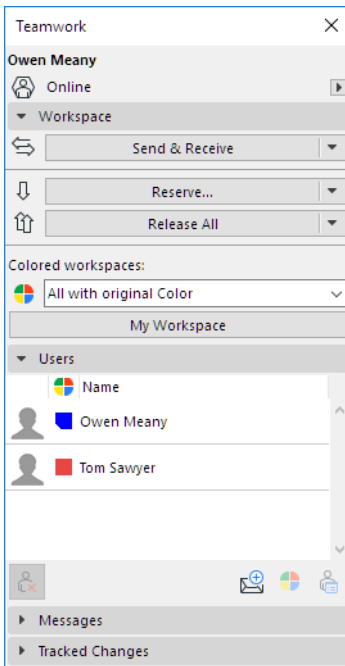
Teamwork Interface

Teamwork-related commands are available in these locations:

Teamwork Palette

- Teamwork > Teamwork Palette
- Window > Palettes > Teamwork Palette

For information on each control, see [Teamwork Palette](#).

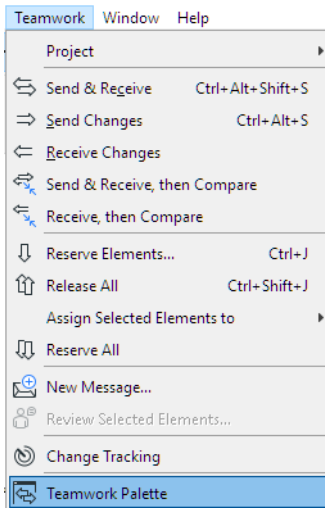


Teamwork Toolbar

- Window > Toolbars > Teamwork



Teamwork Menu



Teamwork Data Location

Choose a default **Teamwork Data** Folder location at:

- Teamwork > Project > Local Data Manager, then click Options.

[See also *Manage Local Teamwork Data*.](#)

Libraries in Teamwork

- File > Libraries and Objects > Manage BIMcloud Libraries
- File > Libraries and Objects > Library Manager

Reserving Elements in Teamwork

To modify or delete an element, attribute or other data type in a Teamwork project, you must first reserve it. Reservation occurs automatically whenever possible.

Note: Automatic reservation not available in BIMcloud Basic.

Topics in this section:

[Overview of Teamwork Reservation](#)

[Reserve Elements by Selection \(Teamwork\)](#)

[Reserve Elements by Criteria \(Teamwork\)](#)

[Reserve Library Objects \(Teamwork\)](#)

[Reservation Results \(Teamwork\)](#)

[Feedback on Element Ownership \(Teamwork\)](#)

Overview of Teamwork Reservation

Anyone can **create new** elements or data; no reservation is needed to create something new. For example, anyone can create a slab, or save a new view, and send it to the server.

However, to **modify or delete** an element, attribute, or other data type from the shared project, that element/data type must first be reserved. Reservation can be done at any time as you work in Archicad; you must be online.

New elements created by you are reserved by you automatically.

As soon as you **reserve** an element or project data, the reservation data is registered on the server, and all other users receive feedback that the element is now reserved by you; the reserved element will not be free for others to use until you **release** it.

Typically, you will reserve the elements only when you need them, then release them when you are done working on them.

Reservation status is always up-to-date as long as you are online; there is no need to send and receive to ensure that reservation data is current.

In Teamwork, you can reserve:

- 1. Elements.** There are two ways to reserve existing elements:
 - Reserve Elements by Selection (Teamwork)
 - Reserve Elements by Criteria (Teamwork)
- 2. Other Project Data.** You can reserve many Archicad data types, such as attributes and certain Navigator items, that are not elements.

[See *Reserving Teamwork Project Data \(Non-Elements\)*.](#)

Auto-Reservation of Elements

Note: Not available in BIMcloud Basic.

Whenever possible, existing elements are reserved by you automatically after you select them and begin a modification. In case it is not possible – because one or more selected elements are reserved by someone else, or if you must first Receive unsend changes – then a warning will appear telling you what to do.

The Auto-reserve function is checked by default at **Options > Work Environment > Data Safety & Integrity**.

If Auto-reserve should slow down your work, you will be prompted to uncheck this feature. You can then Reserve elements as needed, by clicking the Reserve button in Teamwork Palette or from context menus.

Conditions for Reserving

To reserve something, the following conditions must be true:

- 1.** You are online.
- 2.** The element/data type is not currently reserved by any other user.
- 3.** Your role includes access rights to the element or data type.
- 4.** The project is in updated condition on your machine.

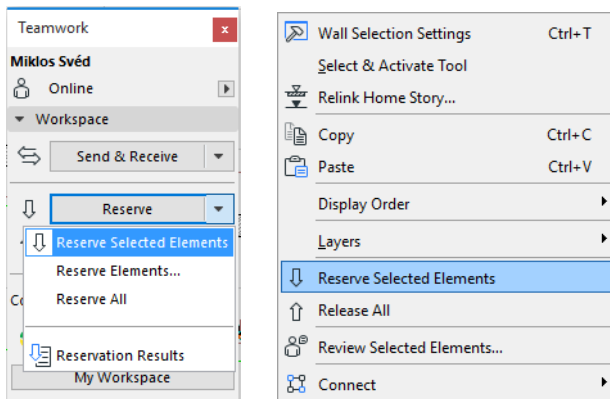
Note: If you do try to reserve elements that are not up to date on your client machine, you will be alerted that a Receive is necessary before you Reserve.

Reserve Elements by Selection (Teamwork)

Note: You must be online in order to reserve.

Follow these steps:

1. Explicitly select the element(s) which you want to reserve.
2. Click the **Reserve** button from the Teamwork Palette, or click **Reserve Selected Elements** from the window's context menu:



Note on Marquee Selection: To reserve elements by selection in Teamwork, you must explicitly select them, so that they show selection dots. Then you can use the **Reserve Selected Elements** command. Placing a Marquee is not enough, because Marquee selection does not explicitly select elements.

You can, of course, use the Marquee to help explicitly select the items you need. For example, draw a Marquee, then use **Select All (Ctrl/Cmd +A with Arrow tool selected)** to select all elements inside the Marquee.

If it turns out that some of the elements you designated could not be reserved - because of ownership conflicts - then the Reserve button changes to **Request**, and lets you send a Request message to the elements' owners, asking that they grant the element(s) to you now.

[See also *Request Elements or Project Data \(Teamwork\)*.](#)

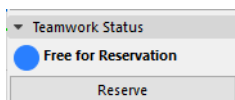
[See also *Reservation Results \(Teamwork\)*.](#)

Reserving Marker Elements: Sections/Elevations/IEs and Details/Worksheets

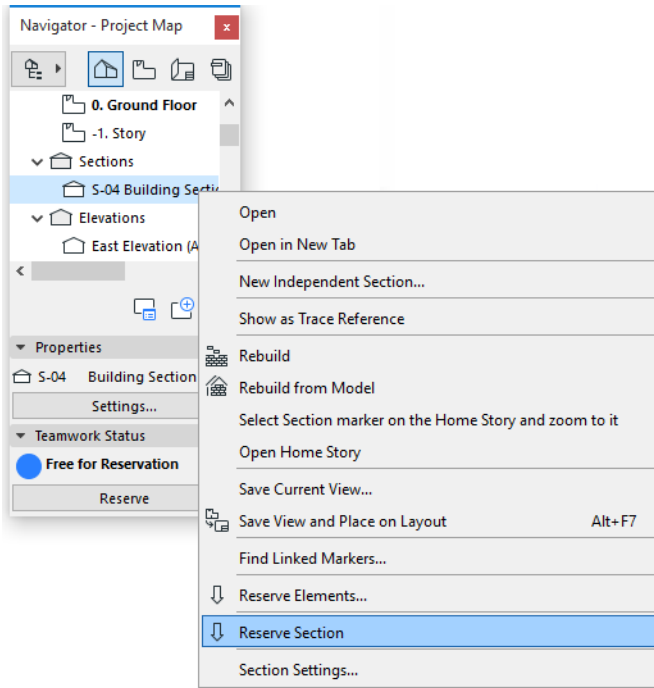
Important: Reserving a source-type marker element means that you reserve the right to adjust the marker's location and Settings. It does *not* mean that you reserve elements shown in the associated viewpoint.

To reserve a marker-type element - e.g. a Section line - select the Section marker, then reserve it using one of these methods:

- Use the Teamwork Palette's **Reserve** command
- Use the **Reserve Selected Elements** command from the window's context menu.
- Select the Section item in the Navigator and click the **Reserve** button at the bottom of the Navigator.



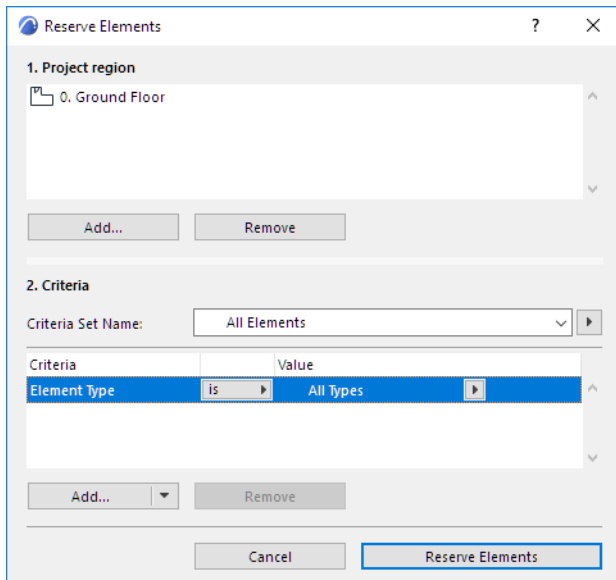
- Select the Section item in the Navigator and click the **Reserve** button from its context menu.



Reserve Elements by Criteria (Teamwork)

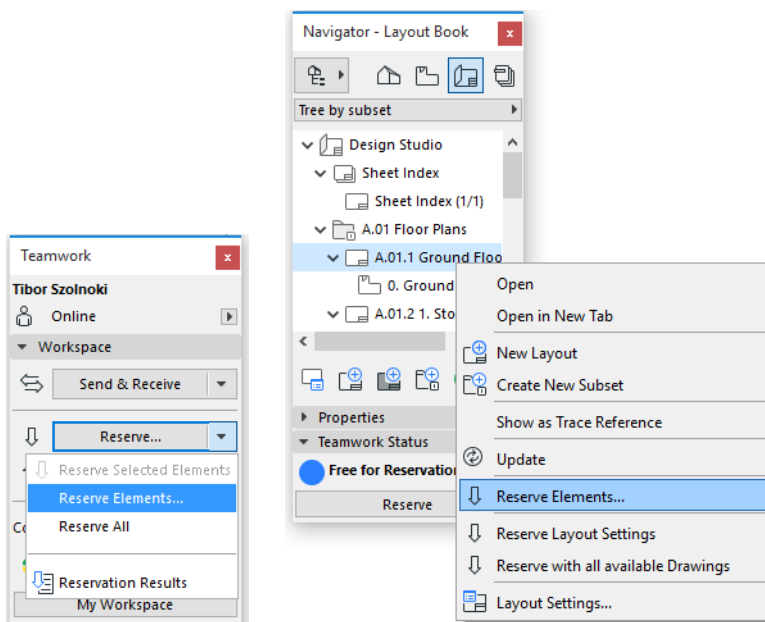
Use the **Reserve Elements** dialog box to reserve all the elements which are located in a given region of the project, and which fit the criteria you define.

For example: reserve every element on particular stories.



To bring up this dialog box, click the **Reserve Elements** or **Reserve** command from one of these locations:

- the Teamwork Palette
- the Teamwork menu
- the context menu of the active window
- the context menu of a Navigator item



In the **Reserve Elements dialog box**, you define

1. the **Project Region**; and
2. **Criteria** for which elements to reserve, within the Project Region

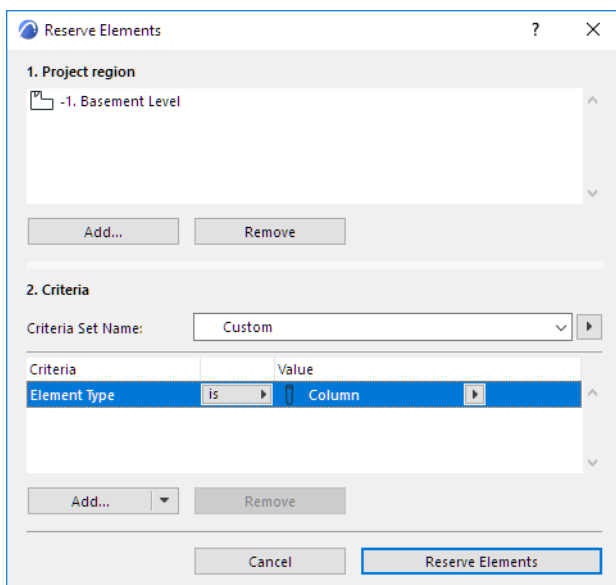
Reserving Elements by Criteria will NOT select the elements; the criteria act as a filter for reserving elements in Teamwork.

In the active window:

If you clicked “Reserve Elements” from the context menu of the active window, the current window is listed as the Project region.

In the Navigator:

If you clicked “Reserve Elements” by right-clicking a Navigator item, that item will be listed in the Project region.



- If you right-clicked the **Layout Book**, all Layouts and Master Layouts will be listed in the Project Region.
- If you right-clicked the **View Map**, all views of the Public View Map will be listed in the Project Region.
- If you right-clicked any Navigator **Publisher Set**, all views and/or layouts linked to this Publisher Set will be listed in the Project Region.
- If you right-clicked any **Navigator Group** (e.g. Subsets; Stories; Elevations), all elements of that group (e.g. all Layouts in a Subset, as in the image below) are listed in the Project Region.

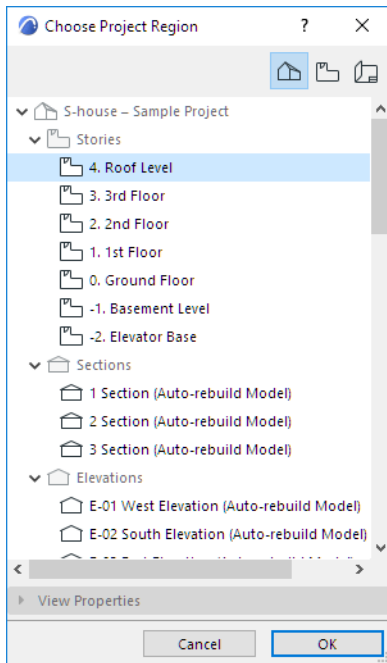
Add/Remove Items from Project Region

The Project Region can contain the following:

- Any **viewpoint**
- Any **view**
- Any **Layout** or **Master Layout**

To remove any listed item from the project region of the Reserve Elements dialog box, select the item and click **Remove**.

To expand the project region to include other viewpoints, views or layouts, click **Add**. This brings up the **Choose Project Region** dialog box, listing eligible Navigator items to add as the Project Region.



Click one or more of the Navigator items, then click **OK** to add them to the Project Region.

Define Reservation Criteria (Teamwork)

In the **Criteria** part of the **Reserve Elements** dialog box, choose the criteria with which to filter the elements that fall within the Project Region defined at the top of the dialog box.

[See *Predefined Criteria Sets*.](#)

Reserve Elements

Once you have defined the Project Region and the Criteria, click **Reserve Elements**.

The program now attempts to reserve all elements that (1) are located in the Project region you defined; and (2) fit the criteria you defined.

If this reservation attempt was unsuccessful, or only partially successful, the Last Reservation Results dialog box appears.

[See *Reservation Results \(Teamwork\)* for details.](#)

Which Elements Will Be Reserved?

- If you reserve elements on a **viewpoint** (without specifying any Layer-related criteria), the reservation will *not* take the current Layer Settings into account (this means, for example, that you may reserve elements that are currently hidden).
- If you reserve elements in a **view** (without specifying any Layer-related criteria), the program *will* take the current Layer Settings into account: only the elements on visible layers will be reserved.
- Defining a “Layer” or “Layer Combination” as a criterion means that the program will find and select the elements which are *visible* according to the given Layer Combination.
- If you reserve elements on a story, the program will reserve only those elements whose Home Story is located on that story. For example, if your Project region is Story 4, and there are roofs visible on that story - yet their Home Story is one story below - those roofs will *not* be reserved.

-
- Reserving elements from a drawing-type viewpoint, such as a Worksheet or Detail, will only reserve 2D Drawing elements - no model elements.
 - If you reserve elements from a Layout or Master Layout, the program will reserve 2D elements only: Drawings and other 2D drawing elements that have been placed on the Layouts/Master Layouts.

Reserve Library Objects (Teamwork)

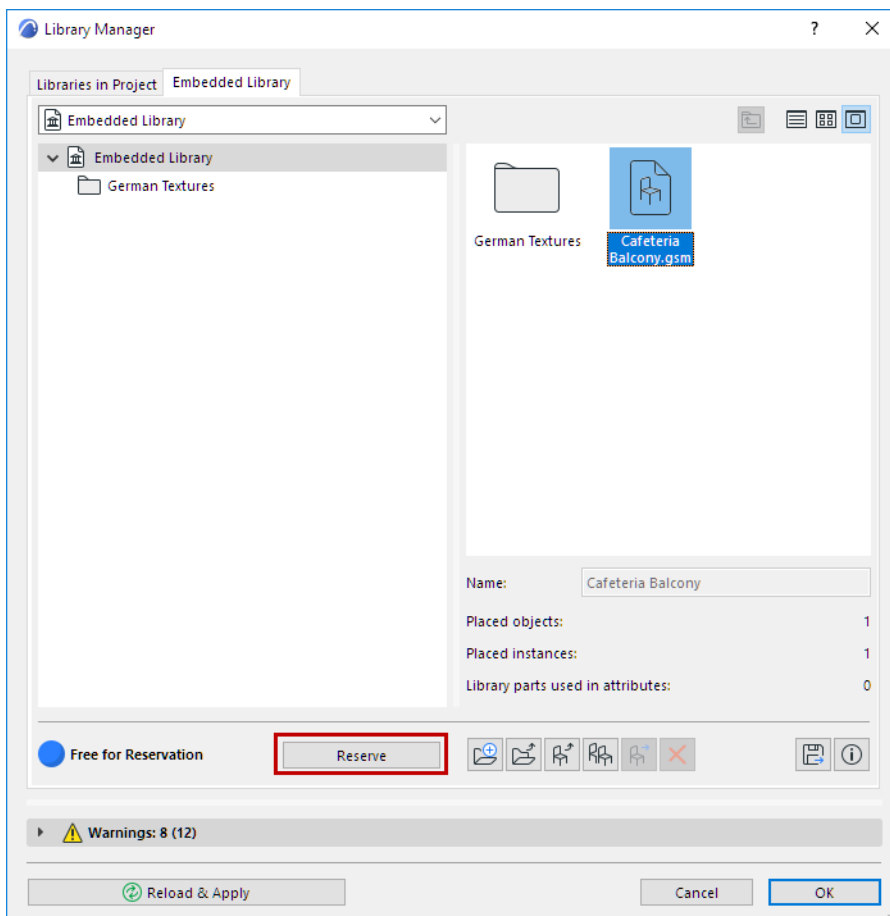
When you place a library object into a Teamwork project, the placed instance of this object is automatically reserved by you: you can graphically move or delete the object.

You can request or reserve other placed object instances, just like other elements in Teamwork.

If the object file is located in the Embedded Library, then you can also reserve this source file, independently of any of its placed instances.

To reserve an embedded object, go to the Embedded Library in Library Manager. Select the object file and click **Reserve**.

See also [Library Manager](#).



You now own the source file of the object: you can modify its parameters (use **File > Libraries and Objects > Open Objects**). Your modifications to the object's source file will be reflected in all placed instances (after Send & Receive), even those not reserved by you.

Feedback on Element Ownership (Teamwork)

The following features give you instant information on which user has reserved which data/element.

Info Tag

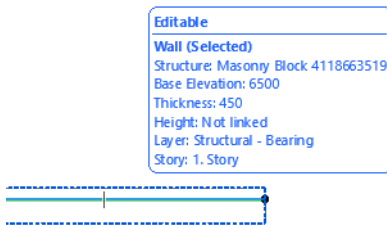
Colored Workspaces

Navigator Icons

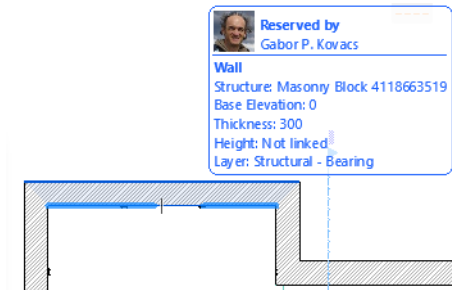
Review Your Workspace (Teamwork)

Info Tag

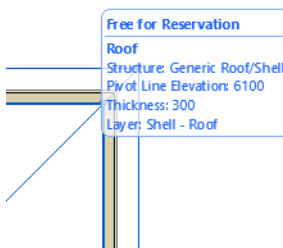
In Teamwork, the Info Tag of an element belonging to you displays **Editable**:



If an element is owned by someone else, the Info Tag indicates **Reserved by** [Owner name]:



If a previously reserved element has been released, the Info Tag indicates that it is again **Free for Reservation**:

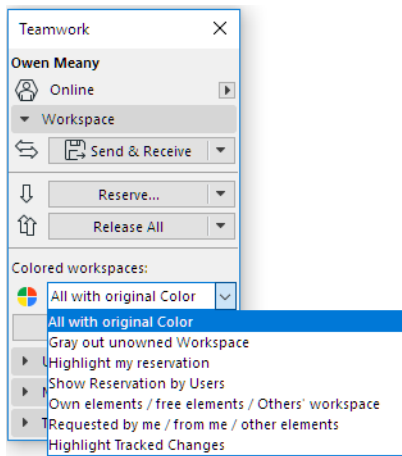


Colored Workspaces

The Colored Workspaces function, in the Teamwork Palette, provides feedback on reserved elements. This function works in the 3D window as well as 2D windows, and in the Interactive Schedule.

From the pop-up Colored Workspaces control of the Teamwork Palette (**Window > Palettes > Teamwork**), choose one of the options to help you easily identify the reservation status of project elements.

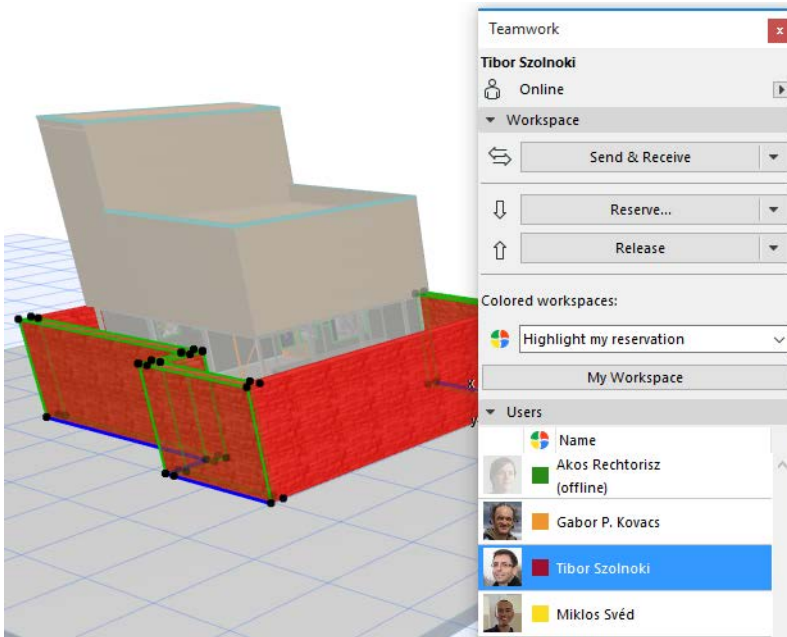
See also [Defining the Colors in Colored Workspaces](#).



See also [Colored Feedback in the Interactive Schedule Window](#).

1. Highlight my reservation

Here, the Colored Workspaces option is set to "Highlight my Reservation:" this means that your own elements are shown using the **Highlight** color; all other elements are shown in the **Passive** color:



Other options in the Colored Workspaces function:

2. **All with original Color:** This is the default option, and essentially means that no reservation-specific colors are used.
3. **Gray out Unowned Workspace:** Everything which is NOT yours is shown in the **Passive** color. (Your own items are shown in their original colors.)
4. **Show Reservation by Users:** Each User's elements are shown in the default user color, defined by the Server Administrator in User Settings.

See the [BIMcloud User Guide for details](#).

All unowned elements are shown in the **Free Elements** color.

5. **Own elements/free elements/other's workspace:** Your own elements in your own color; free elements in the **Free Elements** color; others' workspace in the **Passive** color.

- 6. Requested by me/From me/Other Elements:** Requested by me and From me are shown in those respective colors; other elements shown in the **Passive** color.
- 7. Highlight Tracked Changes:** Display a highlight on your tracked elements.

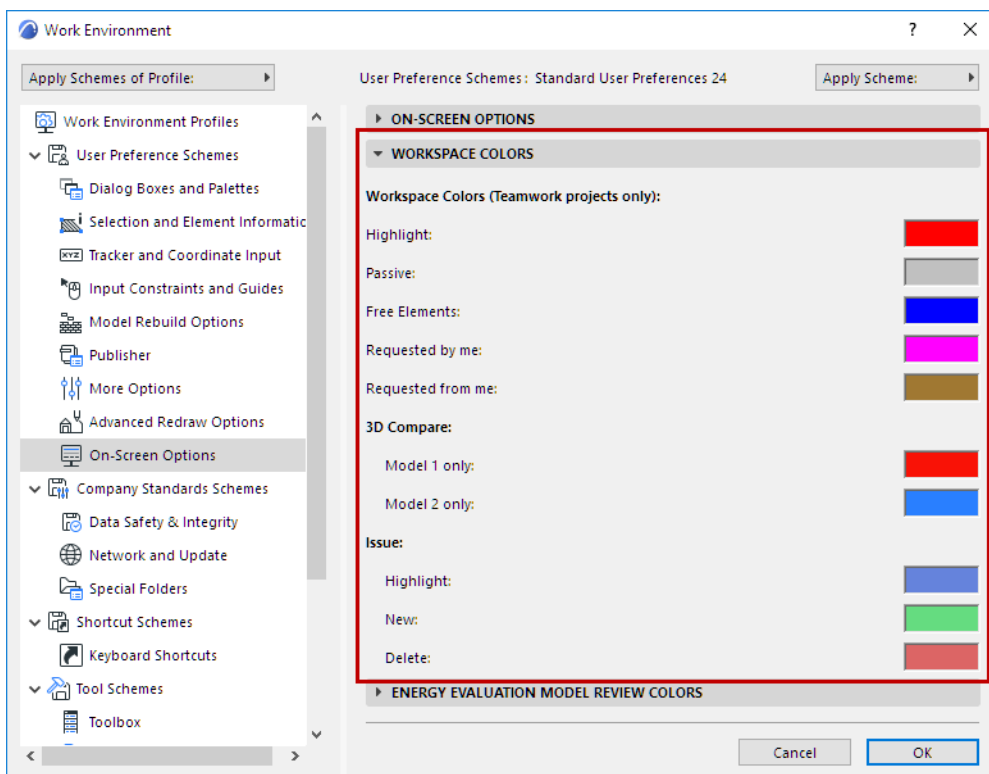
See also [Change Tracking in Teamwork](#)

Defining the Colors in Colored Workspaces

Depending on which Colored Workspace variation you choose, you will use one or more of the following colors:

- Highlight
- Passive
- Free Elements
- Requested by Me
- Requested from Me

These colors are defined in the Workspace Colors section of **Options > Work Environment > On-Screen Options**:



Redefine User Colors Locally

User colors can be helpful:

- to distinguish users of a project from each other, in the Teamwork Palette
- to distinguish their workspaces by color, if you use the Show Reservations by User option.

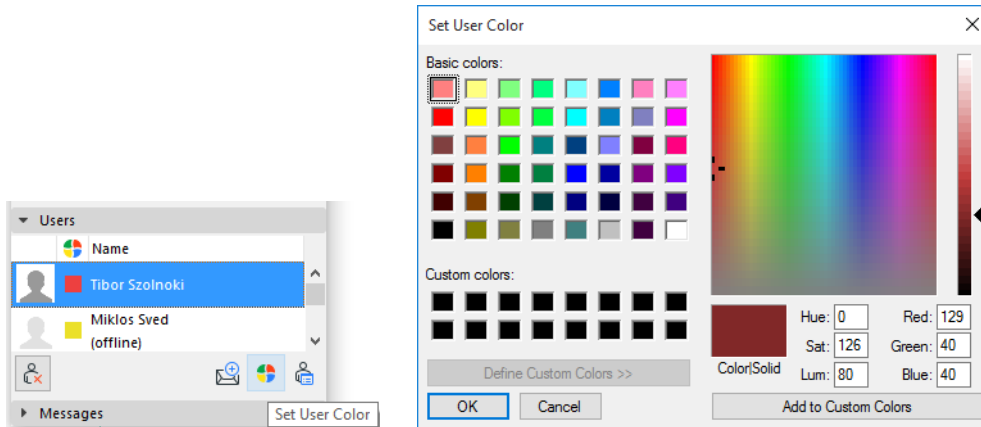
See [Colored Workspaces](#).

For example, if Bob's default color is blue, then all users will see Bob listed with a blue color in the Teamwork Palette, and will see Bob's reservations in blue when using "Show Reservations by User."

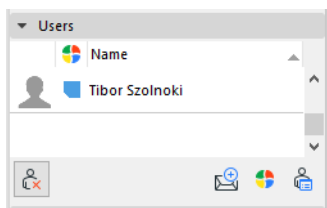
Default user colors are assigned in User Settings; any user can change his/her own default user color in User Settings.

[See the *BIMcloud User Guide* for details.](#)

Any user can also override a user color in Archicad for local purposes (the changed color will be shown on your monitor only). To change a user color, select the User Name in the Teamwork Palette, then click the color icon to bring up the color chooser palette.

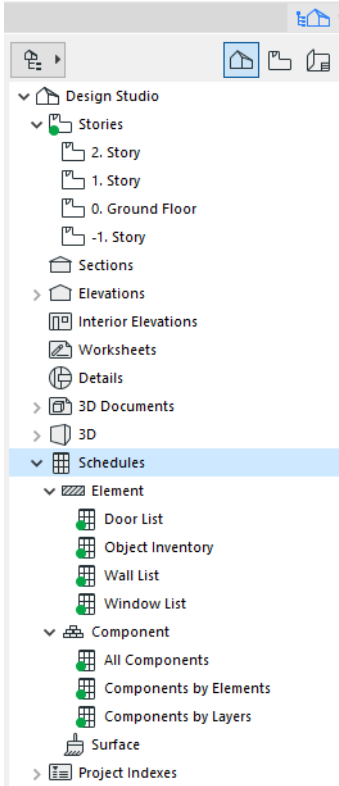


Once you change a user's color on your copy of the project, the color block in the Teamwork Palette is shown as missing a corner, to indicate that this color is different from the global default color.



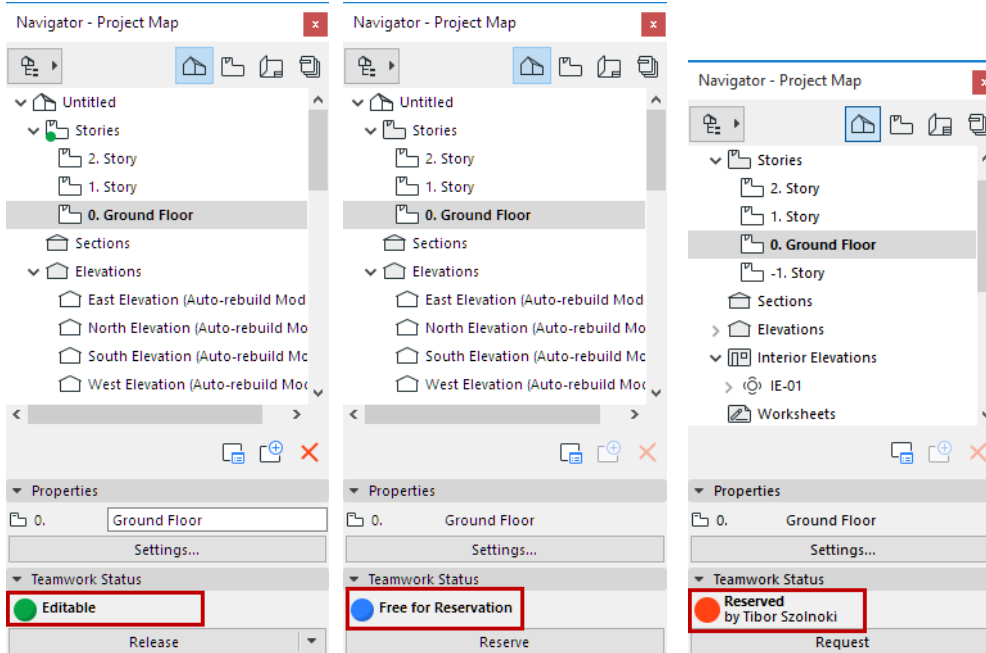
Navigator Icons

In a Teamwork project, a green dot next to Navigator items means that the item is reserved by you.

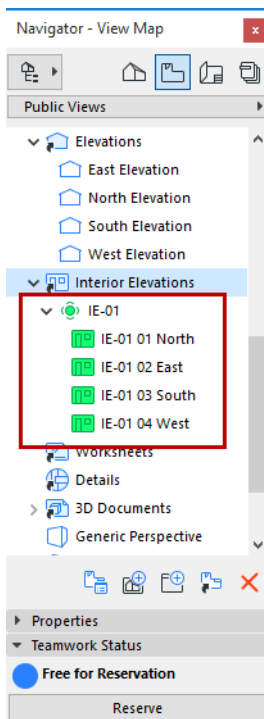


The Teamwork Status panel at the bottom of the Navigator (Palette only) also indicates whether the selected item is

- **Editable** (you can click **Release** if you no longer need it)
- **Free for Reservation** (click **Reserve** if you need it)
- **Reserved** (click **Request** to ask for it)



A newly created item that has not yet been sent to the server is displayed in green in the Navigator.



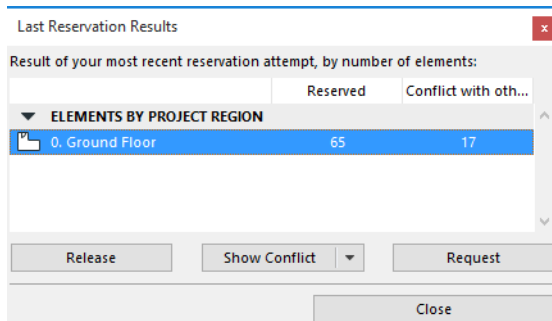
Reservation Results (Teamwork)

When you attempt to Reserve a set of elements or data in Teamwork, it may occur that some of the elements cannot be reserved by you.

Reasons for a failure to reserve elements or project data:

- They are already reserved by another user
- They have been deleted

In such cases, the **Last Reservation Results** palette appears with feedback on the results.



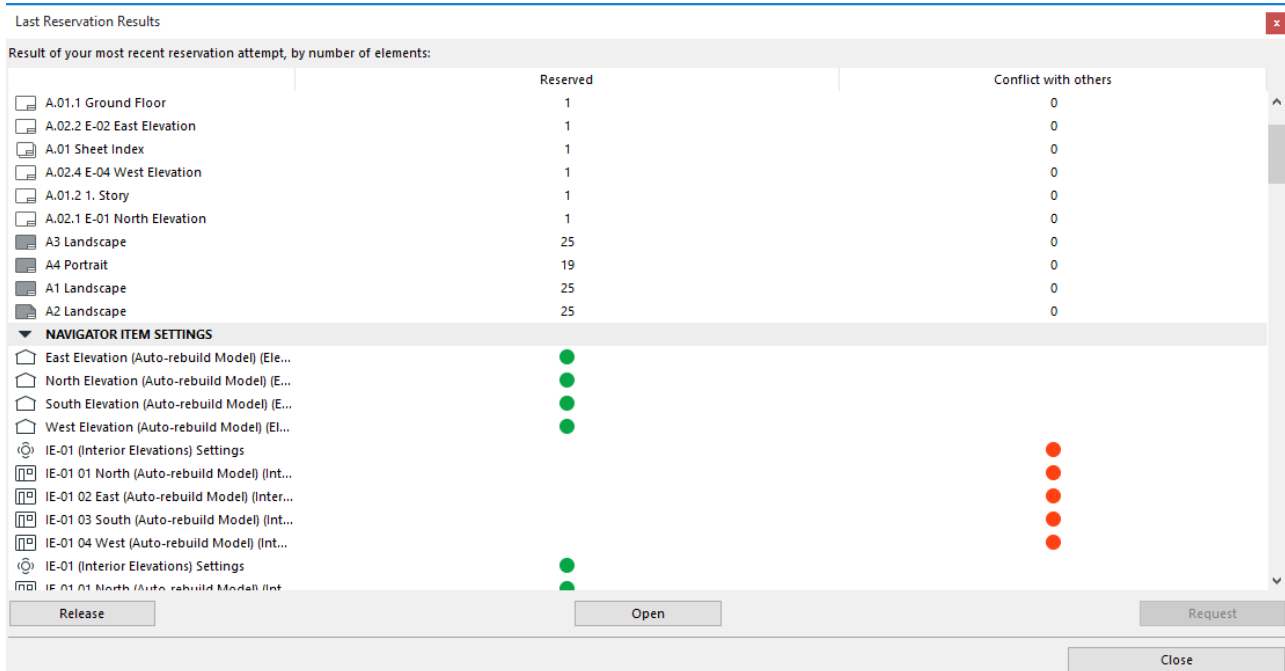
Note: The Last Reservation Results palette also appears if you click “Reservation Results” from the Reserve pop-up button on the Teamwork Palette.

This palette lists the elements (by project region) and the project data that have run into a conflict:

- **Elements by Project Region:** For every Project Region listed, shows the number of elements successfully reserved and the number of elements that were *not* reserved due to a conflict.

For each of the rest of the groups listed in the Last Reservation Results Palette, the reservation result is indicated by a green light in the Reserved column; a red light in the Conflict column; or a gray light to indicate “No Access Right.”

- Navigator Item Settings
- Attribute Types
- Library
- Other Settings

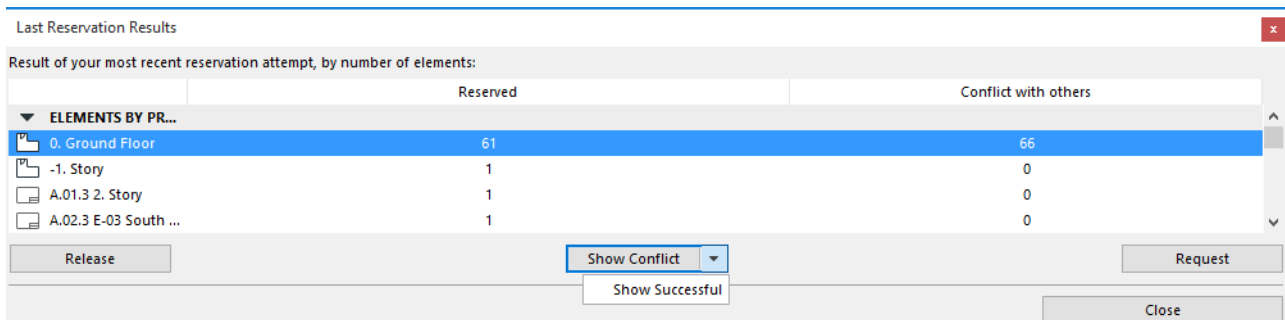


Select one or more item in the list and choose an option:

Release: Release the reservation for the selected region. Reserved elements listed here are now returned to their previous ownership status.

Show Conflict: (Available only if you have selected a single region in the list.) Archicad will open the window of the selected region, and zoom into and select the unreservable element(s).

Show Successful: (Available from the Show Conflict button's pop-up.) Archicad will open the window of the selected region, and zoom into and select the successfully reserved element(s).



Open: (Available only if you have selected a single Navigator Item or Settings item in the list). Archicad will open the selected Settings dialog box.

Request: Click this Request for all the “conflict” elements in the selected region(s). A Request message is created: click Send to send this message to the owners of all of these elements.

[See also *Request Elements or Project Data \(Teamwork\)*.](#)

Reserving Teamwork Project Data (Non-Elements)

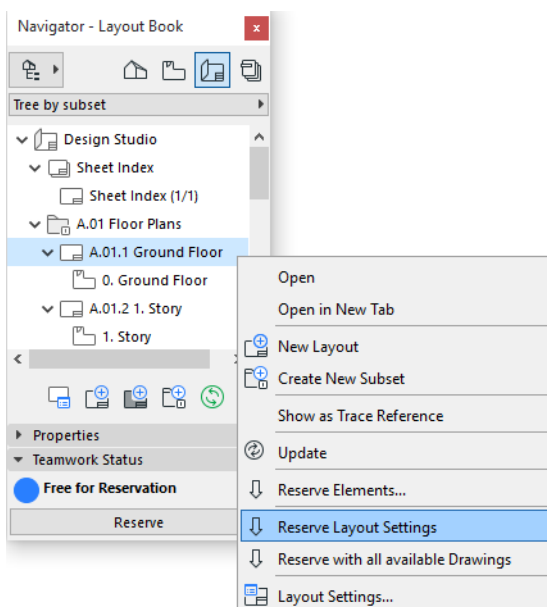
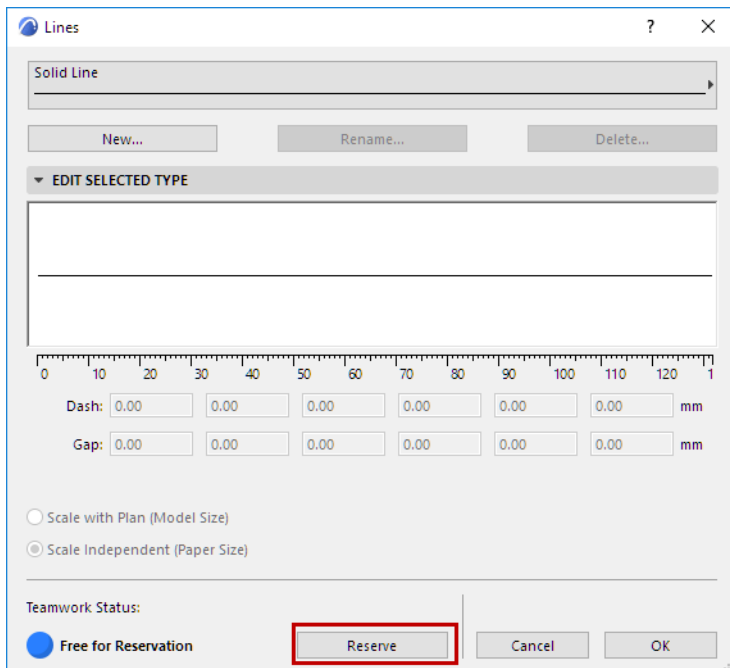
See also [Overview of Teamwork Reservation](#).

Any user, with the appropriate access rights, can create new data and send it to the server. However, to **modify or delete** shared project data, you must first reserve that data type. After reserving an attribute/project data type, the user can modify/delete items of that type.

To reserve project data types, just **go to the relevant dialog box, palette or Navigator item**, and view the Teamwork Status information at the bottom.

For Navigator items, the **Reserve** commands are also available in the context menu.

If the data is available, click **Reserve** to reserve it.



Once you have clicked Reserve in a dialog box, that data type or attribute remains part of your workspace until you release it (even if you click Cancel to leave the dialog box).

When you Release a data type, any changes you have made are sent to the server, even if you then click Cancel to leave the dialog box.

Topics in this section:

[Feedback on Ownership of Teamwork Project Data](#)

[Reservation in Attributes palette; Reserve All Attributes \(Teamwork\)](#)

[Reserve Individual Views and View Map Folders](#)

[Reserve Settings of Layout Book Items \(Teamwork\)](#)

[Moving Views and Layouts in the Navigator: Reservation Requirements \(Teamwork\)](#)

[Reserve Publisher Sets \(Teamwork\)](#)

[Reserve and Delete Layers \(Teamwork\)](#)

[Reserve Find and Select Criteria \(Teamwork\)](#)

[Reservation in Interactive Schedules \(Teamwork\)](#)

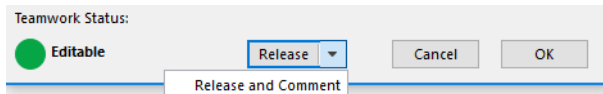
[Reserve List Scheme Dialog Boxes \(Teamwork\)](#)

Feedback on Ownership of Teamwork Project Data

At the bottom of each dialog box (or at the bottom of each Navigator map), the Teamwork Status “control light” is color-coded to indicate the current status of that data type:

- **Blue - Free for Reservation.** The **Reserve** button is available. (Clicking Reserve also means that the reserved data type will automatically be synchronized with the server to ensure it is up to date.)
- **Green - Editable.** This data type belongs to you; you can modify or delete these data as needed, or **Release** them using the button to the right. Changes are sent to the server.

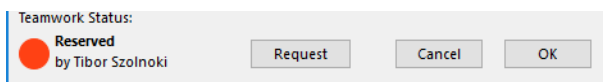
There is an additional option for Releasing:



Release and Comment. The data is released, and the Release event is added, in the form of a comment, to the Activities panel of the BIMcloud. You also have the option to send a message at this time.

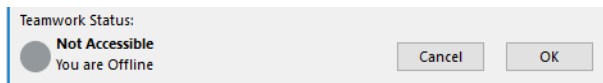
[See Add Comment to Activities \(Teamwork\).](#)

- **Red - Reserved** (by user name). The Request button is available.

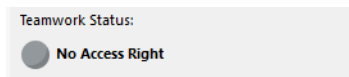


[See Request Elements or Project Data \(Teamwork\).](#)

- **Gray - Not Accessible.** (The server is offline.)



- **Gray - No Access Right:** You have no access rights at all regarding an attribute category (neither “Create” nor “Modify/Delete”).

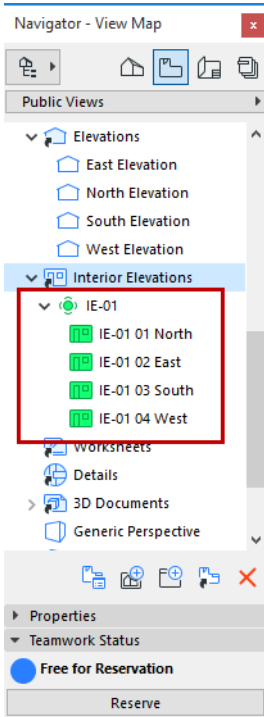


All users joined in to the project see these control lights, which are always up-to-date. (For example, if you reserve the Layer Settings dialog box, your control light turns green; at the same instant, the Layer Settings control lights of the other users turn red.)

A list of the Project Data owned by you is included in the **My Workspace** pop-up of the Teamwork Palette.

[See Review Your Workspace \(Teamwork\).](#)

A newly created item that has not yet been sent to the server is displayed in green in the Navigator.

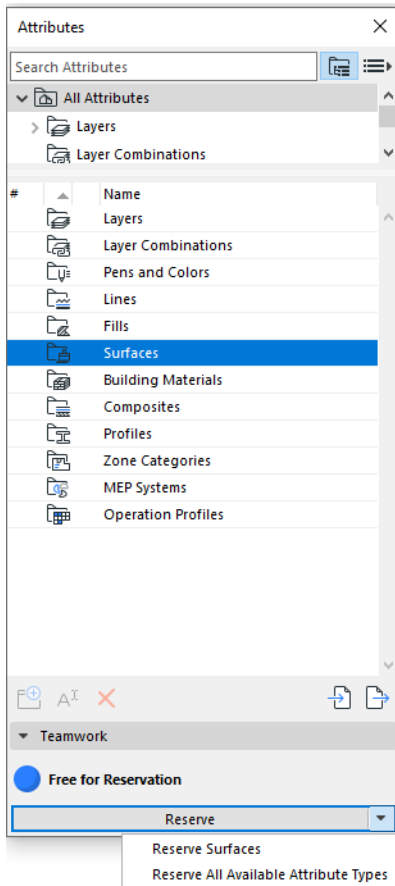


Reservation in Attributes palette; Reserve All Attributes (Teamwork)

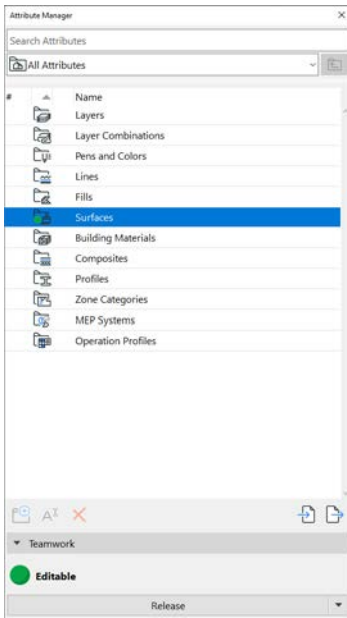
Like other dialog boxes in Teamwork, the Attributes palette has a control light system.

To reserve a particular attribute type:

1. Select its folder in the Attributes palette
2. Click one of the following commands from the bottom of the dialog:
 - **Reserve** to reserve the corresponding attribute settings dialog.
 - **Reserve all available attribute types** to reserve all available settings dialogs.

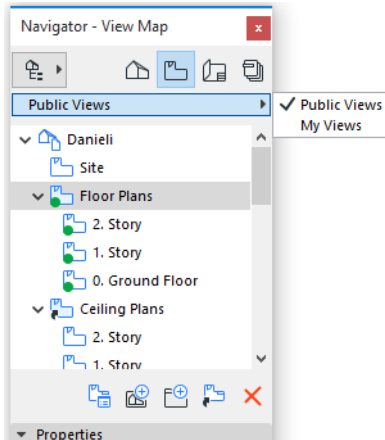


Attributes reserved by you display a green control light on their folder in the Attributes palette, as well as on their settings dialog.



Reserve Individual Views and View Map Folders

In a Teamwork project, there are two View Map modes: Public Views and My Views. Shared views must be in the Public View Map. To switch between View Map modes, use the pop-up at the top of the Navigator Palette View Map:



For more information, see [My Views](#).

Reserve a View

Reserving a **view** means that you reserve the right to:

- modify the View Settings
- delete the view
- move the view in the View Map hierarchy

Note: To move a view, you must also reserve the view's source folder and its target folder.

See: [Moving Views and Layouts in the Navigator: Reservation Requirements \(Teamwork\)](#)

In Public Views, use context menus in the Navigator to reserve the views that you need: right-click the view, then choose **Reserve View Settings**. Alternatively, select the view and click **Reserve** at the bottom of the View Map.

➤ Required permissions:

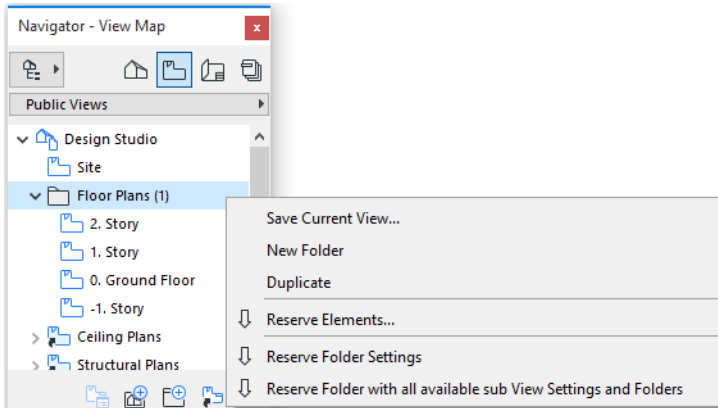
- **Views and Folders - Delete/Modify**

Reserve All Available Views

To reserve all available views in a View Map, or one of its folders, right-click the View Map folder, then choose **Reserve with all available sub View Settings and Folders**.

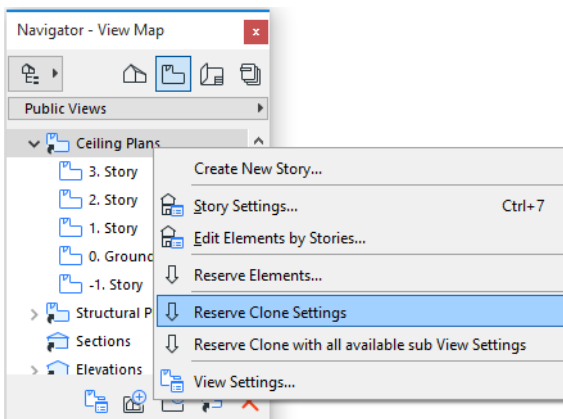
If the selected folder itself is available too, the command is: **Reserve Folder with all available sub View Settings and Folders**. (This will reserve the selected folder, plus all available items that it contains.)

Available items are those not currently reserved by any other user.



Reserve a Clone Folder

To reserve a Clone Folder, select it and choose **Reserve Clone Settings** from the context menu.

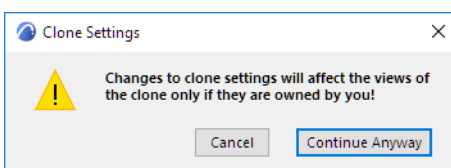


Reserving a clone folder thus allows you to alter the clone folder settings. (As a result, any new views added to the clone folder will take on these settings.)

[See *Clone a Folder in the View Map*.](#)

However, alterations to the clone folder settings will only affect the views in that clone folder which you have already reserved. Settings of cloned views that are owned by others will not change even if you change the clone folder settings.

Note: If you change clone folder settings but have not yet reserved all views in that cloned folder, you receive a reminder to this effect.



My Views

In addition to the Public View Map, you can use the local “My Views” mode in the Navigator Palette View Map. Views in “My Views” are visible and accessible only by you - they are not shared, and they cannot be sent to the BIMcloud with the **Send** command.

To use “My Views,” just click “My Views” from the pop-up at the top of the Navigator Palette View Map.

To add a view to My Views, make sure that the View Map in the Navigator Palette is in My Views mode. Now, whenever you create a new view, it will be placed into My Views.

Alternatively, set up the Organizer with Public Views on one side and My Views on the other, and copy views from one to the other as needed.

Reserve Settings of Layout Book Items (Teamwork)

Note: To reserve the Settings of a Layout, Master Layout or Subset, you must be assigned the role: **Layouts and Subsets - Delete/Modify**. If you do not have this role, the corresponding commands are not available.

Reserve Layout Settings

Use context menus in the Navigator to reserve the Layouts you need: right click the Layout, then choose **Reserve Layout Settings**. (The same applies to Master Layouts.) Alternatively, select the Layout and click **Reserve** at the bottom of the Layout Book.

Reserving a Layout's Settings means that you reserve the right to:

- modify Layout Settings
- delete the Layout
- move the Layout within the Layout Book structure

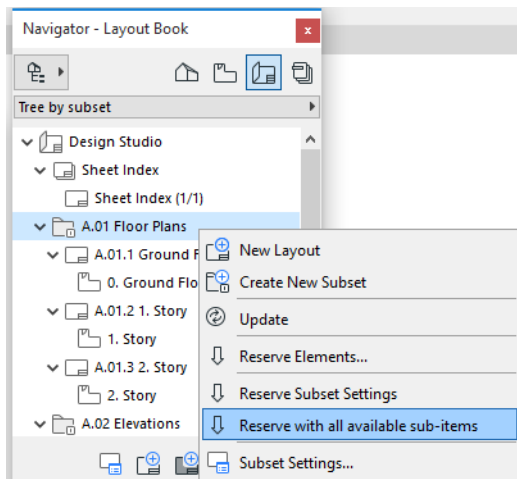
Note: To move a Layout, you must also reserve the Layout's source folder and its target folder.

See: [Moving Views and Layouts in the Navigator: Reservation Requirements \(Teamwork\)](#)

To reserve Settings of **all available Layouts** in the Layout Book, or all Layouts in one of its Subsets or in the Master Layouts folder, right-click the Layout Book name (or the Subset/folder), then choose **Reserve all sub-items**.

If the selected folder itself is available too, the command is: **Reserve with all available sub-items**. This will reserve the settings of the selected folder, plus the settings of all available items that it contains.

Available items are those not currently reserved by any other user.



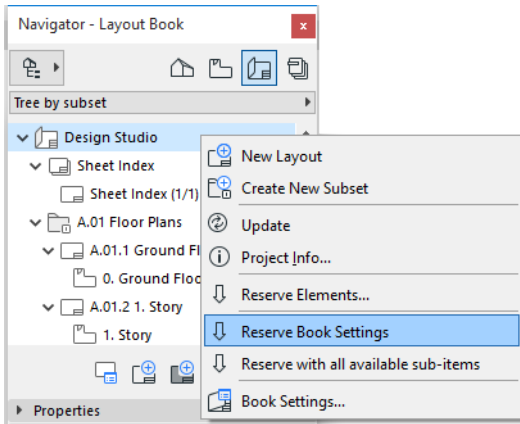
Reserve Subset Settings

Use context menus in the Navigator to reserve the Subsets that you need: right click the subset, then choose **Reserve Subset Settings**. This gives you the right to modify Subset Settings.

Note: Modifications to Subset Settings will affect all Layouts in the Subset, even those owned by other users.

Reserve Layout Book Settings

To reserve Book Settings, right-click the project name at the top of the Layout Book tree structure and click **Reserve Book Settings**.



Reserving the Layout Book gives you the sole right to edit Book Settings.

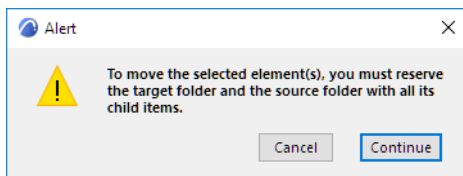
Moving Views and Layouts in the Navigator: Reservation Requirements (Teamwork)

You can use Drag and Drop to rearrange views and Layouts in the Navigator, and to assign a Master Layout to a subset or Layout.

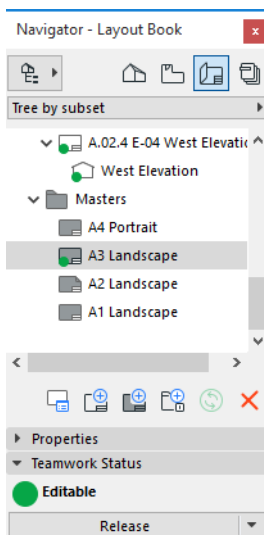
In a Teamwork project, you can only do this if you have reserved:

- the item itself (the one you are moving)
- all the sub-elements of item(s) you are moving
- the target folder to which you are moving the item(s).

If you are missing any of these reservations, you will get the following alert:



If you click **Continue**, the required reservations are carried out automatically (if this is possible).



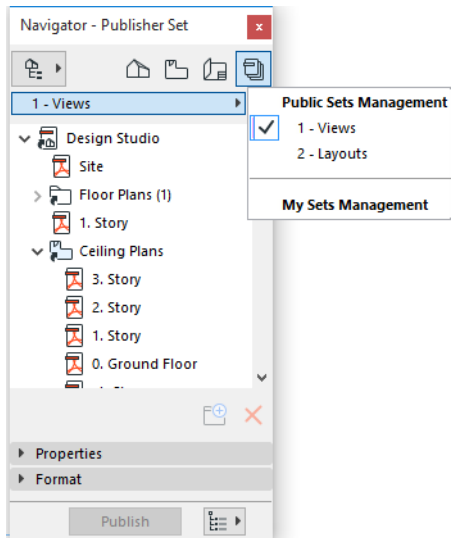
If it is not possible to reserve the needed items (because they are reserved by other users), you are alerted to this.

Note that if you drag a Master Layout onto a Subset in Layout Settings, the automatic reservation (if available) will reserve every sub-element in that Subset (all Layouts and all Drawings).

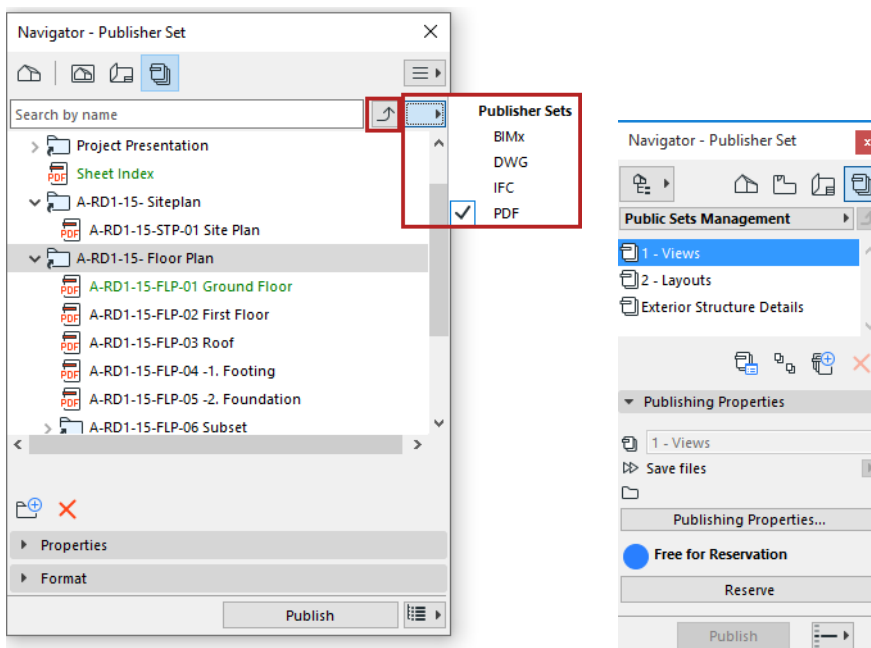
Reserve Publisher Sets (Teamwork)

To reserve a Publisher Set, open the Publisher Set page from the Navigator Palette.

In a Teamwork project, there are two Publisher Set modes: Public Sets and My Sets. Shared Publisher Sets must be placed in the Public Sets list.



Since there can be multiple public Publisher sets, you must specify which one you want to reserve. Navigate to **Public Sets Management** (use the “One Level Up” button) to see the list of public Publisher Sets.



From the **Public Sets Management** list, choose the desired Publisher Set. Click **Reserve** at the bottom of the Publisher Set page to reserve it; or click **Reserve** from the context menu.

Reserve and Delete Layers (Teamwork)

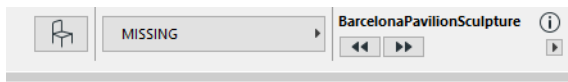
To assist in your design work, you can change layer states and create new Layers or Layer Combinations, even if the Layer Settings attribute has been reserved by someone else.

However, to save and share these changes to the project, or to delete layers, you must reserve Layer Settings.

To reserve Layer Settings, open the Layer Settings dialog box, and click Reserve.

If you delete layers, any of your own elements located on those layers are also deleted.

However, elements on deleted layers are NOT deleted if they are owned by other users. Such elements will be identified as having “MISSING” layers. These elements will be displayed on all outputs; their visibility cannot be changed.



If you find that your elements are located on a deleted (“MISSING”) layer, you should either delete these elements or move them onto another layer.

To easily identify which of your elements are on “MISSING” layers, just hide all project layers: elements whose layers are missing (plus any elements on the Archicad Layer) will be the only elements that are still displayed.

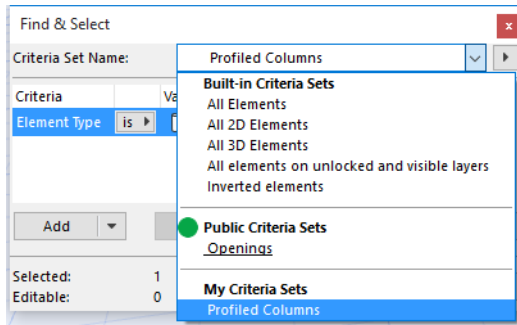
[See also *Delete attributes*.](#)

Reserve Find and Select Criteria (Teamwork)

In a Teamwork project, the Find & Select Palette contains a number of built-in Criteria Sets.

[See *Find and Select Elements*.](#)

Additional Criteria Sets customized by you can be stored either locally (**My Criteria Sets**) or shared and reserved (**Public Criteria Sets**). Click the Criteria Sets pop-up to see which Criteria Sets are available to your project:



The “control light” (green, blue or red) lets you know whether Public Criteria sets are Reserved by you, Available for reservation, or Reserved by someone else.

Use any of these Criteria Sets to Find & Select elements in your project, even if the Public Criteria Sets are reserved by someone else.

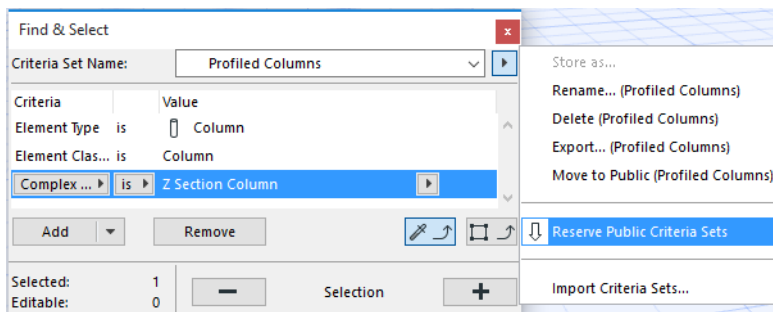
These same Criteria Sets are available in the **Reserve Elements** dialog box.

[See *Reserve Elements by Criteria \(Teamwork\)*.](#)

While any user can use the criteria sets regardless of ownership, you must reserve Public Criteria Sets in order to

- Store a custom Criteria Set as a Public Set
- Overwrite an existing Public Criteria Set
- Delete a Public Criteria Set

To reserve the project’s Public Criteria Set, click this command from the pop-up at the top right of the Find & Select dialog box.



Reservation in Interactive Schedules (Teamwork)

Reserve Schemes for Interactive Schedules

Reserve Elements from IS List Window

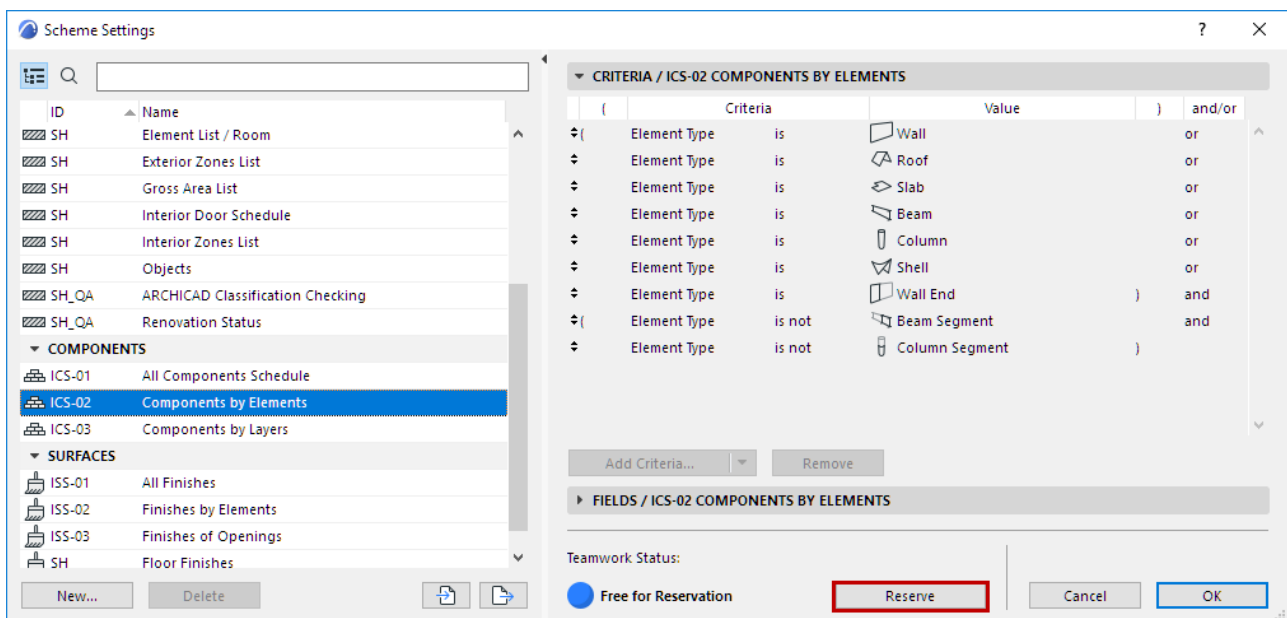
Reserve All Elements of a Schedule

Colored Feedback in the Interactive Schedule Window

Reserve Schemes for Interactive Schedules

Do one of the following:

1. Open Scheme Settings.
2. Select the scheme you wish to reserve.
3. If it is free for reservation, click **Reserve**.



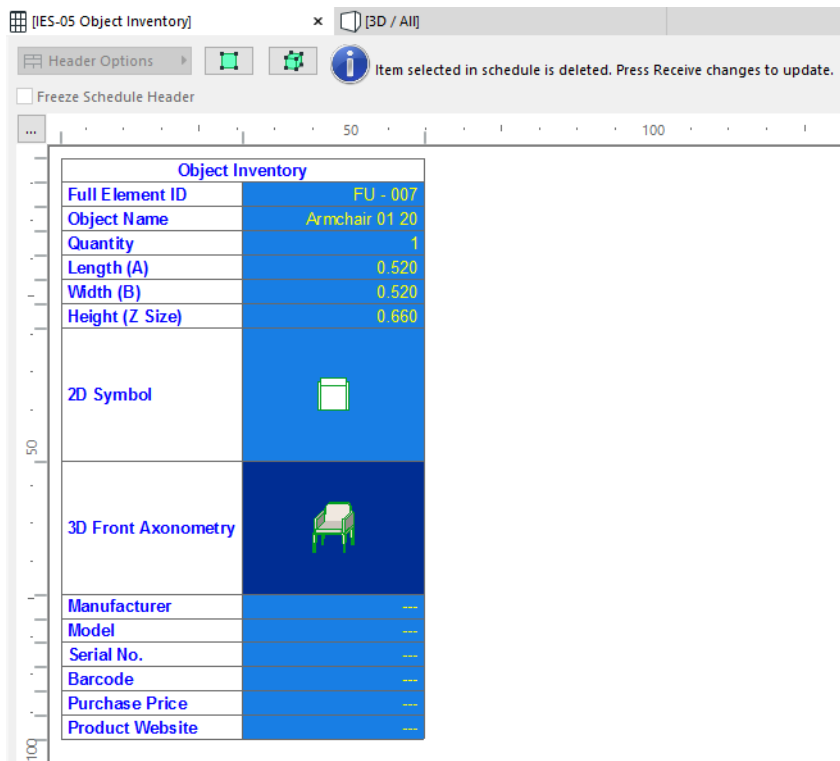
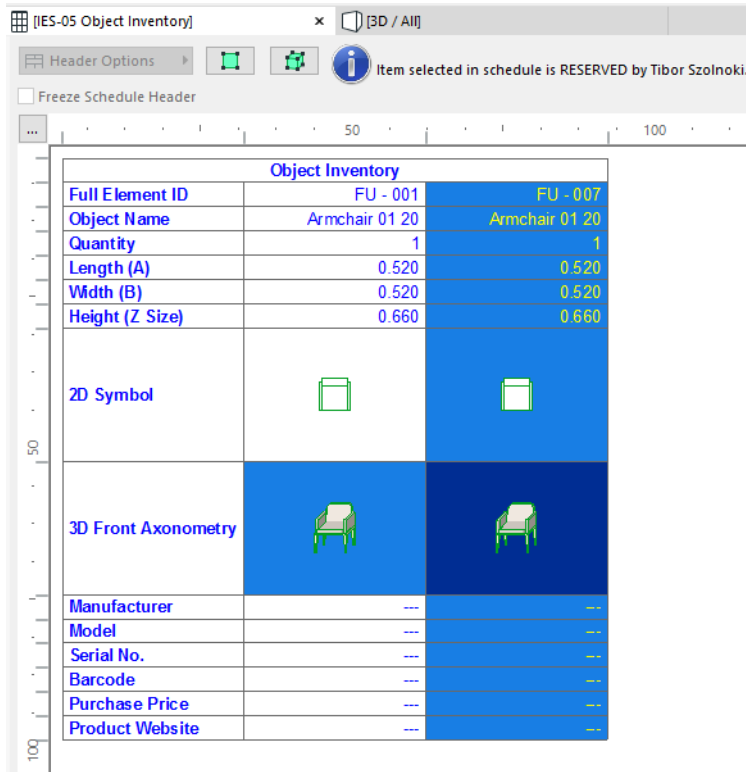
Alternatively:

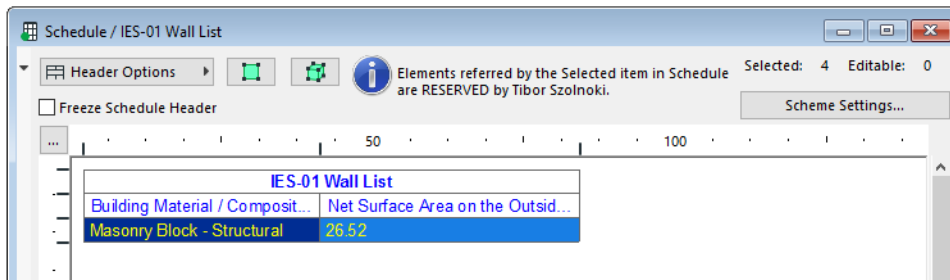
1. Open the Interactive Schedule window.
2. At the bottom left corner, click **Reserve**.

Reserve Elements from IS List Window

You can reserve any elements directly from the Interactive Schedule window.

1. In the Interactive Schedule window, select the element(s) you wish to reserve.
2. A message at the top of the window provides feedback on the elements' reservation status.





“Elements referred by the selected item” means that the selected item does not correspond to a single element. In the picture above, the single element selected in the schedule (Masonry Block) actually refers to four different elements, as indicated by the “Selected: 4” information at the top. Thus, one or more of these four walls is reserved by another user; the Teamwork Palette shows the **Request** command.

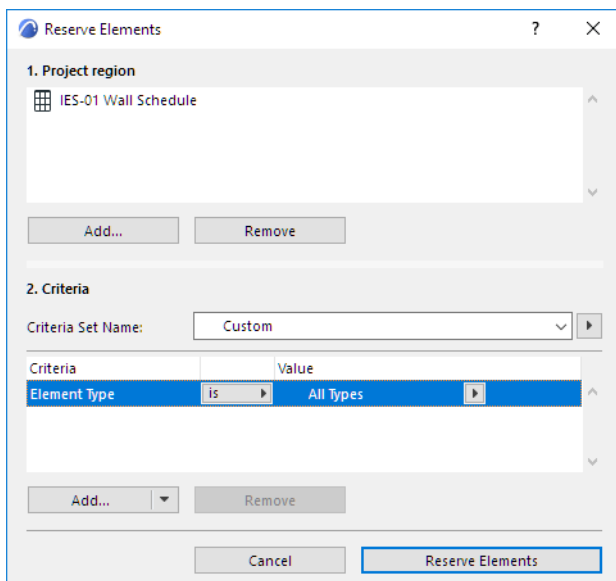
3. If the element is free for reservation, click **Reserve** from the Teamwork Palette. (If it is reserved by another user, you can **Request** it.)

Even if the selected element is free for reservation, or if you have already reserved it, it may still not be possible for you to edit it. In this case, the Interactive Schedule window will provide information about the problem, as in these cases:

- if the element’s layer is locked or hidden. You must unlock or unhide the layer before you can edit the element;
- if you do not have the Access Right needed to edit the element
- if the element is not up to date in your workspace. You must Receive changes before you can edit it.

Reserve All Elements of a Schedule

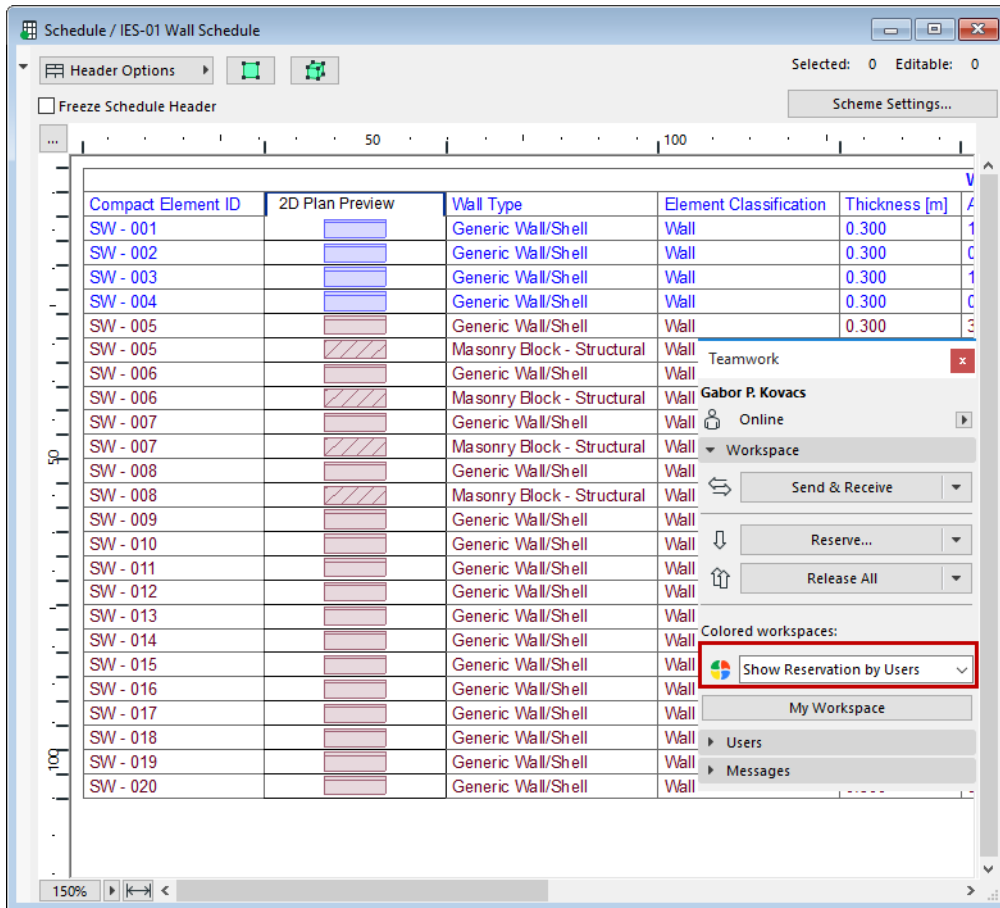
1. Open the Interactive Schedule window.
2. From the Teamwork Palette, click **Reserve Elements** to open this dialog box.
3. The Project Region section contains your Interactive Schedule. In the Criteria section, ensure that “All Types” is chosen as the criterion, then click **Reserve Elements**.



See also [Reserve Elements by Criteria \(Teamwork\)](#).

Colored Feedback in the Interactive Schedule Window

You can use Colored Workspace options to indicate element ownership. (For example, you can show each schedule item is shown in the color of the user who has reserved it, by switching on “Show Reservation by Users” in the Teamwork Palette.)



If a selected schedule item is free for reservation, you can reserve it from the Teamwork Palette, using “Reserve.”

Reserve List Scheme Dialog Boxes (Teamwork)

The following functions of the Calculation List feature can be reserved using the control light interface:

- Set Up List Schemes

Note: List Schemes will function in Teamwork projects only if you save them to the project's Embedded Library.

- Edit Database
- Link Properties to Criteria

For details on the Calculation List feature, see [Calculation \(Legacy\)](#).

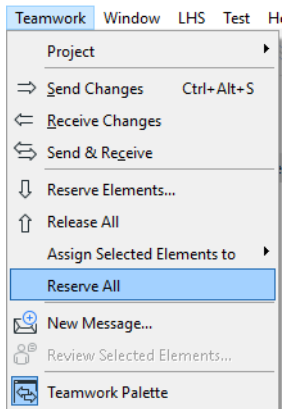
Reserve All Elements and Data (Teamwork)

The **Reserve All** command will attempt to reserve all existing elements and project data in the currently opened Teamwork Project.

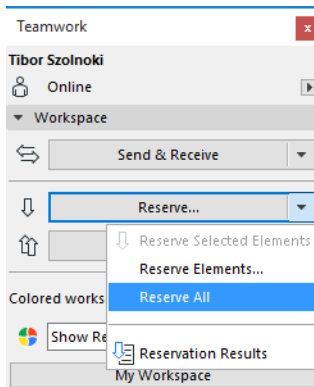
This function can be useful if you expect to be the only user on a project for a given period of time.

The **Reserve All** command is available:

- From the Teamwork menu



- From the Teamwork Palette “Reserve” pop-up:

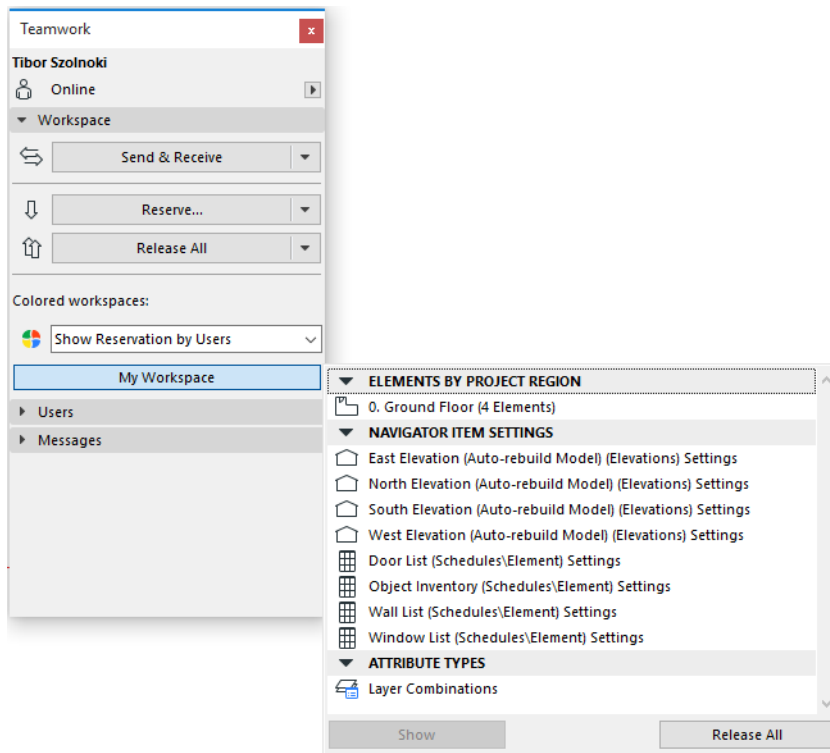


If the attempt to Reserve All runs into conflicts, the Reservation Results dialog box appears.

See also [Reservation Results \(Teamwork\)](#).

Review Your Workspace (Teamwork)

For an overview of all project items (both elements and non-elements) reserved by you, go to the Workspace panel of the Teamwork Palette and click the **My Workspace** button:



Select any item on this Workspace pop-up and click **Show** (to zoom to elements) or **Open** (to open the relevant dialog box), depending on the item you have selected.

Click **Release** if you are ready to release the selected item.

Related Topics:

[Feedback on Element Ownership \(Teamwork\)](#)

[Feedback on Ownership of Teamwork Project Data](#)

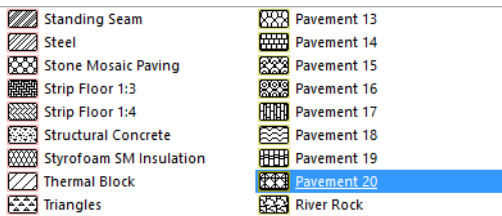
Creating and Sending Project Data (Teamwork)

Create New Data

Regardless of reservation status, any user having the necessary access rights can create and store new project data, and you can modify such data locally.

For example, any user can create a new Fill Type even if he/she has not reserved the Fills dialog box.

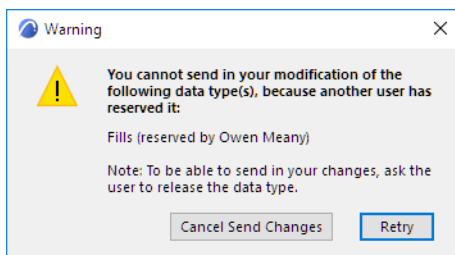
If you create a new attribute or other data type, you are free to modify and delete it. Until the first time you issue the **Send** command, the name of the newly created data type is shown as underlined in the dialog box (like Pavement 20 in this image), indicating that no other users are able to see or use this attribute.



Send New Data

If you create a new attribute or other non-element data type, use **Send** to send it to the server.

If, at the time you want to send changes, the data type is reserved by another user, you will be warned that you cannot send in any changes:



To resolve this situation, you can:

- Get the owner of the data type to release it right now, then **Retry**
- Cancel, then reserve or request the data type (e.g. Fills Dialog Box), then **Send** again
- Delete the new data (e.g. Pavement 20), then Send again

New Data in the Navigator

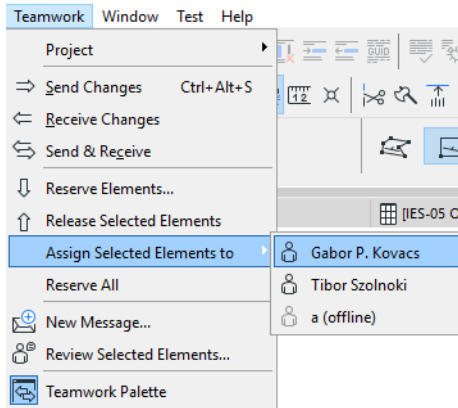
When you create a new Navigator item (e.g. a new view), its Settings are automatically reserved by you. A newly created item that has not yet been sent to the server is displayed in green in the Navigator.

- **Required permissions to create a new view or Public View Map Folder:**
 - **Views and Folders - Delete/Modify**
- **Required permissions to create a new Layout, Subset or Master Layout:**
 - **Layouts and Subsets - Create**

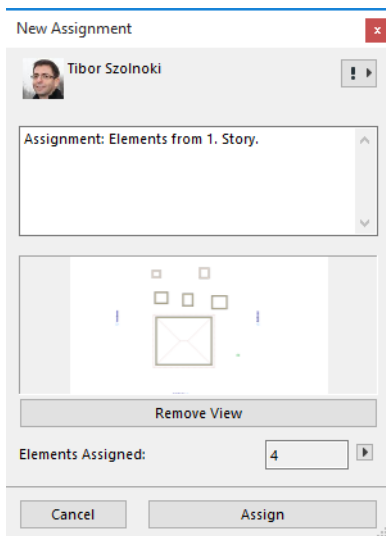
Assign Elements (Teamwork)

You can assign any of your own elements to another user, even if that user has not requested them.

To do this, select the element(s), then the **Assign Selected Elements to** command from the context menu or the Teamwork menu. Choose the user to whom you will assign the elements.

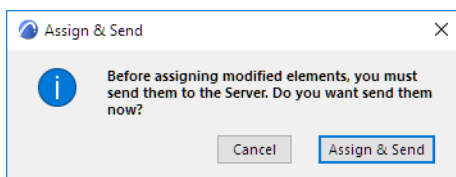


In the appearing dialog box, add any message, then click **Assign** to complete the operation.

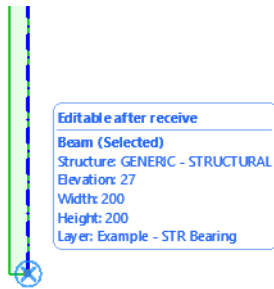


The recipient is automatically notified of this assignment via a message.

If the assigned element is not in updated condition, you are prompted to Send your changes to the server first.



It is possible for you to be assigned an element that is not yet in updated condition on your client machine. In this case, you cannot edit your new acquired element until you do a Receive. This is indicated in the element's Info Tag:



Related Topic:

[Assign Issue to User](#)

Request Elements or Project Data (Teamwork)

Use Request if the element you wish to reserve is currently owned by another user.

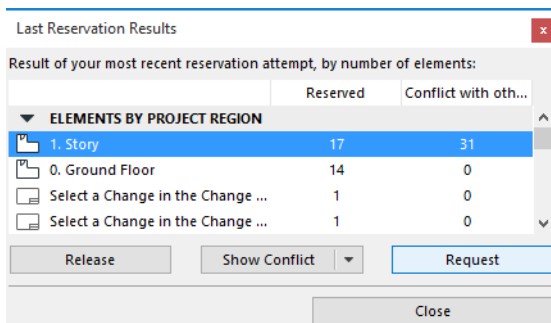
You must be on-line to Request.

The **Request** command is available from the following locations:

- in the Teamwork Palette
- in the context menu of selected Navigator items
- in the context menu of selected elements
- in the dialog box of project data (for example, Model View Options)
- from the Last Reservation Results dialog box.

See [Reservation Results \(Teamwork\)](#).

Clicking the Request button here means you request the “Conflict” items from their owners.



Click **Request**.

The **Ownership Request** dialog box appears, which includes:

- an automatically generated message about the requested items and their location
- an attached model view, which includes camera position, cutting planes and layer settings as applicable
- the number of requested elements. You can still change this set of elements before sending the message: reselect elements, then update the request message by clicking the pop-up next to the number of elements currently attached
- a preview of this attached view

If you don't want to include this view, click **Remove View**.

You can add and modify the text as needed; then click **Request**.

See also [Request Elements or Project Data \(Teamwork\)](#).

Whenever you send a Request message, it is filed in your Pending Requests list.

Concurrently, the Request message is delivered to the owner's To Do list.

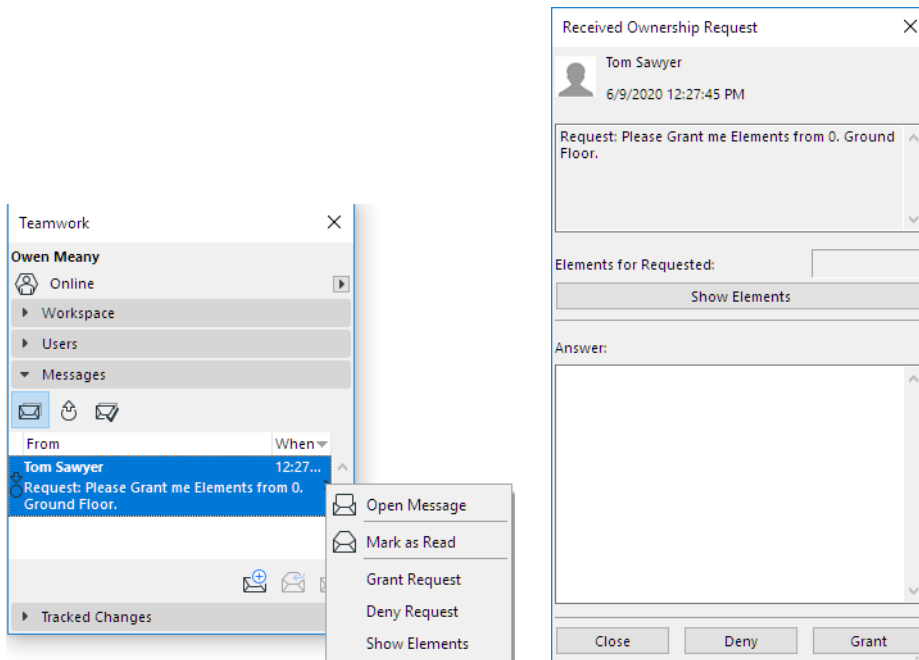
If the owner is not currently on-line, the Request will be delivered as soon as the user goes on-line.

If the owner releases the requested item independently of the Request message, the requesting user automatically receives the item.

If you open one of your Pending Requests, you get a status report: how many of the requested items have been granted/denied and how many are still pending. You can opt to revoke a pending request: click **Revoke Request**.

Received Ownership Request

The owner receives the message; the user can respond to the request via the Teamwork palette, by clicking the triangle to show relevant commands, or double-clicking it to open the request:



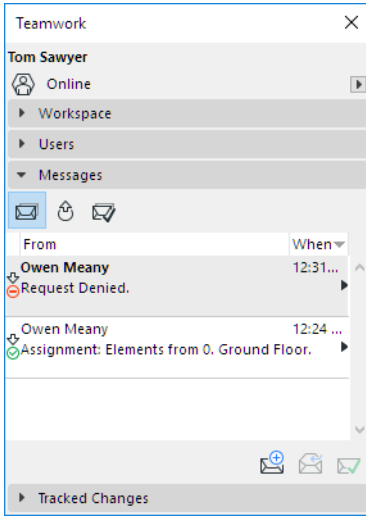
- **Open Message:** Same effect as double-clicking the item, or clicking on the fly-over message notice.
- **Mark as Read:** Unopened messages are listed in bold typeface. You can remove the bold typeface using the **Mark as Read** command. (Double-clicking the message to open it will also mark it as read.)
- **Show Elements/Dialog:** Zoom in to view the requested element(s) or to open the corresponding dialog box.
- **Deny Request.** Click this if you do not want to release the elements. The elements remain in your workspace.
- **Grant Request:** Click to approve the request. If you grant a Request, this means you have released the item, which is now instantly and automatically reserved by the user who sent the request.

In the **Received Ownership Request** message:

- **Apply View:** Click this to open the attached view (if any) of the requested element(s). (This command only appears if a view is attached by the sender.)
- **Restore View:** This button appears if you have clicked Apply View (see above). Restore View will return you to the previous model view.
- **Show Elements:** This command zooms in to and selects the requested elements.

Once the owner grants or denies a Request, the associated request message is automatically moved to the user's Completed list.

At the same time, the user who issued the Request receives the response in their own To-Do list: (the message is concurrently removed from their Pending list).



Release Elements or Project Data (Teamwork)

Release Project Data

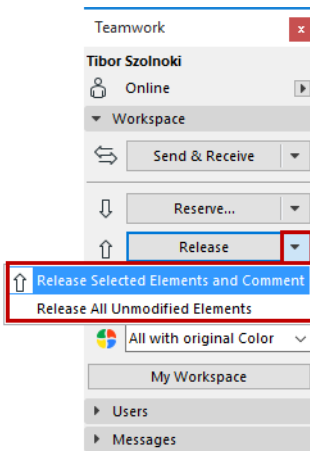
To release project data owned by you, open the relevant dialog box (or select the item in the Navigator), then click the **Release** command from the bottom of the dialog box (or from the Navigator item's context menu).

When you Release a data type, any changes you have made are sent to the server, even if you then click Cancel to leave the dialog box.

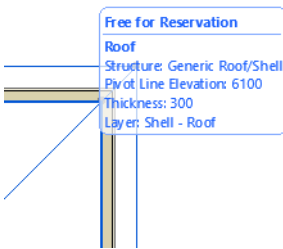
Release Selected Element(s)

To release an element currently owned by you, select it, then click the **Release Selected** command from one of these locations:

- the Teamwork Palette (choose either the **Release** button or the **Release Selected and Comment** option from the pop-up)
- the Teamwork menu
- the context menu of the selected element



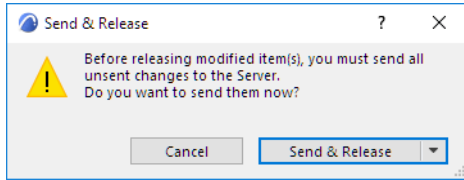
Once released, an element is shown as “Free for reservation” in the Info Tag.



It can now be reserved by another user.

If any user has already sent in a request for this element, then he/she will automatically receive the element (and a message) as soon as it released.

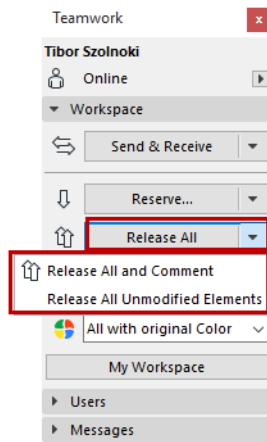
Before releasing a modified element, you must send your changes to the server:



Release All

If nothing is selected, the button in the Teamwork Palette changes to “Release All.” This will release all of your reservations: elements and project data alike.

Click the pop-up button to access other **Release** commands:



Release with Close

See [Keep or Release Your Reservations with Send & Save](#).

Release and Comment

Release commands on the Teamwork Palette and in project data dialog boxes include an option to “Release and Comment.” Choose this to add a comment to the Activities panel of BIMcloud (with or without a message to other users) to report the Release event.

See [Add Comment to Activities \(Teamwork\)](#).

Messaging (Teamwork)

Messaging is integrated in the Teamwork Project: its context-driven functions are tied to elements and actions, so that many messaging functions are automated, reducing the amount of user-initiated communications needed.

For example, when requesting an element, you don't need to explain which one you need: just select it and click "Request." The owner immediately and automatically receives a message (with built-in response commands) that displays exactly which element you have requested. This way, responding to messages is both easy and productive.

Every time you receive a message, an alert appears on your screen.



Note: If you do not want to see these alerts, uncheck the "Show New Message Alert in Teamwork" box at **Options > Work Environment > More Options**.

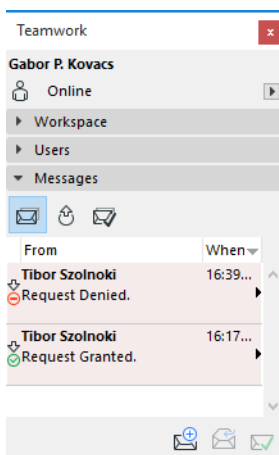
You can click this alert to view the message and respond to it as needed, or ignore it and keep working until you take the time to deal with the new messages.

You can communicate with users who are not online at the moment: they will receive their messages the next time they are online.

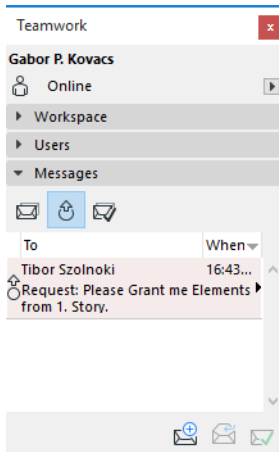
Teamwork's messaging system supports all types of textual input. Any keyboard character can be entered in messages, including special language characters. Graphical data, attachments and hyperlinks are not supported.

The **Messages** panel, available from the Teamwork Palette, has three tabs.

- The **To Do** list shows you items which await action from you. New messages arrive here, including notification of the outcome of your Requests. (for example, "Granted request: Elements on the 1. Story;" or "Denied request.")



- The **Pending Requests** list stores request messages sent by you, which are still pending.



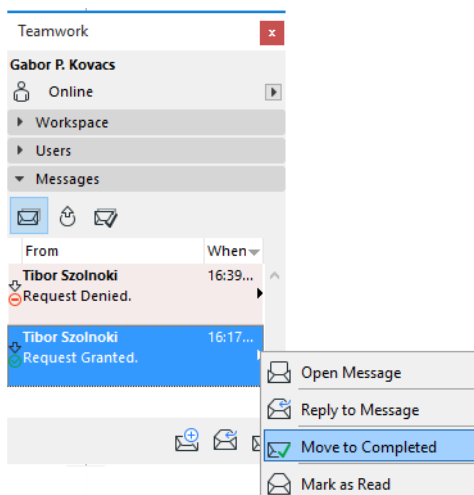
If a pending request (or a part of it) is completed, then it automatically moves to the To Do list, letting you know that you can start working with the elements. (Such items are automatically removed from the Pending Requests list.)

If you sent a request to more than one user, then each response will appear as a separate message on the To Do list. Your request will remain in the Pending Requests list until the last answer arrives.

If you open one of your Pending Requests, you get a status report: how many of the requested items have been granted/denied and how many are still pending.

If you requested an element from someone who simply released the element without responding specifically to your request, the element is then automatically given to you. Your request is then moved from the Pending Requests list to the To-Do list, in the form of a "No pending elements" message.

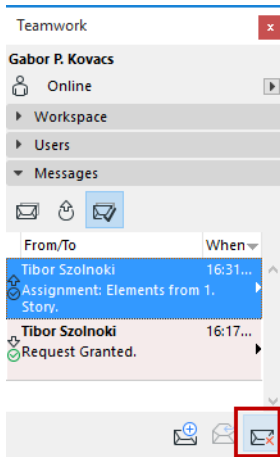
- The **Completed** list is a repository of messaging actions that are no longer pending and no longer require action by you. You can also move an item from the To Do list to the **Completed** list manually:
- Select the item, then the **Move to Completed** command from the context menu or from the drop-down menu (if available).
- Click the **Move to Completed** icon the bottom right corner.



Exception: If a message is expecting a response from you, then you cannot move it to the Completed list; you must first answer that message. (These messages are typically Requests, requiring an answer of

Grant or Deny.) Once you answer with a Grant/Deny, then the message automatically moves to the Completed list.

- To delete one or more messages from the **Completed** list, select them and click the Delete item at the bottom of the list.



Related Topics:

[Request Elements or Project Data \(Teamwork\)](#)

[Create a New Message \(Teamwork\)](#)

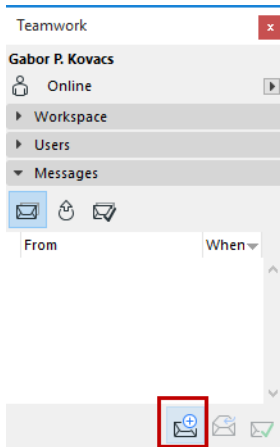
[Receive and Open Message \(Teamwork\)](#)

[Review Elements \(Teamwork\)](#)

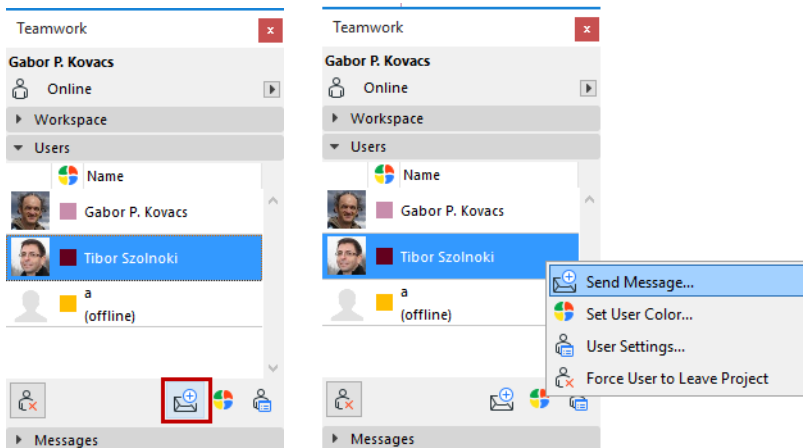
[Change Tracking in Teamwork](#)

Create a New Message (Teamwork)

To create a new message, click the **Send New Message** icon at the bottom of the Messages Panel of the Teamwork Palette, or use **Teamwork > New Message**.

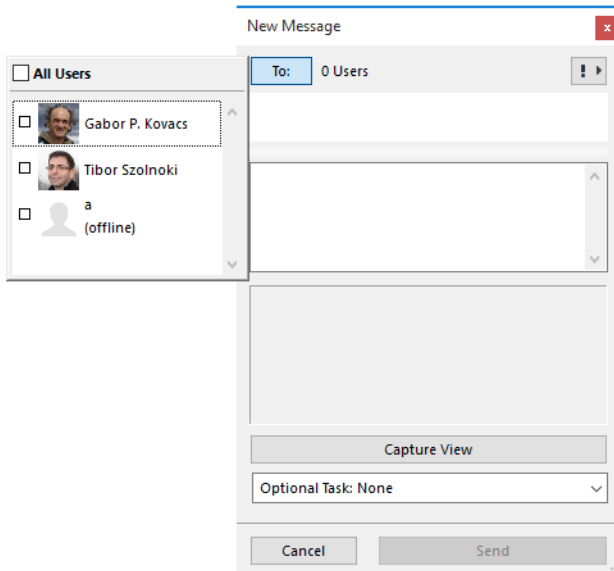


Alternatively, select one or more user names from the Teamwork Palette's Users panel, or from the context menu, and click the same icon there. The selected names will appear in the message's To: field.

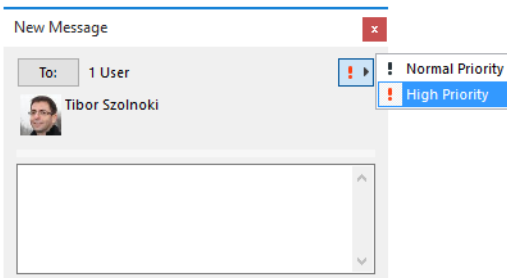


Choose Recipients

Click the **To:** button to bring up a checklist of users who are logged in to the project (whether currently online or not). Choose the recipients of your message, or click “All Users” to include everyone joined in to the project.



For extra emphasis, you can choose to add a High Priority icon to your message from the pop-up at the top right. High priority messages are shown in the recipient’s To Do list with a colored background.



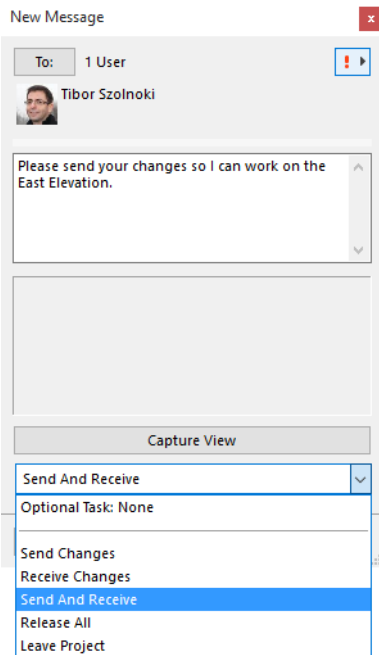
Type in the desired text message, and click **Send**.

Add a Task Request

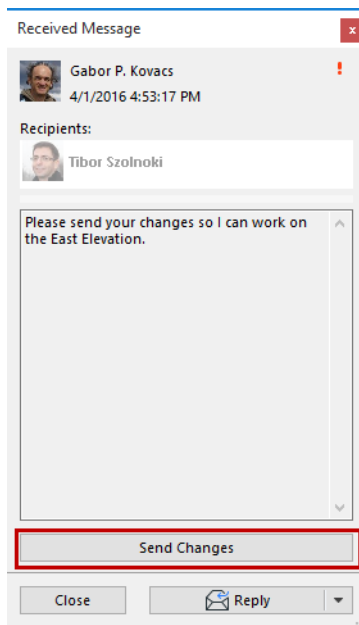
You can make it easier for the message recipient if you add a pre-programmed task request as part of your message.

Before sending your message, you can optionally add a task button, so that the recipient’s copy will contain not only the message you typed, but also the specified command button, to facilitate a quick response.

Click the **Optional Task** pop-up in the New Message dialog box:



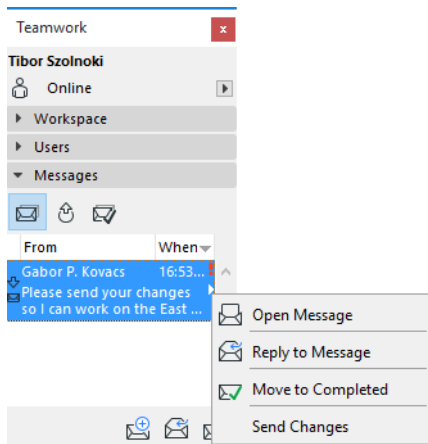
For example, if your message asks the recipients to send in all their work, you can attach a **Send Changes** button to your message. In complying with your request, the recipient needs only to click this **Send Changes** button from the message.



Receive and Open Message (Teamwork)

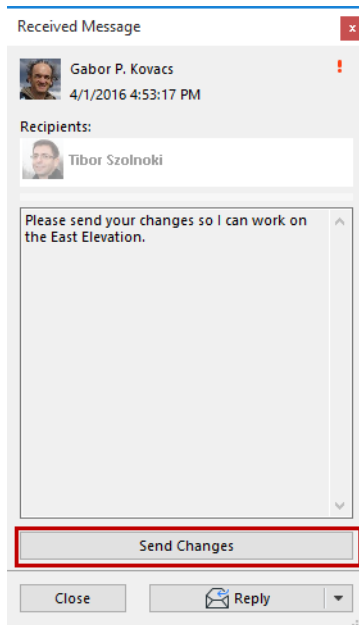
To view a new message and related options, do one of the following:

- Click on the on-screen message alert
- Click the pop-up of the new message in your To-Do list, and choose one of the options:



If the sender has included a Task as part of the message, the related command is shown here: click to complete the Task.

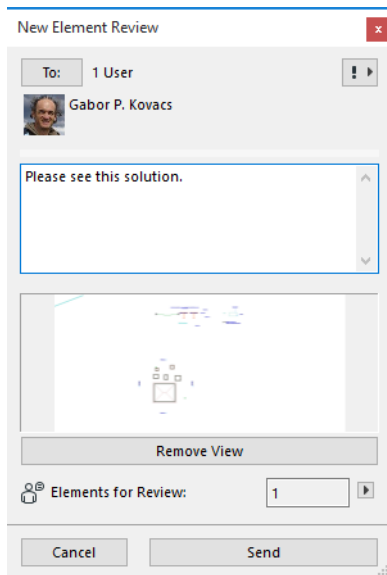
The same options are available if you open the message.



Review Elements (Teamwork)

If you want to discuss certain element(s) with a fellow user, it often easiest to ask them to take a look. In Teamwork, this is easy:

1. Select the element(s) in question.
2. From the context menu or Teamwork menu, choose “Review Selected Elements.”
3. Select the user(s) who you want to review the elements.
4. The **New Element Review** includes an attached view, plus the number the elements in question.
5. Click **Send**.



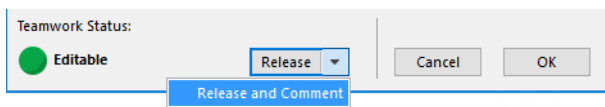
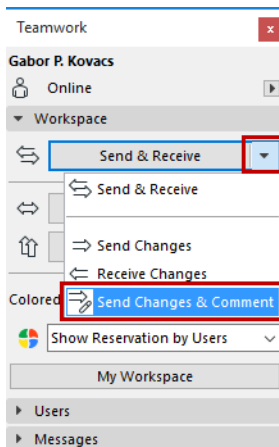
6. The recipient clicks “Show” to see the elements under review.

Add Comment to Activities (Teamwork)

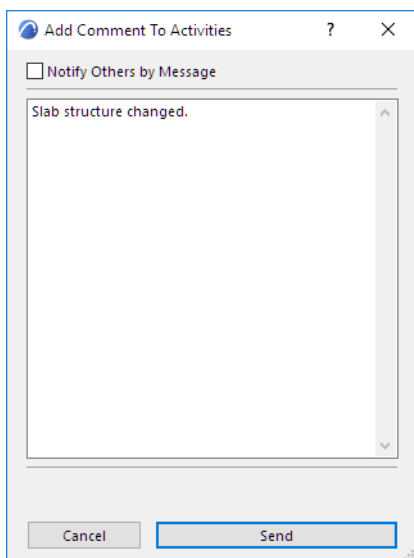
Comments are optional texts that are recorded in the Activities panel of the BIMcloud Manager.

To add a comment to the Activities log, do one of the following:

- Use **Teamwork > Project > Add Teamwork Comment**
- Choose **Send Changes and Comment** or **Release and Comment** from the pop-up or context menu of the regular Send and Release commands.



The appearing Add Comment to Activities dialog box may contain an automatically generated text about the action you have just taken. (If you just issued the **Add Teamwork Comment** command, the field is empty.)

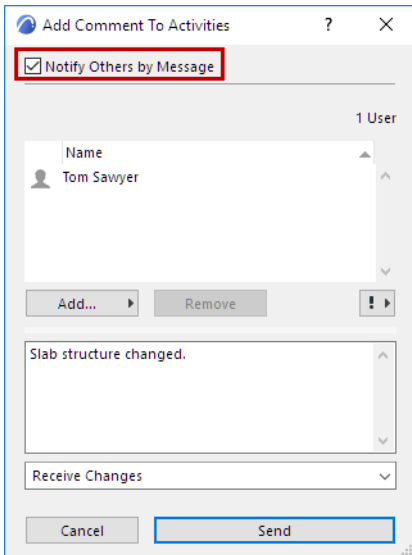


Type any text into this field. When you click OK, the comment will be added to the Activities panel of this project in BIMcloud Manager.

Notify Others by Message

In addition to sending the Comment to the Activities panel, you can notify any other user at the same time, by sending a message.

When the Add Comment dialog box appears, check the “Notify Others by Message” box:



Define the message recipients and add an optional task as with regular messages.

See [Create a New Message \(Teamwork\)](#).

Libraries in Teamwork

A Teamwork Project uses two types of libraries: the Embedded Library plus the BIMcloud Libraries. Just like in any Archicad project, these libraries are listed in the Library Manager.

[See also Library Manager.](#)

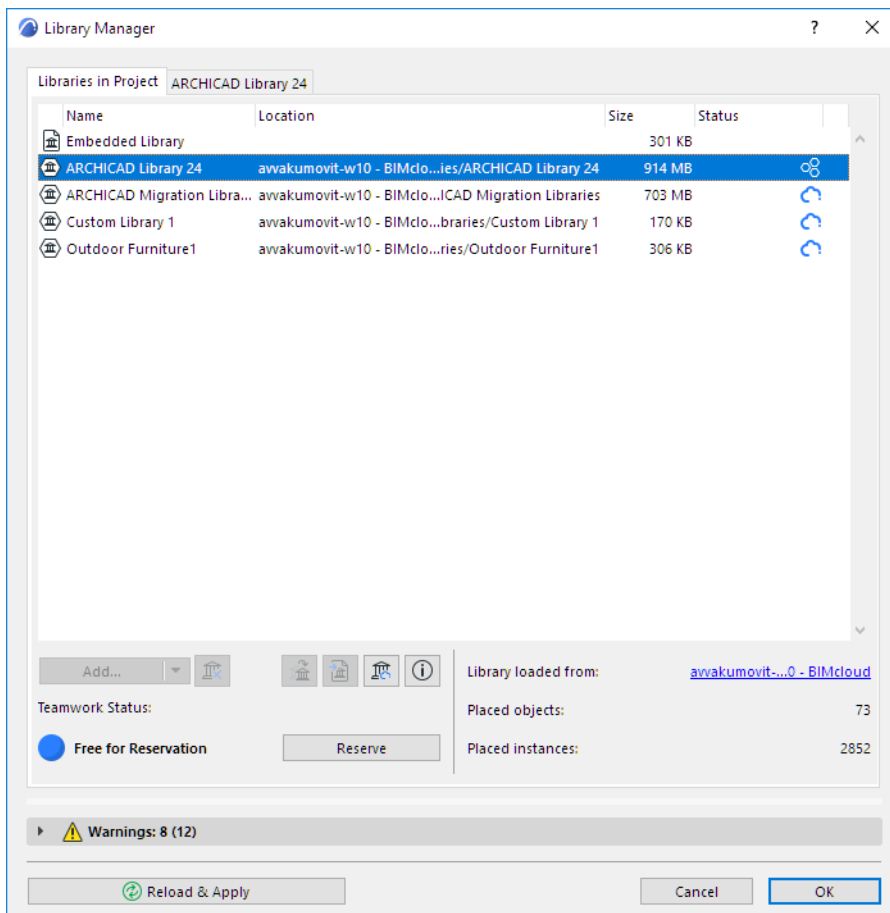
In order to use a BIMcloud library in a Teamwork project:

- the library must first be uploaded to a BIMcloud Server;
- and then it must be added to the project using Archicad's Library Manager.

[See Upload a Library to BIMcloud.](#)

[See Add BIMcloud Library.](#)

Linked libraries cannot be used by Teamwork projects; this limitation is necessary to ensure that all users have access to all library objects, even when working offline.



Therefore, the process of Sharing a Teamwork project includes uploading the project's linked libraries to the server.

[See Click Libraries to open the dialog and decide how to handle the project libraries..](#)

Embedded Objects in Teamwork

Embedded objects are added to the Embedded Library of Teamwork projects, just like solo projects. If a user adds an embedded object, all other team users - after they do a Receive - can access and place the object. Such object instances can be reserved in Teamwork just like any other element.

If you work in Teamwork, you must embed objects created by Add-ons, like trusses, otherwise editing these elements will not be possible. It is highly recommended to embed custom objects and custom door/window components.

If you want objects to be accessible to all Teamwork users and avoid editing conflicts, you **MUST** either save them to the Embedded library (when you initially create them), or add them to the Embedded library (if they have been saved to another location).

Reserve Placed Object vs. Reserve Embedded Object

You can reserve a placed object in the project, and you can also reserve an Embedded Object from the Library Manager (or Object Editor).

These two reservations are entirely unrelated. When you reserve a placed instance of an object in the project, you can then edit its Object Settings, either in place or in the Settings dialog box. Your edits will affect only the particular object instance you have reserved.

It is possible to reserve an Embedded Object itself (as opposed to its placed instance), either in the Library Manager, or in the Object Editor. (Both locations will indicate the object's reservation status; they are in sync.) You can then edit its script in the Object Editor. Editing the script will affect all instances of this object in the project, regardless of who has reserved those instances.

Hotlink Modules and XREFs in Teamwork

You can work with Hotlinks in Teamwork just as in solo projects, if you have access rights and reservations.

Place New Hotlinks

- You must have the “Hotlink and XREF - Manage” access right.

Relink a Hotlink Source File

- You must have the “Hotlink and XREF - Manage” access right.
- You must have the “Hotlink - Instances” access right.

Update a Hotlink

- You must have the “Hotlink - Instances” access right.

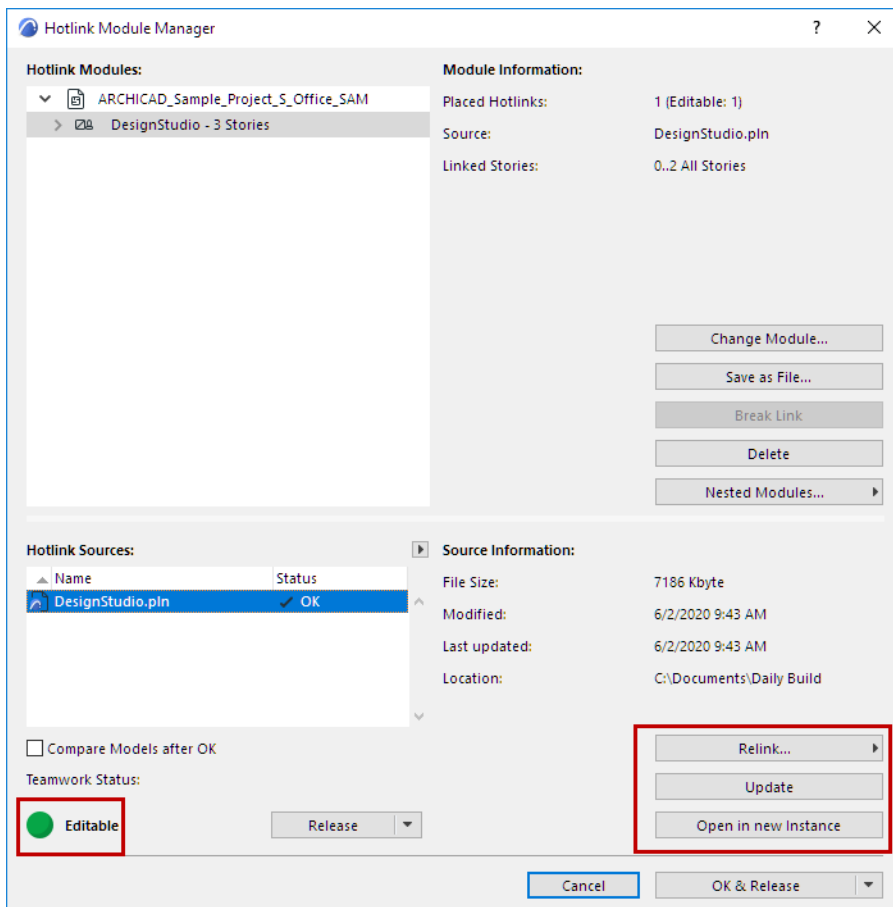
Update a Hotlink Source File

- You must have the “Hotlink and XREF - Manage” access right.

How to Reserve Hotlink Module Manager

When updating a Hotlink, the Hotlink Module Manager will be reserved automatically in the background, if it is available, and then released after the update.

Go to the Hotlink Module Manager (**File > External Content > Hotlink Module Manager**) and click **Reserve**.



Updating a Hotlink will update *all* module instances, regardless of who owns the instances.

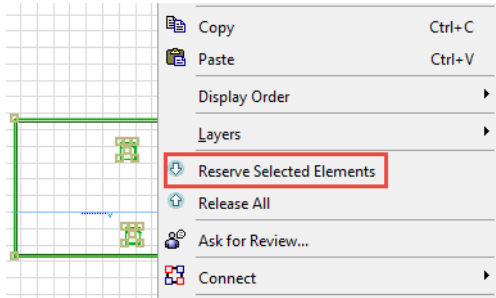
If you **Change** or **Delete** or **Relink** a Hotlink module in the Hotlink Module Manager, this action will affect only those instances of the Hotlink module which you have reserved. (Hotlink Module Manager will be reserved automatically in the background, if it is available, and then released after the action.)

The same is true for deleting or replacing XREF instances.

How to Reserve a Hotlink Module Instance or XREF Instance

By placing a Hotlink Module instance, you automatically reserve it.

A placed Hotlink Module or XREF instance can also be reserved or requested by others like any element, using the commands of the Teamwork Palette and the context menus.



Reserving a Hotlink Module or XREF instance means that you have the right to move it or delete it. You can also modify its settings (e.g. its orientation/rotation, Master Layer, height and elevation/story values) and replace it with a different Module.

Attach New XREFs

- You must have the “Hotlink and XREF - Manage” access right.
- You must reserve the Xref Manager.

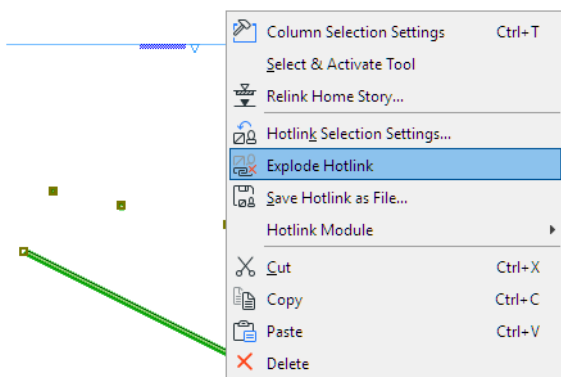
Attach or Modify Existing XREF References

- You must have the “XREF - Instances” access right.

Explode Hotlink Module Instance in Teamwork

The Break command is not available for Hotlink Modules in Teamwork. However, you can explode individual Hotlink instances.

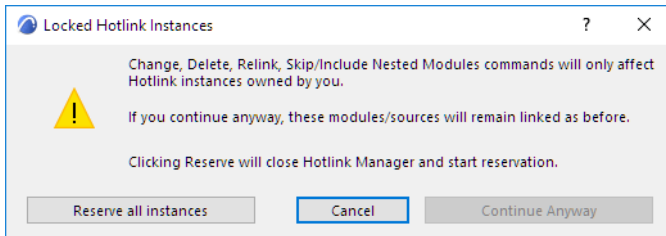
1. Select the Hotlink on the Floor Plan.
2. Use the **Explode Hotlink** command from the context menu, or from **File > External Content**.



Note: With this Explode command, any dimensions that were added to elements in the module will be lost. Also, Solid Element Operations connections will be lost.

Relink from Teamwork Project

When you relink Hotlink Sources, you must first reserve all the Hotlinks you want to relink. If this is not the situation, then a dialog will prompt you to reserve individual Hotlinks:



- **Continue anyway:** (Available only if one or more Hotlinks of the source file are not reserved by you.) If you select this option, then only the instances reserved by you will be relinked to the new source. Hotlinks not reserved by you will remain linked to the original source.
- **Reserve all instances:** If you select this, the Hotlink Module Manager will close, and Hotlink instances will be reserved. After this process is finished, open the Hotlink Module Manager again, and start relinking again.

Hint: Reserve the Hotlink modules you want to relink before you begin Relink from Hotlink Module Manager.

Hotlink Modules

Topics in this section:[About Hotlink Modules](#)[Place Hotlink](#)[Edit Hotlink Module Content](#)[Hotlink Settings](#)[Hotlink Module Manager](#)[Create Module \(.mod\) File](#)[Save Selection in PLN Format](#)[Hotlink Modules and XREFs in Teamwork](#)[Compare Archicad Models Before and After Hotlink Update](#)**Related Topics:**[Properties in Hotlinks](#)[Hotlink Rhino File](#)[Hotlink 3D RVT Model](#)[Hotlink IFC File](#)[Hotlinks and Design Options](#)

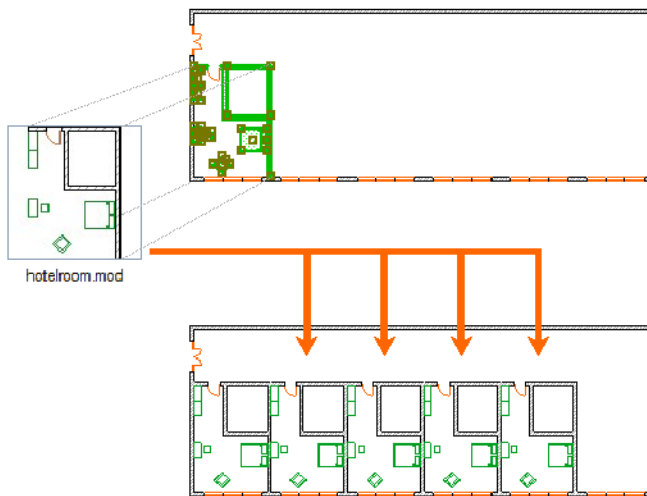
About Hotlink Modules

Hotlink Modules allow you to insert the contents of external files (sources) into the currently open Project (host).

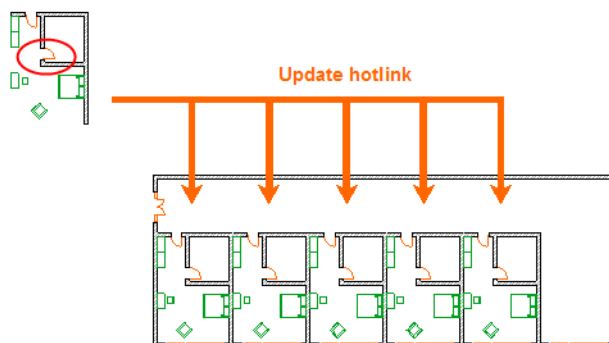
Hotlink Modules can be used, for example, to manage the repetitive structures of buildings such as hotels or offices with a large number of identical rooms: if rooms are placed as a Hotlink module, you can modify all the rooms in one step, by just updating the module's source file. Moreover, the same structures can be used in multiple projects.

When updating a Hotlink Module, you have the option to launch the Model Compare window, which helps you visualize exactly what has changed as a result of the update.

See [Compare Archicad Models Before and After Hotlink Update](#).



Change in module



Which files can you place as Modules?

Archicad file types:

- Solo Project (.pln)
- a Module-type file (.mod)
- a Teamwork Project from a BIMcloud (of the same Archicad version as your host project)
- Archicad 2D Lines (.2dl)

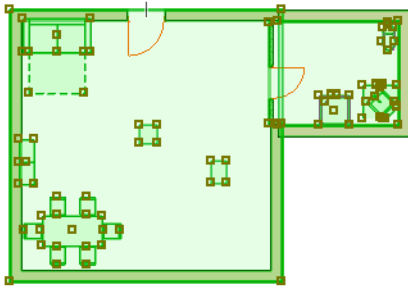
Other file types that can be placed as Modules:

- an IFC file

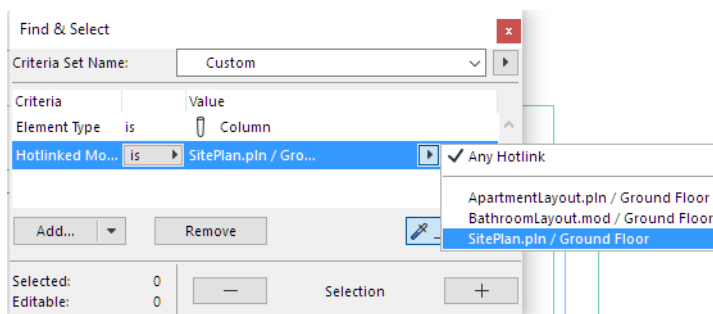
- a Rhino file (.3dm)
- an RVT file

Characteristics of Hotlink Modules

- Modules are stored in the host project. Even if the Source file is not currently available, the module is still present and visible. However, the module can't be updated as long as the referred source file is unavailable.
- The placement parameters (e.g. orientation, Master Layer, Design Option) of each Hotlink can be different even though they share the same Module source.
- The placed Hotlink can only be edited as an entity. Its elements cannot be edited as long as they remain part of a Hotlink.
- Elements belonging to a selected Hotlink have hollow square marks on their hotspots. If several Modules are selected, their selection dots will have different colors to help you distinguish them.



- You can use **Find & Select** command to locate Hotlink Module elements.



Attributes of Hotlink Modules

For Archicad-based modules:

- If the elements of the Hotlink Module use attributes with the same names as attributes of the Host Project, the Host attributes will be applied.
- If no attributes of that name exist in the Host, new attributes will be added to the Host Project.
- Once an attribute is part of the host file, that attribute will not be updated by any attribute modifications in the source file.
- Attributes imported with Hotlink Modules will not be removed when deleting Hotlinks or Modules.

See also [Master Layer](#) for more information about Hotlink Module layers.

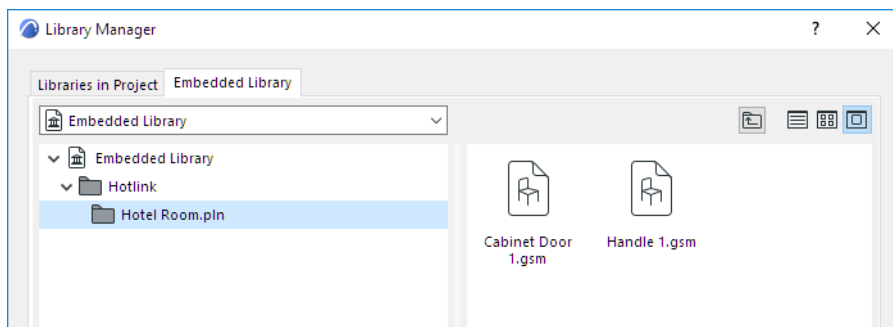
Notes: The above information applies to Archicad-based Modules only.

- **IFC Hotlink Module** attributes are managed according to the template file of the Translator for Import used to place the IFC hotlink. [See *Hotlink IFC File*](#).
- **Rhino-based Hotlink Module** attributes (layer and surface settings) are set at **Import Options**, when selecting the module. [See *Import Options for Rhino 3D Model*](#)

Library Parts in Hotlink Modules

Note: Applies to Archicad-based Modules only. For handling of imported (hotlinked) IFC library parts, see [Library Parts Location](#).

- **Linked Library Parts:** Placing a Hotlink module does not import any linked Library Parts. Make sure that the Library Parts used by the source are also available to the host, by doing one of the following:
 - Use the same Libraries for both source and host
 - Add the source’s Libraries to the host file’s libraries
- **Embedded Library Parts:** Placing a Hotlink module adds the source file’s entire Embedded library to your host file’s Embedded library. The newly added Embedded library can be seen in your Library Manager, inside the automatically created “Hotlinks” folder of your Embedded library. In this folder, each Hotlink source file is listed separately along with all of their embedded objects. These objects will be updated in your host project if they are updated in the source project.



Important: It is *not* recommended to place any these objects into your project separately. These objects will be deleted from the Embedded Library if the Hotlink module is deleted from the project, or if all placed instances of the module are deleted. (You will notice the deletions after you re-open the host file.)

Design Options in Hotlink Modules

[See *Hotlinks and Design Options*](#).

Properties and Classifications in Hotlink Modules

[See *Properties in Hotlinks*](#).

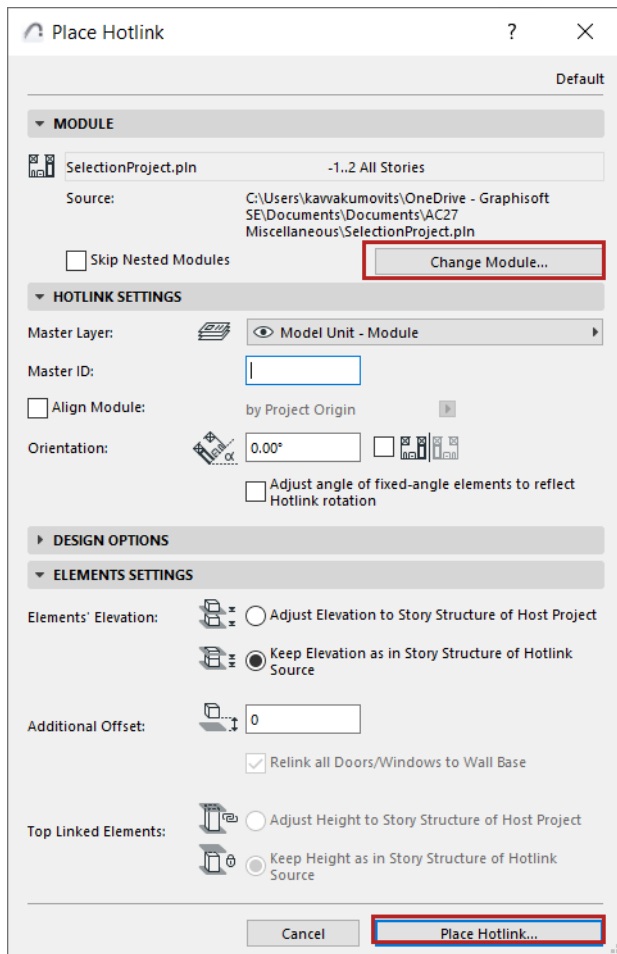
Place Hotlink

1. Choose **File > External Content > Place Hotlink**.

The **Place Hotlink** dialog appears.

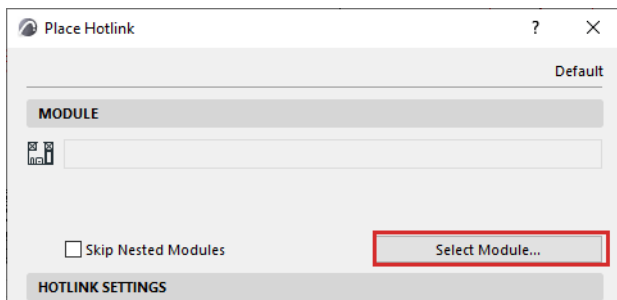
If the project already contains one or more Hotlinks:

- The last placed Hotlink is listed. Click **Place Hotlink** to place it again, or **Change Module** to choose another one.



If no Hotlink has been placed yet:

- Click **Select Module** to define one.



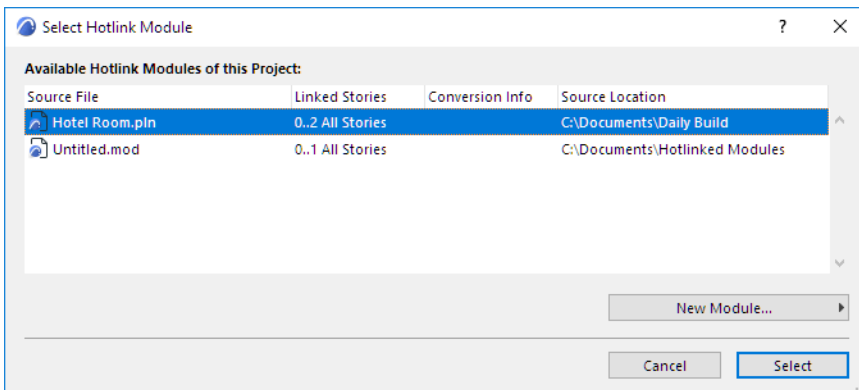
2. Select Hotlink Module

The **Select Hotlink Module** dialog box appears. Hotlinked Modules in the project (if any) are listed with information on their linked stories and the source file location.

- For IFC-based Modules, the Conversion Info column indicates the Template and Translator files used in converting the file to Archicad.
- For Rhino-based Modules, the Conversion Info column is the timestamp identifying the module.
- For RVT-based Modules, the Conversion Info column contains the 3D View and timestamp.

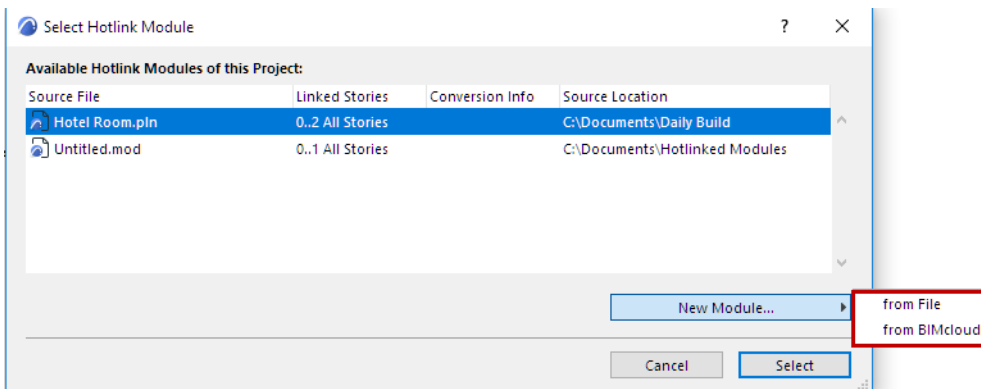
3. Do one of the following:

- Select one of the existing modules of this project, shown in the list, or
- Click **New Module** to add a new Hotlink Module to the current project.



If you choose New Module:

Choose the source of the new Hotlink: either **From File**, or **From BIMcloud**.



- **From File:** A directory dialog box will appear. Select the desired file to use as the module.
- **From BIMcloud:** Choose a BIMcloud, then choose the Teamwork project. If needed, log in to the BIMcloud. [See Log in to BIMcloud.](#)

Note: The source Teamwork project must have the same version number as the host project.

If you choose IFC file format:

See [Hotlink IFC File](#) for additional options.

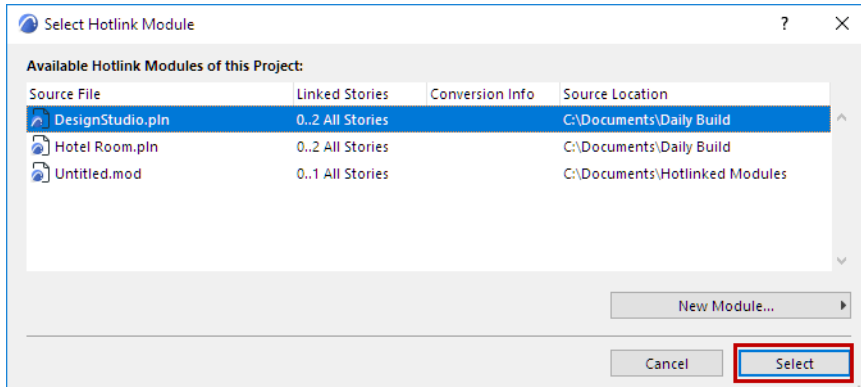
If you choose Rhino (.gsm) file format:

See [Hotlink Rhino File](#) for additional options.

If you choose Revit (.rvt) file format:

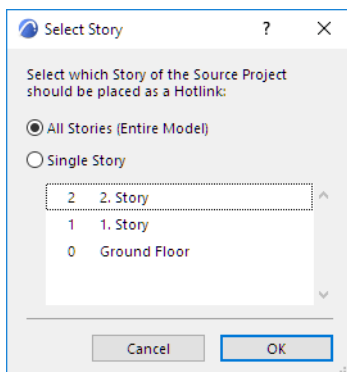
See [Hotlink 3D RVT Model](#) for additional options.

4. Click **Select**.



5. If the selected PLN or .mod file has multiple stories, the **Select Story** dialog box appears.

Note: **Select Story** does not appear for IFC modules (which are placed with all stories) or Rhino-based modules.

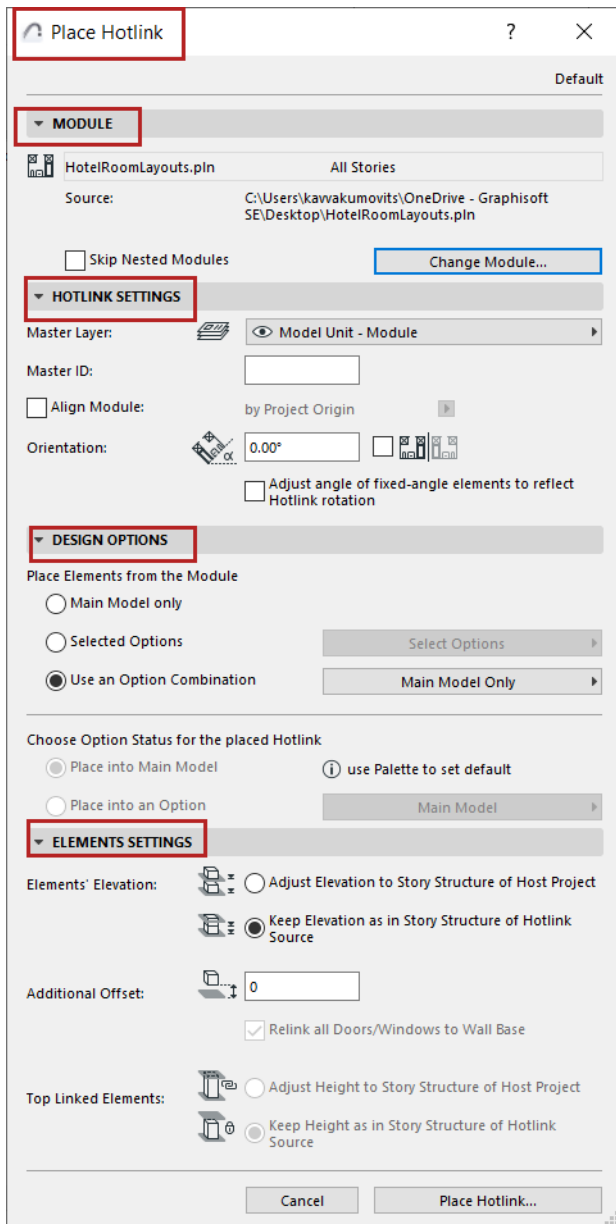


Select an option:

- **Single Story:** Click on one of the listed stories to place as Hotlink.
- **All Stories (Entire Model):** Place all stories of the module file into the host project.

6. Click **OK**.

7. Click **Select** to return to the **Place Hotlink** dialog box:



- set Master Layer and Orientation
- optionally Align Module at either the Project Origin or Survey Point
 - Available at Hotlink placement only, for module formats that use Survey Point: PLN, MOD, IFC, RVT
 - If this is checked, then the Orientation control is grey
- assign an optional Master ID

8. Set **Design Options** for the Hotlink:

- Choose Design Options to place from the source file. (The Main Model is always placed, but you can choose particular Options or an Option Combination.)
- The Hotlink will be added to the current default Option of the host file.

[See *Hotlinks and Design Options*.](#)

9. Define how to handle elements' elevation and top linked elements

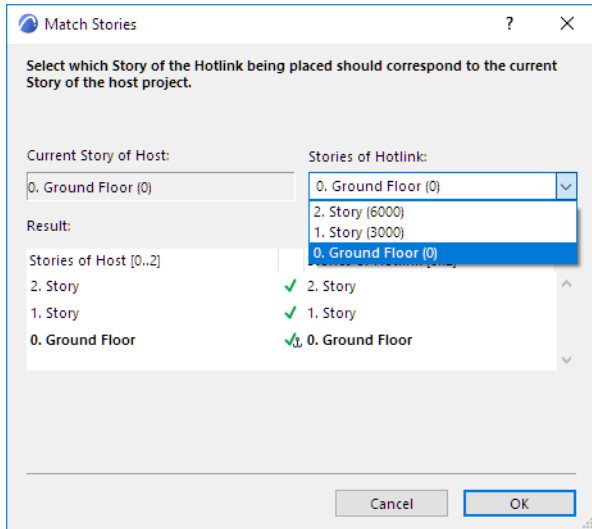
Note: When hotlinking an IFC file, make sure you set the Elements' Elevation option to **Keep Elevation!** Otherwise, the imported model may “fall apart” when its elements are placed in a different story structure.

For details, see [Hotlink Settings](#).

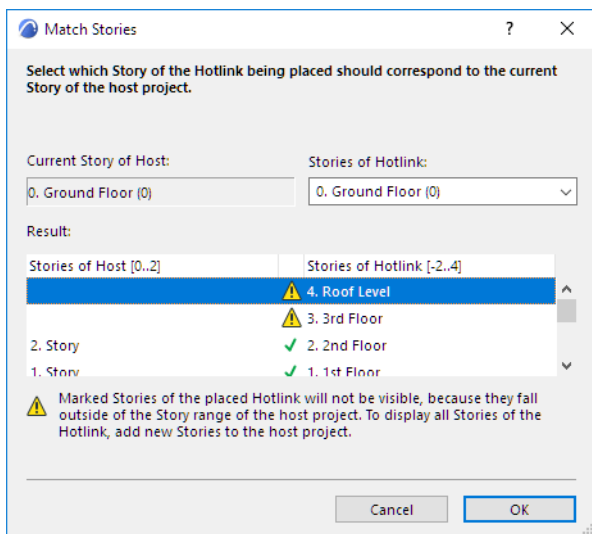
10. Click **Place Hotlink**.

11. **Match Stories** (for Hotlink Modules with multiple stories):

In the **Match Stories** dialog box, select which story of the module to place onto the current story in the host project. (Not relevant for Rhino-based modules).



If you try to place a multistory module into a host project that contains fewer stories than the module you are placing, the module stories which do not “fit” in the host project will not be visible.



To resolve this, create additional stories as needed in the host project.

12. Click **OK** to place the Hotlink.

Edit Hotlink Module Content

This section describes three workflows:

Edit Module in Separate Archicad

Break Hotlink (Embed Elements)

Save Hotlink as File...

Edit Module in Separate Archicad

Not available for IFC- or Rhino-based Hotlink Modules.

1. Do one of the following:
 - In Hotlink Module Manager, choose the desired source file from the Hotlink Sources list, then click **Open in New Instance**.
 - Select the placed Hotlink, then use the context menu: **Hotlink Module > Edit Module in Separate Archicad** from the context menu.
2. Edit the source file as needed, then save it.

Hint: If you want to see the Floor Plan context of the placed module while editing its source in the second Archicad, use the “Trace” function. ([See View Context of Hotlinked Module while Editing Module Content.](#))
3. Return to the first (host) project.
4. Go to **Hotlink Module Manager**. In the Hotlink Source list, note that the Status of the edited module’s Hotlink Source now reads **Modified**. (If needed, click **Refresh Status**.)
5. Click **Update**.
6. Click **OK** to carry out the update on all of the module’s Hotlinks and close Hotlink Module Manager.

Note: In Teamwork, you must reserve Hotlink Sources in the Hotlink Module Manager to be able to update.

Break Hotlink (Embed Elements)

Use this command to break this Hotlink and embed the elements into the host project.

Use this if you have placed a Hotlink, but you now want to edit it locally in the host project, rather than via its source file.

For example: you have placed multiple hotlinks of a module of a standard hotel room. One of the rooms doesn’t fit into the plan. You can break the hotlink of that one instance, then edit its content to fit its particular location.

1. Select the module on the Floor Plan.
2. From the context menu, use **Break Hotlink (Embed Elements)**.

Or use **File > External Content > Break Hotlink (Embed Elements)**

Note: To break *all* the placed Hotlinks of a particular module, use the **Break Link** command from Hotlink Module Manager.

Note: In Teamwork, the Break Hotlink function is not available. Instead, you can Explode the Hotlink Module. ([See Explode Hotlink Module Instance in Teamwork.](#))
3. Edit the elements as needed. (Use **Edit > Grouping > Suspend Groups** to enable editing.)

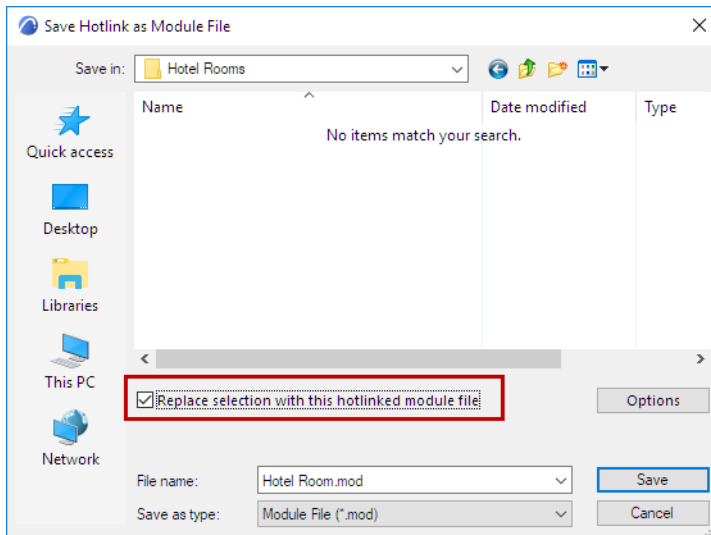
Save Hotlink as File...

Available from Hotlink Module Manager, and the context menu of a placed Hotlink Module.

Use this command to save the Hotlinked Module elements as a new file in module format (.mod).

For example: you have placed multiple hotlinks of a module of a standard hotel room (Room A). You want to change one of the room layouts (calling it Room B), but without embedding its elements into the host project (because you will want to place more Room B hotlinks later).

1. Select a Hotlink. (In Hotlink Module Manager, or on the Floor Plan.)
2. Use the **Save as File** or **Save Hotlink as File** command.
3. Check the box to **Replace selection with this hotlinked module file**.

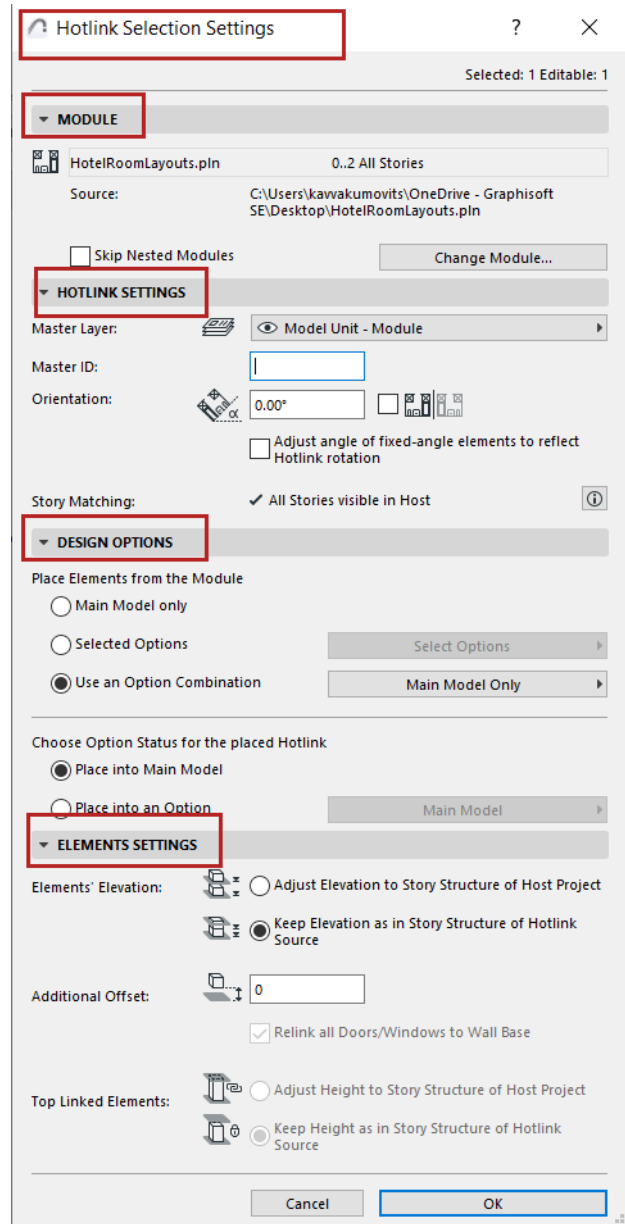
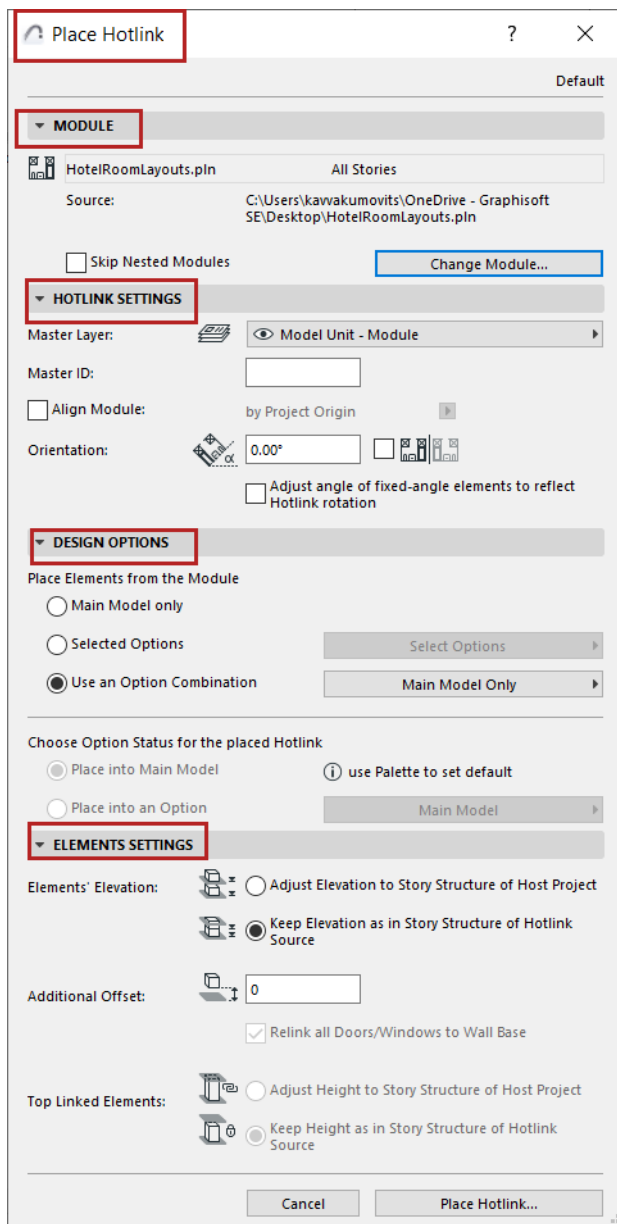


You have created a separate new module (Room B) which you can edit independently of the original Room A module.

Hotlink Settings

Hotlink Settings are available:

- in the **Place Hotlink** dialog box
- in **Hotlink Selection Settings**, available for a selected Hotlink module:
 - Choose **Hotlink Selection Settings** from the module's context menu, or
 - Use **File > External Content > Hotlink Selection Settings**



Module Panel

Skip Nested Modules (Hotlink Settings)

It is possible to have several levels of Hotlink Modules nested into each other.

If you do not want include the nested modules in this Hotlink, go to the Module panel of Hotlink Settings and check **Skip Nested Modules**. This will apply to the current Hotlink only.

Note: IFC- or Rhino-based Modules cannot have nested Modules. However, IFC- or Rhino-based Modules can be nested inside a PLN hotlink.

Note: To skip or include nested modules in *all* Hotlinks of this module, use Hotlink Module Manager.

See [Nested Modules \(Hotlink Module Manager\)](#).

Change Module

Select a different module to replace the selected module.

Hotlink Settings panel

Master Layer

The Master Layer controls the visibility and locking of the Hotlink as a whole. This Master Layer is an ordinary layer of the host file and can include any other elements.

- Different Hotlinks of the same module can be assigned to different Master Layers.
- The elements themselves keep their element-specific layer assignments and are placed on the corresponding layers of the current Project, if layers with these names exist. If necessary, new layers will be created.

See also [Hotlink IFC File](#).

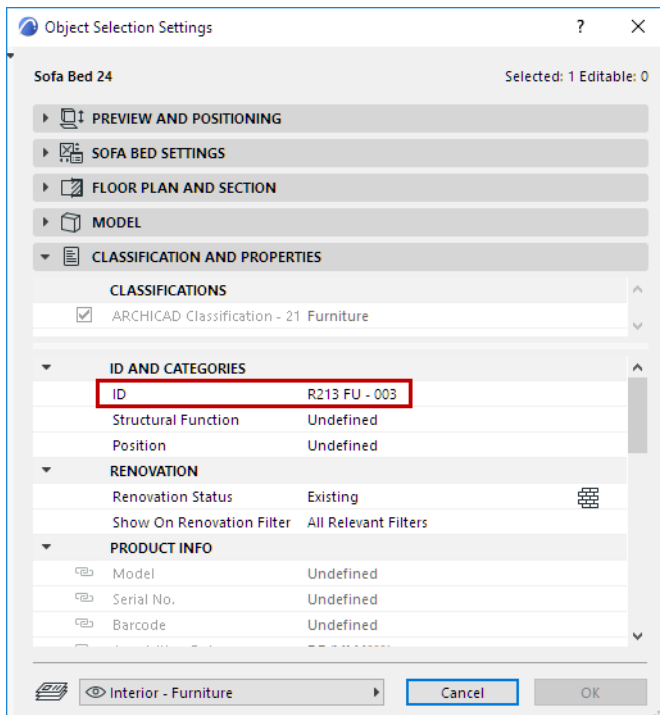
- Visibility of individual elements within the module depend on element-specific layers.
- Layer Combinations of the source file are not imported with the modules.

Note: For best results, use the same layer structure in both the source and the host file.

Master ID

Define an optional, custom Master ID for this Hotlink: this is a tag added to the Element ID of each Hotlink element.

For example: when hotlinking a hotel room, define its Master ID to correspond to its room number. Each element of the Hotlink now has an Element ID extended by this room number (e.g. R213).

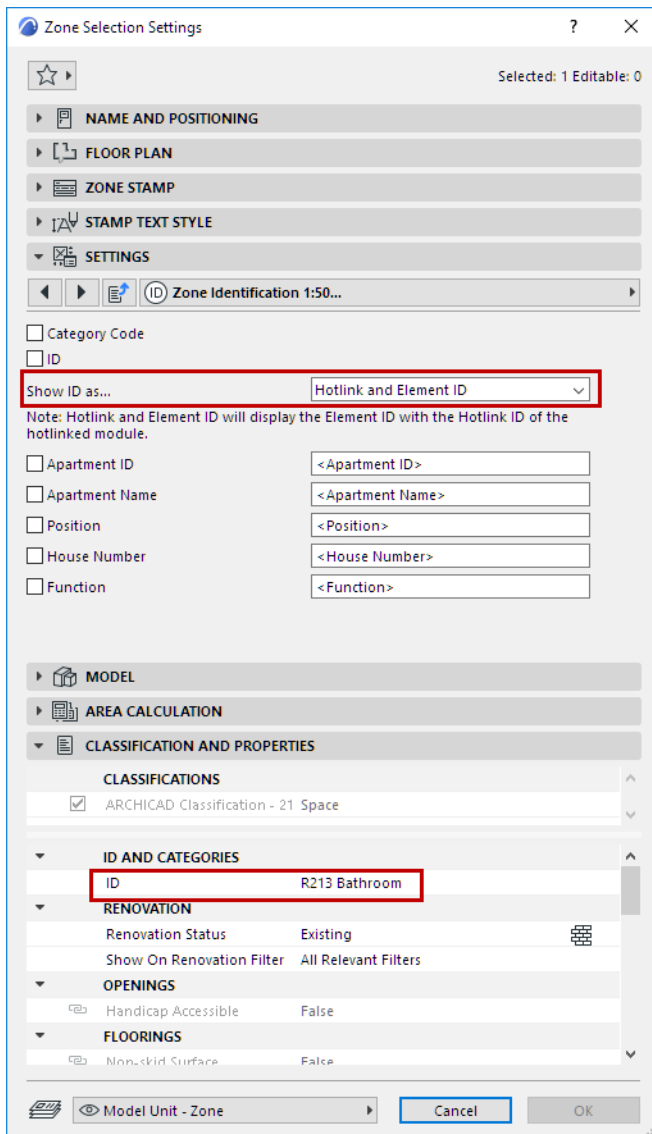


Extend Zone Element ID with Master ID

In another example: extend the Zone Element ID by the Master ID of the Hotlink it belongs to.

- For example, a Zone with an Element ID of Bathroom in the source file is placed into the host file as part of a Module having a Master ID of R213. Its Zone ID will then be R213 Bathroom.

Note: To display this full Zone ID, make sure that the Zone Settings are using the “Hotlink and Element ID” display form.



Orientation

- Type a value in the **Orientation** field if you want to place the Hotlink at an angle, compared to its orientation in the source file.
- Check the box next to the Orientation field to place a **mirrored** Module.

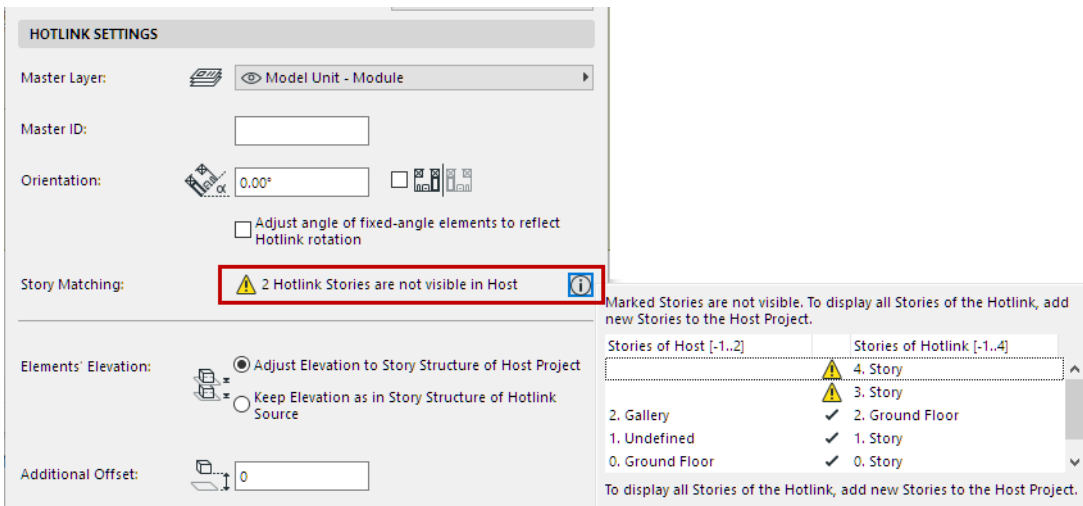
Adjust angle of fixed-angle elements to reflect Hotlink rotation:

Check this if you want to rotate these fixed-angle elements (e.g. Labels or Zone Stamps) to reflect the Hotlink's new orientation.

Story Matching

If you have placed a multistory module into a host project that contains fewer stories than the placed module, the module stories which do not “fit” in the host project will not be visible.

Hotlink Settings shows feedback on the situation:



To resolve this, create additional stories as needed in the host project.

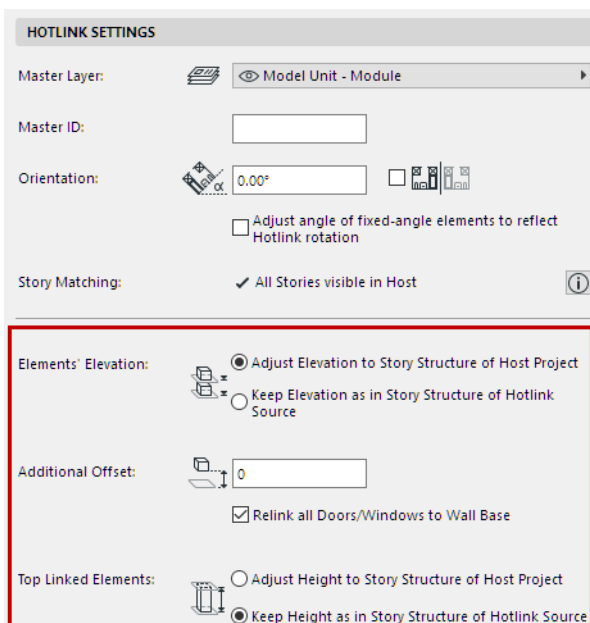
Design Options

- Choose Design Options to place from the source file. (The Main Model is always placed, but you can choose particular Options or an Option Combination.) You can vary the Option content for each placed instance of the hotlink.
- At placement, the hotlink is automatically added to the current default Option of the host file. After placement, you can redefine the Option Status of the placed hotlink.

[See Hotlinks and Design Options.](#)

Elements Settings

These controls are relevant if the Hotlink module’s story structure is not the same as the Host project’s story structure.



- **Adjust Elevation to Story Structure of Host Project:** Element elevations will follow the shift in Story Levels in the Host Project. Therefore, Hotlink Elements may shift vertically.

Note: If you choose this option, make sure to review additional options for Top Linked Elements and Door/Window elevations (see controls below).

- **Keep Elevation as in Story Structure of Source File:** All Hotlink elements will retain their original vertical positions relative to the Hotlink insertion point.

Note: When hotlinking IFC files, make sure you use the **Keep Elevation** option! Otherwise, the imported model may “fall apart” when its elements are placed in a different story structure.

Additional Offset

Enter an optional Elevation offset value. This modifies the elevation value of elements in the Hotlink.

Relink all Doors/Windows to Wall Base

If you check this: Doors/Windows whose sill or header was linked to a particular story in the source file will lose the story link, and instead will be linked to the Wall base in the host file.

Otherwise, the sills/headers of the Doors and Windows will be linked to the stories in the host file.

Top Linked Elements

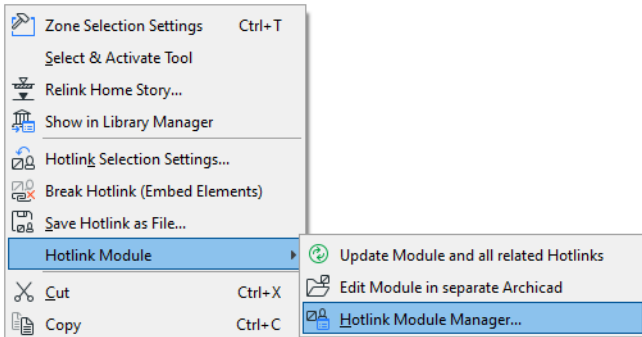
- **Adjust Height to Story Structure of Host Project:** Top Linked elements will stretch or shrink to match the corresponding story structure of the host file. Available if “Adjust Elevation” option is chosen above.
- **Keep Height as in Hotlink Source Story Structure:** Top Linked elements will not stretch (they will retain their original heights.)

Hotlink Module Manager

Use this dialog box to manage Hotlink Modules and Hotlink Sources in the project.

Open it from:

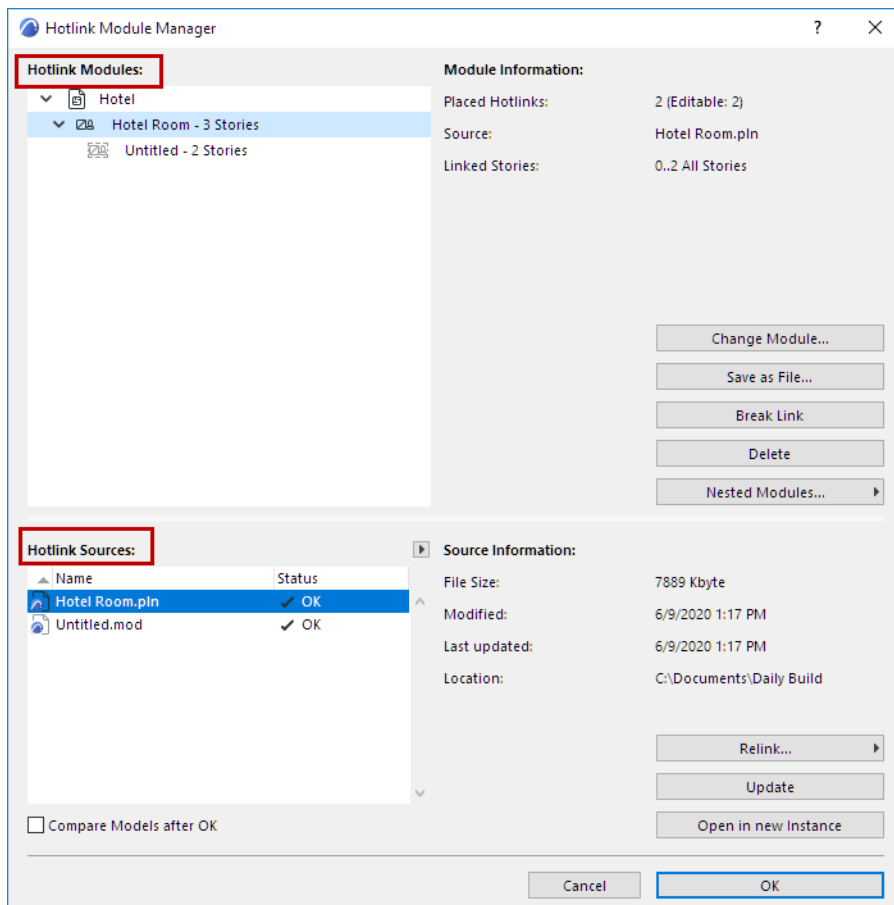
- **File > External Content > Hotlink Module Manager**, or
- the context menu of any selected Hotlink in the project, under Hotlink Module



- You can also check Hotlink Module/Source status and access Hotlink Module Manager through the Action Center. (**File > Info > Action Center**).

The “Hotlink Modules” list (in the top half of the dialog box) shows the hierarchy of modules hotlinked into the current project.

- The list is hierarchical, indicating any nested modules at a lower level in the tree structure.



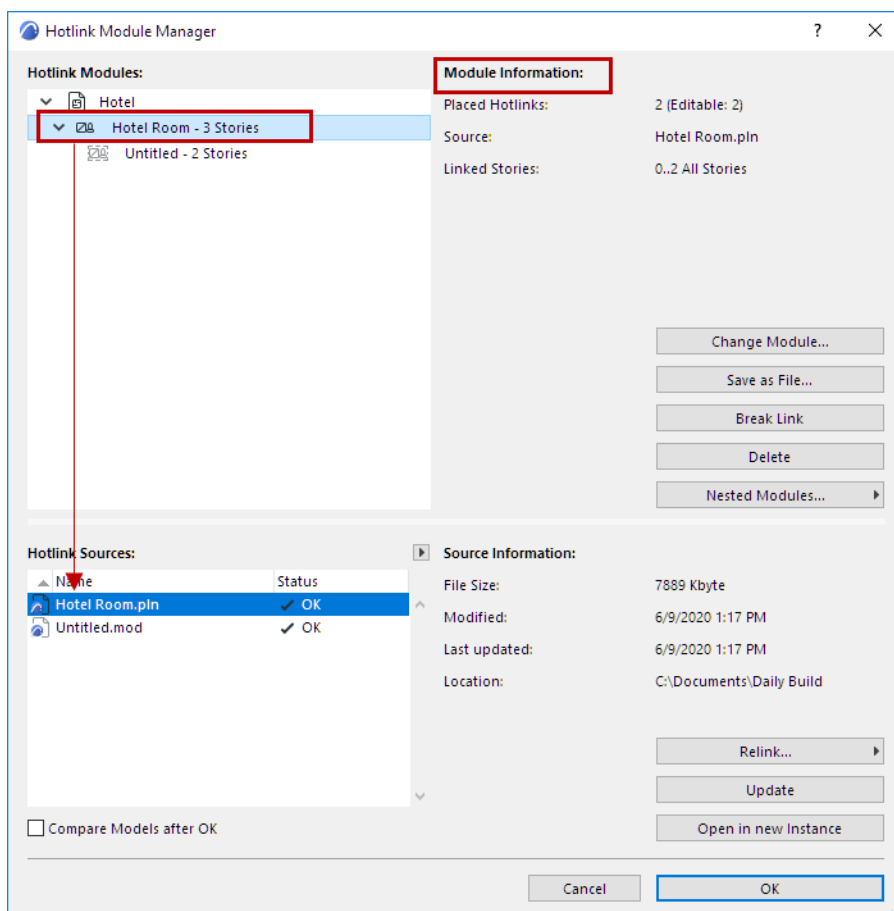
Module Information

Select any module in this list. Module Information includes:

- the number of placed Hotlinks from this module
- the name of the source file
- the linked story (or stories) of this module.
- **Conversion Info**
 - For IFC-based Modules: the Template and Translator files, and the template file's location
 - For Rhino-based Modules: import options for this module (e.g. layer and surface conversion)

You can also hover the cursor over a listed Module to see information about it, including the module's location in the host file.

Note that selecting a module in this list highlights the name of its source file in the "Hotlink Sources" list below.



Change Module

To replace the selected module with a different one, click **Change Module**.

Note: Not available for Nested Modules.

The appearing **Select Hotlink Module** dialog lists the available Hotlink Modules of this project. Select one to change the module.

See [Select Hotlink Module](#).

Save as File

See [Save Hotlink as File...](#)

Break Link

Check this box to break *all* the Hotlinks of this module and embed their elements into the host project. The elements will be regular editable Archicad elements, grouped together but no longer part of a module and no longer linked to any other source file.

This command is not effective on Hotlinks which are on a locked layer.

Note: To break any *single* placed Hotlink, select the Hotlink and use the **Break Hotlink (Embed Elements)** from Hotlink Settings.

See [Break Hotlink \(Embed Elements\)](#).

Note: The **Break** command is not available for nested modules.

The Break Link function can be helpful if, for example, a project is nearing completion and you want to create a few internal renderings. You can save a copy of the project visualization purposes: in the copy, you use **Break Link** to embed all the hotlinks of a module. This allows you to access all of the elements individually, so you can use Surface Painter much more easily.

Delete

Deletes the Hotlink Module and all of its placed Hotlinks from the project.

Hotlinks or elements which are on a locked layer will not be deleted.

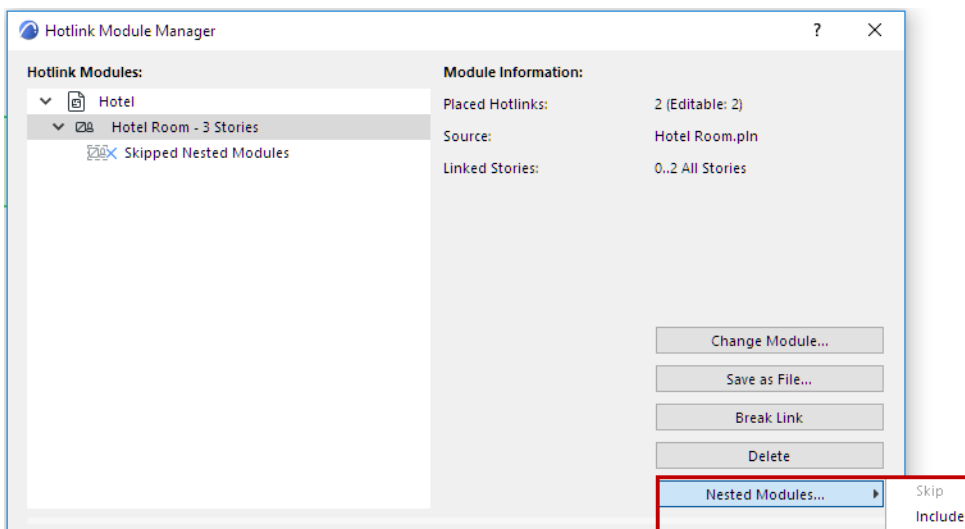
Note: The **Delete** command is not available for nested modules.

Nested Modules (Hotlink Module Manager)

Nested Modules (if any) are listed in the Hotlink Module Manager according to their location in the module hierarchy. Use the Nested Modules pop-up menu to skip or include them in the placed Hotlink:

- **Skip:** The nested modules in all Hotlinks of the selected module will be skipped (they will not be shown on the Floor Plan). They will, of course, still be present in the source module.
- **Include:** All nested modules in all Hotlinks of the selected module are shown.

To re-display a skipped nested module, select its parent module again from the Hotlink Module Manager, then choose **Include** from the Nested Modules pop-up.



If you select a nested module in the Hotlink Modules list, you cannot change or break the Hotlink (these commands and the module name itself are grayed), but you can save it as a new file.

Notes:

- IFC- or Rhino-based Modules cannot have nested Modules. However, IFC- or Rhino-based Modules can be nested inside a PLN hotlink.
- To skip or include Nested Modules for any single Hotlink, use **Hotlink Selection Settings**.

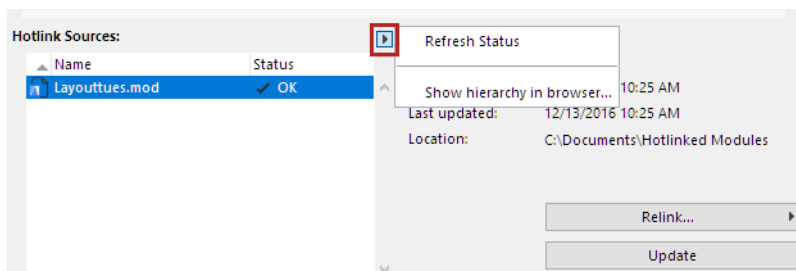
See [Hotlink Settings](#).

Hotlink Sources

Lists the source files of the Hotlink Modules in this project, by Name and Status.

- Click the pop-up arrow to refresh the Status as needed.

Note: By default, Archicad automatically checks the status of all Hotlinks each time you open Hotlink Module Manager. If this process slows down your work, uncheck the “Automatically Check Status” box at **Options > Work Environment > Data Safety and Integrity**.



See also [Hotlink Source File Status Messages](#).

- To sort data by column, click the headers
- Use the splitter bar to edit the column widths
- Select any file in this list to view, on the right:
 - **File size:** Size of the source file.
 - **Modified:** Last time the source file was modified (at its Location. The path is shown below).
 - **Last updated:** Last time the source file was updated (in the current host project). If this date is older than the “Modified” date, your source is out of date and should be updated.
 - **Location:** Path of the source file.

For any selected file in the list, you can do the following operations:

Relink

Update

Open in New Instance

Relink

Use **Relink** if any of your Hotlinked Sources are shown with “Missing” status.

In the appearing “New Hotlink” directory dialog box, provide the location of the Hotlink Source (which is probably missing because it was moved to a new location.) As long as a Hotlinked Source is missing, Archicad cannot update its module instances.

All Hotlinks based on the original source will now be replaced, based on the new source file.

Note: An IFC source file must be relinked to another file in IFC format.

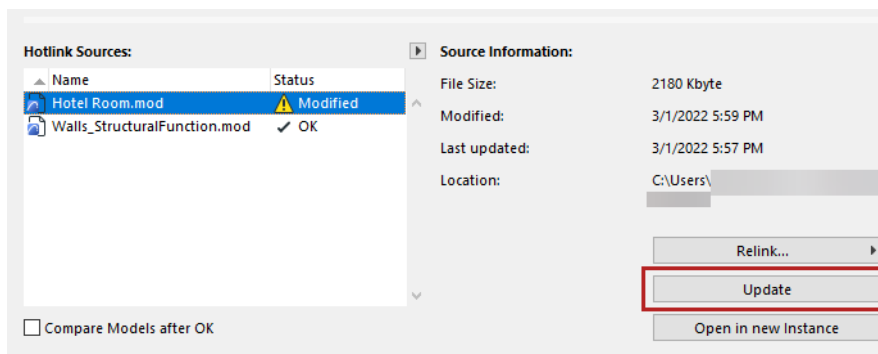
Warning: Carrying out these actions means that your Archicad project's entire Undo queue will be cleared!

If the new source file has a different story structure than the host file, you will be warned that some dimensions and labels may be lost as a result of relinking.

Note: Relink is available only for source files of top-level Hotlink Modules; you cannot relink a nested Hotlink. If a nested Hotlink is shown as "Missing," open the file into which the nested Hotlink is placed, and relink the source file.

Update

If the source file of a Hotlink module has been modified at its original location, its status in the Hotlink Sources list is shown as "Modified." You must update the source file and thereby update all of the module's placed Hotlinks: select the source file and click **Update**.



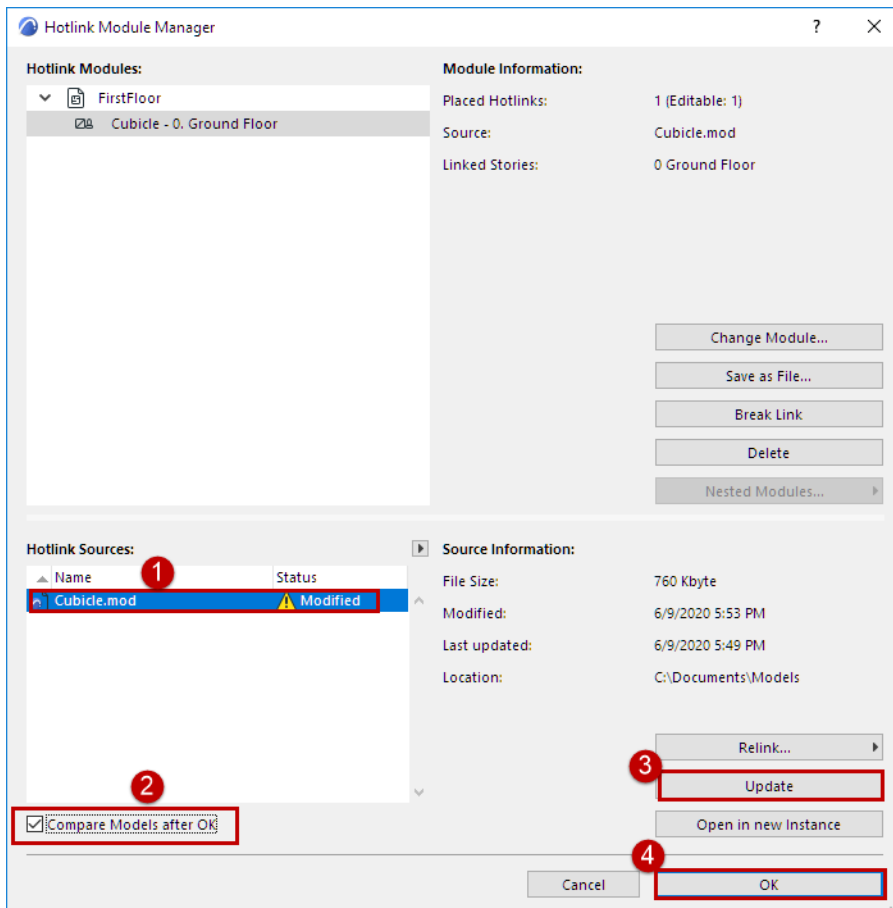
This command is also available from the context menu of a placed Hotlink, as: **Hotlink Modules > Update Module and All Related Hotlinks**.

Note: To set update preferences when opening or publishing a file containing Hotlinks, use the drop-down options at **Options > Work Environment > Data Safety & Integrity > Hotlink Update**.

See [Hotlink Update](#).

Update, then Compare

When updating a Hotlink Module, you can launch Model Compare in one step, which helps you visualize exactly what has changed as a result of the update.



See [Compare Archicad Models Before and After Hotlink Update](#).

Open in New Instance

Note: Not available for IFC or Rhino source files.











Opens the selected source file in a new instance of Archicad.

See also [Edit Module in Separate Archicad](#).

Hotlink Source File Status Messages

- **OK:** This Source file's modules are up to date.
- **Modified:** Source file has been changed. To update the modules in your project that originate in this source file, you must use Update.
- **Missing:** You have placed modules from this file in your project, but Archicad cannot locate the source file. To enable the updating of these modules, you must first locate the source file (use **Relink** to define the missing file's location).
- **Not Accessible:** The source file is a legacy Teamwork project located on a BIMcloud. You do not have access to this source, because you are using either a Demo version of Archicad, or a non-Teamwork network key.

- **Legacy:** Legacy source files are those saved in an Archicad version prior to Archicad 17. To enable these modules to be updated, you must first open the legacy source file and re-save it in Archicad version 17-26.

Module Status	Icon
Normal	
Nested	
Modified	
Updated	
Deleted	
Change Module	
Link Broken	
Relink Source File	
Skip Nested Modules	
Missing	

Create Module (.mod) File

A Module format file (.mod) is a simplified type of Archicad file, which can be placed as a Hotlink Module or merged into a project.

- You can save all project elements or just selected elements and selected stories in the module file
- A .mod file contains only model and Floor Plan data
- A .mod file includes Embedded objects, but no information from Section/Elevation/IE, Worksheet/Detail, or 3D Document data
- A .mod file refers only to attributes actually used by their elements. Thus, a .mod file is much smaller than a .pln file
- The **Save Hotlink as File** always saves the hotlink in .mod format.

Note: It is also possible to split a project into multiple PLN files (as opposed to Module-format files). [See *Save Selection in PLN Format*](#).

Topics in this section:

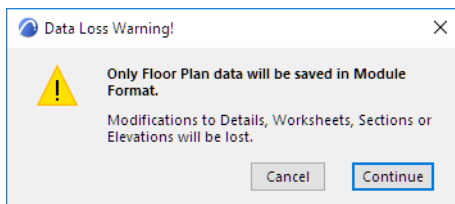
[Save as .mod file - from Floor Plan](#)

[Save a view as a .mod file - from Publisher](#)

[Module Save Options](#)

Save as .mod file - from Floor Plan

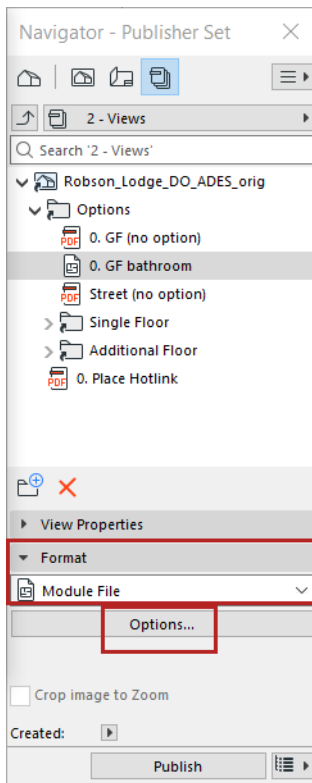
1. In the Floor Plan, use **File > Save As**, and choose the .mod file format.
2. You receive a warning that the module file will contain only Floor Plan data.



3. Click OK to create the module file and close the dialog box.

Save a view as a .mod file - from Publisher

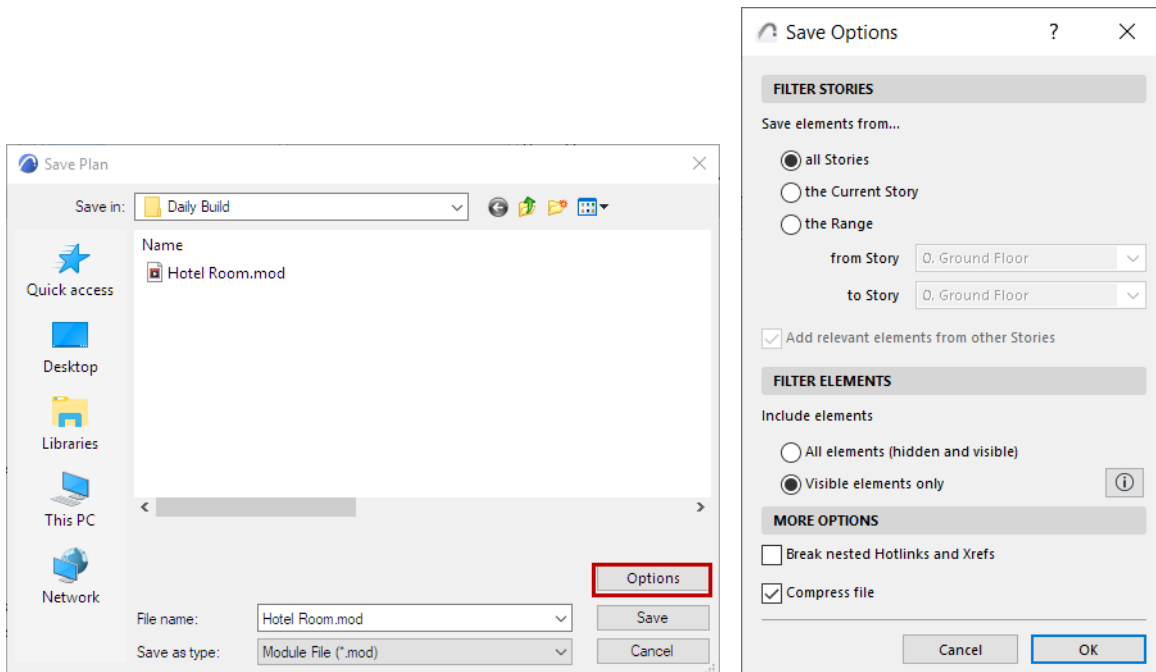
Available for views saved from the Floor Plan.



Click **Options** to set which stories and elements should be part of the module. (See below.)

Module Save Options

Click the **Options** button in the Save As dialog to bring up **Save Options**.



Filter Stories

- The default choice is **All Stories**

- Alternatively, choose just the **Current Story**, or a specified **Range** of stories to include in the module file. Any element that falls within this range will be included in the module.
 - **Add relevant elements from other Stories:** If you choose either **Current Story** or a **Range**, check this to include all elements that are displayed on these specified stories (because their Show on Stories control is set to Relevant Stories), even if their home story is out of the module's story range.

Filter Elements

Choose to save all elements of the Floor Plan, or just the visible ones.

- **All elements (hidden and visible):** The saved module will contain all project elements from the Floor Plan - regardless of their Renovation status or Option status.
- **Visible elements only:** The saved module will only contain elements which are
 - on visible layers
 - visible due to partial structure display settings
 - are on **active** Design Options
 - are **existing** based on their Renovation status

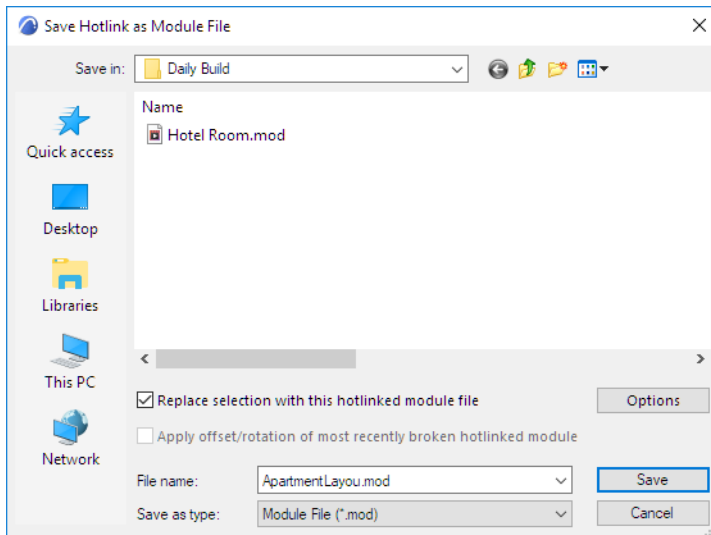
Break nested Hotlinks and Xrefs

If the module file contains any nested Hotlinks and/or Xrefs, check this box to break those links, if the source files may become unavailable or are no longer needed.

See [Place Hotlink](#).

Save Selection as Module

1. Select elements in the Floor Plan. (To select elements from several stories, use the Marquee tool's multistory option - see [Single Story or All Stories.](#))
2. Go to **File > External Content > Save Selection as Module** to open the following dialog box:



3. Enter or choose a file name.
4. Choose the Module File (.mod) format.
5. Optionally: Check **Replace selection with this Hotlinked Module file** to immediately replace the selected elements with the module file.

Note: In Teamwork, the “Replace selection...” checkbox is NOT available.

- Also check **Apply Offset and Rotation of Edited Module** to retain the custom offset/orientation settings, if you are replacing an original module with a new one. (This option is relevant when several hotlinks of the edited module were placed using different rotations and offsets.)

6. Click **Save**.

Notes:

- If you have selected certain elements and copied them to the clipboard, you can save a module that will include those elements only: use **File > Save As**, then choose the **Module File from Clipboard** file format.
- If the selection contains elements that cannot be modified (e.g. they are locked, located on a locked layer or reserved by another user), an alert will warn you. You will then have the choice of either creating the module with the editable elements only, or canceling the operation.
- You can also save selected elements in PLN format. See [Save Selection in PLN Format.](#)

Save Selection in PLN Format

Use this feature to split an Archicad model into multiple PLN files, which can act as Hotlink Modules.

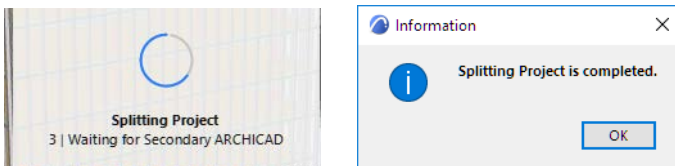
For example, you can save the building facade as a separate file from the rest of the building, or save two wings of the building in separate files.

Each separate PLN file retains the full 2D content as displayed in the Navigator - including Layouts, Worksheets, and Details. The two parts of the original model are thus “split”.

(In contrast, saving a selection in .mod format saves model and Floor Plan data only.)

Save Selection as Module

1. Open the original model, and select the part you want to save as a separate PLN file.
2. Use **File > External Content > Save Selection as Module**:
3. Choose the Archicad Solo Project (.pln) format.
4. Click **Save**.
5. Archicad automatically splits the project. The original PLN file and the saved PLN selection are opened in two separate Archicad's. The feedback lets you know what is happening.



Use the checkboxes to hotlink the two projects to each other in one step, as described in the following:

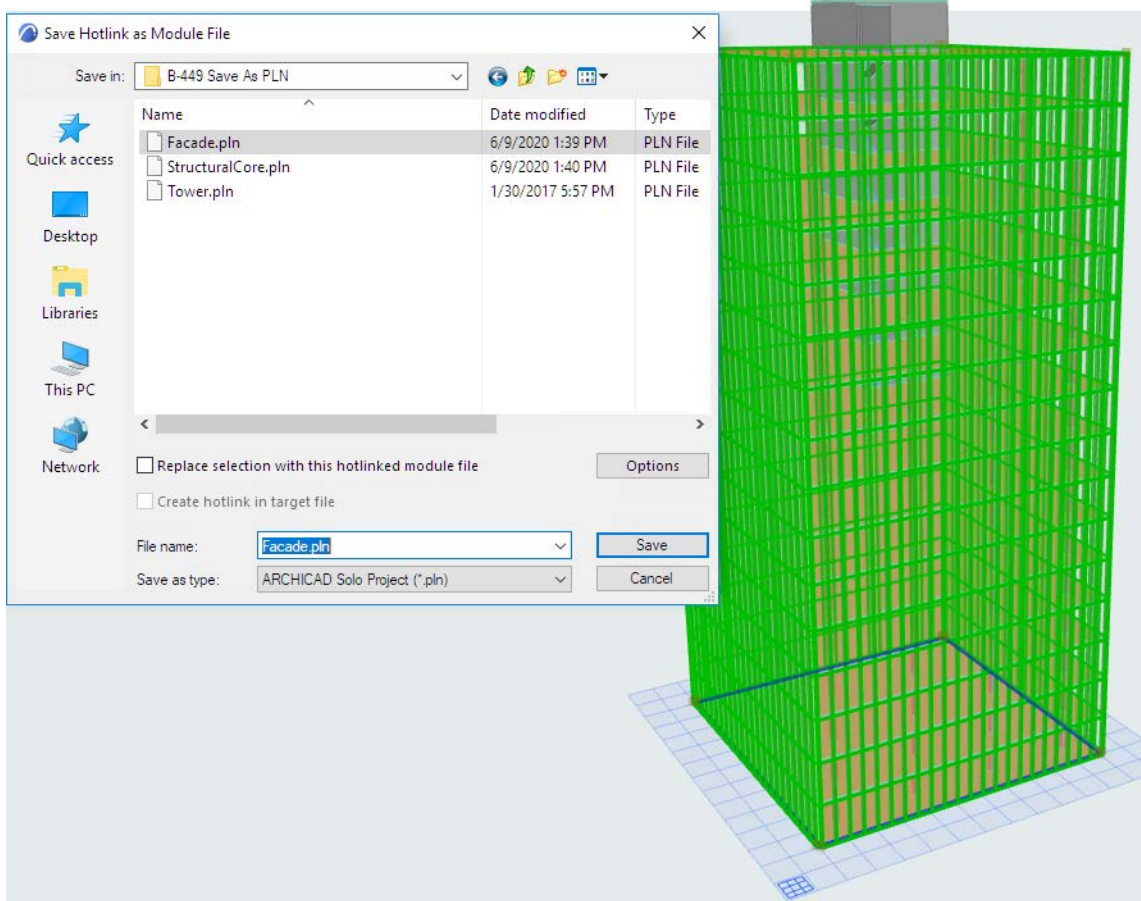
Split and Hotlink Model Parts

In this example, the facade of a highrise is selected and saved as a PLN file, and hotlinked with the original tower file.

1. Open the model file (Tower).
2. Select the facade elements (the Curtain Walls) in Floor Plan or the 3D window.

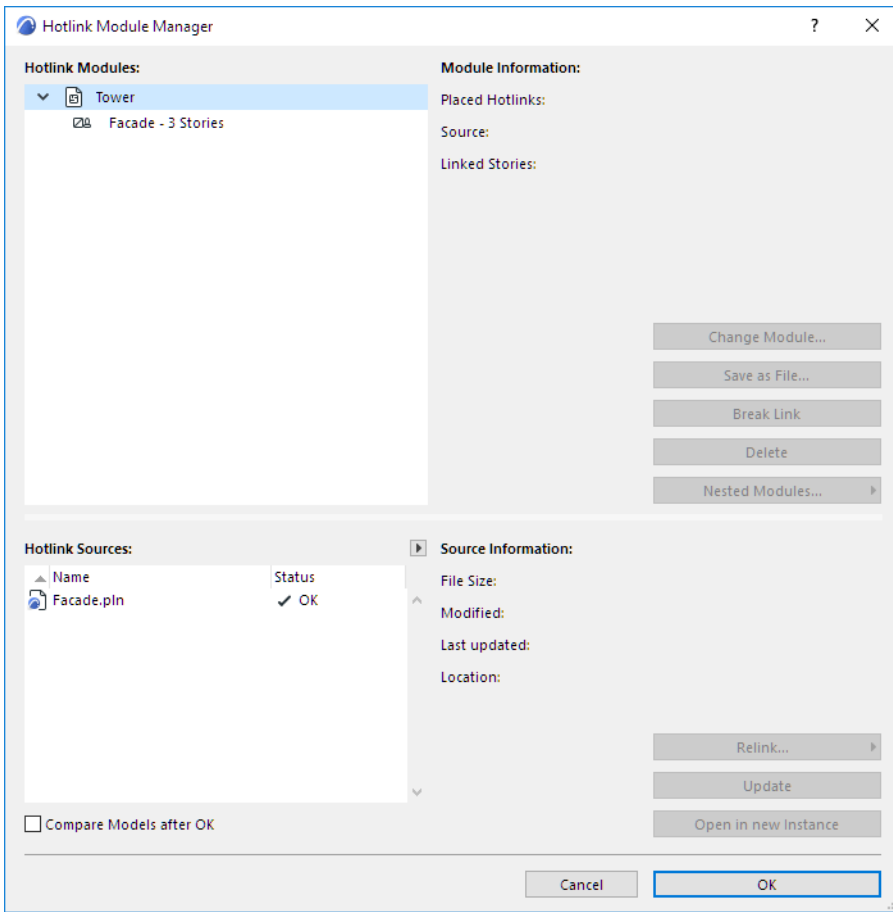
Note: When selecting on Floor Plan, use the Marquee tool's multistory option to include elements from several stories - [see *Single Story or All Stories*](#).

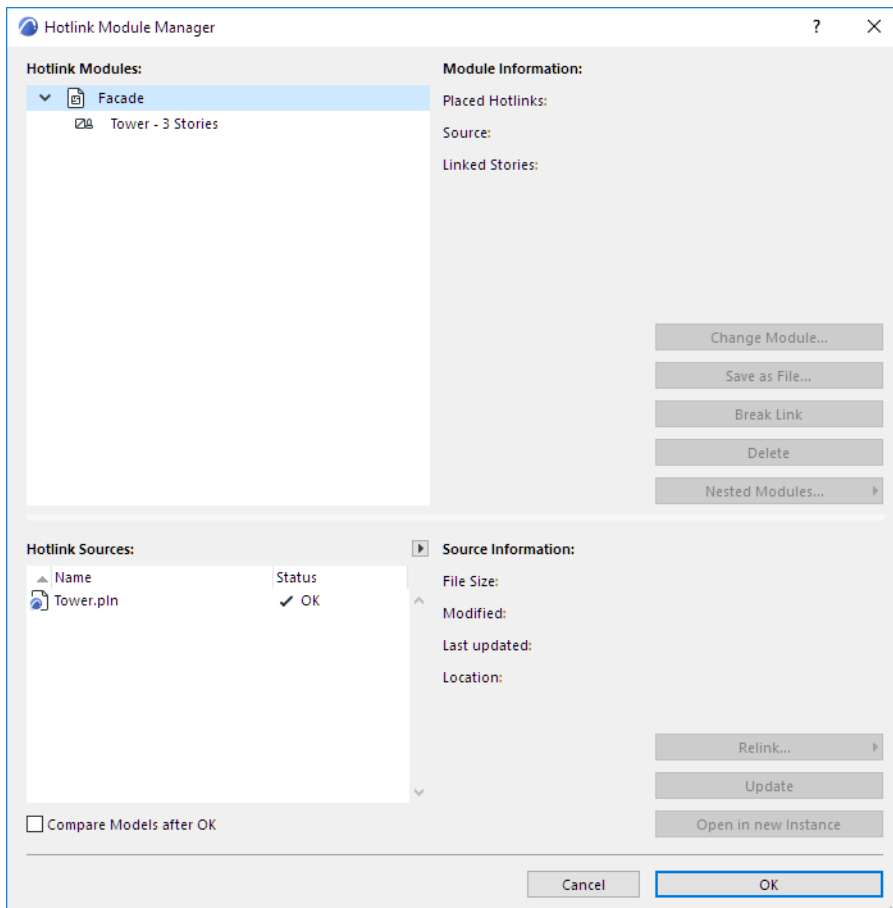
3. Go to **File > External Content > Save Selection as Module** to open the following dialog box:



4. Enter or choose a file name.
5. Choose the Archicad Solo Project (.pln) format.
6. Check both options:
 - **Replace selection with this hotlinked module file**
 - Create hotlink in target file
7. Click **Save**.
8. Archicad splits the project and opens the saved selection (the Facade) in a second Archicad instance.

- 9. Now you have two Archicad projects running: the Tower (with the Facade hotlinked in), and the Facade (with the Tower hotlinked in).





You can save any number of PLN selections from the original Archicad model and hotlink them automatically to the original model.

However, if you also want to hotlink the resulting “secondary” PLN files to each other, you must do so manually.

Collision Detection

Use this function to identify collisions among 3D elements in the model. A collision occurs when two or more elements physically intersect.

Collision Detection works by checking for collisions between any two groups of elements. The groups are defined by criteria chosen by you.

For example, you might run Collision Detection:

- Between construction and MEP elements. (MEP elements can come from hotlinked or merged external MEP IFC files, or else modeled inside Archicad.)
- Between concrete and steel construction elements
- Between elements classified as different product/element types (by Uniclass, OmniClass, UniFormat or any other classification system)
- To check the headroom clearance for escape routes or access for the disabled

Collision Detection is available from the Floor Plan and 3D windows only.

Access from:

- Design > Model Check > Collision Detection
- or click the Collision Detection icon from the Model Check Report

Topics in this section:

[Understanding Collisions](#)

[Run Collision Detection](#)

[Define Collision Groups](#)

[Set Volume/Surface Tolerances](#)

[Example: Collision Detection with Volume/Surface Tolerances](#)

[Example: Headroom Collision Check](#)

[Exempt Building Material from Collision Detection](#)

Understanding Collisions

- Only elements on visible Layers are considered
- Elements that are just touching are not collisions
- Solid Element Operations and Priority-Based Connections are already resolved in the model and do not cause collisions
- 3D Cutting Planes are ignored (the entire model is considered regardless of Cutting Planes)
- You can define Tolerances for colliding Volumes and/or surfaces: if the colliding volume/surface area does not exceed the defined Tolerance value, it does not count as a collision

[See *Set Volume/Surface Tolerances*.](#)

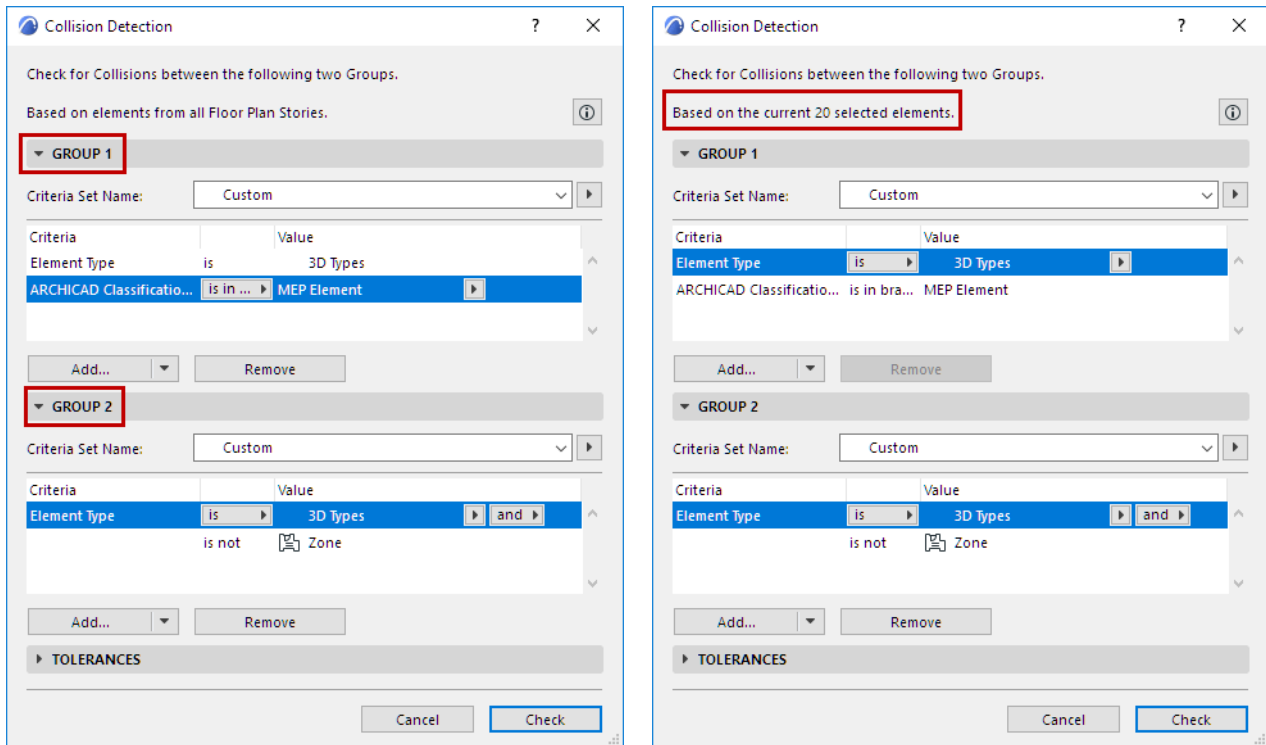
- The status of Building Materials (“Participates in Collision Detection” property) is considered

[See *Exempt Building Material from Collision Detection*.](#)

Run Collision Detection

1. Go to the Floor Plan or 3D Window.
2. Go to **Design > Model Check > Collision Detection**.
3. In the dialog box, define criteria for the two groups of 3D elements that you want to check.

- If any elements are selected, Collision Detection will check only selected elements! See the top of the dialog for feedback on any current selection.



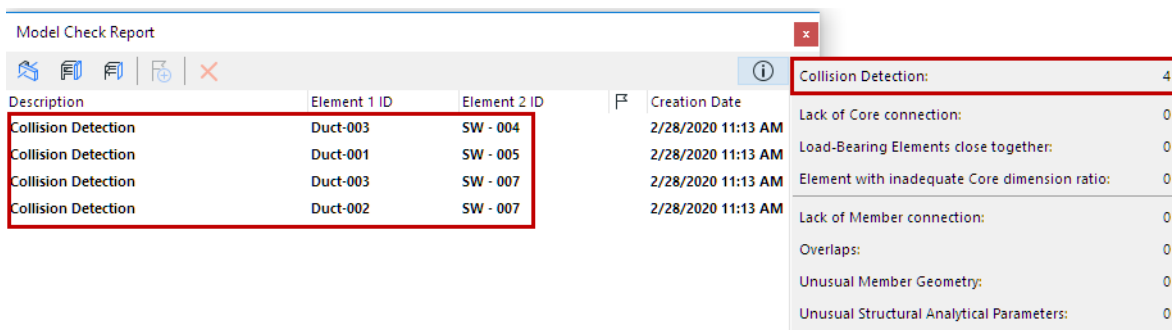
See [Define Collision Groups](#) for more information.

4. Set Surface and/or Volume Tolerance values as needed.

See [Set Volume/Surface Tolerances](#) for more information.

5. Press **Check** to run Collision Detection.

6. The **Model Check Report** opens to show results. Collisions-type problems are listed as such in the Report. Click Info to see the number of Collisions.



As with other problems listed in the Model Check Report, you can view the related elements in the model and create Issues.

See [Model Check Report](#).

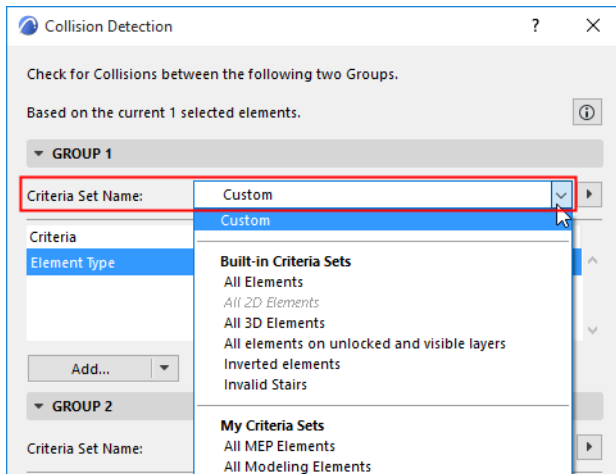
Related topic:

[Surface Criteria in Graphic Override Rules and Collision Detection](#)

Define Collision Groups

Under Group 1 and Group 2, use the criteria interface to define the collision groups.

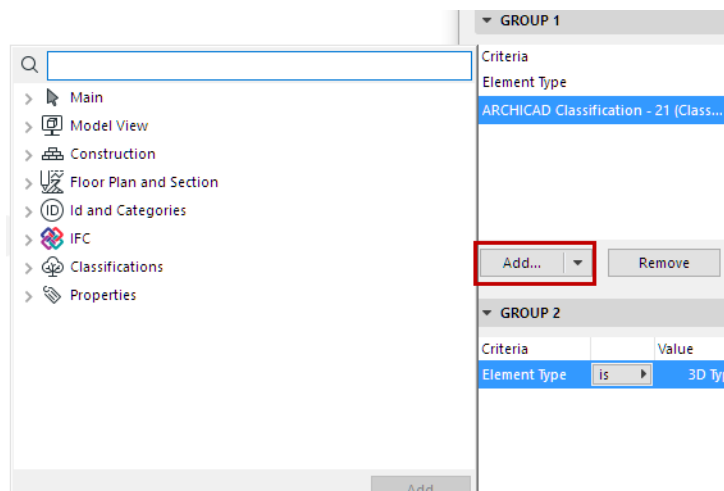
- To check collisions between all element types, use the same criteria for both groups.
- To check collisions between 3D elements and MEP elements only, limit one of the groups to elements classified as MEP.
- To use a saved Criteria Set to define a Collision Group: click the **Criteria Set Name** arrow to list the project's criteria sets:



- If you pick a Criteria Set here, its contents will overwrite the Collision Group's current criteria
- Listed Criteria Sets may be disabled if the set includes criteria not applicable to Collision Detection (e.g. 2D element types, or IFC properties)

See also [Predefined Criteria Sets](#).

- Click **Add** to bring up a dialog to choose any Properties and Parameters by which to limit the group.

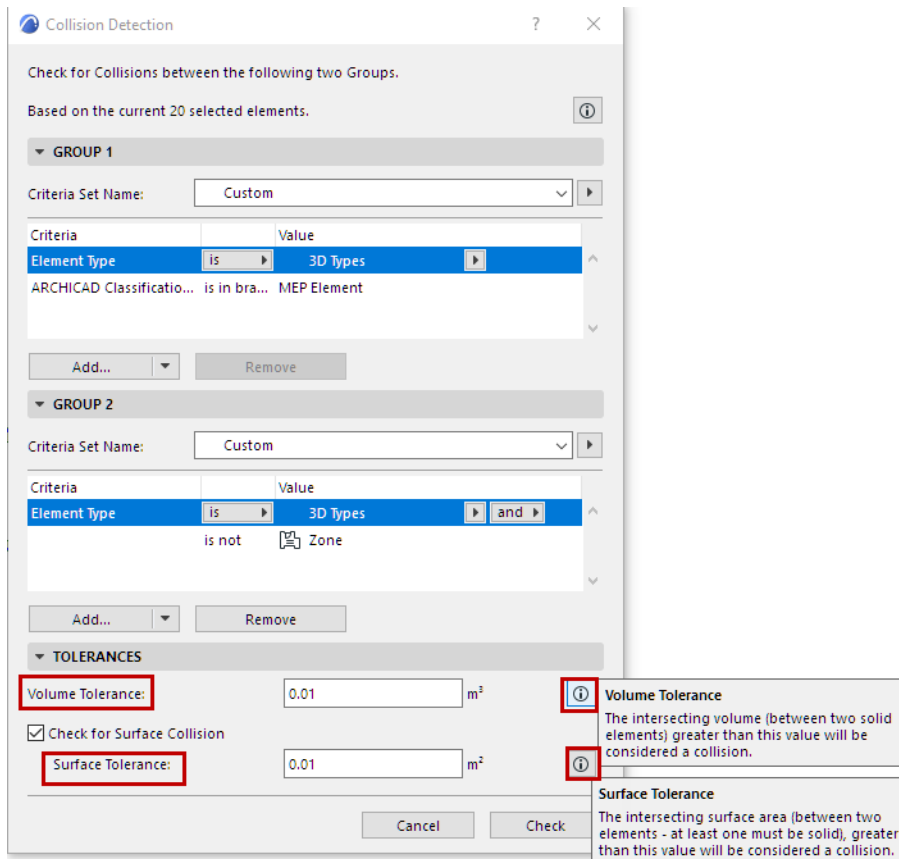


Set Volume/Surface Tolerances

Optionally, define Tolerances for the volume and surface of elements' colliding parts.

Elements are considered to collide if the colliding parts' volume or (if enabled) the colliding parts' surface area exceeds the Tolerance value.

Note: The units here are those of the project's Working Units (Options > Project Preferences).



Volume Tolerance

Collision Detection always considers colliding elements in terms of the **volume** of the colliding parts.

You can set an optional Volume Tolerance: If the volume of the two elements' colliding parts exceeds this tolerance value, the elements are treated as colliding.

Check for Surface Collision

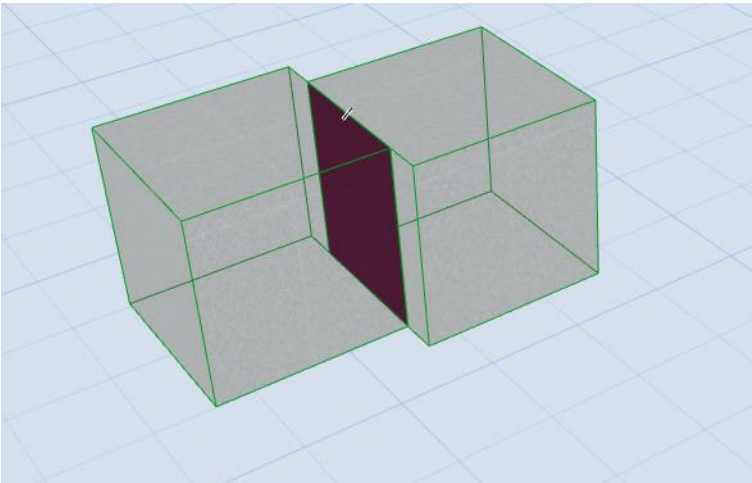
As an additional option, the Collision Detection function also checks the surface area of the colliding parts. If you enable Surface check and set a tolerance, then elements are treated as colliding if the colliding surface area exceeds this Surface Tolerance value.

Note: If Surface Check is on, it is possible that two elements will be considered as colliding even if the colliding volume doesn't exceed the Volume Tolerance - because the colliding surface area exceeds the Surface Tolerance. See the example below.

Example: Collision Detection with Volume/Surface Tolerances

Run Collision Detection with the Tolerance settings shown here.

The following two cubes are one cubic meter each. They collide slightly; the overlapping volume is very slight (it doesn't exceed Volume Tolerance), but the overlapping surface is considerable (it exceeds Surface Tolerance). As a result: Collision Detection counts these two elements as colliding.

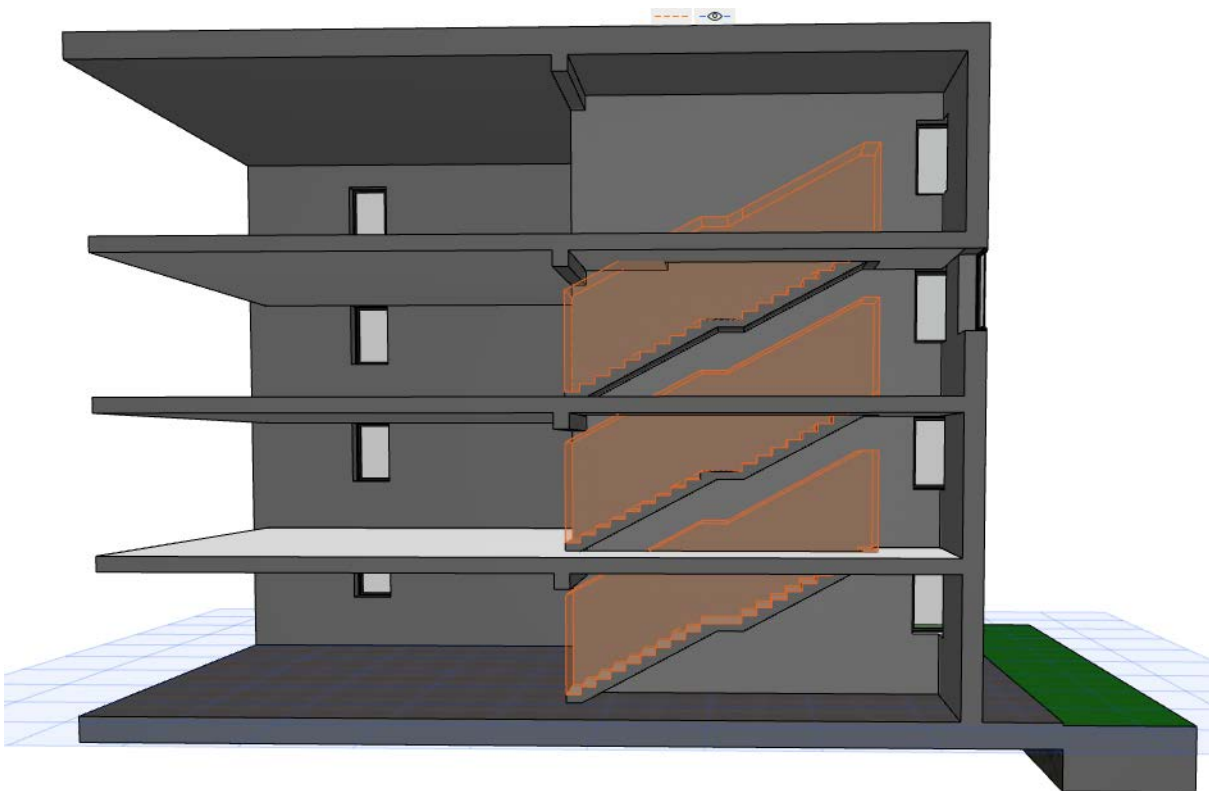


Two Colliding Elements (based on Surface Area of Colliding Part)

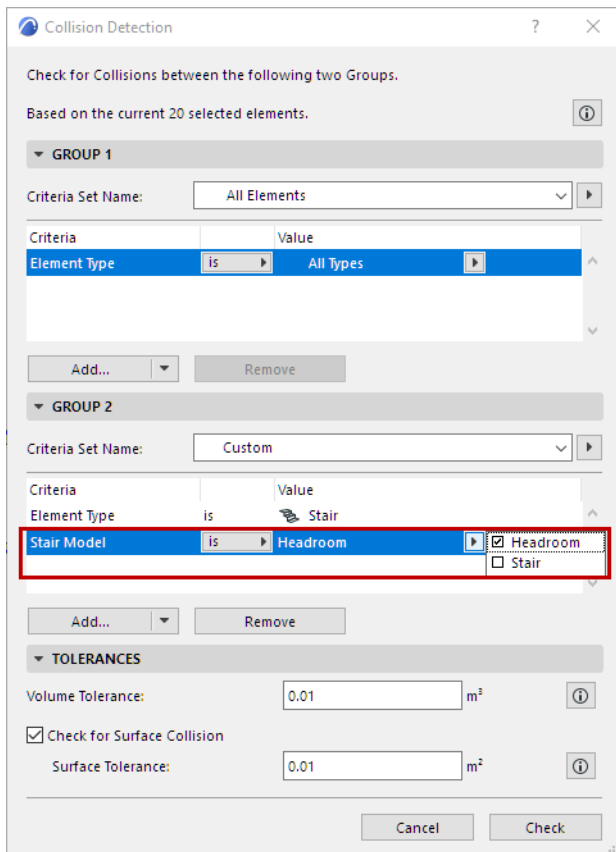
Example: Headroom Collision Check

We will check for Headroom collision in this 3D view of the model.

Note: The headroom visibility is turned on at **Document > Model View > Model View Options** (Stair Options).



1. Go to **Design > Model Check > Collision Detection**.
2. In the dialog box, define the collision criteria.



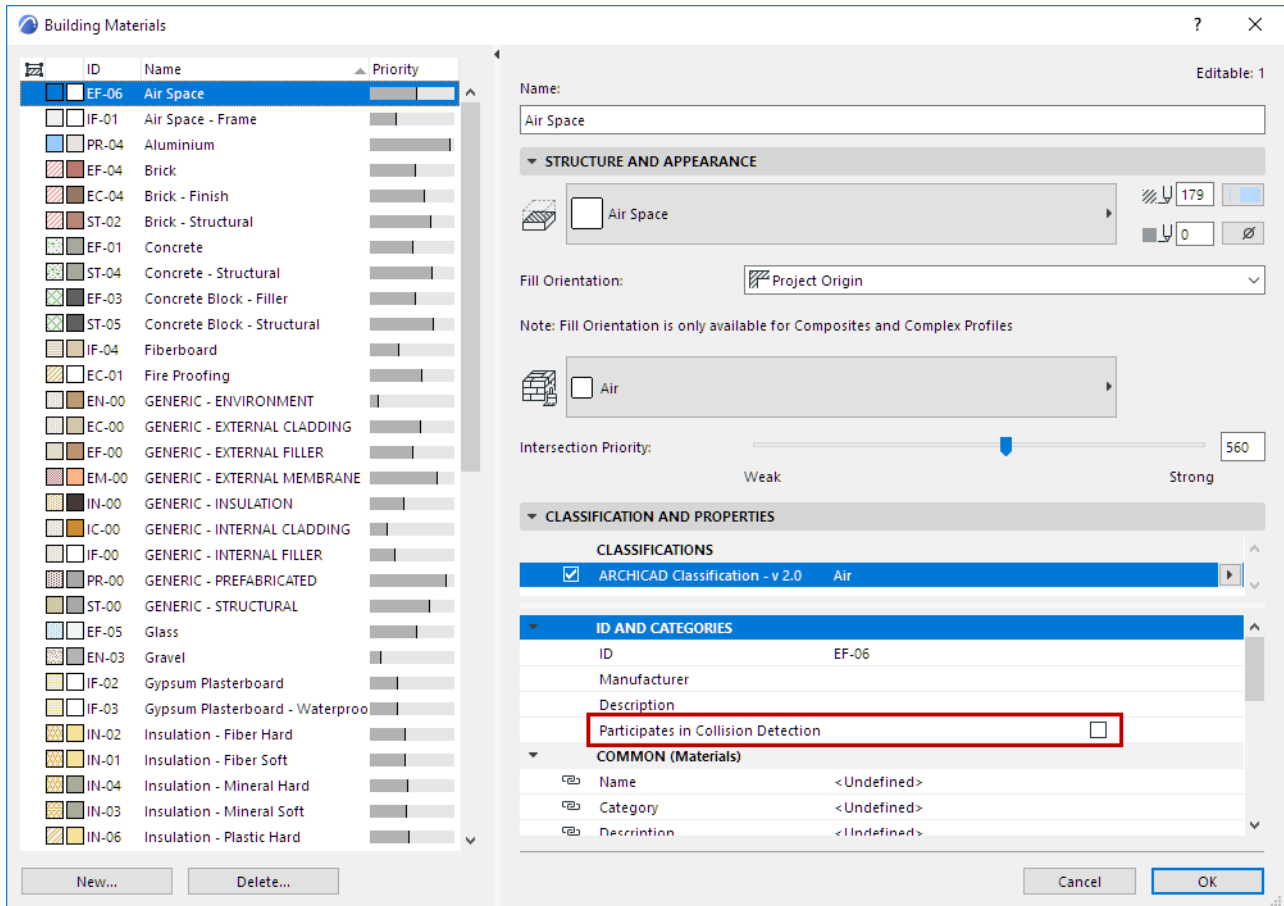
- For **Group 1**: all 3D elements
 - For **Group 2**: Stairs, considering the Headroom part of the Stair Model.
3. Press **Check** to run Collision Detection.
 4. The **Model Check Report** opens to show results. Here, for each result you can view the related elements in the model and create Issues as needed.

[See Model Check Report.](#)

Exempt Building Material from Collision Detection

Collision Detection takes the elements' Building Materials into account. Building Materials can be set to participate in Collision Detection, using the checkbox "Participates in Collision Detection" in the Classification and Properties panel of Building Materials, at **Options > Attributes > Building Materials**.

If this checkbox is turned off, then the Building Material will not participate in Collision Detection.



For example, an MEP element (e.g. duct) that runs through a Building Material used as an air gap will not be considered as a collision.

Archicad Design Checker

Design Checker enables users to capture errors as they design, right inside Archicad. Use it at any stage of the design process to validate and improve the quality of your design model geometry and data - a kind of “BIM spell checker”.

Design Checker utilizes a cloud service called Solibri Inside, developed by Nemetschek sister company Solibri. You can use basic functions of Design Checker from Archicad without any Solibri account.

Availability

To use Archicad Design Checker, you need:

- Archicad 26 Update 2 (build 4019) or newer
- Sign-in with Graphisoft ID
- Accept the Conditions of use (this dialog appears when you start Design Checker). Your user identification data will be transferred to Solibri, to verify your eligibility to use Design Checker.

Benefits and basic functions

- No installation needed to use the basic functions of Design Checker, available in Archicad out of the box.
- Since Design Checker works as an Archicad command, there is **no need to export files** to an external application and import the results, which greatly simplifies your quality assurance workflow.
- You can perform design quality checks **early in the design process** to guarantee the quality of your design model geometry and data.

Additional functions

To access additional functions of Design Checker:

- Register for Solibri Inside and/or upgrade your account. Please see [Solibri Inside](#).
- Archicad subscribers may also be eligible for a Solibri Inside promotional offer. Please check the [Graphisoft website](#) for details.

How it works

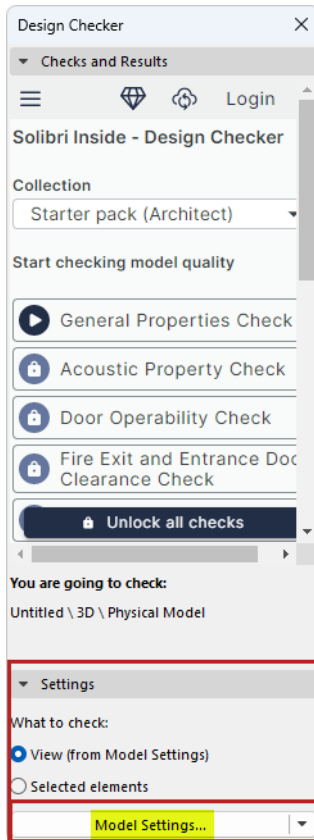
1. Open Design Checker with one of the following:
 - Click the Design Checker **on the main toolbar**



- **Design > Model Check > Design Checker**
2. To continue, check the box to agree to the conditions of use. Your user identification data will be transferred to Solibri, to verify your eligibility to use Design Checker.

The “Design Checker” Palette opens, which lists the available Design Checks. The set of available checks is updated from time to time.

3. In the Settings panel at the bottom of the Palette, do one of the following
 - Select a view (all elements in that view will participate in the Check),
 - Select elements to run the Check on



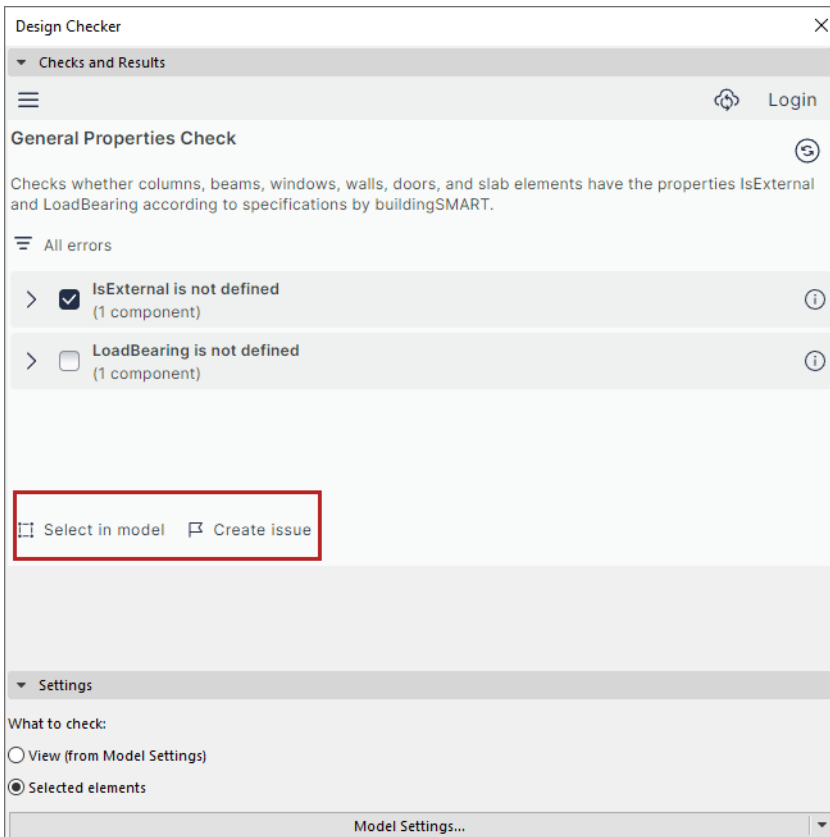
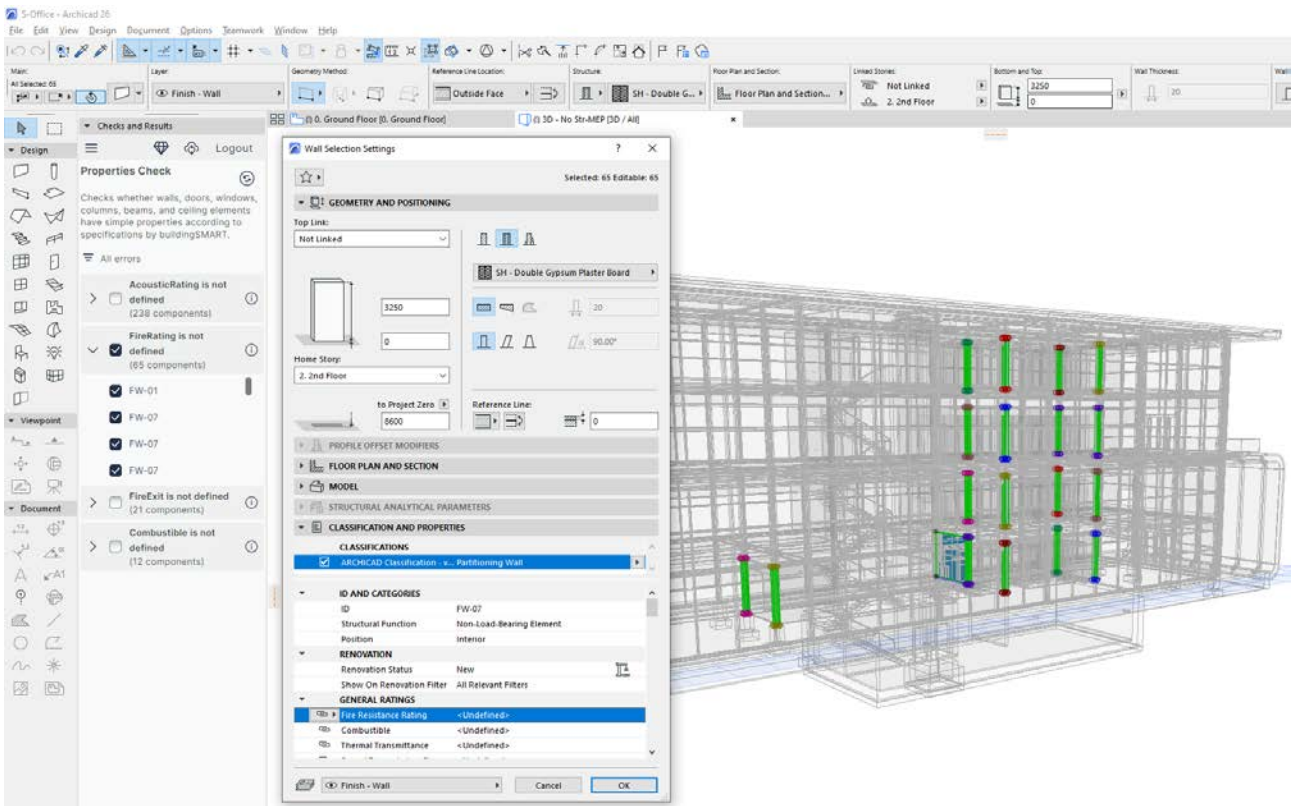
4. Click Model Settings.
5. In the appearing Settings dialog, specify an IFC Translator for exporting data to Solibri Inside. We recommend the “IFC4 Reference View-based Export” translator.

Note: If you are using an IFC2x3 translator, make sure that the “Material preservation mode (IFC2x3 only)” field of the “Geometry Conversion for IFC Export” Dialog is set to the “Never explode elements, preservation is not guaranteed”. (See [Material preservation mode \(IFC2x3 only\)](#))
6. In the Design Checker dialog, click any of the available Checks to run it (for example, “General Properties Check”).

Design Checker generates the required data and sends it to Solibri Inside for checking operations.

The Palette displays the resulting issues, along with their related elements.

In the following image, the “Properties Check” has found that selected elements have several of their basic Properties set to “Undefined”.



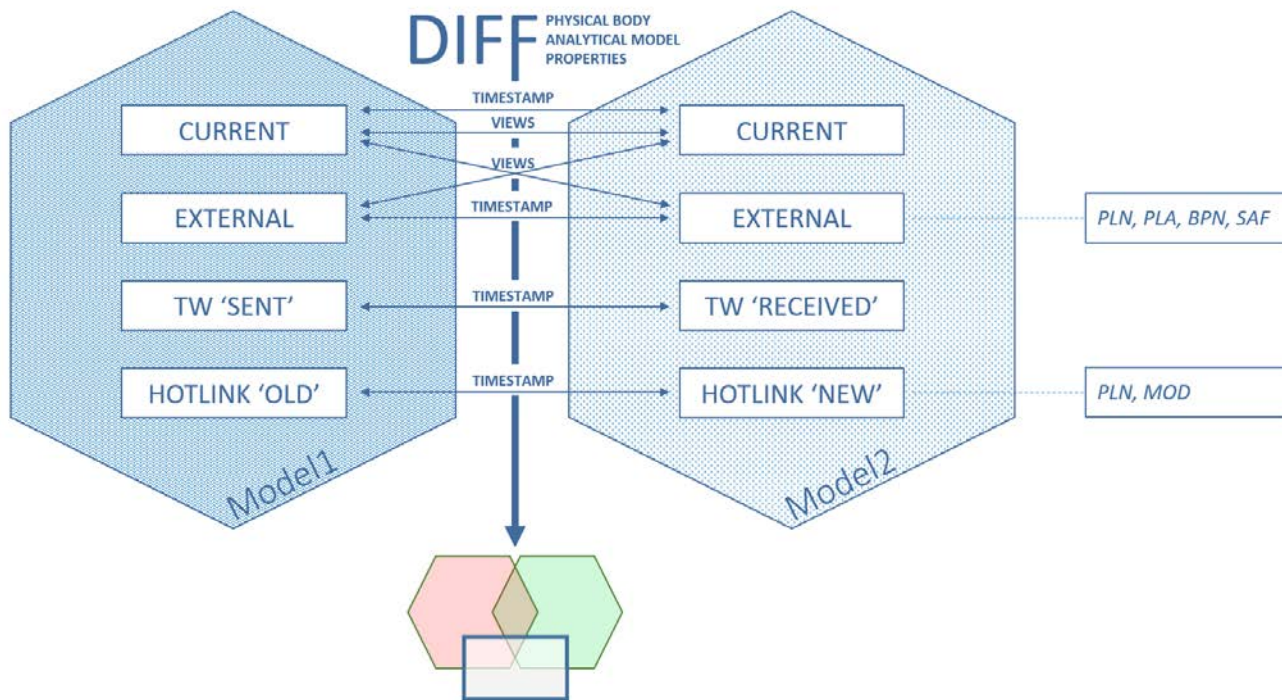
- You can select the identified elements to visualize and correct the problems.
- You can also create an Issue in Archicad’s Issue Organizer.

Model Compare

Use Model Compare to easily compare and visualize changes between two models. Changes are identified based on “new”, “modified”, and “deleted” 3D elements. 2D elements are not considered.

Model Compare compares 3D views from any two models, which can include:

- Current physical or structural analytical model (even two different timestamp versions of the same current model)
- External file (PLN, PLA, or SAF)
- Teamwork model: compare the two versions before and after Receive Changes
- Current model: compare the two versions before and after Hotlink update
- IFC model (This is an experimental feature. [See IFC Structural Analysis Model Import/Export - Experimental Feature.](#))



Model Compare offers numerous options to visualize changes between two groups of 3D models

In each workflow, the Compare command generates a view of the two compared models in a separate “Model Compare” window.

Use the Model Compare Palette to filter, list, and zoom to changes, and to see which parameters have changed.

You can create an Issue out of any of the changes.

Notes:

- The Model Compare function compares two models based on a fixed timestamp for each. Therefore, if you later change either model (e.g. modify view setting, delete an element), the previous Model Compare results cannot be updated. Run the Model Compare again, based on the updated models.

- Model Comparisons of large and complex projects may take a while, but the process runs in the background. You can continue to work uninterrupted on the current model.
- The Model Compare tab gives you feedback while the process is underway and also when it is finished.
- You can run multiple Model Compare processes simultaneously.

Topics in this section:

[Compare Teamwork Model Versions \(Before and After Receiving Changes\)](#)

[Compare Two Saved Versions of Current Model](#)

[Compare Two Views of a Single Project](#)

[Compare Archicad Models Before and After Hotlink Update](#)

[Compare Structural Analytical Models](#)

[Select View for Compare](#)

[Filter Changes Shown in Model Compare Palette](#)

[Model Compare: Review the Changes](#)

[Workspace Colors for Model Compare](#)

Related Topic:

[Create Issue from Model Compare Change List](#)

Compare Teamwork Model Versions (Before and After Receiving Changes)

Use this Compare workflow in Teamwork after receiving changes, to see what has changed in the model. There are two “Compare” commands in the Teamwork context:

Send and Receive, then Compare

Send and Receive as usual. This results in a new local cache (after Receive): the “New Model”. This new model is compared to your previous local cache (prior to the Send), which is the “Old Model”.

Receive, then Compare

As in the above scenario: you receive changes made by others, creating a new local cache (“New Model”), and then compare it to your old local cache (“Old Model”).

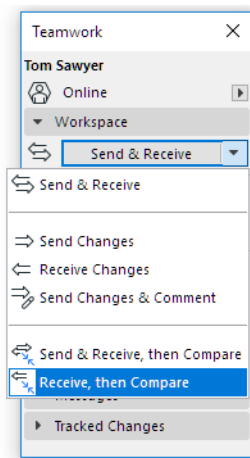
The only difference is that the first command shares your own work to the BIMcloud, the second command does not.

Example Workflow

In this example, two architects are using Teamwork to collaborate in Archicad. User 1 makes changes and sends them to the server.

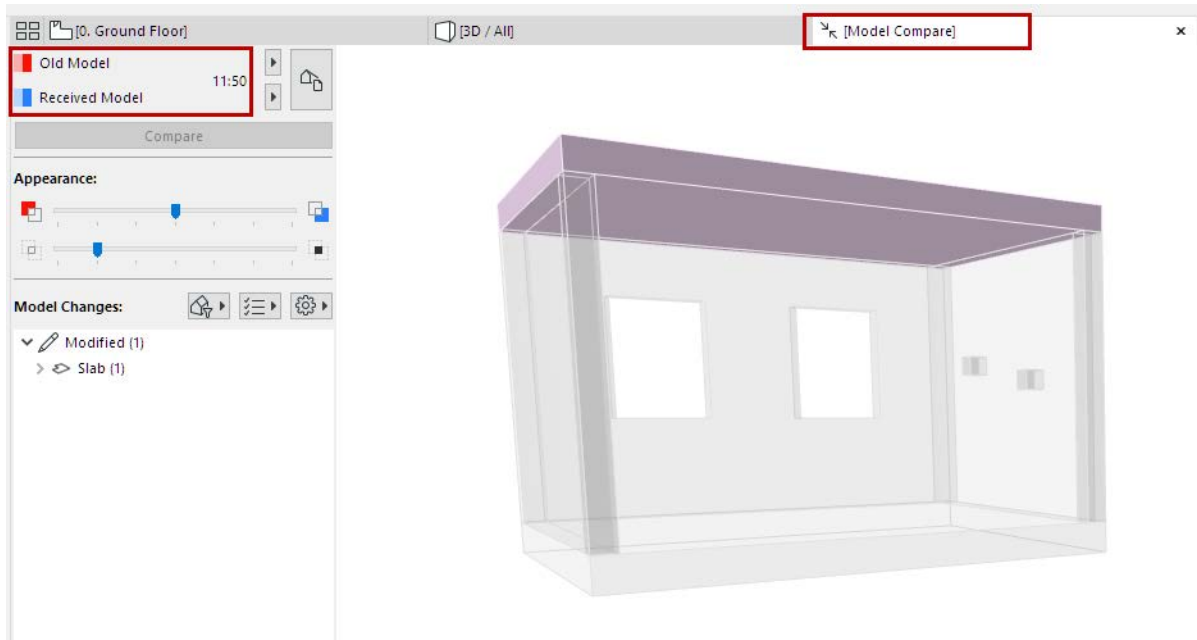
User 2 wants to see what has changed.

1. From the Teamwork menu or Teamwork palette, the user clicks **Receive, then Compare**.



2. User 2 receives changes from the BIMcloud.
3. The Model Compare begins in the background. The Model Compare tab shows that generation is underway, then a green checkmark when it is done.

4. Open the Model Compare window. The palette indicates that - after receiving changes from BIMcloud - the new version ("Received Model") is compared to the previous version ("Old Model", as it existed prior to Receive).



5. Check the results in the Model Compare window.
Review the Model Compare palette to see what has changed.

[See *Model Compare: Review the Changes.*](#)

6. As needed, create Issues.

[See *Create Issue from Model Compare Change List.*](#)

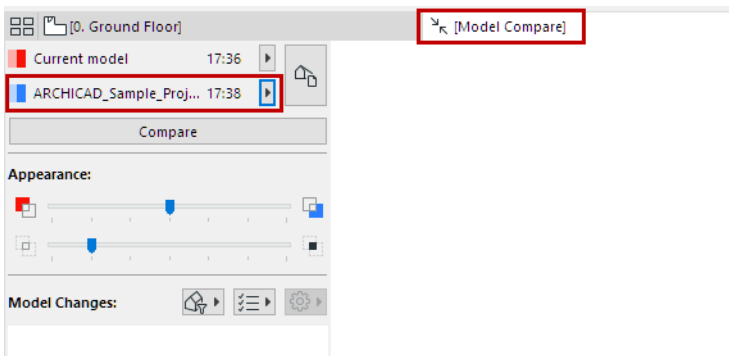
Compare Two Saved Versions of Current Model

This workflow is recommended for the following scenarios:

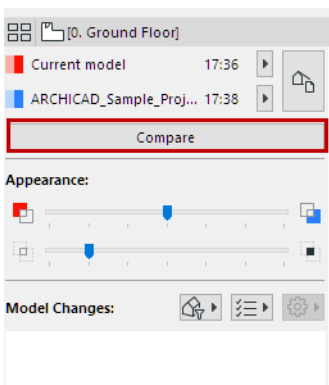
- Create design options, then compare them
- Compare views based on Renovation status
- View today's progress: in the afternoon, compare the current model version to what it looked like this morning.

Example Workflow

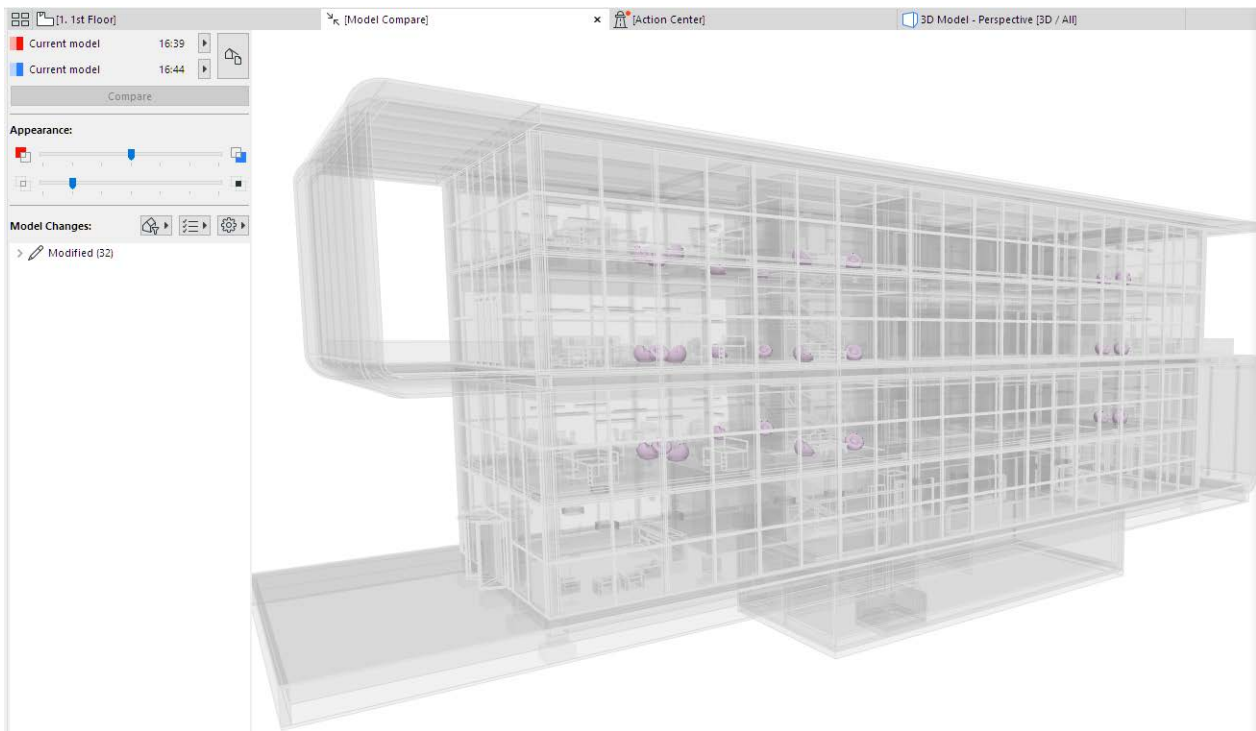
1. Use **Design > Model Compare**.
2. The **Model Compare window** opens.
3. From the Model Compare palette, click **Select Model 1**, and choose **Current model**. The timestamp is shown.
4. Click on Select Model 2, and browse for an earlier saved version.



5. When comparing two different views, current View settings can be significant. [See Select View for Compare.](#)
6. Click **Compare**.



7. Check the results in the Model Compare window.
Review the Model Compare palette to see what has changed.
[See Model Compare: Review the Changes.](#)



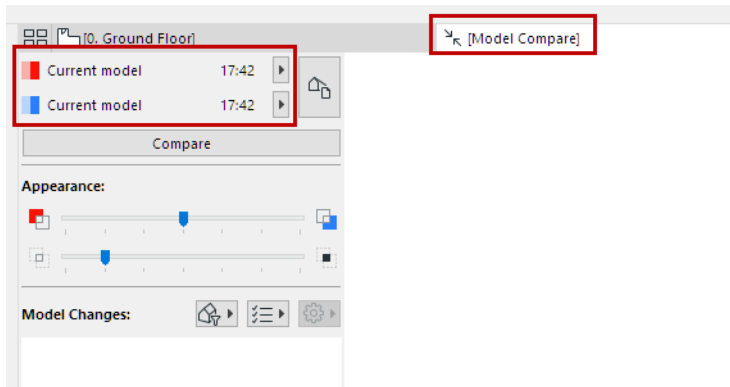
8. As needed, create Issues.

See [Create Issue from Model Compare Change List](#).

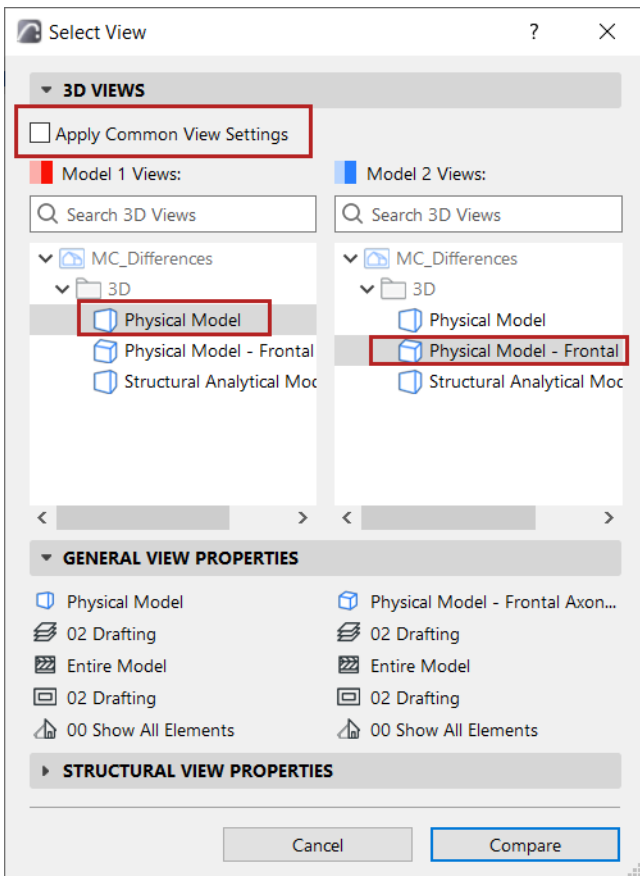
Compare Two Views of a Single Project

This workflow is recommended e.g. to compare two views in a single Archicad project. For example, compare two different 3D views of the existing vs. proposed site plan.

1. Use **Design > Model Compare**.
2. From the Model Compare palette:
Select **Current model** for both Model 1 and Model 2.



3. Click the **View Selector** button.
4. From the Select View dialog, uncheck “Apply Common View Settings.” The 3D views of each model version are now listed separately, side by side.
5. Here, choose “Existing Site” (from Model 1) and “Proposed Site” (from Model 2). These are two different views of the same project.



See also [Select View for Compare](#).

6. Click **Compare**.
7. Check the results in the Model Compare window.
Review the Model Compare palette to see what has changed.

See [Model Compare: Review the Changes](#).

8. As needed, create Issues.

See [Create Issue from Model Compare Change List](#).

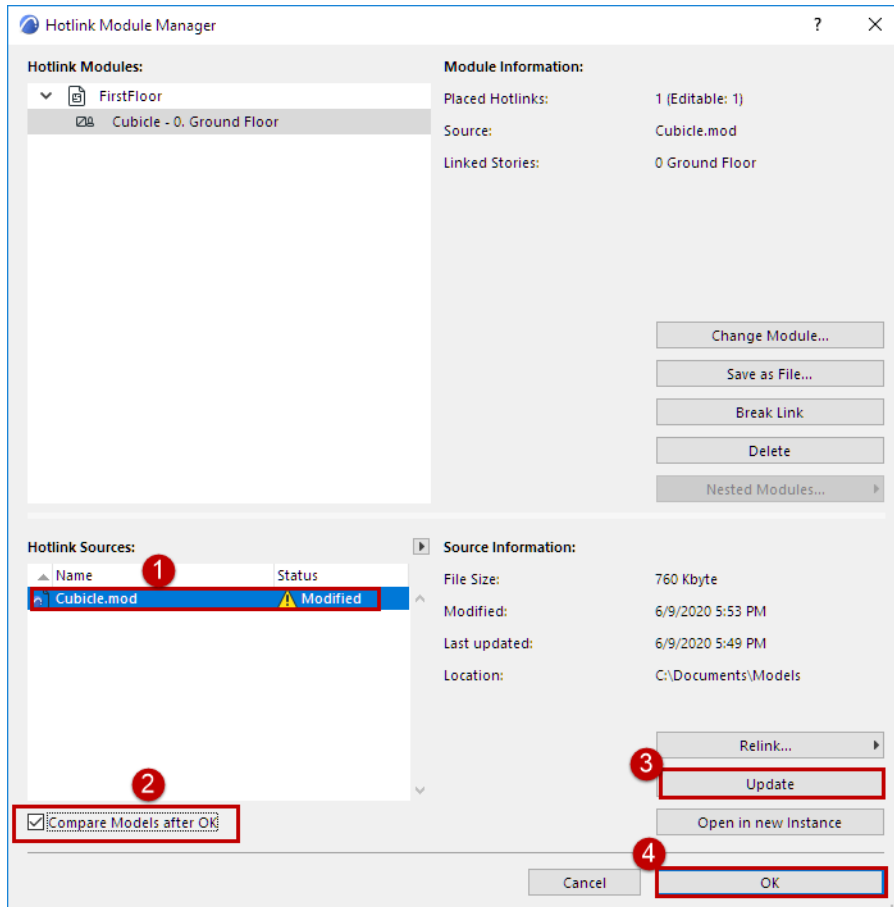
Compare Archicad Models Before and After Hotlink Update

This workflow can be useful when you update a Hotlink in your project.

As usual, the Update command refreshes the Hotlinks in your model to reflect changes in the Hotlink source. But if you also check the optional “Compare Models” checkbox in Hotlink Module Manager, you will be able to visualize what has changed as a result of the update.

Example Workflow

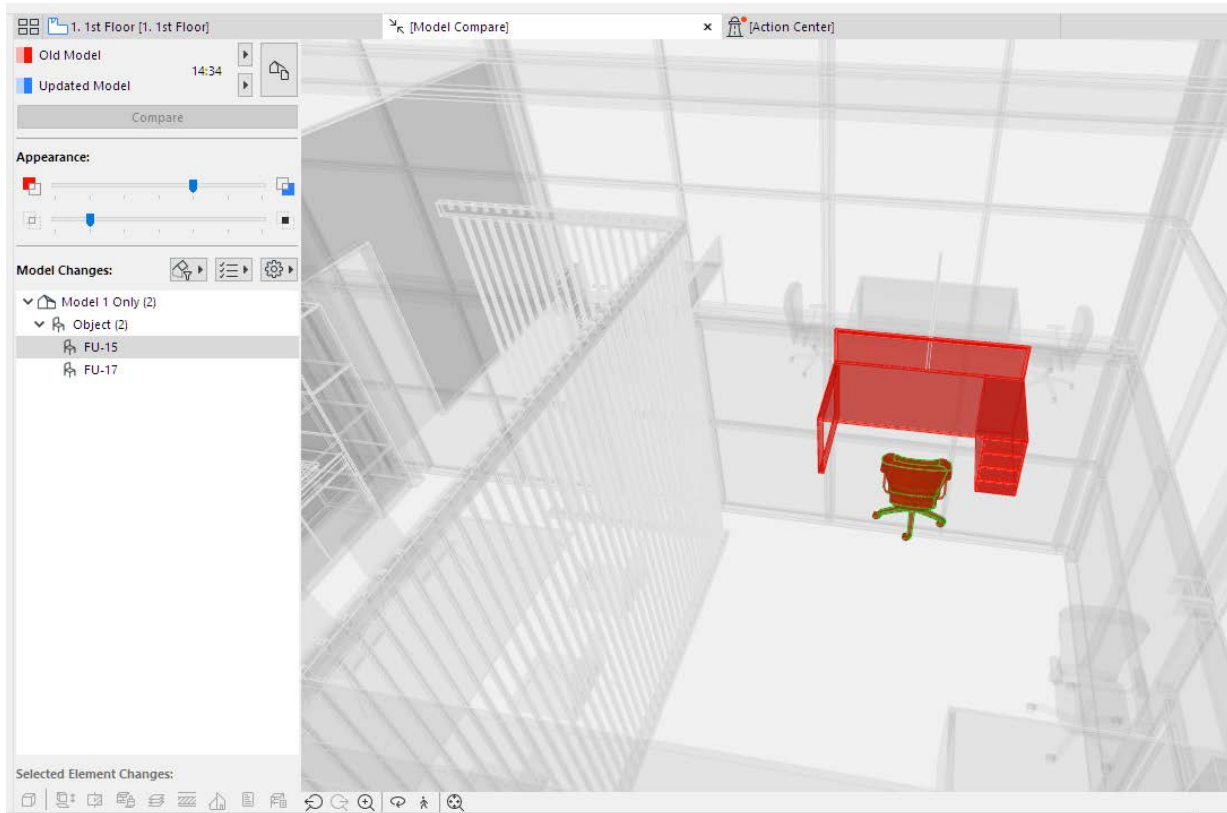
In Hotlink Module Manager:



1. Select the Hotlink Source which has changed (it is shown with Modified status).
2. Check the box to **Compare Models After OK**.
3. Click **Update**.
4. Click **OK**.
5. The Hotlink Module is updated in the model.
6. The Model Compare window opens to compare the “Old Model” and “Updated Model”.
Review the Model Compare palette to see what has changed.

[See Model Compare: Review the Changes.](#)

Here, you can see that a desk/chair has been deleted from the cubicle module.



7. As needed, create Issues.

See [Create Issue from Model Compare Change List](#).

Compare Structural Analytical Models

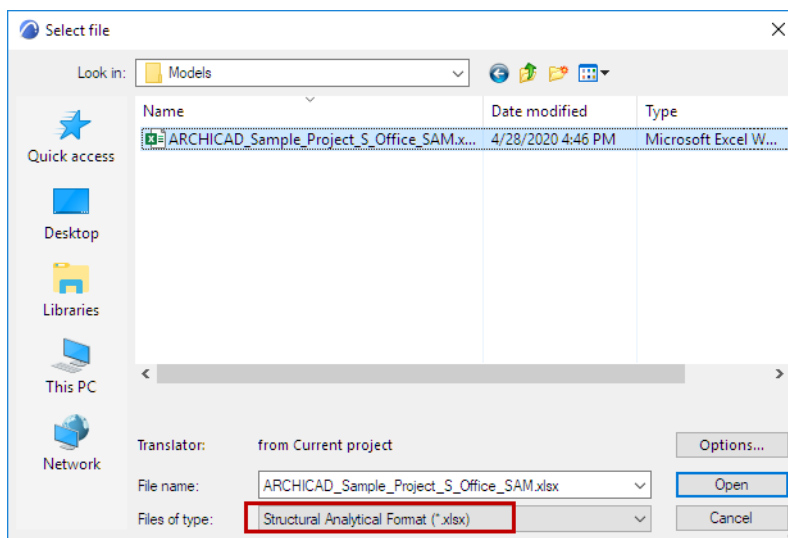
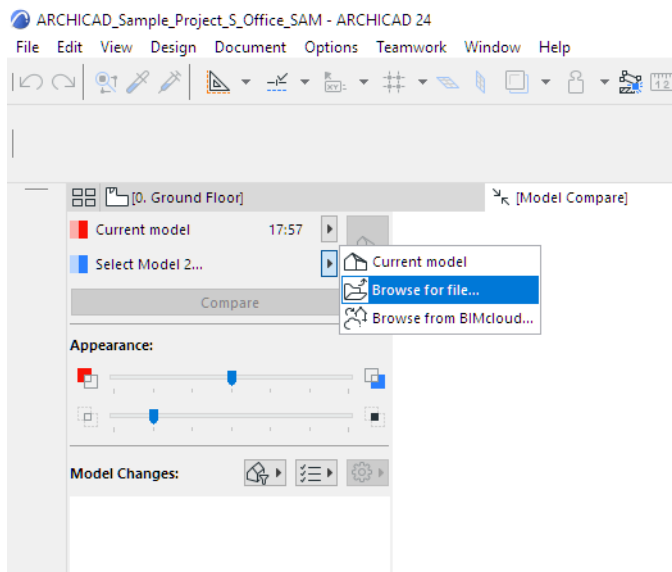
Use this workflow to compare views of a Structural Analytical Model.

In this workflow, you must *enable the Structural Model* in the “Select View” dialog, as described below.

Example Workflow

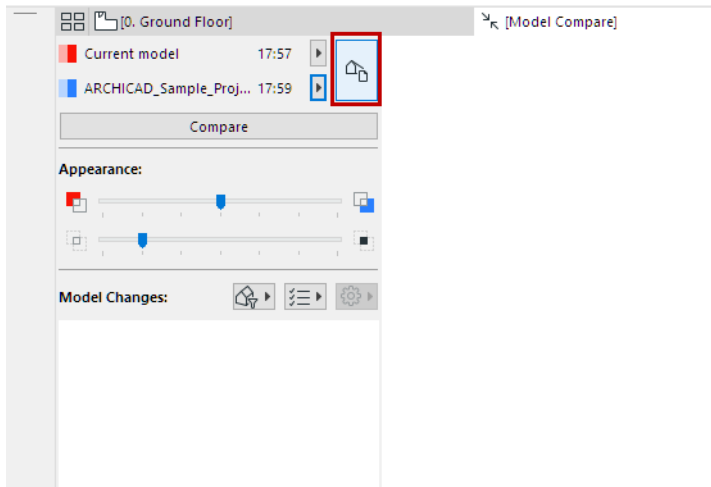
In this example, the structural engineer has revised the structural analytical model, and is doing Model Compare to assign issues.

1. Use **Design > Model Compare**.
2. From the Model Compare palette:
 - Select **Current model** for Model 1. (You can browse for any other pln or SAF as needed.)
3. For Model 2, browse for the SAF file (format: Structural Analysis File .xlsx).

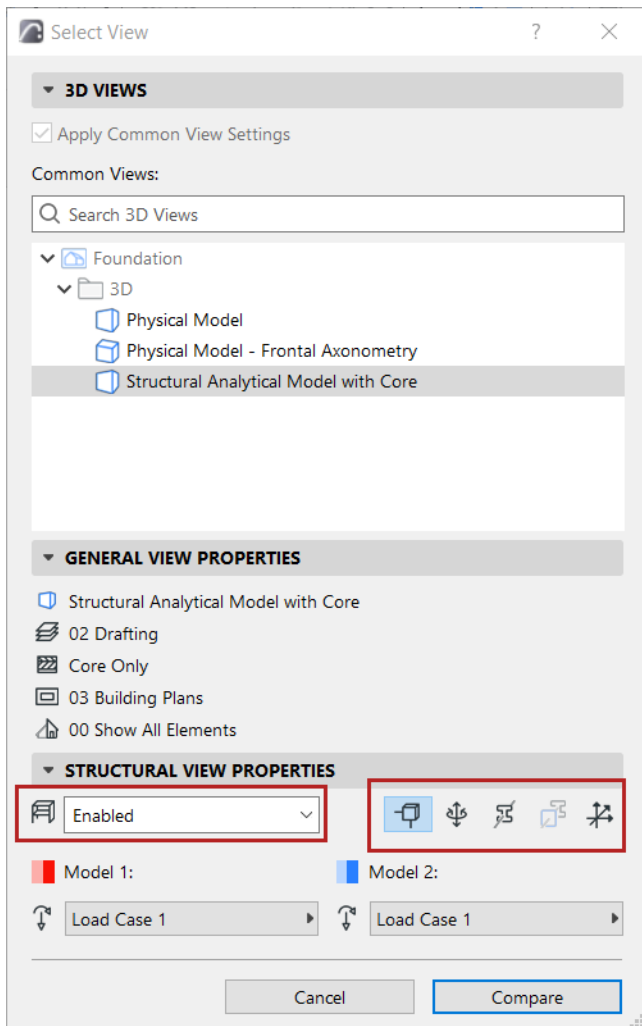


Note: When you open a SAF file using Model Compare, the SAF Translator *settings of the current project* are applied. Make sure the current project’s SAF Translator settings are the ones you need. To change them, import a different Translator file to the current project. See [Import/Export SAF Translators](#).

4. Click Open.
5. From the Model Compare palette, click the View Selector button.



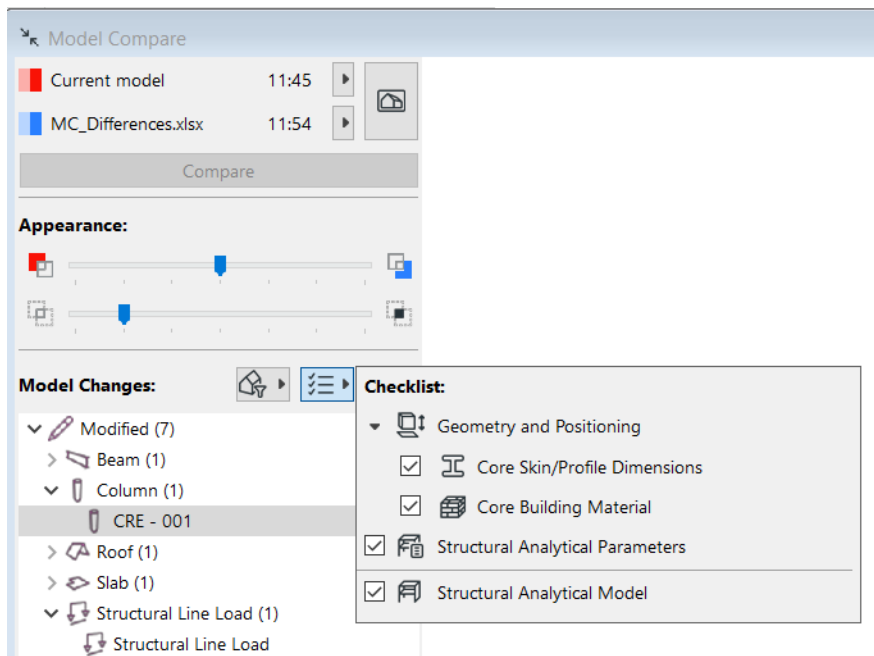
6. The **Select View** dialog opens.
7. Select 3D View to compare in the 3D Views panel. (See also [Select View for Compare](#).)
Note: In comparing pln with SAF models, only common views are available, since the SAF file does not contain Views.
8. In the **Structural View Properties** panel, enable the Structural Analytical Model option.



- Click any of the icons (nodes, end releases, profile, replacement profile, local coordinate system) to show these Structural components in the Compare window.
- Optionally, choose to compare particular Load Cases of each view. (See [Load Cases in Model Compare](#), below.)

Note: Visibility settings are stored in the selected view, but these icons and Load Case options allow you to temporarily override them.

9. Click **Compare**. While the Model Compare results are being generated, you can navigate to any other tab or window.
10. Check the results in the Model Compare window.
11. Filter the parameters you want to see. Available parameter sets are specific to the Structural Analytical Model.

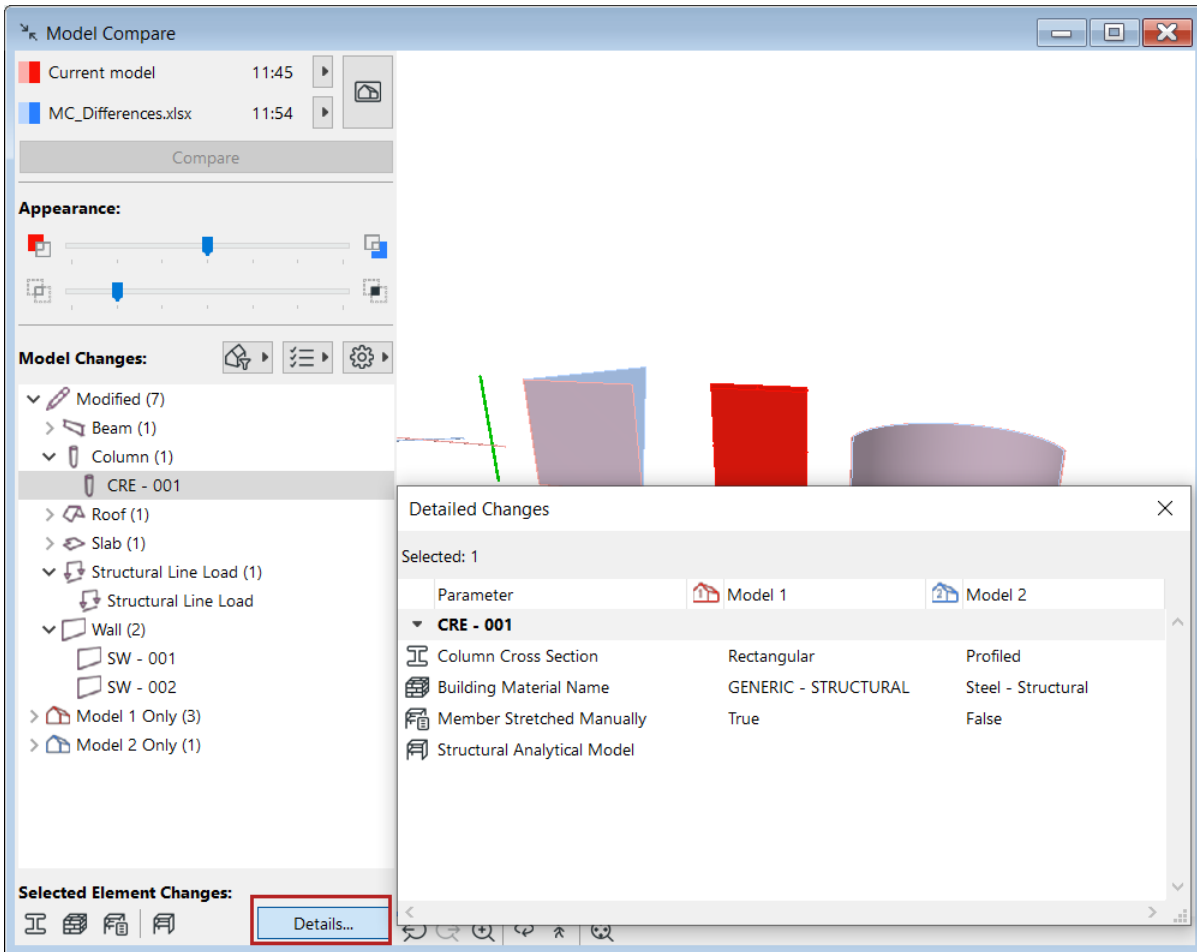


12. Review the Model Compare palette to see what has changed: changed elements that exist in both models (“Modified”; and those that exist only in Model 1 or in Model 2.

[See *Model Compare: Review the Changes*.](#)

13. Click **Details** for more information on structural-related changes in modified elements. (Available for Structural view compare only. Not available in Teamwork.)

Where available, you can see Model 1 vs. Model 2 values for specific parameters.



Note: You can also open Detailed Changes from the Structural Analytical toolbar, or from Window > Palettes.



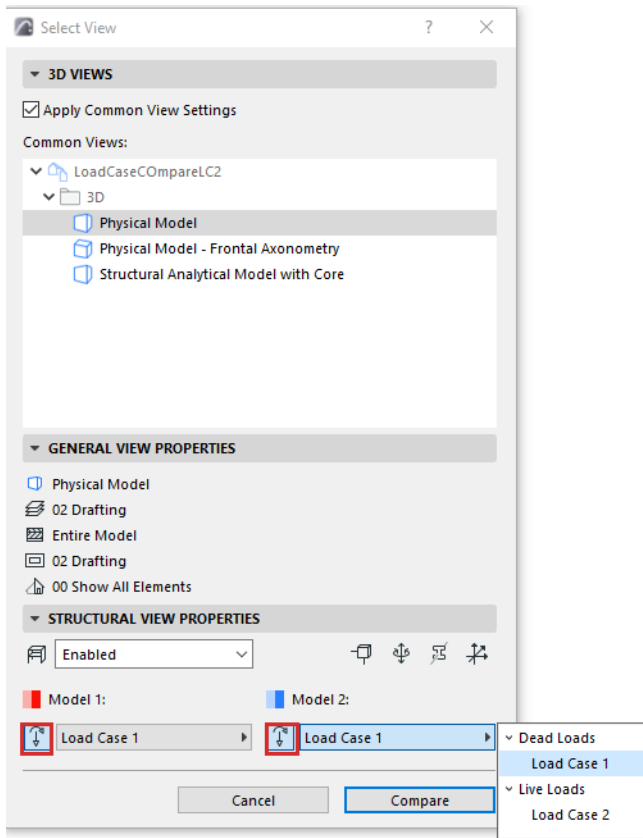
14. As needed, create Issues.

See [Create Issue from Model Compare Change List](#).

Load Cases in Model Compare

In the Select View dialog (Structural View Properties panel):

- For both models, check the Load Case controls. The model’s saved Load Case (part of View Settings) is displayed.
- Optionally, enable or disable Load Cases for either or both models. Disabling the Load Case means Load-related information will be ignored for that model.
- For either or both models, you can choose a different Load Case to consider in Model Compare.



Select View for Compare

Model Compare compares 3D views from two different models or versions - either the physical model or the Structural Analytical Model.

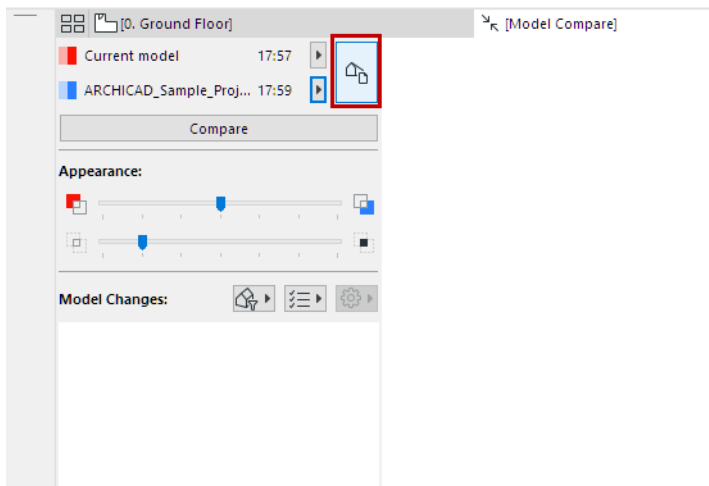
To define exactly which 3D views, use the **Select View** dialog. You can choose a 3D view that is common to both of the models, or a different view from each.

View selection can be significant for Model Compare:

An element can be visible in one view but not in another (e.g. due to Renovation status, Design Options, Layer Settings, Partial Structure Display, etc.)

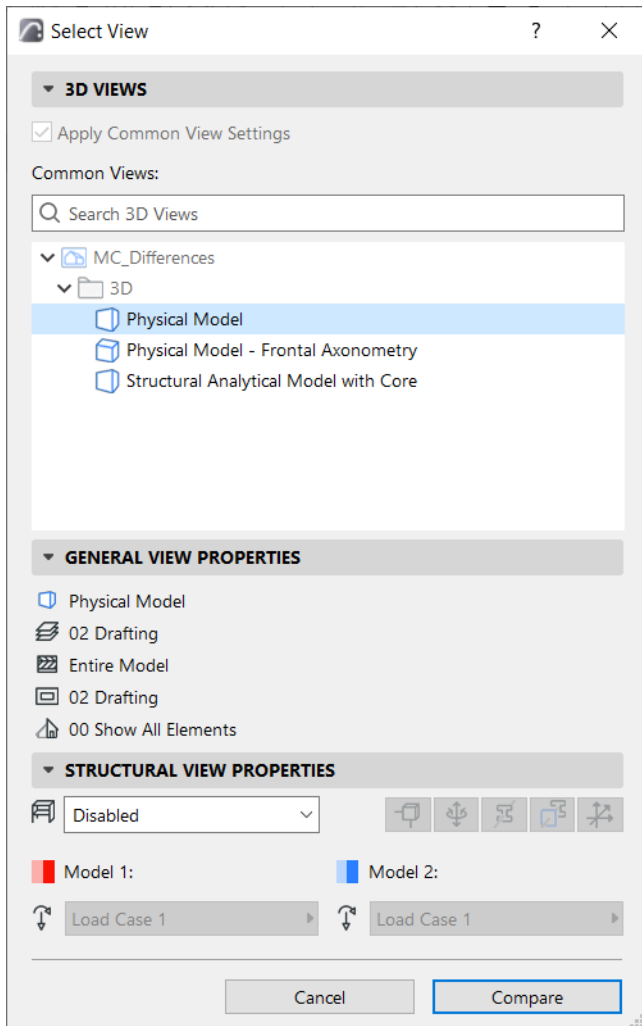
- Intersecting elements can be considered modified in one view, yet not modified in another view, depending on the view's Layer Combination.

1. Use **Design > Model Compare** to choose the two model versions.
2. From the Model Compare palette, click the View Selector button.



3. The **Select View** dialog opens, showing the combined View Maps of the two selected projects.

4. Select a view (here, a view that is common to both compared models), then review its **General View Properties** in the panel below.



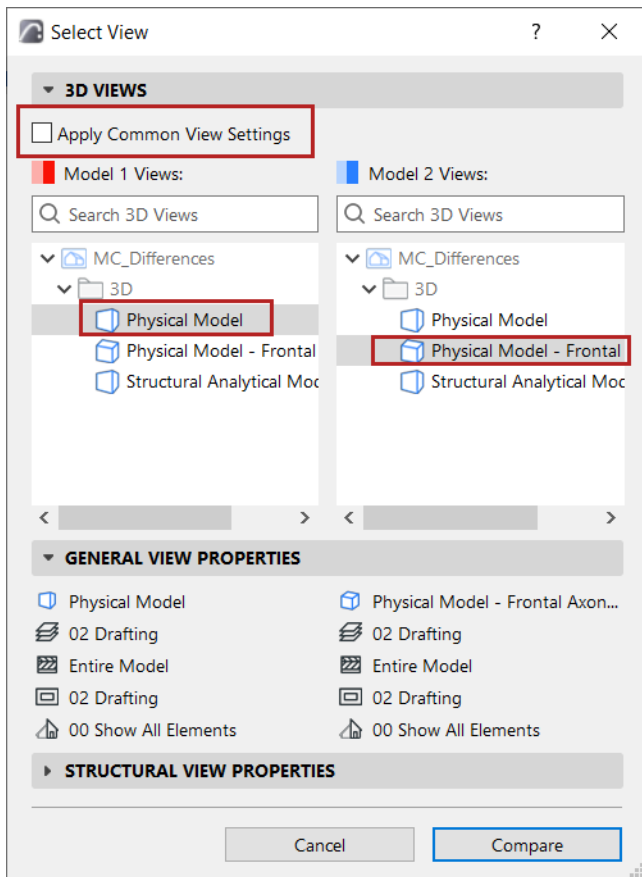
Physical vs. Structural Analytical Model

By default, Model Compare considers views of the Physical Model. To compare views of the Structural Analytical Model, enable this in the Structural View Properties panel.

For details, see [Compare Structural Analytical Models](#).

To compare two different 3D views:

1. Uncheck “Apply Common View Settings.”
2. The 3D views of each model version are now listed separately, side by side.



3. Review their **General View Properties** in the panel below.
4. For each model, select a 3D view, then click **Compare**.

Related Topic

[Compare Structural Analytical Models](#)

Filter Changes Shown in Model Compare Palette

By default, the Model Compare palette lists *all* 3D model changes resulting from the Compare process. Use the Filter buttons to narrow the list of changes shown here. You can also set your preferred list format.

Topics in this section:

[Filter by Element Criteria](#)

[Filter by Subscription](#)

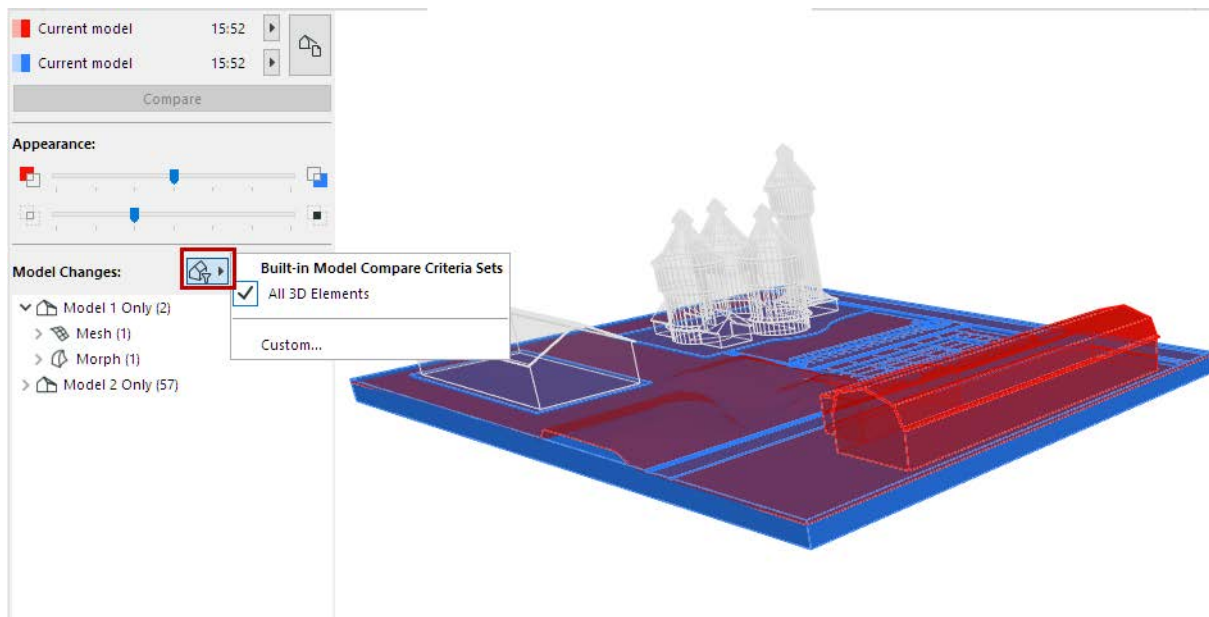
[Filter by Parameter Sets](#)

[List Format](#)

Filter by Element Criteria

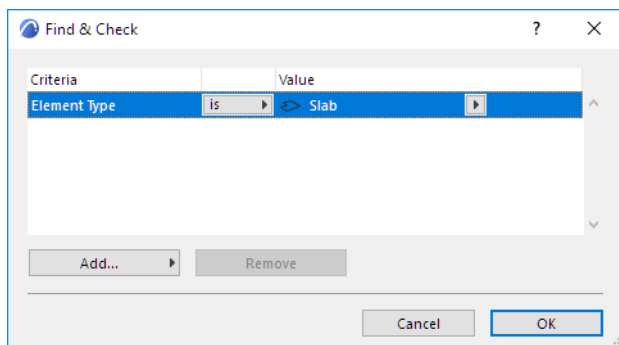
Click the **Filter by Criteria** button to access the criteria set pop-up.

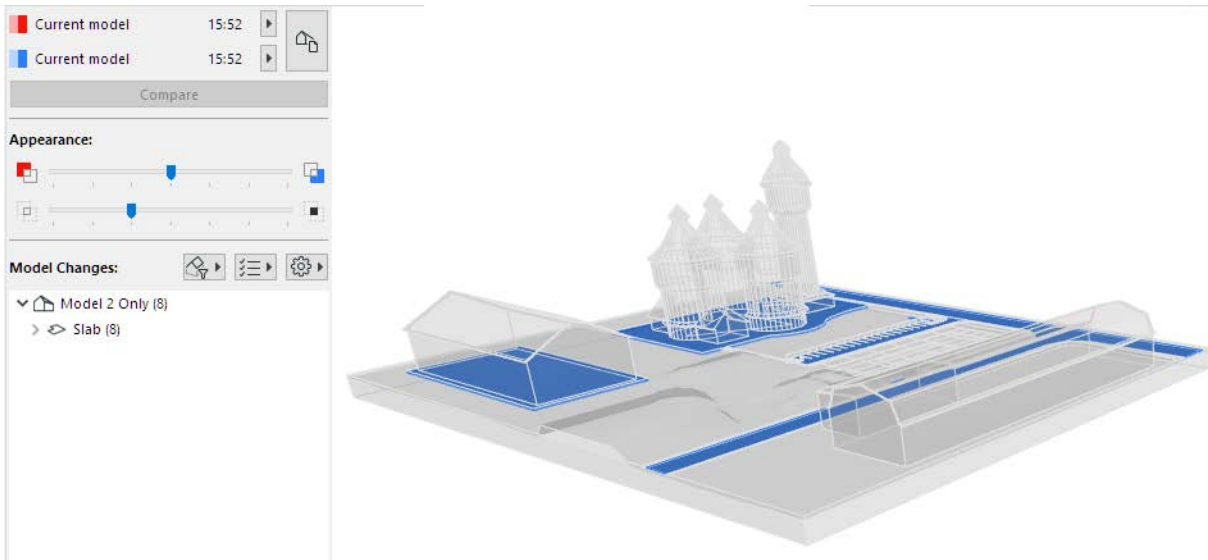
By default, **All 3D Elements** is active: All elements are listed in the Model Changes list.



To narrow the list of elements, click **Custom** to define elements by criteria.

- The **Find & Check** dialog opens.
- Set criteria for listing elements: for example, filter the list to show only the slabs.

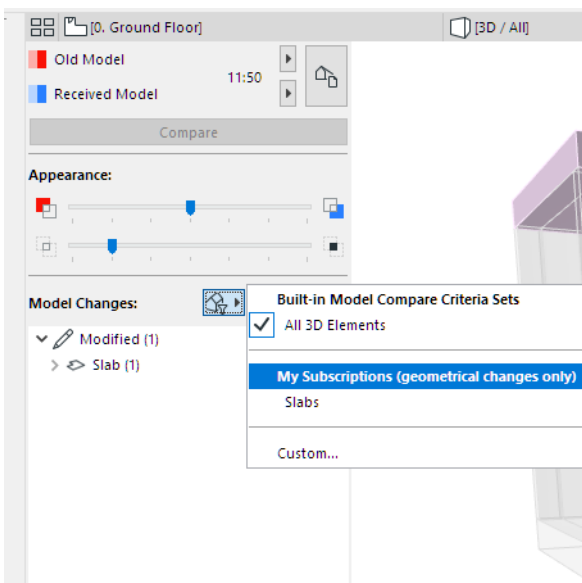




The custom criteria set used as a filter cannot be saved.

Filter by Subscription

If you are using Project Change Tracking (in Teamwork), you can filter to see element changes by Subscription.



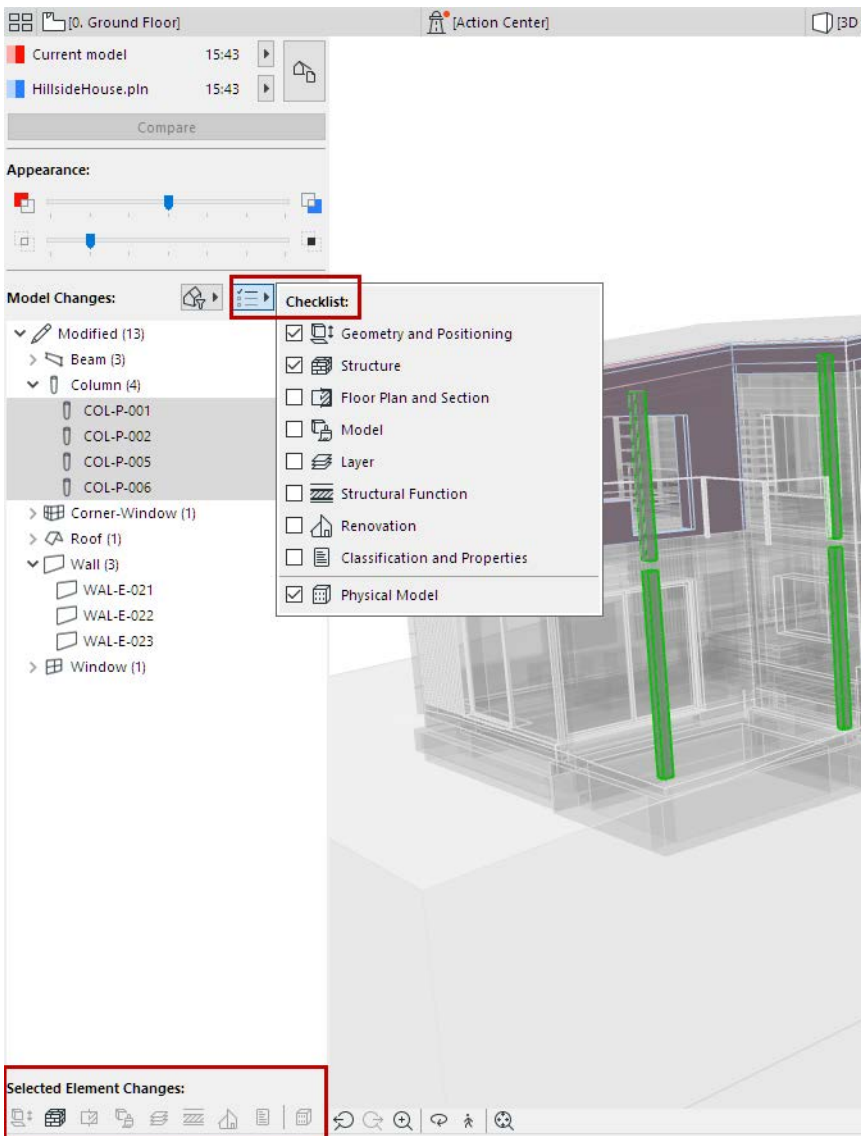
[See also Change Tracking in Teamwork.](#)

Filter by Parameter Sets

Check/uncheck the type of modifications that you are interested in. Items that have changed *based on the checked parameters* are listed.

Available parameter sets vary depending on the Model type chosen in the Select View dialog: either Physical Model or Structural Analytical Model.

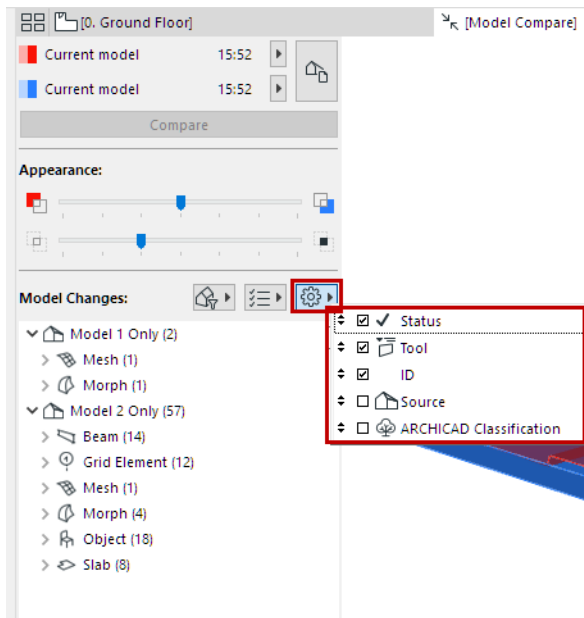
The parameter sets in this checklist are the same as the “Selected Element Changes” feedback at the bottom of the list.



List Format

Click the Settings button.

In the pop-up, check/uncheck boxes to define your preferred list format.



Drag and drop the items to change their order.

- Status (e.g. New/Modified/Deleted)
- Tool
- ID
- Source
- Classification

Model Compare: Review the Changes

After the Compare command is run, the Model Compare window displays the model with colored highlights and other feedback on changes.

The **Model Compare palette** is docked at the left edge of the window.

Topics in this section:

[List of Model Changes](#)

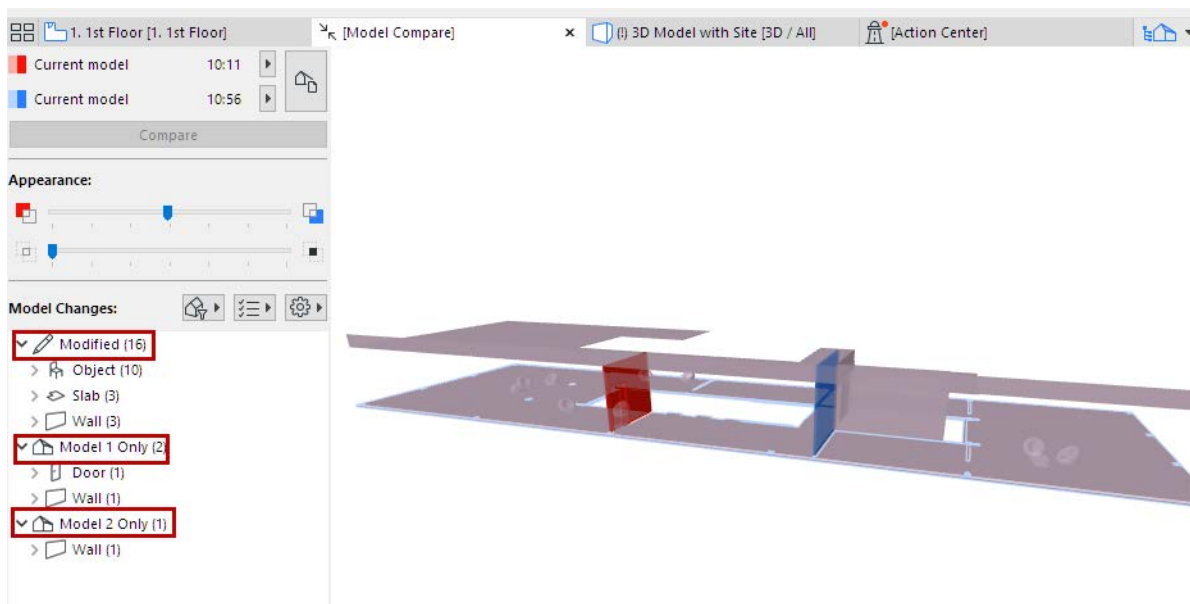
[Select Changed Elements](#)

[Set Visibility of Compared Models](#)

[Changes By Parameter Set](#)

List of Model Changes

By default, the Model Changes list shows all changed elements, grouped by Status (Modified), Model 1 Only, and Model 2 Only.



To customize this list, use the filter and settings buttons.

[See *Filter Changes Shown in Model Compare Palette*.](#)

Select Changed Elements

Select any item from the Model Changes list to select the corresponding element in the Model Compare window, and vice versa.

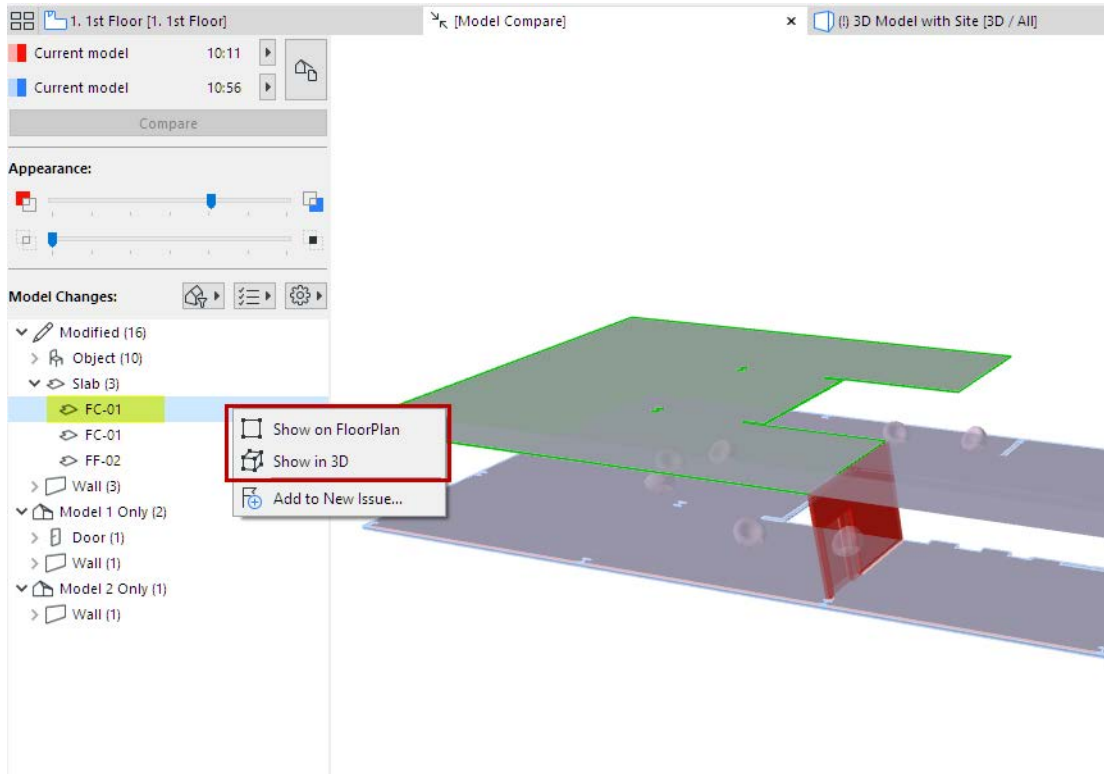
Note: In the Model Compare window, you can only select elements that have changed.

Select Elements By Group

Select a parent item in the list to select all its member elements in the Model Compare window.

Show Changed Element in Model

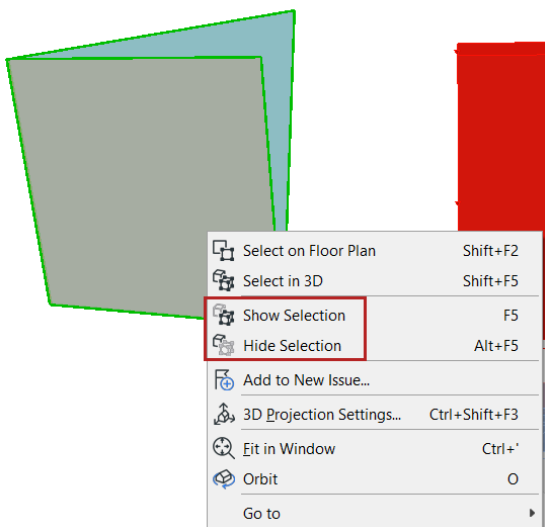
For a selected item in the Model Changes list: use the context menu commands to **Show on Floor Plan** or **Show in 3D**.



Show or hide selection

For a selected element in the Model Compare window:

- Use **Show Selection** or **Hide Selection** to get a better view of the changed element and/or its environment.
- Reverse this command with **Show All**.



Set Visibility of Compared Models

The “Model 1 only” and “Model 2 only” elements are shown in their respective colors.

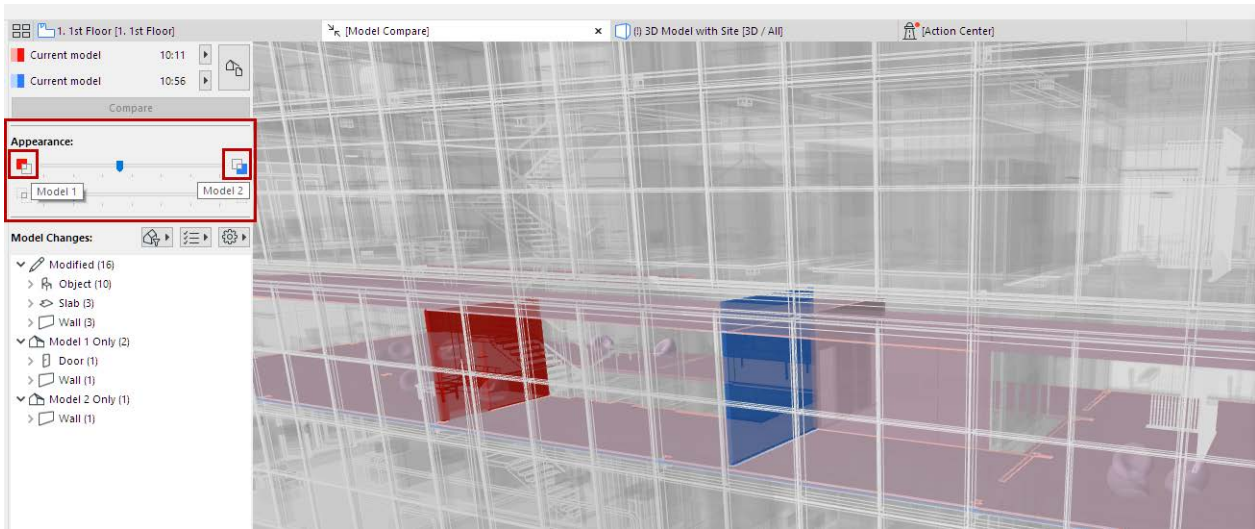
In this example:

- The red wall is “Model 1 only”, because it was deleted.
- The blue wall is “Model 2 only”, because it is a new element.
- The slabs and chair objects are shown in a blended (purple) color, because they have been modified, while remaining in their original position.

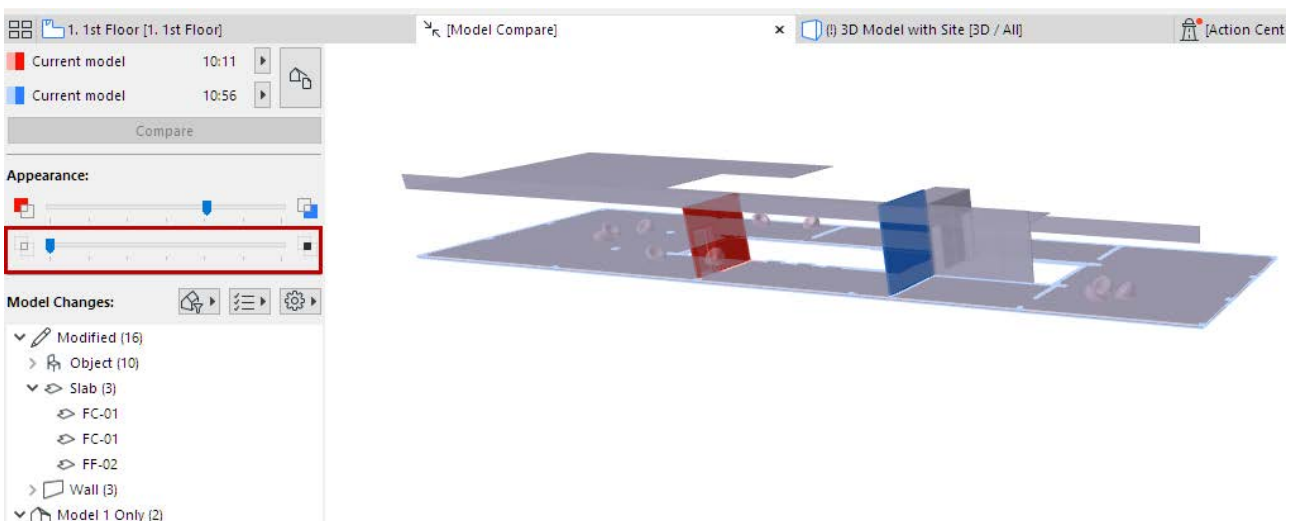
[See also](#)

In the **Appearance** section of the Model Compare palette:

- Use the top slider to graphically emphasize Model 1 or Model 2.



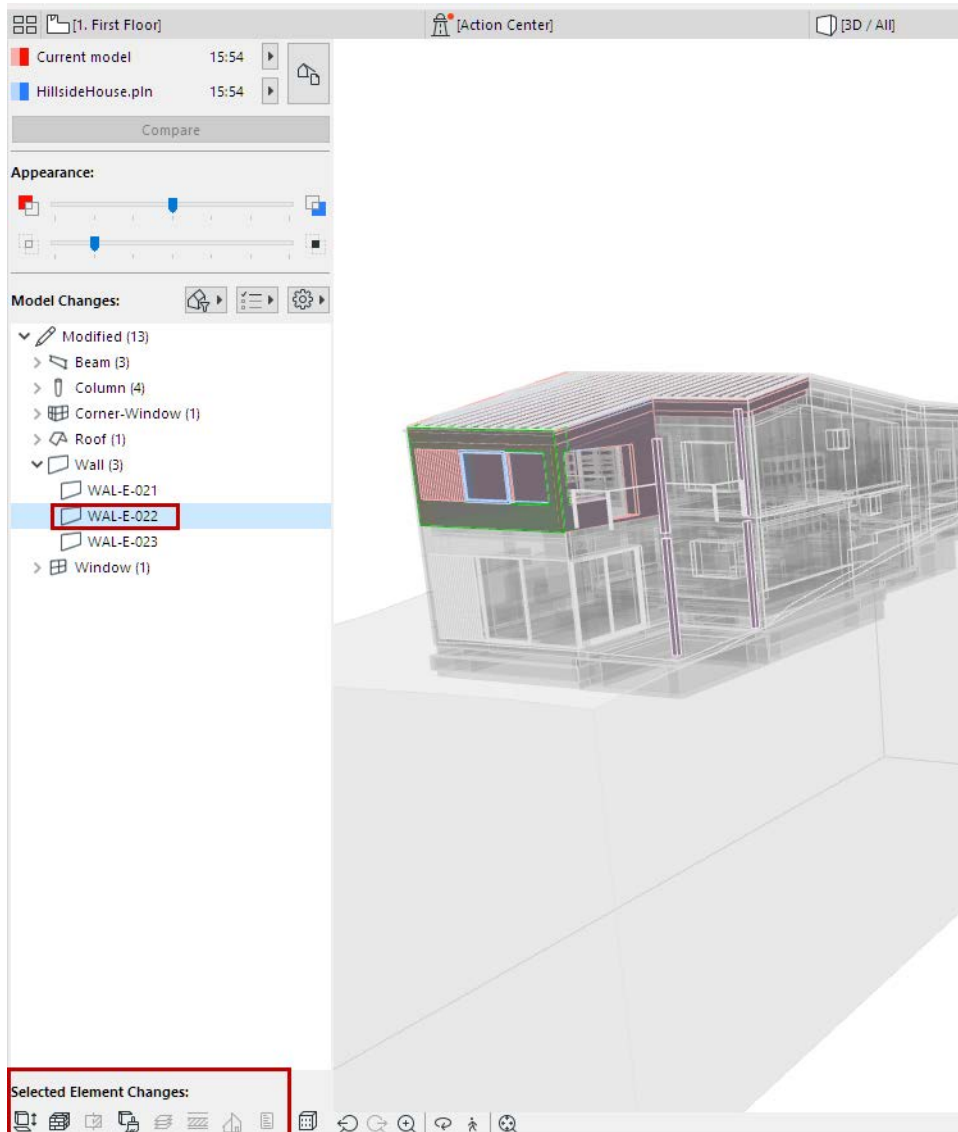
- Use the bottom slider to set the relative visibility of the unchanged model. Here, the bottom slider is set to zero. The unchanged model is not shown at all, and you see only the changes:



Changes By Parameter Set

For a selected item in the Model Changes list:

The “Selected Element Changes” icons indicate which parameters have changed. (These parameter sets are the same as in the “Checklist” used to filter the change list in the Model Compare window.)



Detailed Changes in Structural Model

If the Model Compare involves a Structural Analytical view, additional change details are available.

[See Compare Structural Analytical Models.](#)

Related Topic:

[Filter by Parameter Sets](#)

[Create Issue from Model Compare Change List](#)

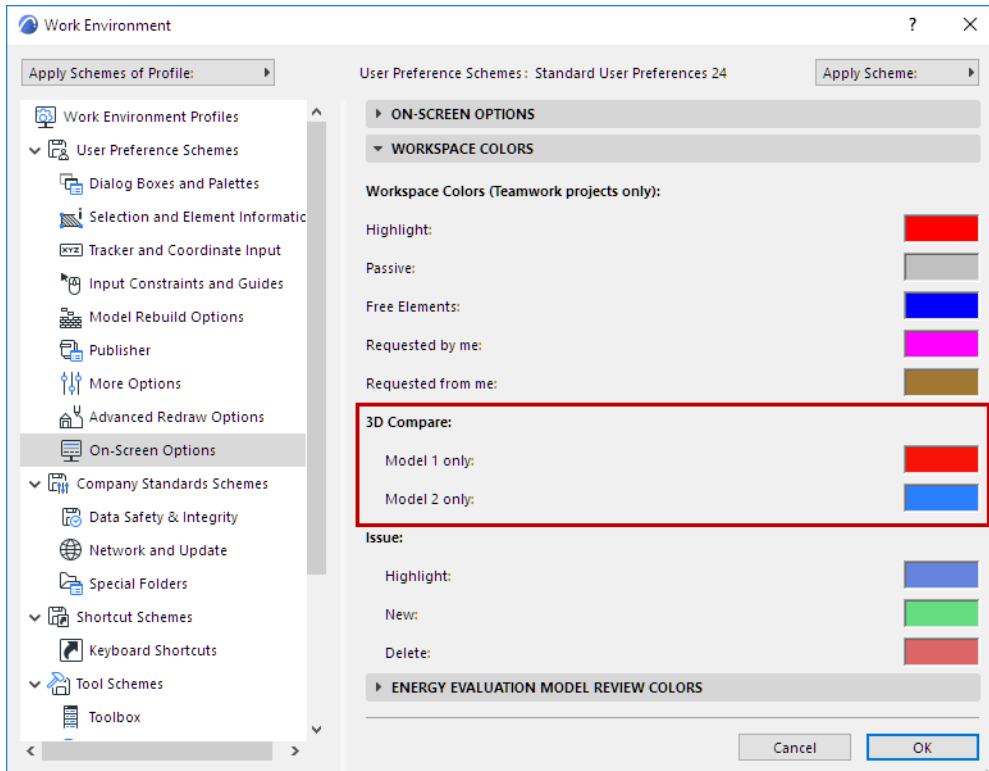
[Workspace Colors for Model Compare.](#)

Workspace Colors for Model Compare

Go to **Options > Work Environment > On-Screen Options**.

In the Workspace Colors panel:

Set the colors used to display “Model 1 only” and “Model 2 only” elements in the 3D Model Compare window.



Color of Modified Elements

- Elements that have **moved from one place to another** are shown in both the original and the new positions, in a lighter version of the Model 1 color (original position) and in a lighter version of the Model 2 color (new position).
- If an **element remains in place**, but another of its parameter has changed (e.g. surface color): the element is shown as modified, combining the Model 1 and Model 2 colors.



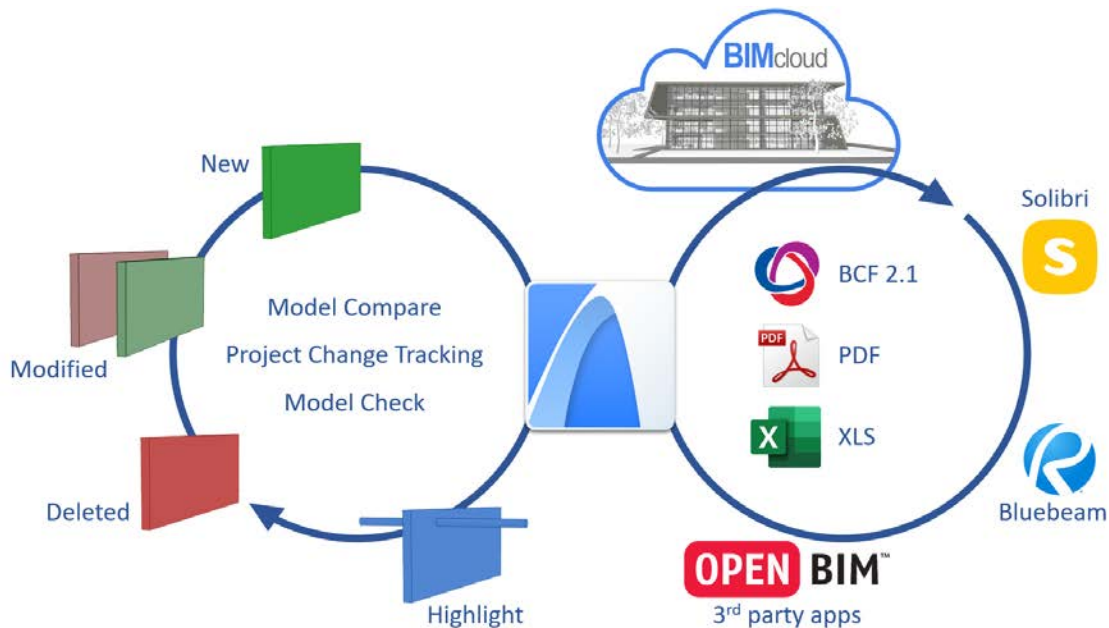
Issue Handling

Archicad's Issue Management workflow supports an extensive collaborative workflow. It replaces the Markup tool from earlier Archicad versions.

Use this workflow to:

- Create Issues that highlight model elements and/or views, with optional markup
- Make element-based suggestions (New, Modify, Delete)
- Create Issues from the discrepancies or modifications that are flagged by Archicad (the results of Model Compare and Model Checks)
- Add Comments
- Define Issue details (e.g. Status, Priority, Type, Labels, Description)
- Assign Issues to any user in Teamwork
- Use Issue Organizer to sort through long Issue lists in flexible formats

Archicad supports the latest BIM Collaboration Format (BCF 2.1), as well as all earlier BCF versions.



Topics in this section:

[Add Selected Elements to New Issue](#)

[Add Views to Issue](#)

[Issue Manager](#)

[Suggestion Elements in Issue Manager \(Highlight, New, Modify, Delete\)](#)

[Assign Issue to User](#)

[Accept Issue Element\(s\)](#)

[Issue Management Workflow: Example](#)

[Issue Organizer](#)

[Issue Details](#)

[Create Issue from Model Check Report](#)

[Create Issue from Model Compare Change List](#)

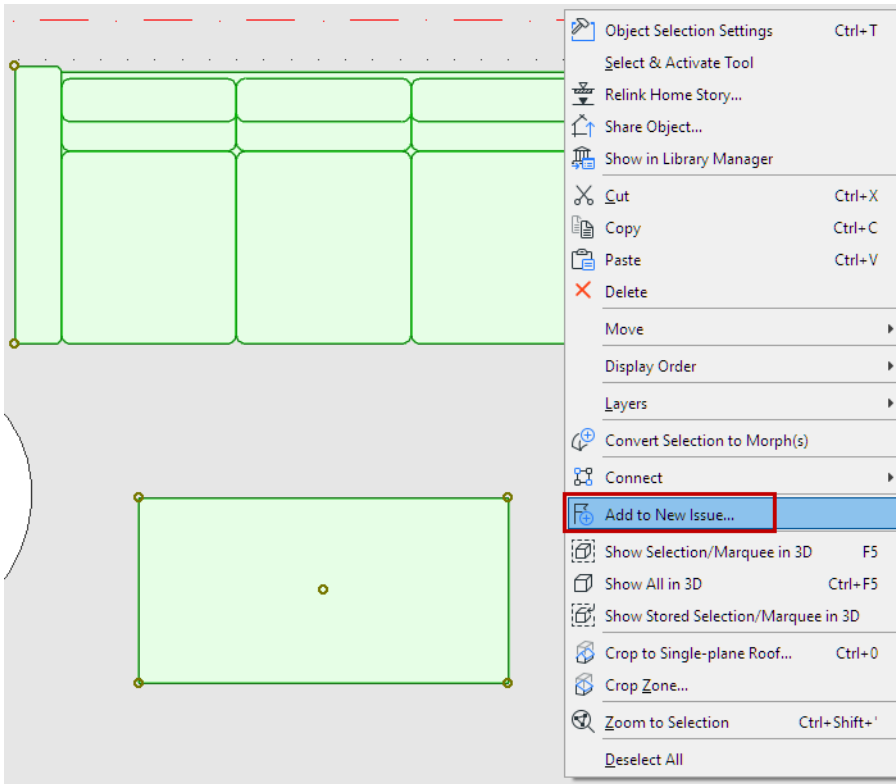
[Import Issues \(BCF format\)](#)

[Export Issues \(BCF format\)](#)

[Export/Import Issues Using PDF Markup](#)

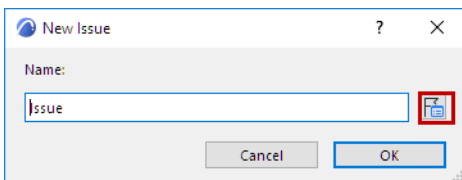
Add Selected Elements to New Issue

1. Select one or more elements. Anything in the model that has Settings can be added.
2. Right click: from the context menu, choose **Add to New Issue**.



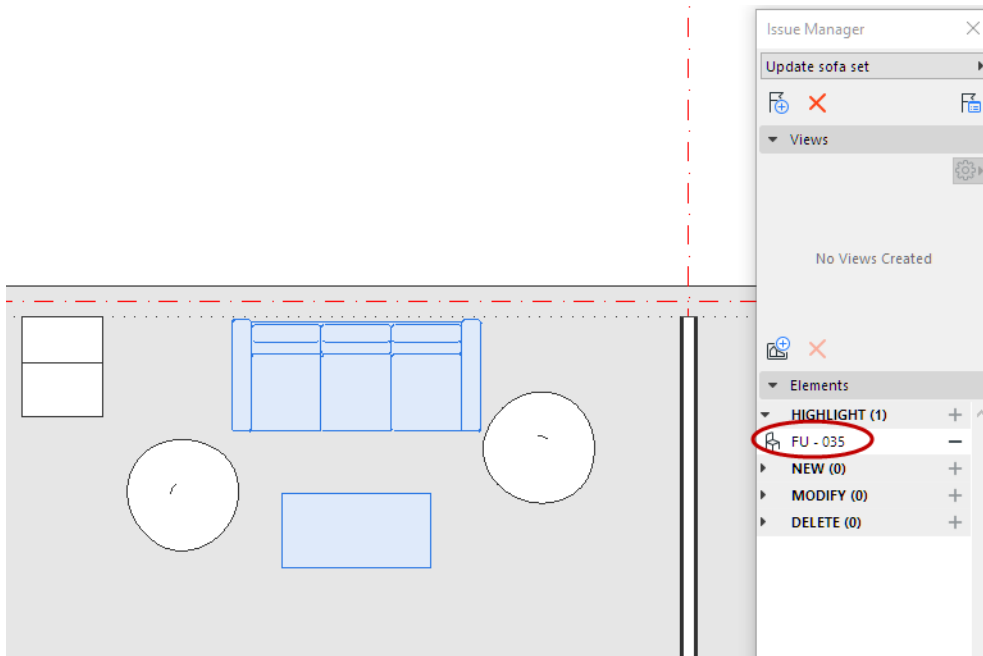
The **New Issue** dialog appears, and the **Issue Manager** palette opens.

3. In the New Issue dialog:
 - Enter a name
 - Click the icon to open **Issue Details**: define Status, Priority, Type, Description as needed, then click OK.



4. The new Issue is now open in **Issue Manager**.

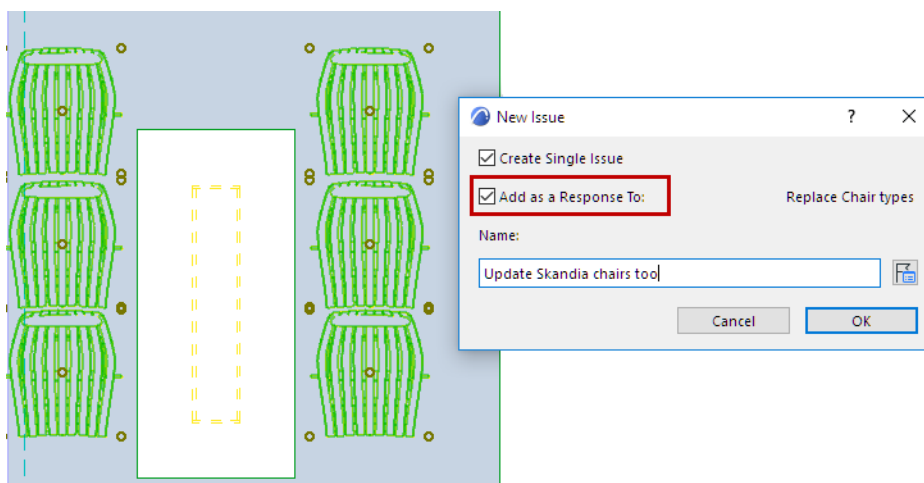
By default, the Issue elements are placed the Highlight list of Issue Manager, and are highlighted in the model.



Add Element(s) to New Issue - As Response

You can add a new Issue as a response to the current Issue which is open in Issue Manager.

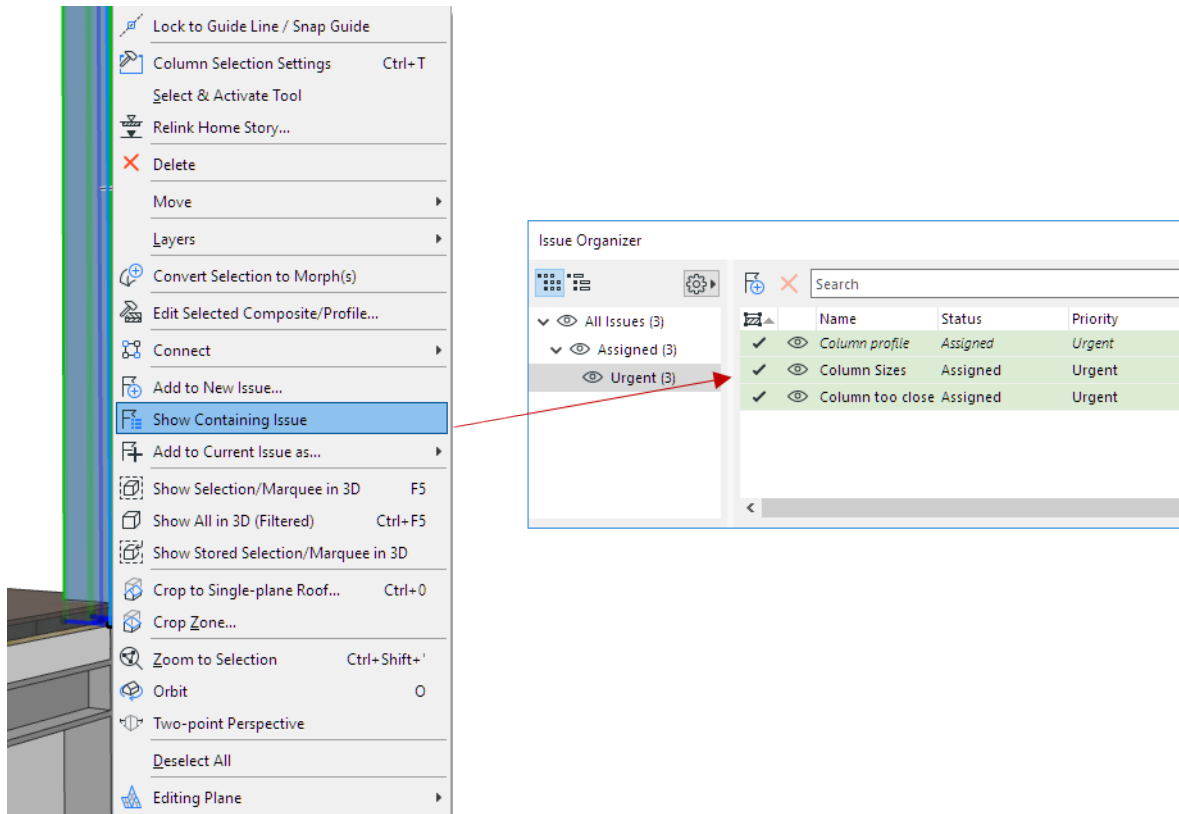
1. Select the model element(s).
2. Right click: from the context menu, choose **Add to New Issue**.
3. In the New Issue dialog, check **Add as a Response to**



See also [Suggestion Elements in Issue Manager \(Highlight, New, Modify, Delete\)](#).

Show Containing Issue

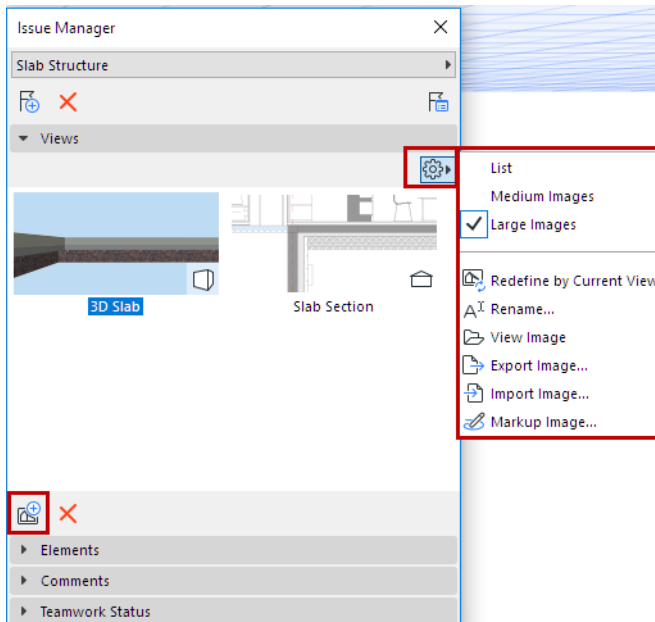
For a selected element: check whether the element is part of any existing Issue, then filter for the relevant Issue(s) in Issue Organizer.



Add Views to Issue

Add any number of snapshot views to an Issue.

Views are typically related to the Issue's elements, but Issues do not have to contain any elements. You can just create a new Issue, add one or more views, and mark them up if needed.

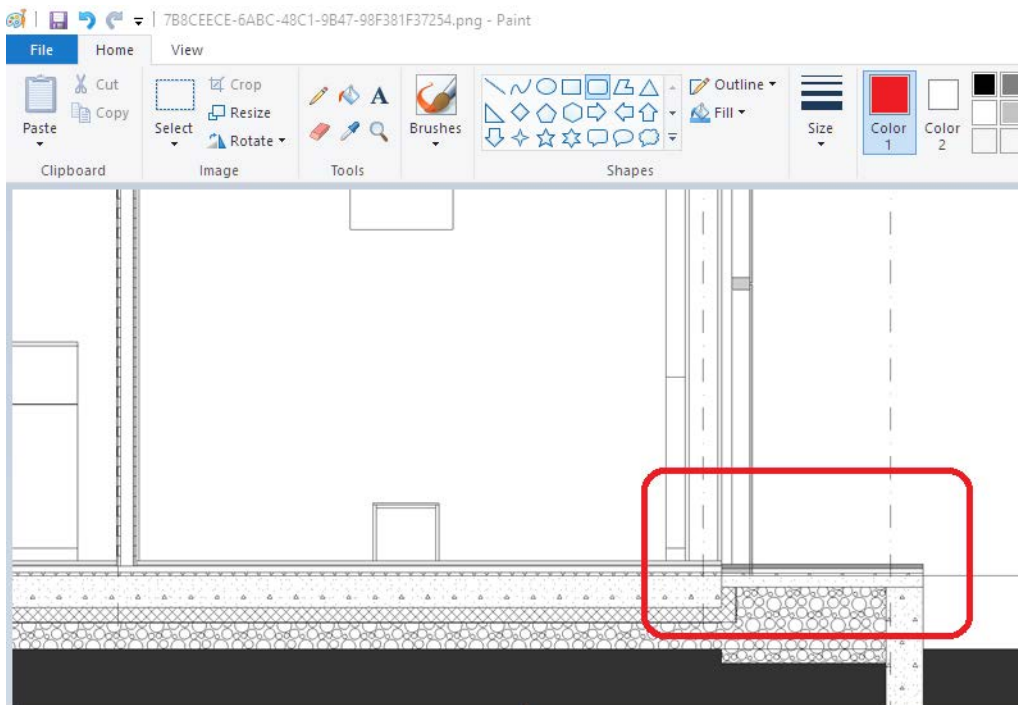


Create and Manage Views in Issue

1. Open Issue Manager.
2. Set up the view in any model window.
3. Click the **Add new view** icon.
4. Optionally, name the view.
5. Use the Settings pop-up to edit or redefine any selected view.

Use the Settings pop-up to manage any single selected view.

- **Redefine by Current View:** Updates the list of visible elements, the camera position, the cutting planes, and the snapshot.
- **View Image:** Opens the snapshot in a new tab
- **Markup Image:** Open and edit the saved view in the default image editor (usually Paint or Preview)

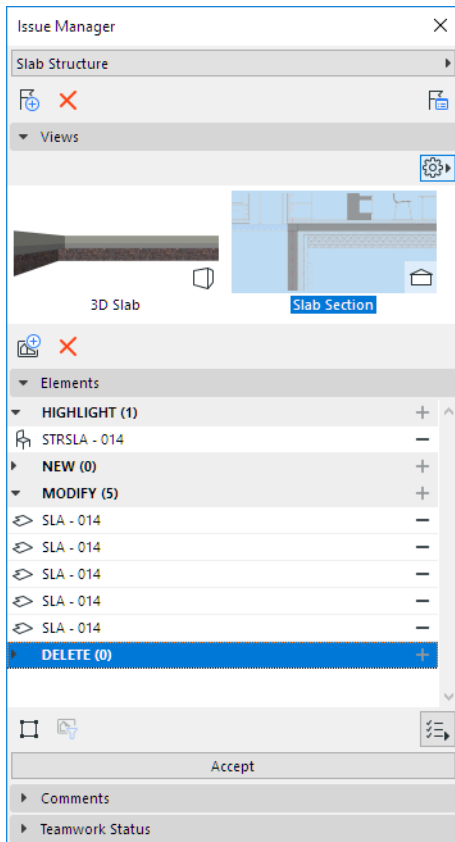


- Export/Import Image: Export or import the view in the chosen image format.
Note: When exporting an Issue in BCF format: if the Issue contains a Floor Plan view, it will generate a 3D axonometry top view, with cutting planes at the top and bottom level of the floor.

Issue Manager

Issue Manager is a palette with controls for managing a **single, opened Issue**, with the following functions:

- Add views (images), or import/export images
- Manage Issue elements (categorize them as Highlight, New, Modify or Delete)
- Accept the suggestions in the Issue (e.g. new or modified elements)
- Add optional Comments.



Open Issue Manager

Issue Manager opens automatically when you create a new Issue.

You can also open it with one of the following:

- Use its toolbar command



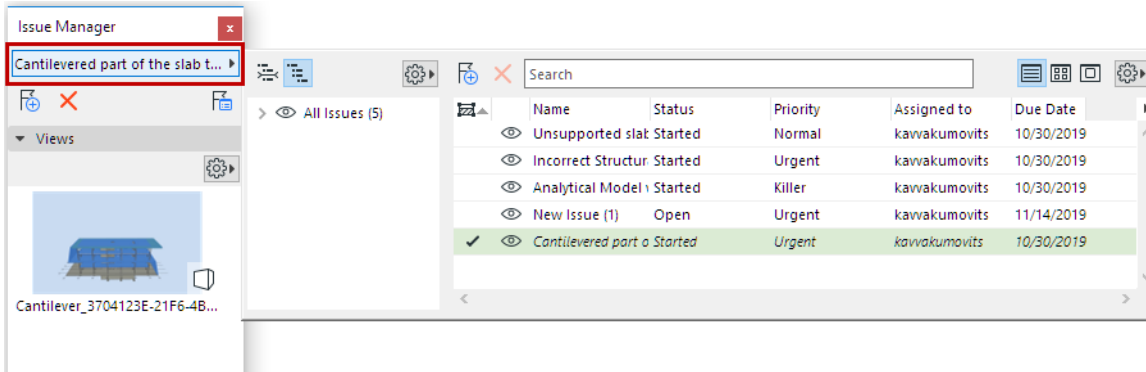
- Double-click a listed Issue in Issue Organizer
- **Document > Issue Manager**
- **Window > Palettes > Issue Manager**

Switch to a Different Issue

From Issue Manager:

Click the Issue name pop-up to open the Issue Organizer list.

Choose a different Issue to work on.



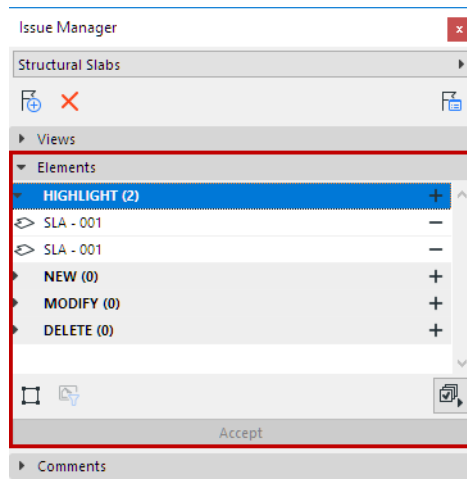
Related Topics:

[Suggestion Elements in Issue Manager \(Highlight, New, Modify, Delete\)](#)

[Add Views to Issue](#)

Suggestion Elements in Issue Manager (Highlight, New, Modify, Delete)

Use the Elements panel of Issue Manager to manage the elements contained in any given Issue.



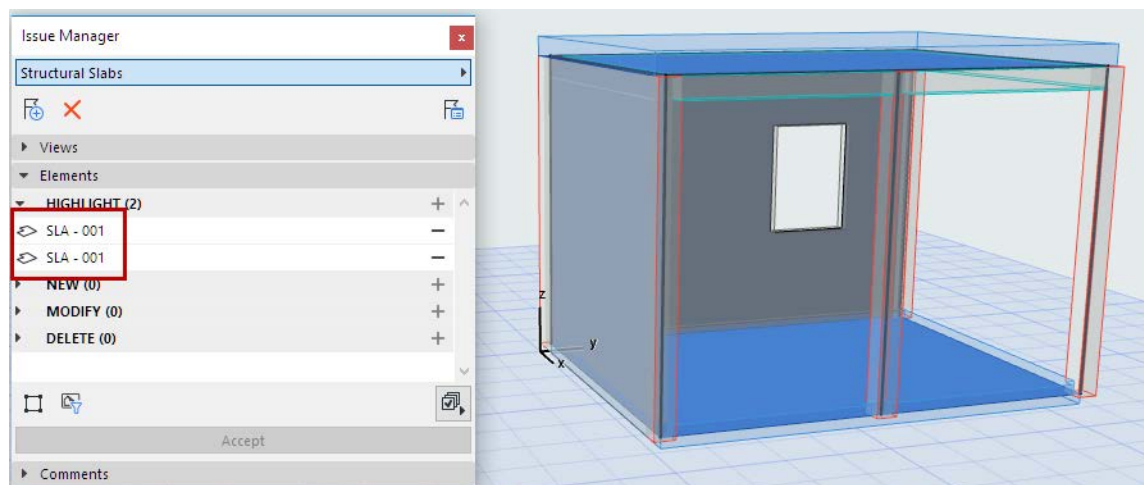
Highlight

This is a simple markup. Any element that has its own settings in Archicad can be highlighted in one or more Issues.

Elements of newly created Issues are shown as “Highlight” by default in Issue Manager, and they are highlighted in the model.

For example, when you create an Issue from the Model Check Report, the elements contained in the Issue are all listed in the Highlights section.

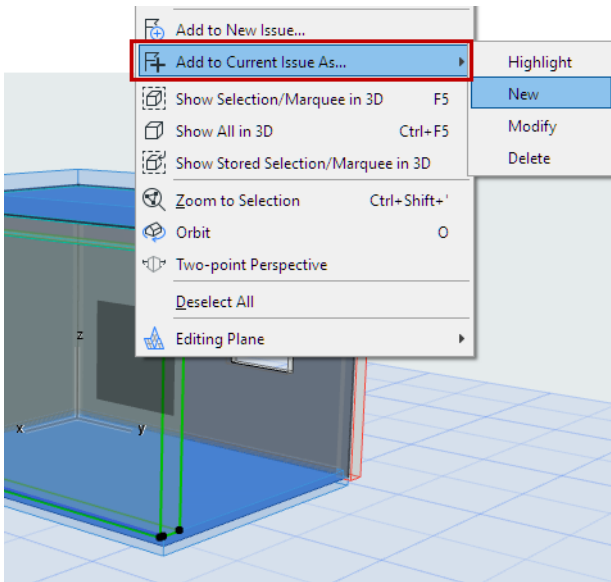
[See Create Issue from Model Check Report.](#)



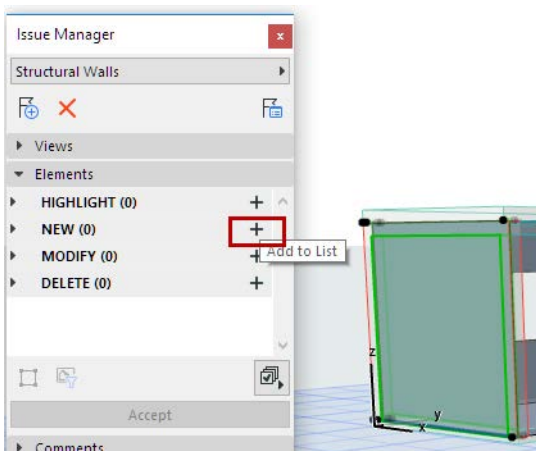
Make Suggestions with Issue Elements: New, Modify, Delete

To add element(s) to an Issue that is currently open in Issue Manager:

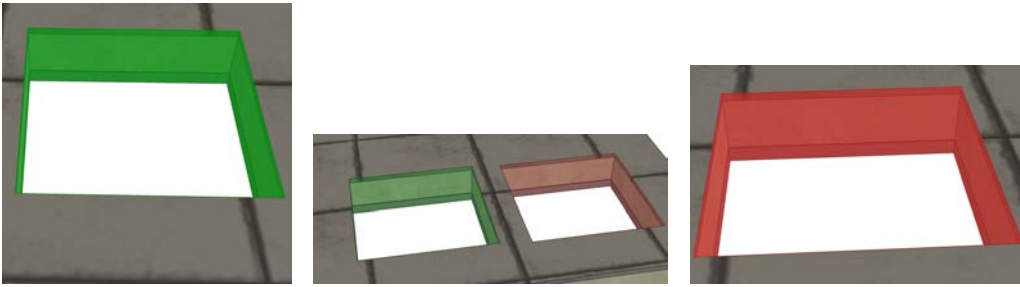
1. Select the model element(s).
2. Do one of the following:
 - Use the context menu command “Add to Current Issue as”



- In Issue Manager, click one of the Plus buttons to move the selected element to Highlight, New, Modify or Delete.



- “Highlight” elements can be moved to New, Modify, or Delete
- “Delete” elements can be moved to Highlight, New, or Modify
- “New” or “Modify” elements cannot be moved to a different category, because they do not exist in the model (in their suggested new form) until you accept them. Once they are accepted, they go to the Highlight list.



Suggestions for element changes - New, Modify and Delete

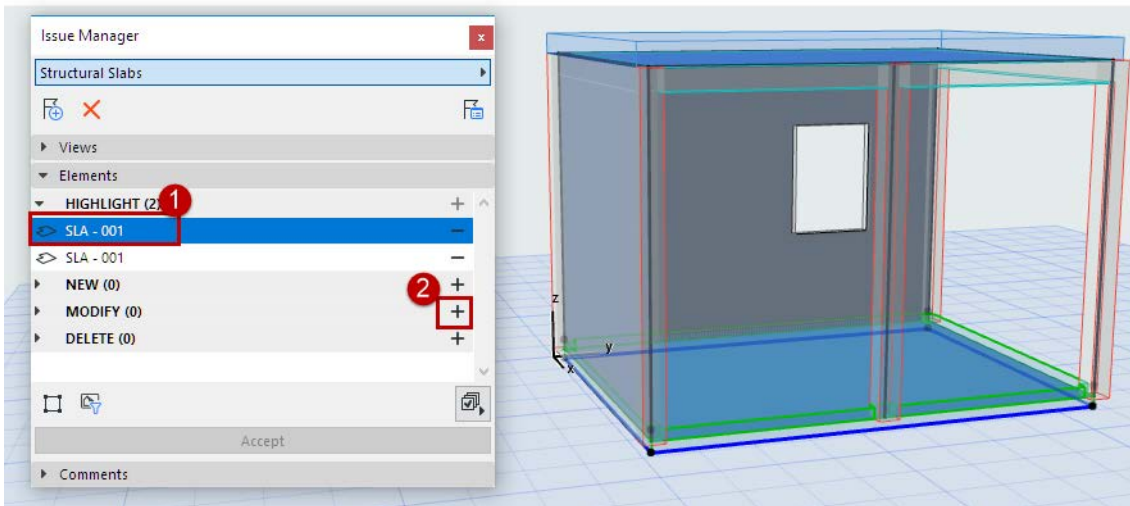
What Happens if a Door/Window/Skylight is “New” or “Modify”?

When you suggest a Door/Window/Skylight as “New” or “Modify”, Issue Manager lists its parent element (Wall or Roof), rather than the opening element you suggested. (Reason: In Archicad, you can’t create or change a Door/Window/Skylight separately from its parent element.)

Example: Suggest Element as “Modify”

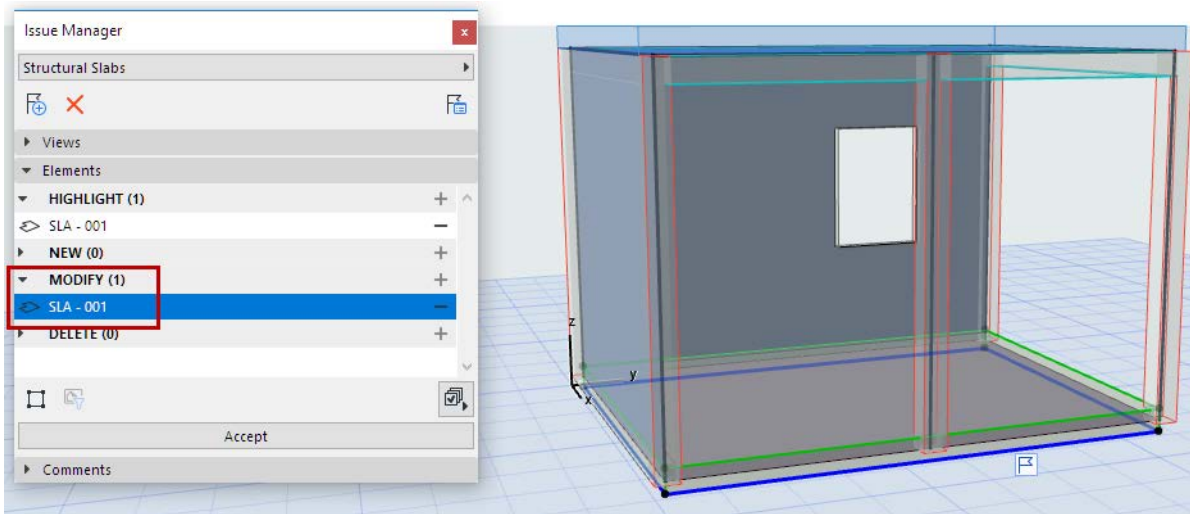
In Issue Manager:

1. Double-click the element in the “Highlight” list. This selects the element in the model.
2. Click the **Plus** sign of the status where you want to move it: for example, “Modify”.



The selected Issue element is now in “Modify” status. Although it is a single element in the Issue, it exists as temporary element pair in the model (“Modified New” and “Modified Deleted”).

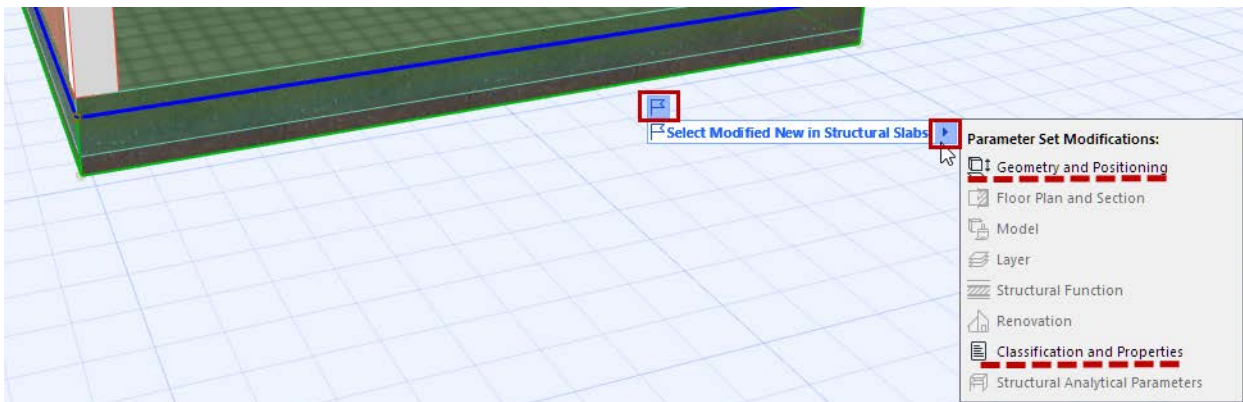
On the Floor Plan, it is displayed with a Marquee, so it is easy to locate.



For a selected “Modify” element in the model:

Use the flag pop-up

- to select its pair in the model (“Modified New” or “Modified Deleted”)
- to see the name of the Issue(s)
- to see which element parameters will be affected by the proposed change, if it is accepted.



Remove Element from Issue

Click the Minus sign to remove any element from the Issue.

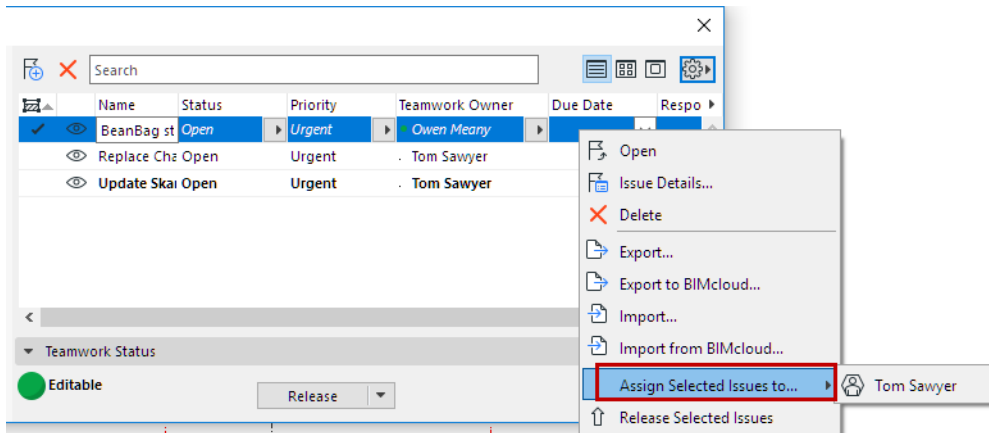
Workspace Colors of Issue Elements

Issue elements are displayed in the model using the colors defined at Options > Work Environment > Workspace Colors.

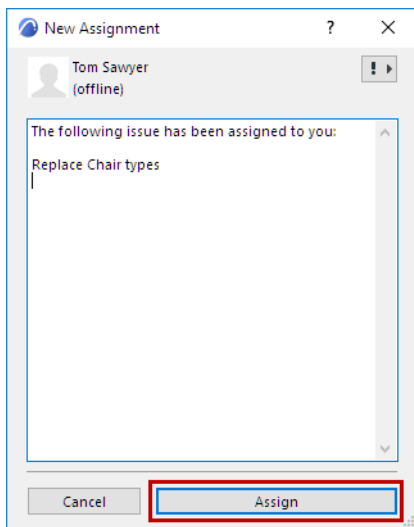
Assign Issue to User

In Teamwork, every suggestion (via its containing Issue) can be assigned to any other user, who is instantly notified.

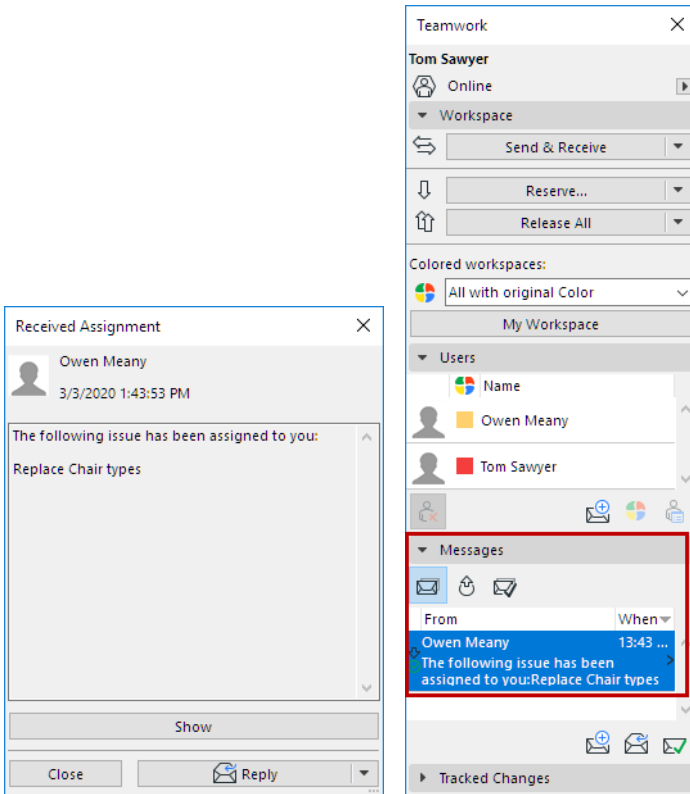
1. Open **Issue Organizer**
2. Select the Issue(s) to assign
3. Do one of the following:
 - Choose **Assign to** from the context menu
 - In the **Teamwork Owner** field, click the pop-up to choose a user.



4. In the appearing dialog, add or edit the message as needed, then click **Assign**.

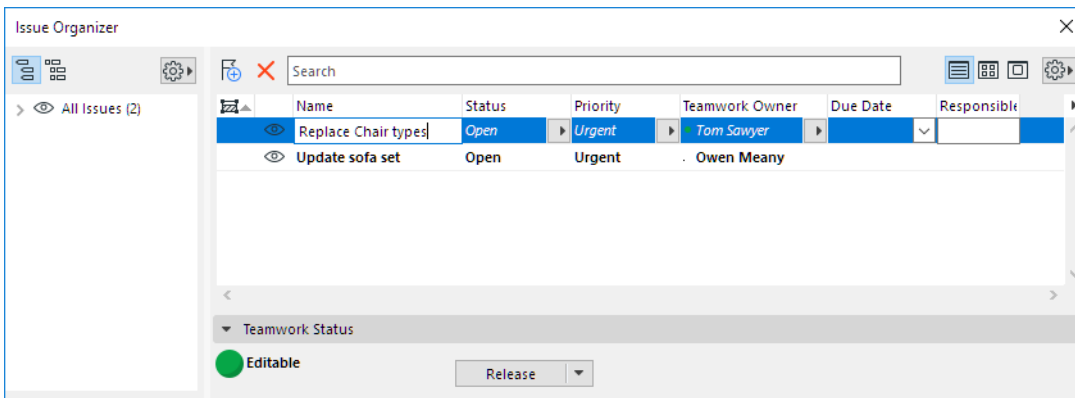


5. The receiving user is notified (via Teamwork Messaging)



The Issue is now reserved by the user to whom it has been assigned.

6. The new Issue owner can click **Show** to see Issue Organizer, and handle the Issue as needed.



Accept Issue Element(s)

Suggested design changes in any Issue (the “New”, “Modify” and “Delete” elements) are not carried out in the model until the assigned user accepts it.

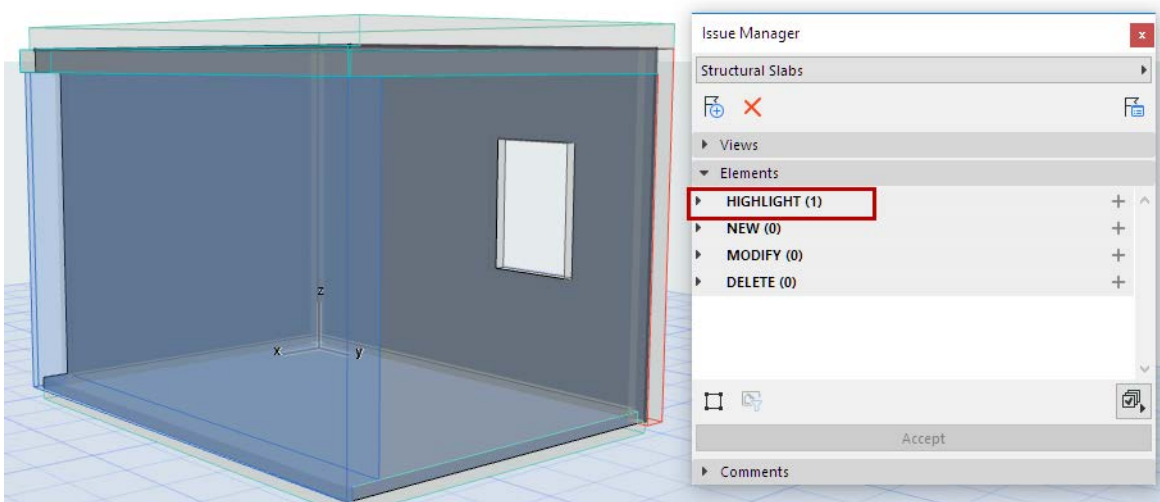
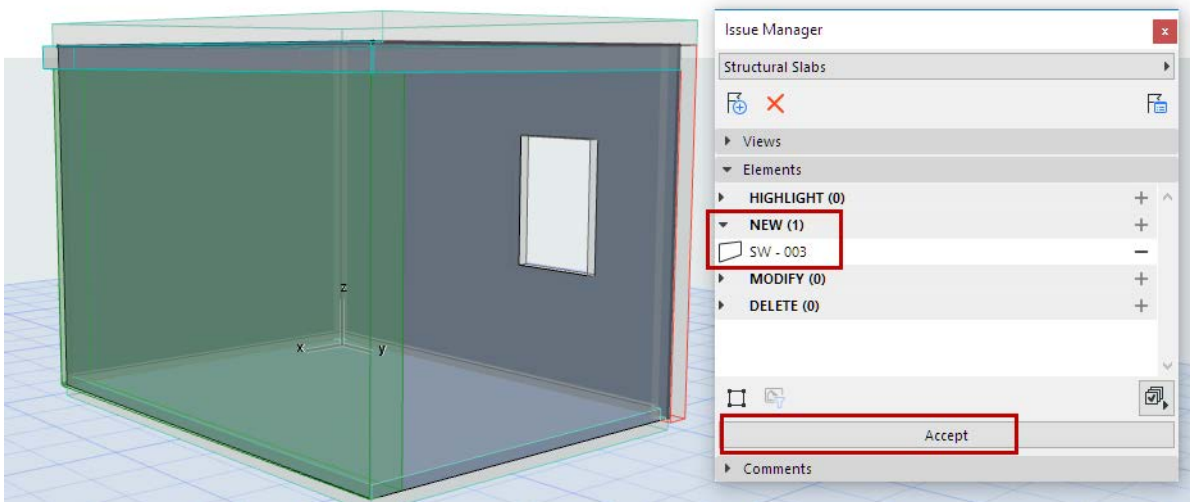
1. In Issue Manager, select the suggested New/Modify/Delete items which you agree with.
(If you do not select any, they will all be accepted.)

2. Click **Accept**.

Once a suggestion is accepted fully or partially:

- “Delete” elements, once accepted, are removed from the project and no longer listed in the Issue
- “Modify” elements/element parameters replace the previous instances in the model, and move to the Highlight list of Issue Manager
- “New” elements become part of the model and move to the Highlight list of Issue Manager

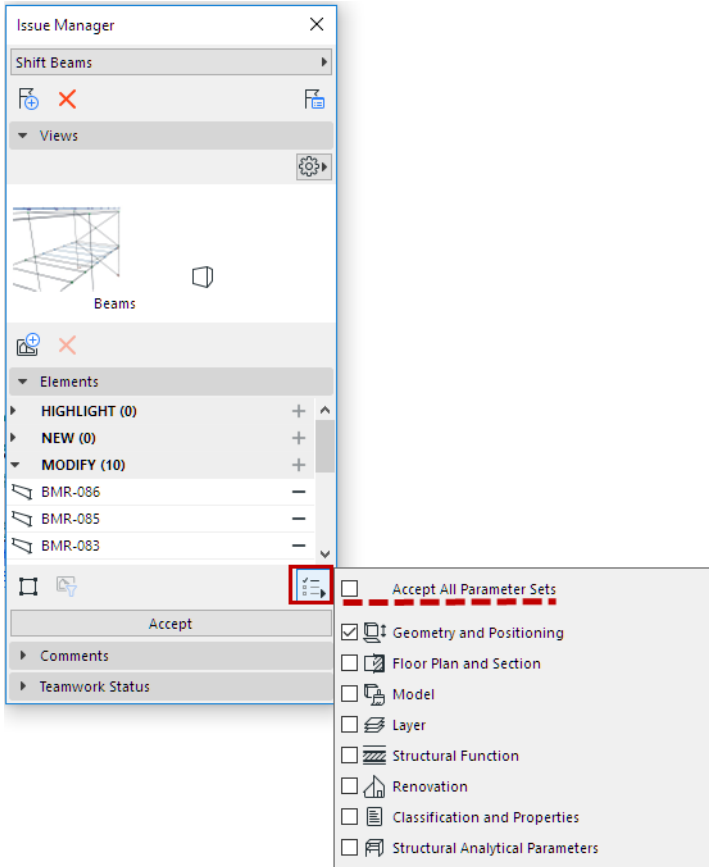
For example, a Wall that is a “New” Issue element does not become part of the model until you click Accept in Issue Manager. Then the element moves to the Highlight part of the Elements list.



Accept Limited Parameter Changes

When receiving an Issue with proposed element changes, you can decide to accept only changed parameters.

1. In Issue Manager: At the bottom of the Elements list, open the Parameter Sets pop-up.
2. Uncheck the box for **Accept All Parameter Sets**.



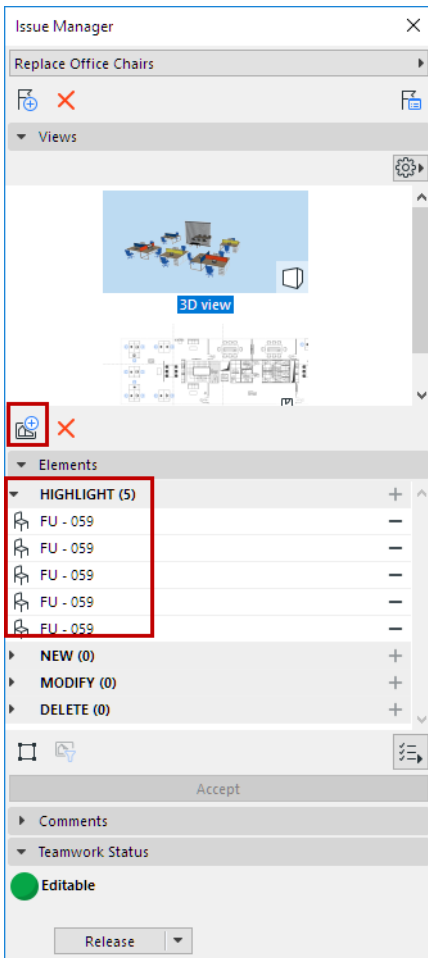
3. Now check just the parameters that you want to accept in the modified element (e.g. accept the modified geometry, but not the changed classification).
4. Click **Accept**.

Issue Management Workflow: Example

In this example, two Teamwork users use Issues to manage the replacement of office furniture.

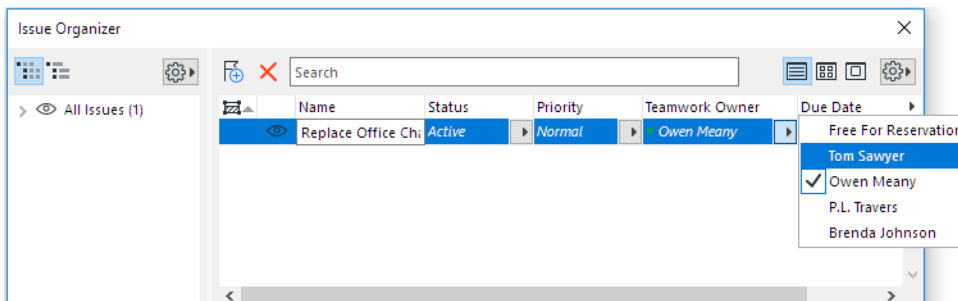
User 1 Creates the Issue

1. Select the problematic objects.
2. Create and name the new Issue. (See [Add Selected Elements to New Issue.](#))
3. The Issue opens in Issue Manager: Issue elements have been added to the Highlights panel.



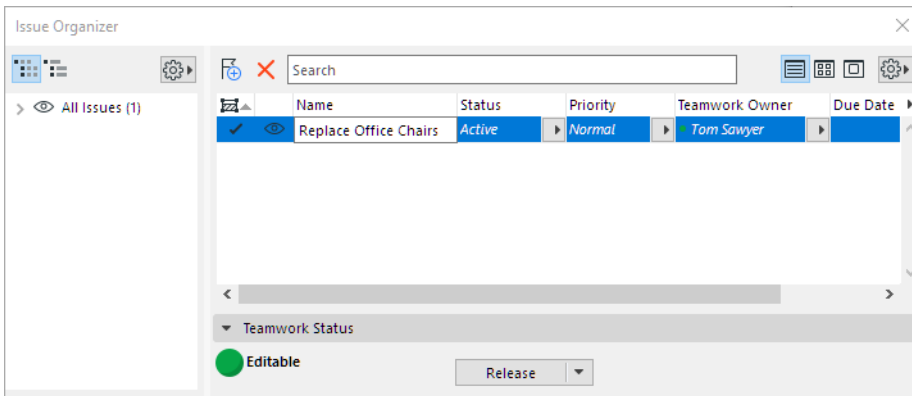
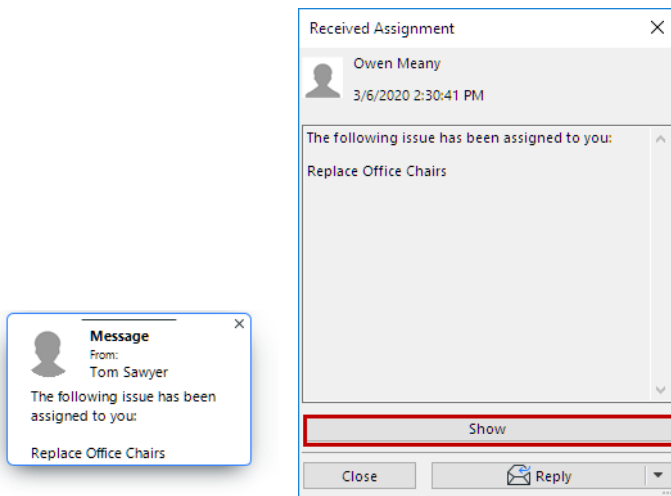
User 1 Gives the Issue to User 2

1. Open Issue Manager
2. In the Issue item's Teamwork Owner column, assign the Issue to User 2. (See [Assign Issue to User.](#))

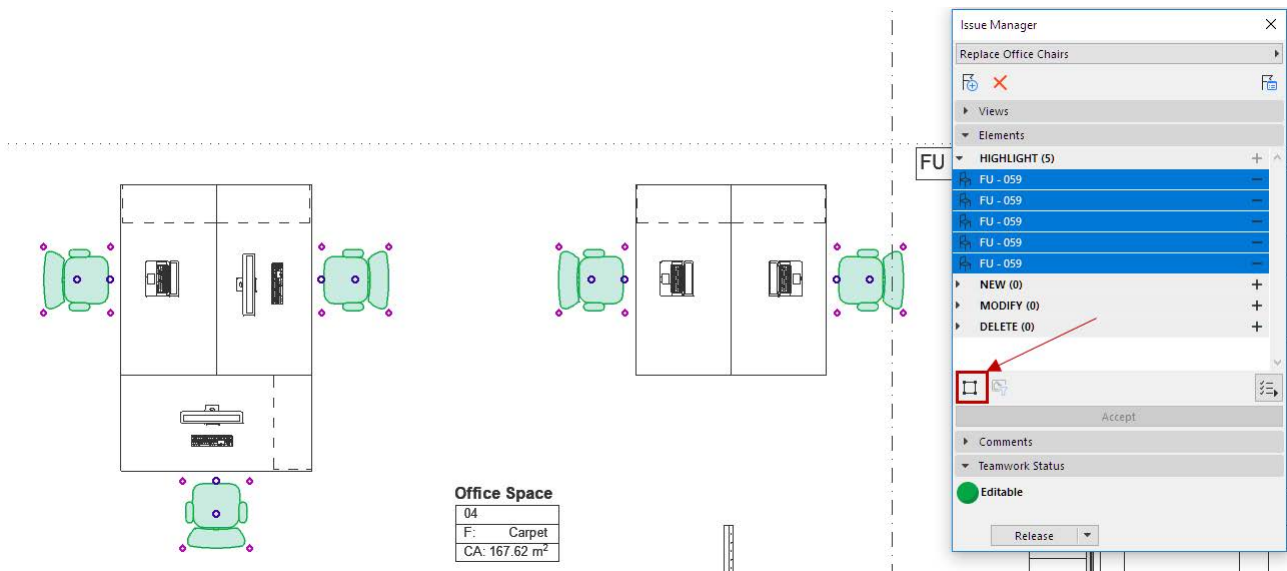


User 2 Gets The Issue

1. User 2 gets the alert and clicks to open the message.
2. From the message, click **Show**: The Issue is listed in Issue Organizer.



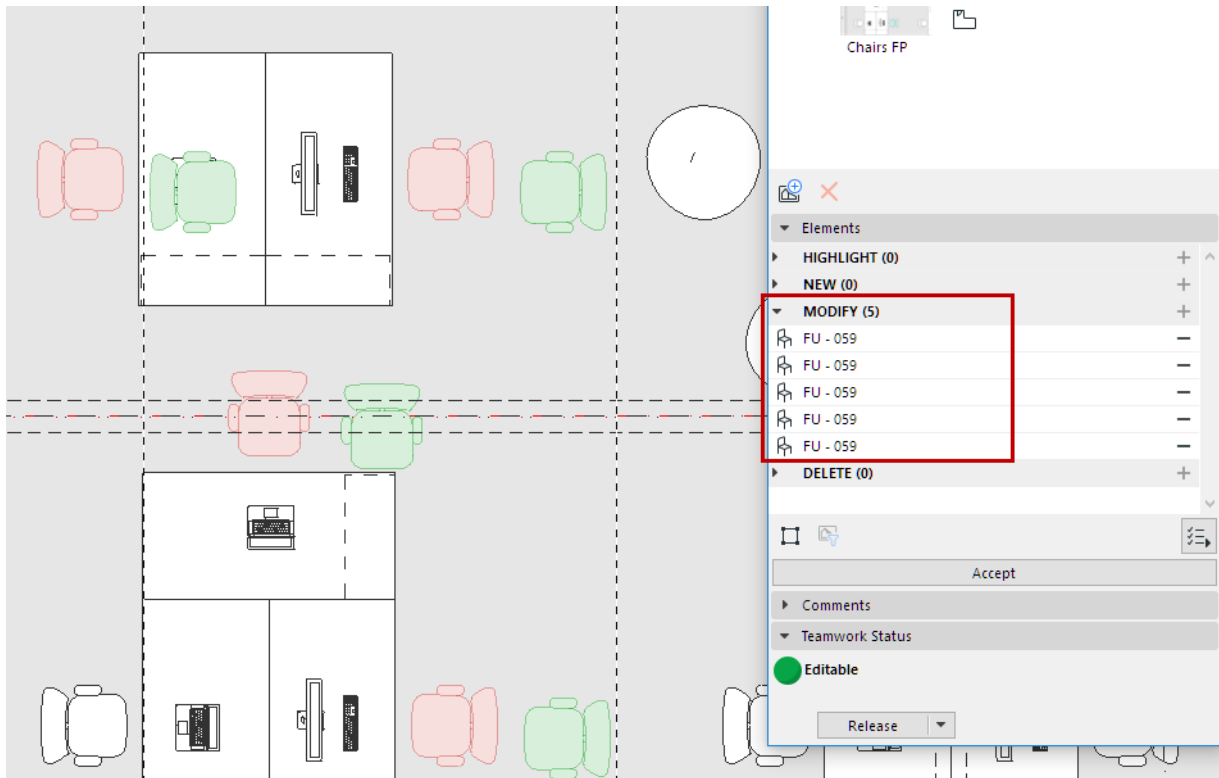
3. Double-click the Issue to see its content in Issue Manager.
4. Click the **Select All Elements** icon to see the highlighted Issue elements close up.



User 2 Modifies the Elements

User 2 decides to update the chairs.

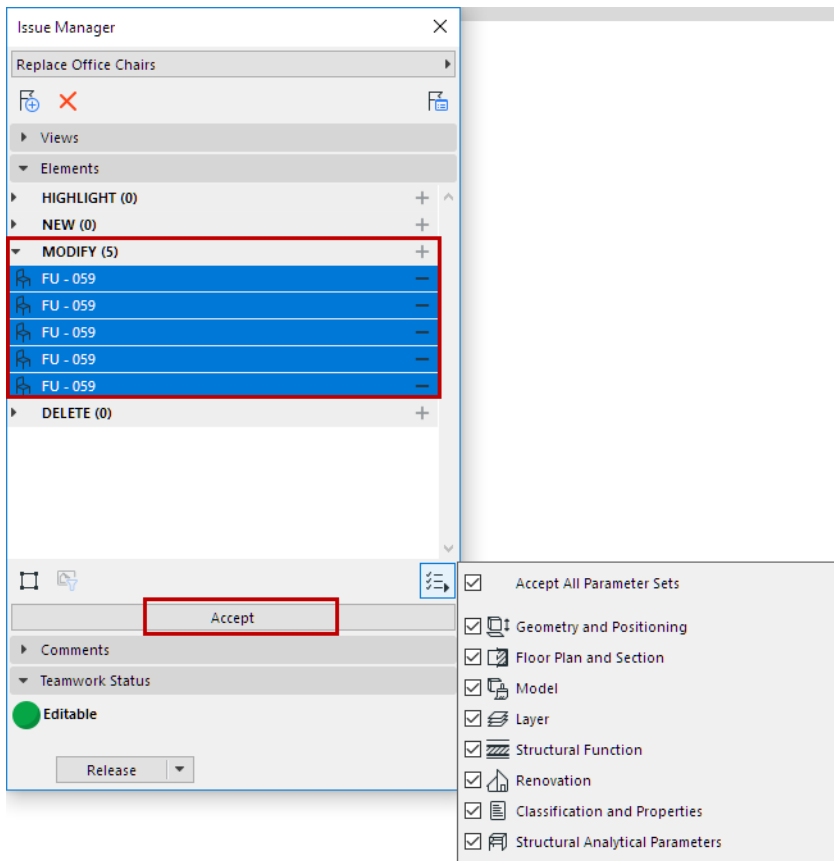
1. Reserve the Issue elements (if needed.)
2. Move the elements to the Modify part of the list in Issue Manager. (See [Suggestion Elements in Issue Manager \(Highlight, New, Modify, Delete\)](#))
3. Select the elements in the model and move/change them to a more modern chair type.



4. Reassign the Issue to User 1. (See [Assign Issue to User.](#))

User 1 Gets The Issue and Accepts the Changes

1. User 1 gets the alert and clicks to open the message.
2. From the message, click **Show**: The Issue is listed in Issue Organizer.
3. Double-click the Issue to see its content in Issue Manager.
4. Zoom in to see the modifications in the model.
5. In Issue Manager, select the elements you want to Accept.
6. Click **Accept**.
 - You can opt to accept only certain parameter sets: Uncheck the parameters you want to skip.



See [Accept Issue Element\(s\)](#).

Issue Organizer

Issue Organizer lists **all the Issues** in your Project.

Open Issue Organizer

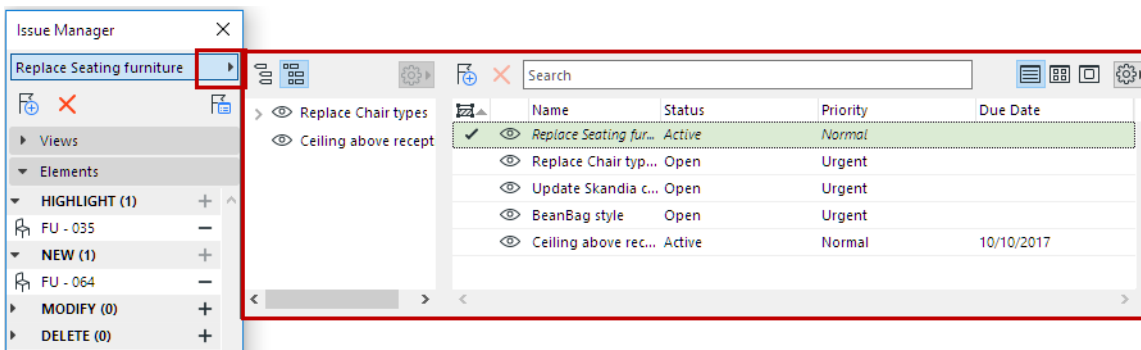
Do one of the following:

- Use its toolbar command



- Document > Issue Organizer
- Window > Palettes > Issue Organizer

You can also access Issue Organizer as a pop-up from the top of Issue Manager.

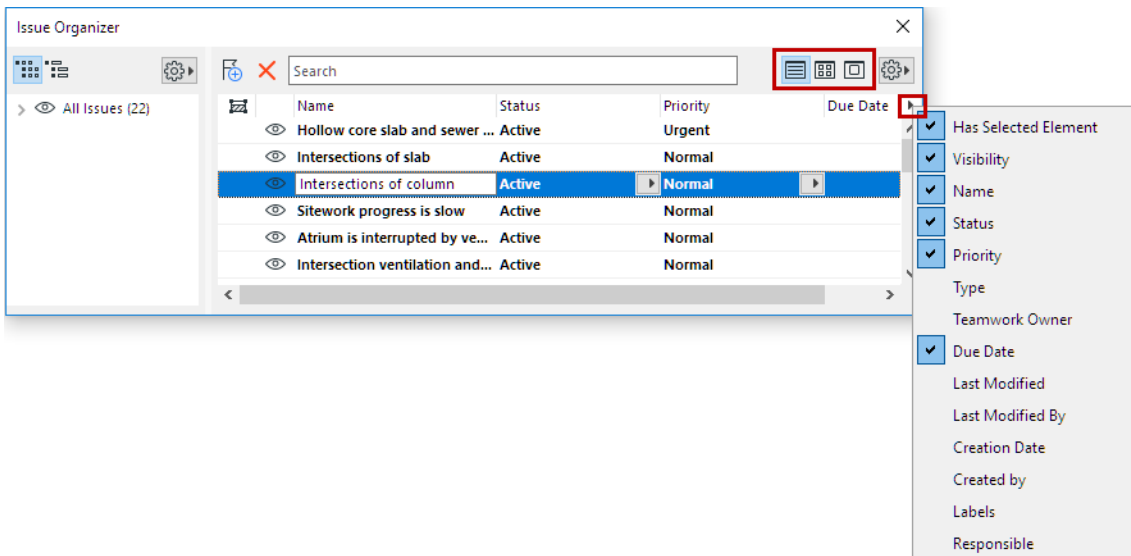


Issue Organizer pops out from Issue Manager

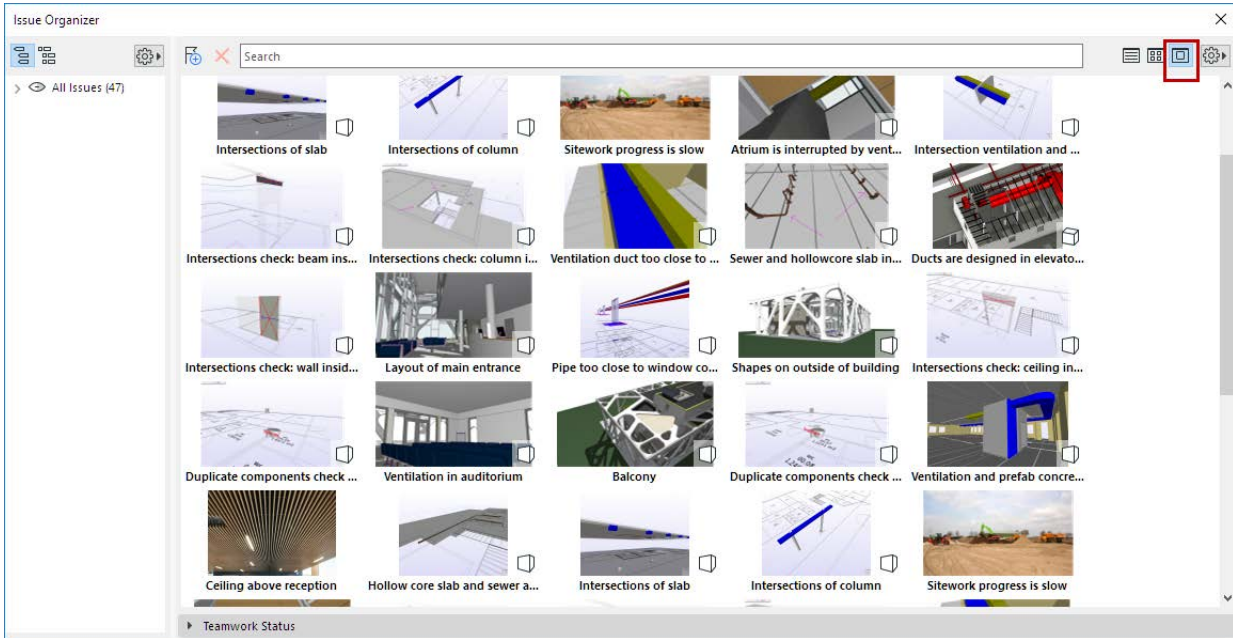
Set Preferred List View and Column Headers

Use the icons and pop-out at the top right of Issue Organizer:

- In List view (default):
 - Choose the column headers you need
 - Click any column header to sort Issues based on that column.



- Or display Issues using medium or large thumbnails:

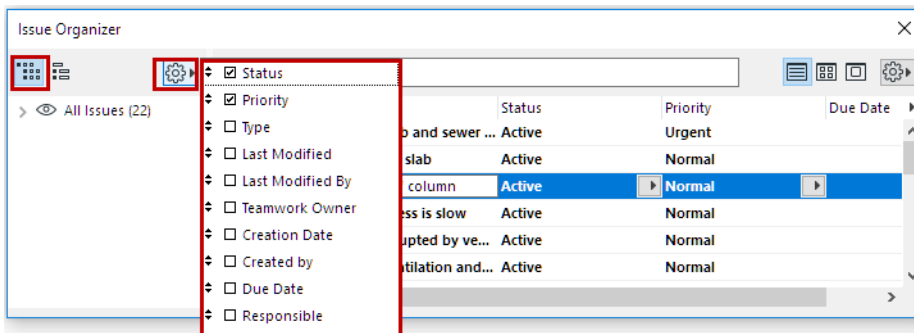


View Issues as Thumbnails

Sort Issue List

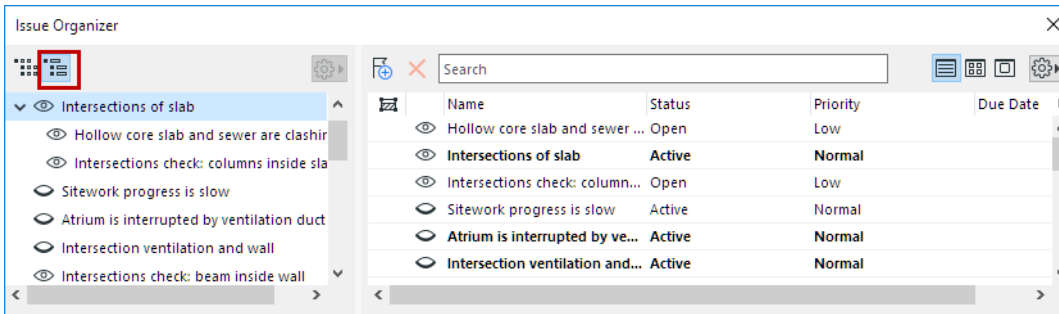
Sort by Tag

1. Choose the left-side icon to sort the listed Issues by tags.
2. Click the settings pop-up to see the available Tags.
3. Define which tags to consider in listing the Issues, and in what order. (Here: by Status, and within that by Priority).



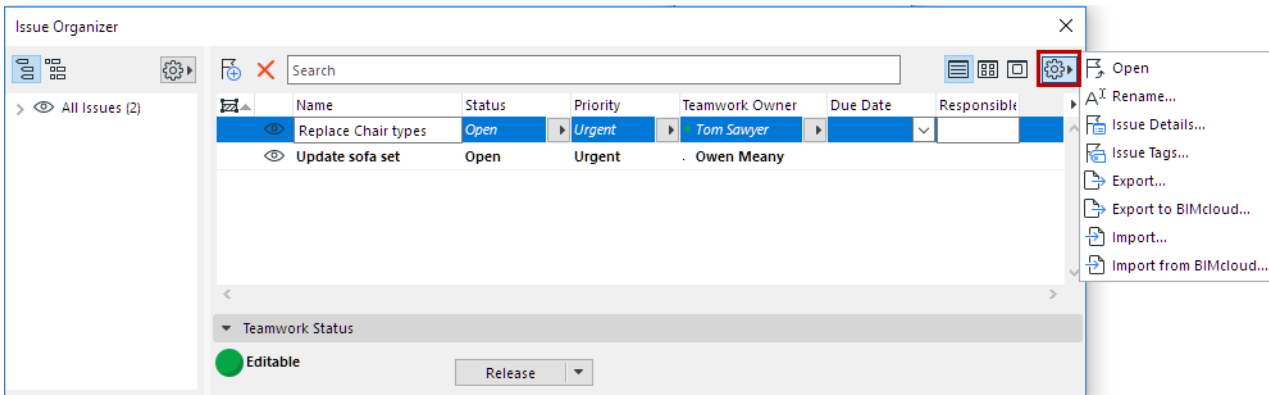
Sort by Thread

Choose the right-side icon to sort by thread.



Within the tree structure, you can drag Issues to manually rearrange the thread hierarchy (provided that you own the Issues you are dragging).

Commands for Managing Issues (Settings pop-up, right side)



- **Open** selected Issue (also via double-click) in Issue Manager
- **Rename** selected Issue
- Open **Issue Details**

See [Issue Details](#).

- **Issue Tags**: Define values for the project's "Priority", "Status" and "Type" Tags. (These values are global definitions, irrespective of current Issue selection)

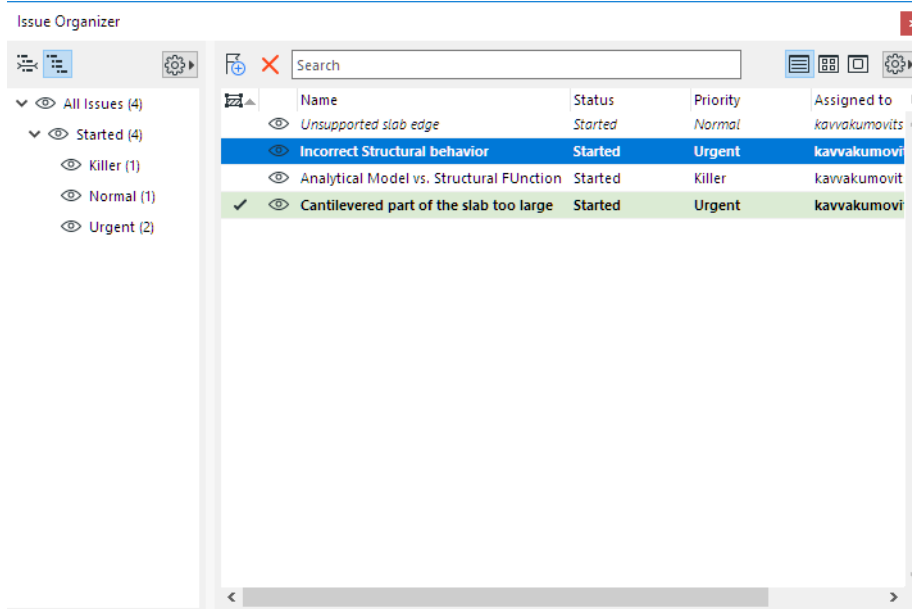
See [Issue Priority, Status, and Type](#).

- **Export/Import** and Export to/Import from BIMcloud
 - Use these commands to export or import BCF files (version 1.0, 2.0 or 2.1)

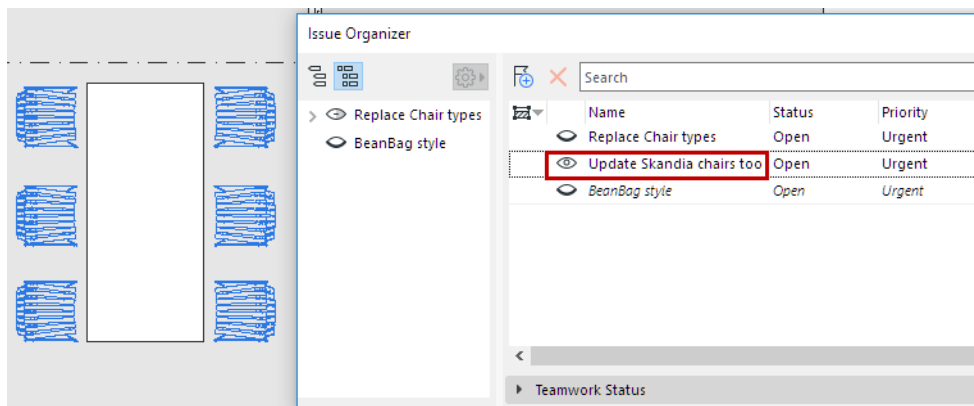
See [Export Issues \(BCF format\) and Import Issues \(BCF format\)](#).

Color Feedback from Issue Organizer

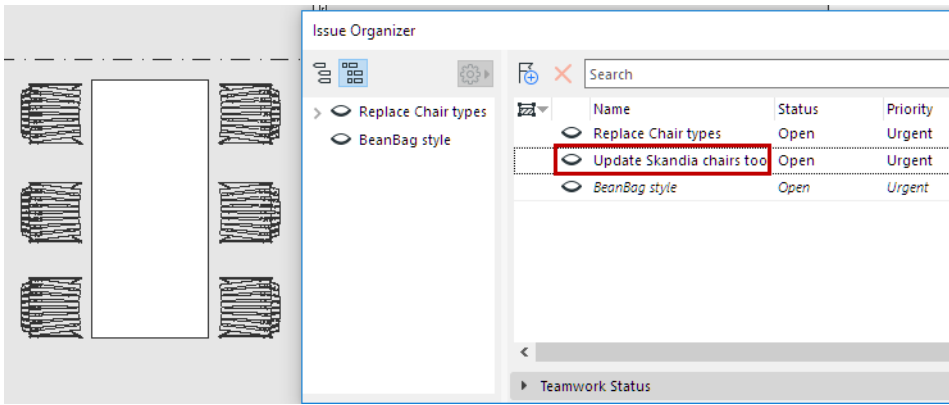
The format of listed Issues helps you keep track of the Issue you are working on.



- *Italics*: This element is currently open in Issue Manager
- **Bold**: This Issue has been imported via BCF, but you have not yet clicked on it.
- Checkmark plus **green** highlight: An element from this Issue is currently selected.
- Eye toggle: Show/hide elements/highlights, **if the Issue is not currently open in Issue Manager**:
 - For **New** elements: show or hide the elements
 - For **Highlight/Modify/Delete** elements: show or hide their highlight colors



Eye toggle: Show Issue elements with highlight

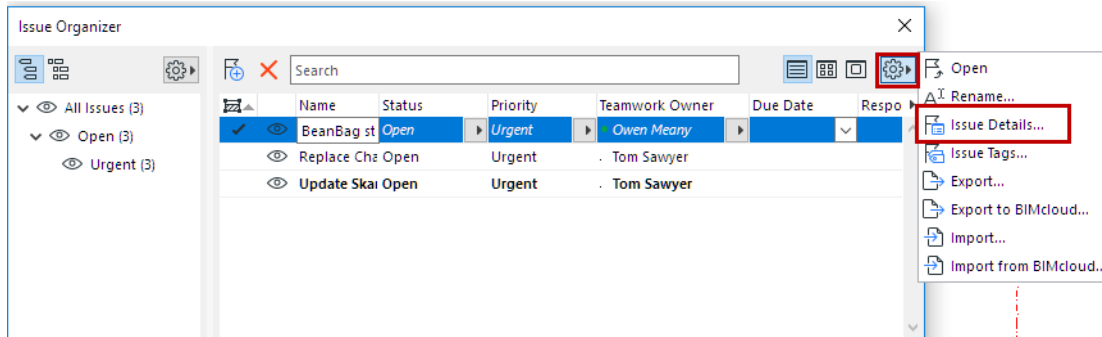


Eye toggle: Hide the highlights on Issue elements

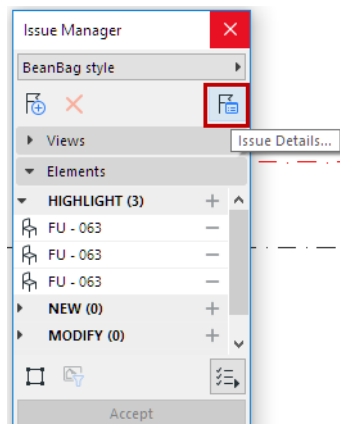
Issue Details

To open this dialog, select an Issue and click Issue Details from one the following:

- Context menu
- Issue Organizer (Settings button)



- Click the **Issue Details** icon from Issue Manager



Use Issue Details to modify any of the following:

- Name
- Status
- Priority
- Type

[See Issue Priority, Status, and Type below.](#)

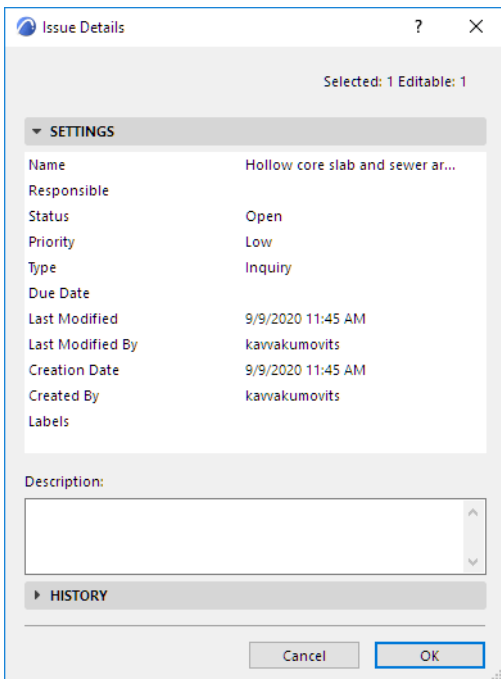
- Due Date

You can also enter text strings in the following fields, which are recognized in BCF-based Issue exchange:

- Responsible
- Labels
- Description

You can use these tags and data fields to configure the Issue list in Issue Organizer.

[See Sort Issue List](#)

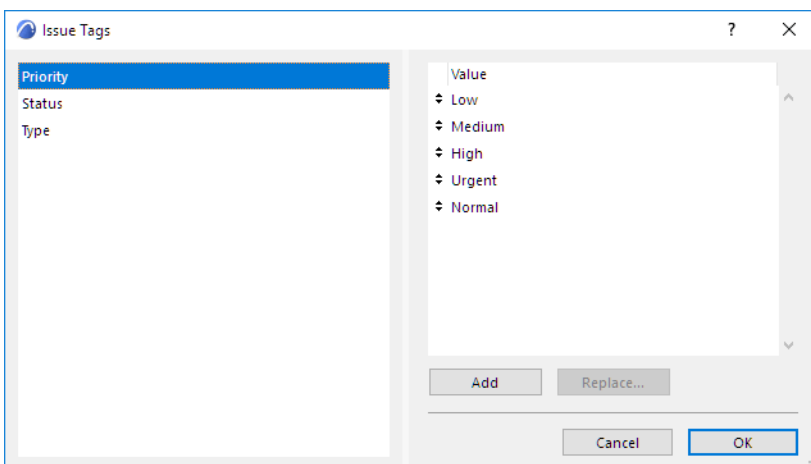
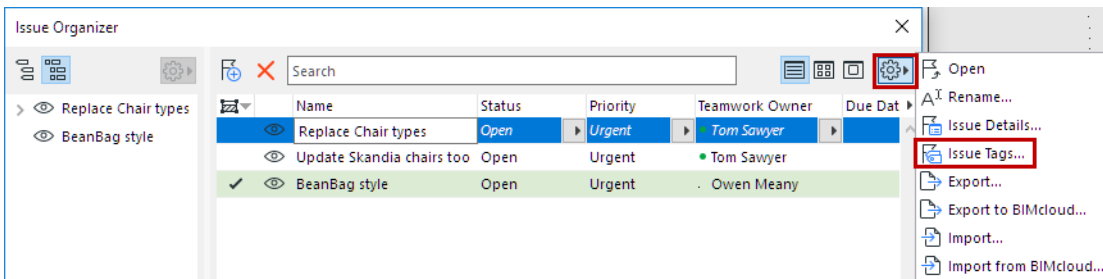


The History panel of Issue Details logs all activities related to a particular Issue.

Issue Priority, Status, and Type

To add or change the available values for Priority, Status and Type:

From the Settings pop-up of Issue Organizer, click **Issue Tags**:



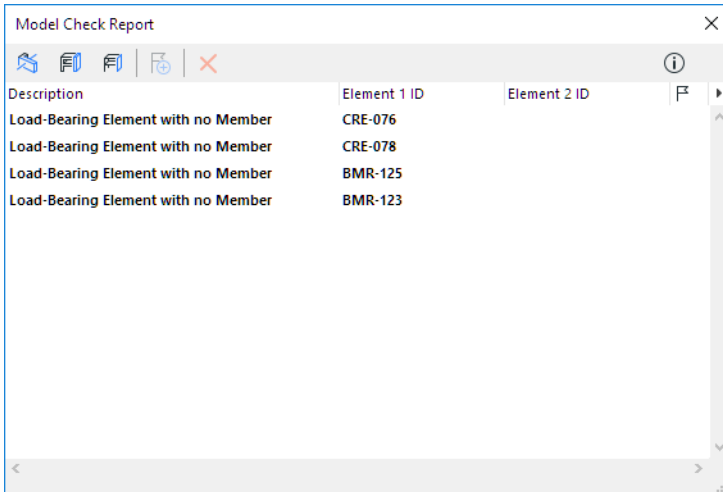
Create Issue from Model Check Report

The Model Check Report lists problems found via Archicad's three checking functions (at **Design > Model Check.**)

See [Model Check - Overview](#).

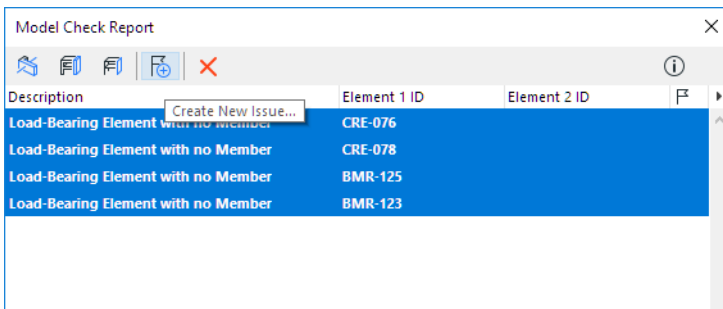
You can add any item listed in the Model Check Report to an Issue for further handling.

In this example, the check shows that four Load-Bearing elements are not part of the Structural Analytical Model, which in this model is a mistake.



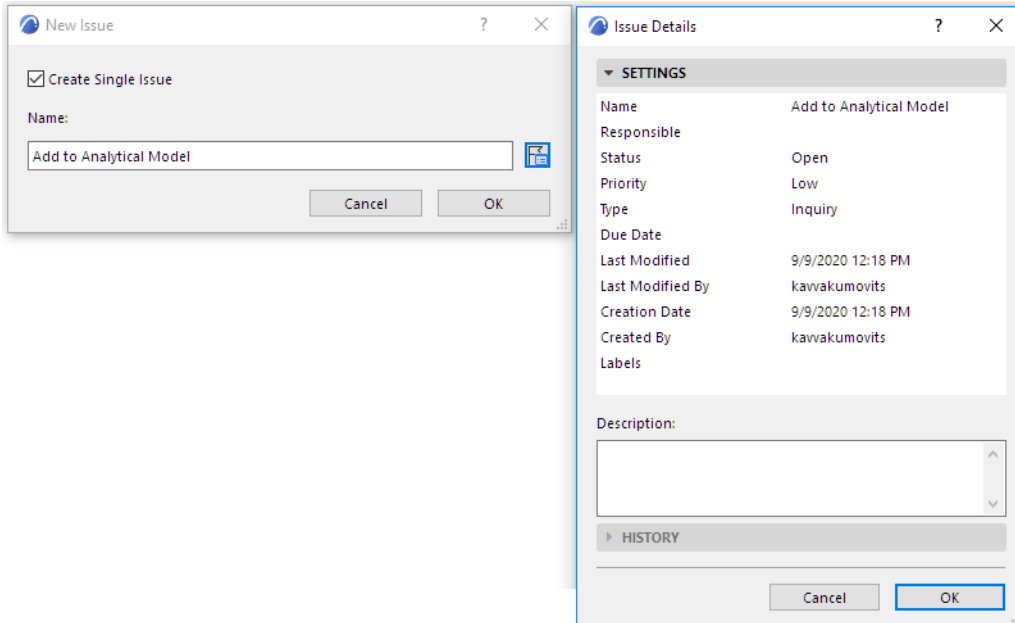
Create one or more Issues out of these listed items.

In the Model Check Report:



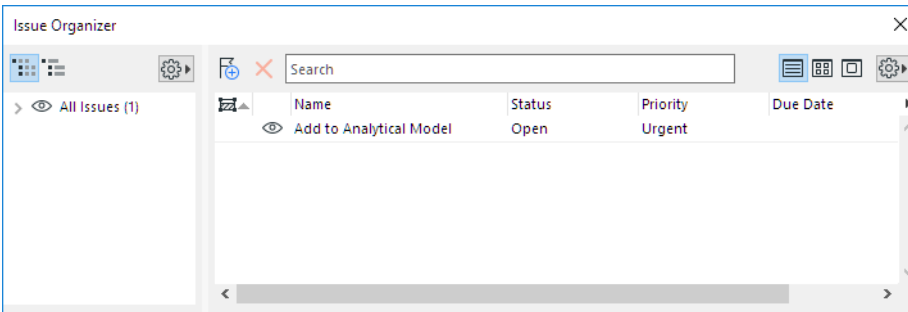
1. Select the problems - here, all four of them.
2. Click the Create New Issue icon at the top of the dialog.
3. In the appearing dialog, add the four items to a single issue.
4. Enter a name: "Add to Analytical Model."

5. Click the flag icon to open **Issue Details**: define Status, Priority, Description as needed, then click OK.



6. Back in the New Issue dialog, click **OK**.

7. The new Issue is added to Issue Organizer.



To manage or accept elements of the Issue:

[See Suggestion Elements in Issue Manager \(Highlight, New, Modify, Delete\).](#)

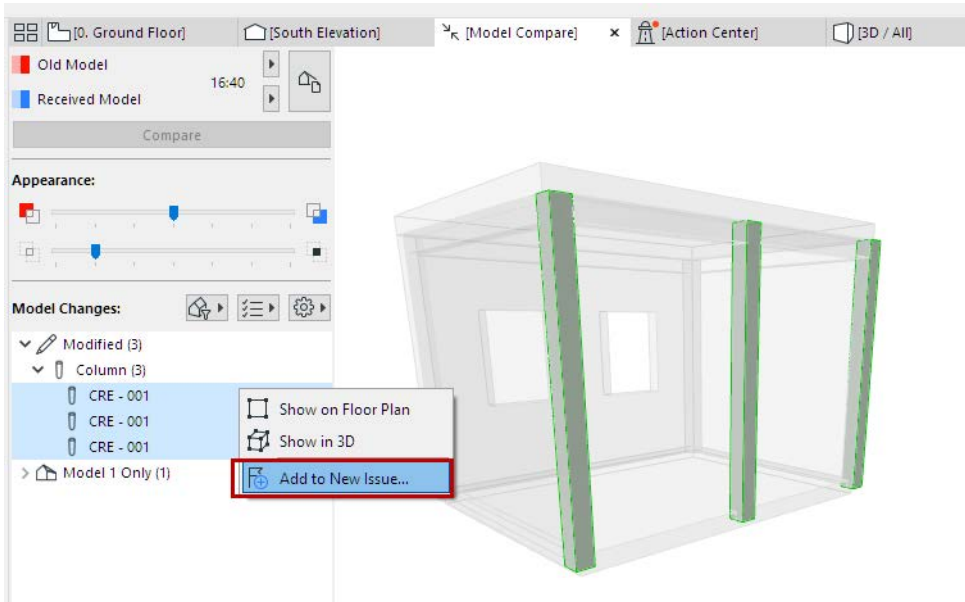
Create Issue from Model Compare Change List

The Model Compare function provides a list of changes between two models. Changes are identified based on “new”, “modified”, and “deleted” 3D elements.

For details, see [Model Compare](#).

From the Model Compare palette's Change List:

1. Select one or more listed elements.
2. Use the context menu to **Add to New Issue**.



3. The new Issue is added to Issue Organizer.

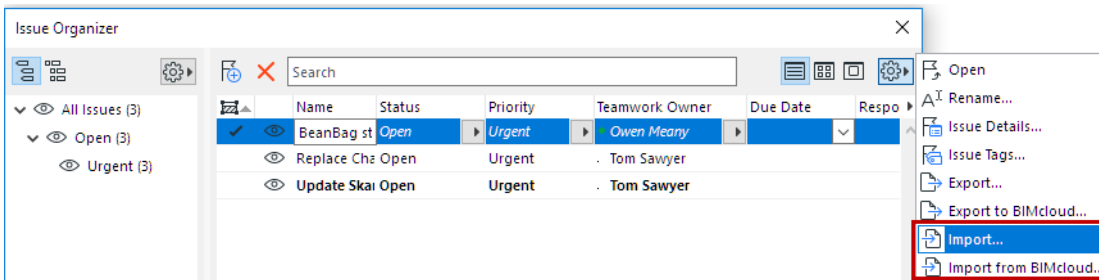
To manage or accept elements of the Issue:

See [Suggestion Elements in Issue Manager \(Highlight, New, Modify, Delete\)](#).

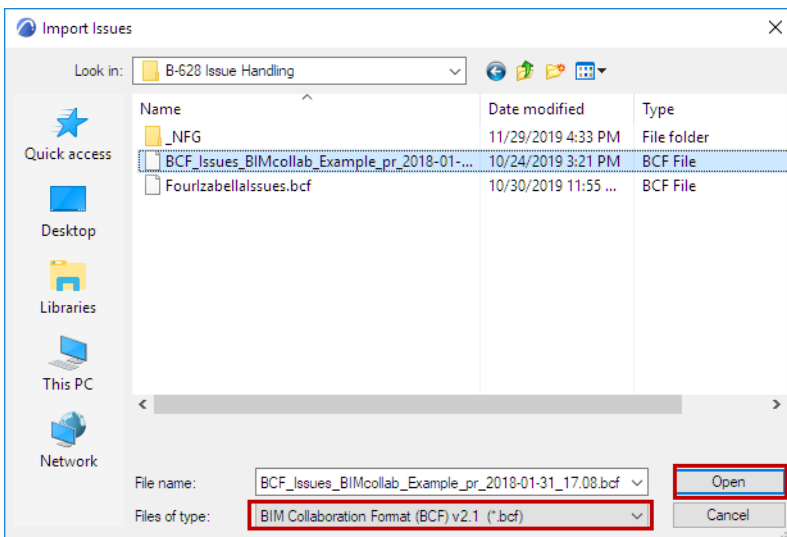
Import Issues (BCF format)

1. Go to Issue Organizer.
2. From the Additional Settings pop-up button:
 - **Import**
 - **Import from BIMcloud**

If needed, log in to BIMcloud. [See Log in to BIMcloud.](#)

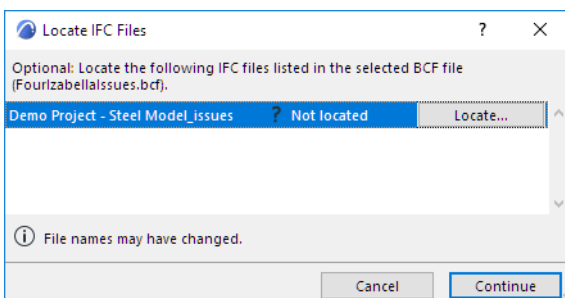


3. Browse for a file in BCF format. (All BCF formats are supported.)
4. Click **Open**.



Note: When importing a BCF file, you don't need to also re-import the IFC file that you created out of your Archicad model. The elements in the BCF file will identify the Archicad elements in their native environment, because the Archicad elements already include a corresponding IFC GUID.

5. The **Locate IFC Files** dialog box appears.

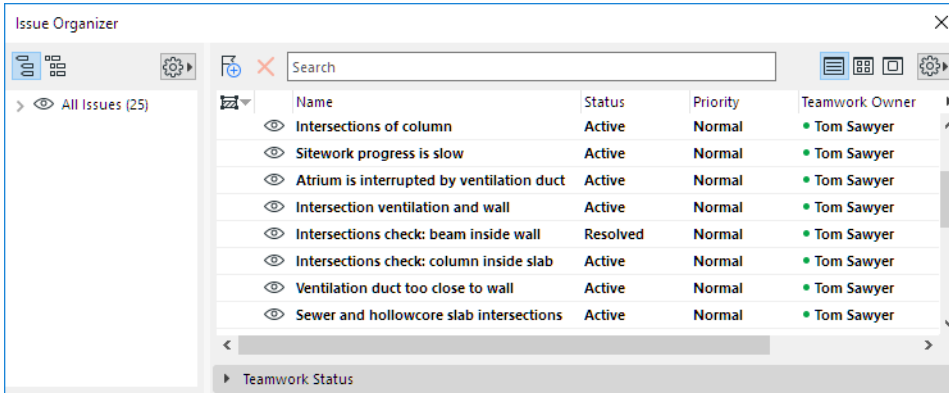


Here, you can optionally browse for and locate the IFC files referenced in the BCF entry. The IFC file names are listed: select one and then click **Locate** to browse for the file location. (IFC files are identified based on their GUID, not their name.)

Note: Locating the referenced IFC files is optional. Click **Continue** to skip this step. However, you may wish to locate the referenced file if, for example, composite elements were exploded at IFC export and thus were assigned different GUIDs: locating the original file here will allow the BCF file to identify the original elements.

6. Click **Continue**.

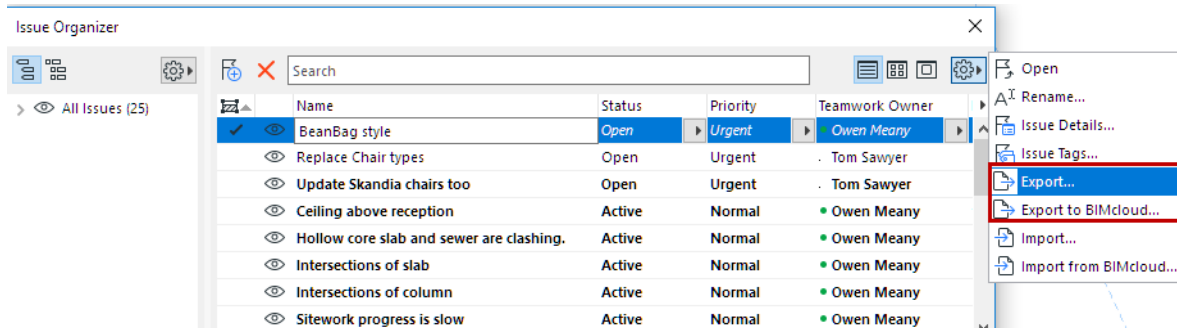
7. The imported Issues are added to Issue Organizer. Each is shown in bold type, until you open it.



Export Issues (BCF format)

1. Go to Issue Organizer.
2. Select one or more Issues.
3. From the Additional Settings pop-up button:
 - **Export**
 - **Export to BIMcloud**

If needed, log in to BIMcloud. [See *Log in to BIMcloud*](#).



4. Save the file in BCF format. (All BCF versions are supported; choose a BCF version from the Save as type drop-down list.)
5. Click **Save**.
6. All the exported Issues are saved in a single .BCF file.

Note: When exporting an Issue in BCF format: if the Issue contains a Floor Plan view, it will generate a 3D axonometry top view, with cutting planes at the top and bottom level of the floor.

Export/Import Issues Using PDF Markup

Save or publish Archicad views or layouts as PDFs, then send the PDFs to consultants for markup in a reviewer application (e.g. Bluebeam).

Then import the PDF markups back to Archicad via the Issue Organizer. The imported markups will appear in the right places, on the original source views in Archicad.

Repeat the process as needed. Each time you reimport markups from a PDF, you can decide whether to skip or include duplicate or modified Markups from earlier rounds.

Notes:

- This method works only if the marked up views originate in Archicad.
- On Mac, do not use the Preview function to mark up PDFs for this workflow.

Create PDF Documents for Markup

In Archicad, use the Publisher to create PDF-based Publisher sets or publisher items.

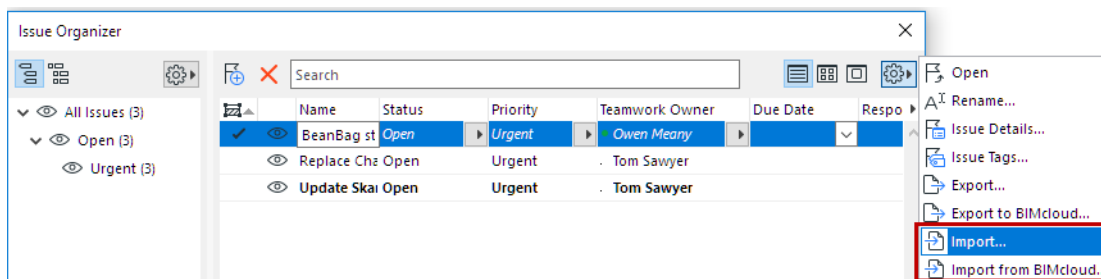
See [Create PDF Output Using the Publisher](#).

Then send the resulting PDF to the consultant for markup in an external application.

Import Issues to Archicad

Once the PDF has been marked up, follow these steps in Archicad to import the Issues:

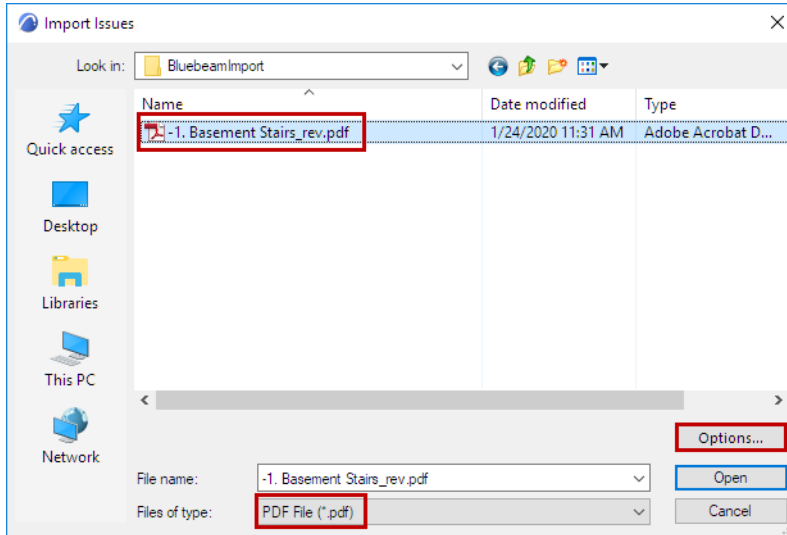
1. Go to Issue Organizer.
2. From the Additional Settings pop-up button:
 - **Import**
 - **Import from BIMcloud**



Browse for a file in PDF format.

If needed, log in to BIMcloud. See [Log in to BIMcloud](#).

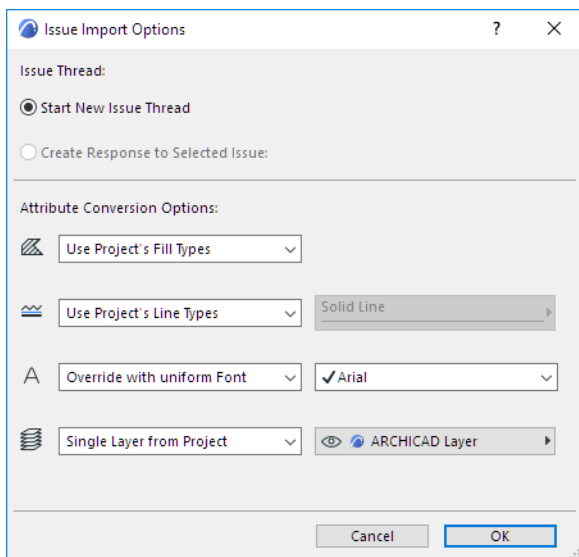
3. Click Options.



The **Issue Import Options** dialog box opens:

Issue Import Options

Choose an option for how the imported Issues should appear in Issue Organizer.

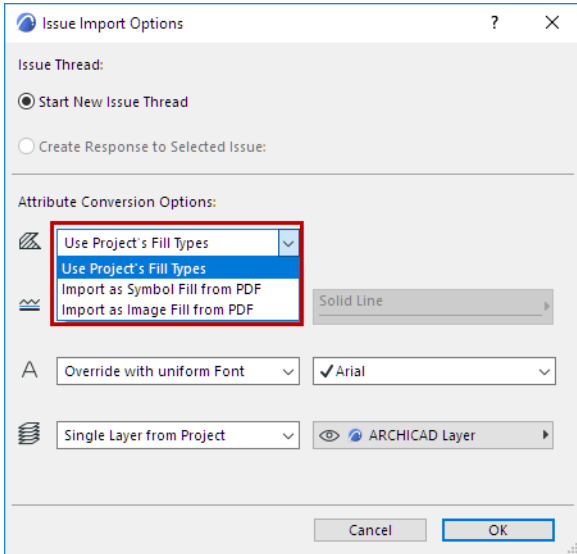


- **Start New Issue Thread:** The imported Issues will constitute their own thread. (In the Issue Organizer, you can sort entries by thread.)
- **Create Response to Selected Issue:** This option is available if an Issue is currently open in Issue Manager. Choose this to list the imported Issues underneath the currently active Issue, as sub-Issues.
Tip: To create a separate “category” for the imported Issues, first create a new Issue in Archicad, and give it a name (e.g. “Imported Issues from PDF”.) With this new entry selected, import the PDF Issues, and use the “Create Response to...” option. The imported Issues will be listed under this Issues (use **Sort by:** Thread).

Attribute Conversion Options

Choose how to convert the fills, lines, fonts and layers of the Issues you are importing.

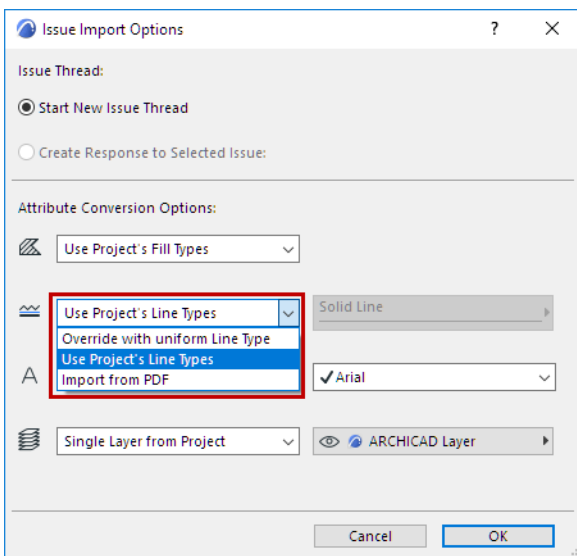
Fills



- **Use Project's Fill Types:** No new fills will be imported from the PDF. Any imported fills that have no counterpart in the project will be transformed into solid fills.
- **Import as Symbol Fill from PDF:** A new vectorial fill attribute will be created in the project and used for imported fills. (For example: "60% PDF Solid Fill")
- **Import as Image Fill from PDF:** A new image fill attribute will be created for any fills, among the drawing primitives, which have no counterpart among the project's attributes.

Note: If an image fill is generated, then this file is added to the project's Embedded Library. If you are working in a Teamwork project, you must have the needed access right: Library Parts – Create.

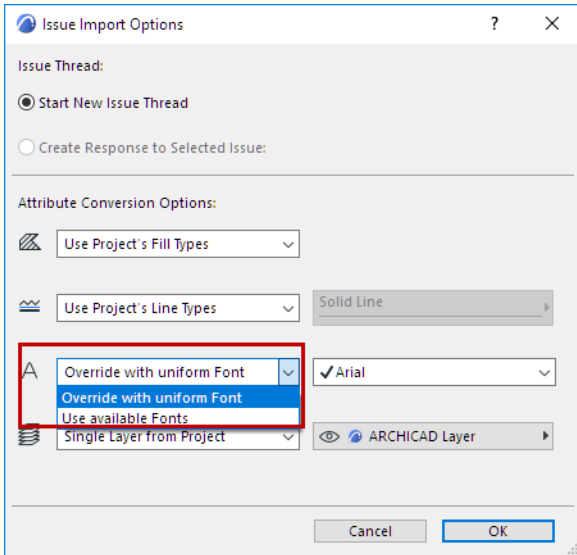
Lines



- **Use Project's Line Types:** No new Lines will be imported. Issues are displayed using similar, existing Archicad lines (e.g. solid, dashed, dotted). If a corresponding Line does not exist in the Archicad project, a solid line is used.

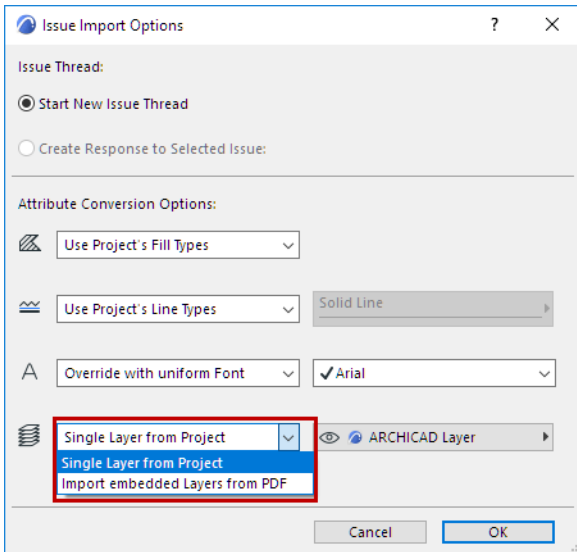
- **Override with uniform Line Type:** Choose one of the project’s Lines. Imported Issues will be displayed using this Line Type.
- **Import from PDF:** Line Types of Issues are imported and added to the Archicad project.

Text Font



- **Override with uniform Font:** Choose one of the project’s font types to display text items on imported Issues.
- **Use available Fonts:** Imported Issue texts are displayed using similar fonts available in the Archicad project.

Layer of Issues

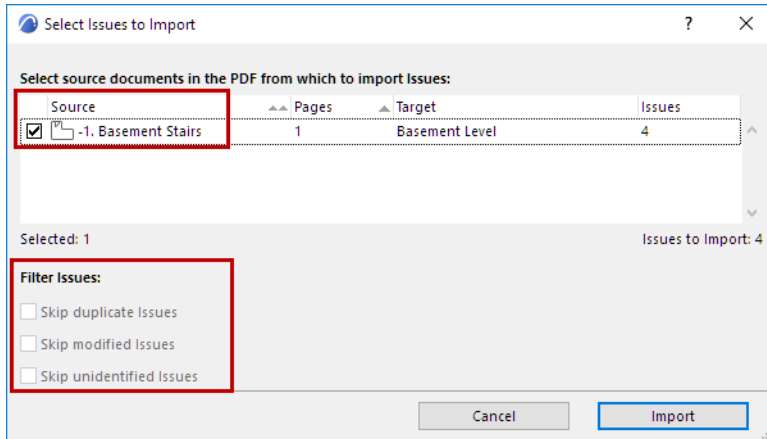


- **Single Layer from Project:** Choose one of the project’s Layers. Imported Issues will be placed on this Layer.
- **Import embedded Layers from PDF:** Issues will be placed on layers corresponding to their embedded layer assignment (if any) in the PDF source. Those embedded layers will be added to your Archicad project.

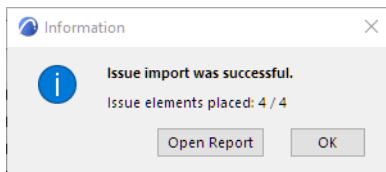
After setting Issue Import Options, click **OK** to return to the Import Issues dialog.

Import Issues

1. From the Import Issues dialog, select the file to import. Click **Open**.
2. The **Select Issues to Import** dialog appears:

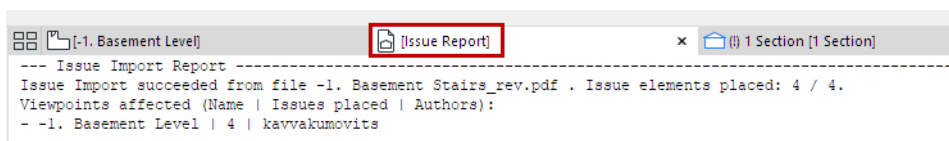


3. Select the source documents in the PDF from which to import the Issues.
4. Filter the Issues as needed:
 - **Skip duplicate Issues:** Duplicate Issues have already been imported to Archicad, and they have not changed in the PDF since then. *Check this to skip importing these duplicate Issues.*
 - **Skip modified Issues.** Modified Issues have already been imported to Archicad, but they were later modified in the PDF. *Check this to skip importing these Issues.*
 - **Skip unidentified Issues:** Some markup applications do not assign unique IDs to the Issues. Thus, these Issues cannot be identified as either new, duplicate or modified. (This is not a problem in the Bluebeam Revu workflow.) *Check this to skip importing these Issues.*
5. Click **Import**. An Info dialog appears:



6. Click **Open Report** to see a summary of Issues that have been added to the original Archicad views, or click OK to skip the report.

The new Issues are also listed in Issue Organizer.



7. Navigate to the source Archicad views or Layouts to handle the imported Issues.

Change Tracking in Teamwork

Note: BIMcloud users only. Not available with BIMcloud Basic.

Use this optional feature in Teamwork to track project changes made by other users, and to receive notifications about such changes.

Change Tracking applies only to elements or data which already exist in the Teamwork project and have been sent to the server. (A newly created element or data type, which has not yet been sent to the server, cannot be tracked.)

To use Change Tracking, create any number of subscriptions. Subscription set-up is flexible, to ensure you are notified about changes that you need to know about.

Notifications appear on screen and in the Teamwork Palette in the dedicated “Tracked Changes” panel and palette. These are changes that have been sent to the server, but not yet received by you. Once you receive the changes, the Tracked Changes list will be deleted.

Define subscriptions for several types of changes:

- Elements by criteria or by selection, optionally filtered by parameter (track only the parameters that are relevant for you)
- Specific project data (such as attributes)
- Further filter any subscription by user or Team

“High-Trust” Collaboration with Teamwork Users

The Change Tracking workflow is not tied to permissions. It is thus a “high trust” workflow that enables Teamwork users to learn about relevant project changes sent by others to the BIMcloud.

For example: the architect, structural engineer, and mechanical engineer alike can potentially modify the same composite Wall. Such a modification constitutes significant information for all three professionals. Change Tracking allows them to track such changes (once they are sent to the BIMcloud) in a timely manner, and to make corrections or comments (via Issues or Messages) as needed.

Modified Element Geometry: What Counts as a Change?

Core Data

A change in element geometry is tracked only if it changes the element’s “core data”. These are geometric values that are defined and visible in the element’s Settings dialog box. For example: Wall height.

Derived Data

In contrast, an element’s “derived data” are *not* tracked as a change in element geometry.

For example, a top linked Wall’s height is derived from the current Story height. A change in Story height will show up in your Tracked Changes list (if your subscription includes Story Settings), but the resulting change in Wall height will not.

Topics in this Section:

[Create New Subscription](#)

[Filter Element Changes by Parameter Type](#)

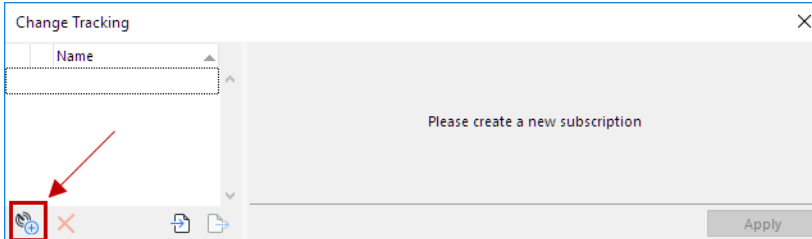
[Filter Change Notifications by Teams and Users](#)

[View and Manage Tracked Changes](#)

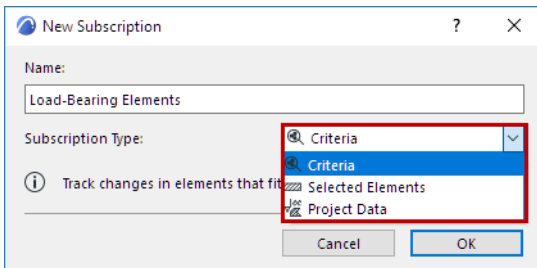
[Manage Subscriptions](#)

Create New Subscription

1. Go to **Teamwork > Change Tracking**
2. In the appearing dialog, click the **New Subscription** icon (or right-click anywhere in the dialog box).



3. Enter a name for the new subscription - for example, "Load-Bearing Elements".
4. Choose one of the three methods to define this Subscription Type:



Each setup is described below:

Subscribe to Changed Elements - by Criteria

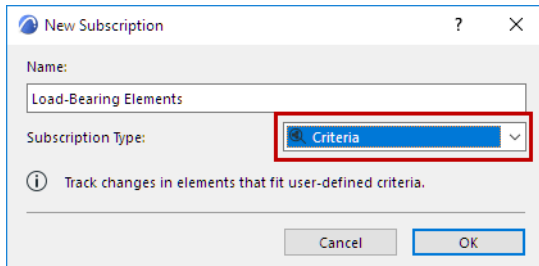
Subscribe to Changed Elements - by Selection

Subscribe to Changes in Project Data

The subscription starts tracking changes and sending notifications from the point at which you create it (changes that were sent to the server earlier cannot be tracked).

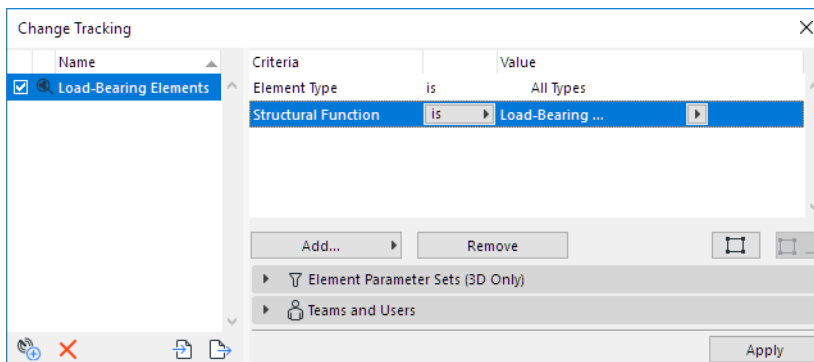
Subscribe to Changed Elements - by Criteria

1. Go to **Teamwork > Change Tracking**
2. In the appearing dialog, click the **New Subscription** icon (or right-click anywhere in the dialog box).
3. In the New Subscription dialog, choose **Criteria**.



4. Click **OK**.
5. In the appearing Change Tracking dialog: use the fields (similar to those in the Find & Select dialog) to define element criteria for the subscription. All elements that meet these criteria will be tracked (subject to any additional filters).

For example, this subscription tracks all elements whose Structural Function is load-bearing.



Note: To quickly set up criteria based on a selected element: use the **Copy Settings** button. [See Copy Settings.](#)

6. In the panels below, you can fine-tune this subscription by narrowing the set of tracked parameters, and/or by Teams and Users.
7. Click **Apply** to activate tracking for this subscription.
8. Change notifications will appear in your Teamwork Palette, heralded by an on-screen pop-up. [See View and Manage Tracked Changes.](#)

Related Topics

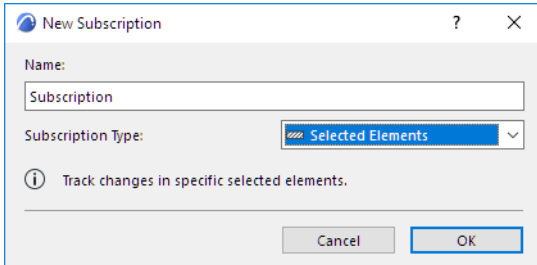
[Filter Element Changes by Parameter Type](#)

[Filter Change Notifications by Teams and Users](#)

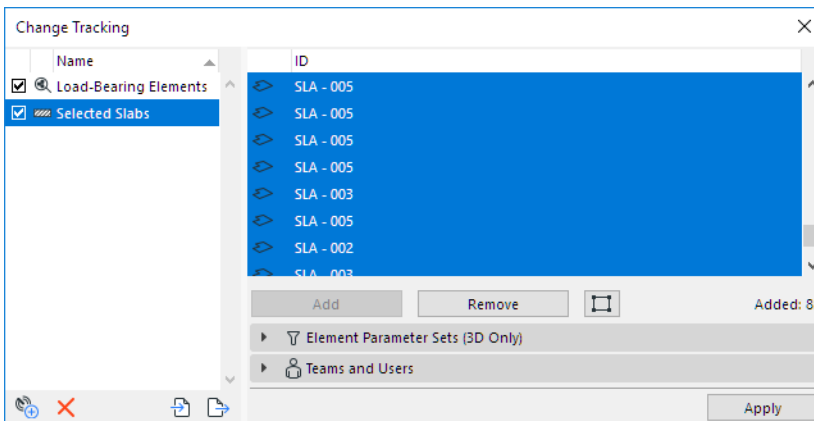
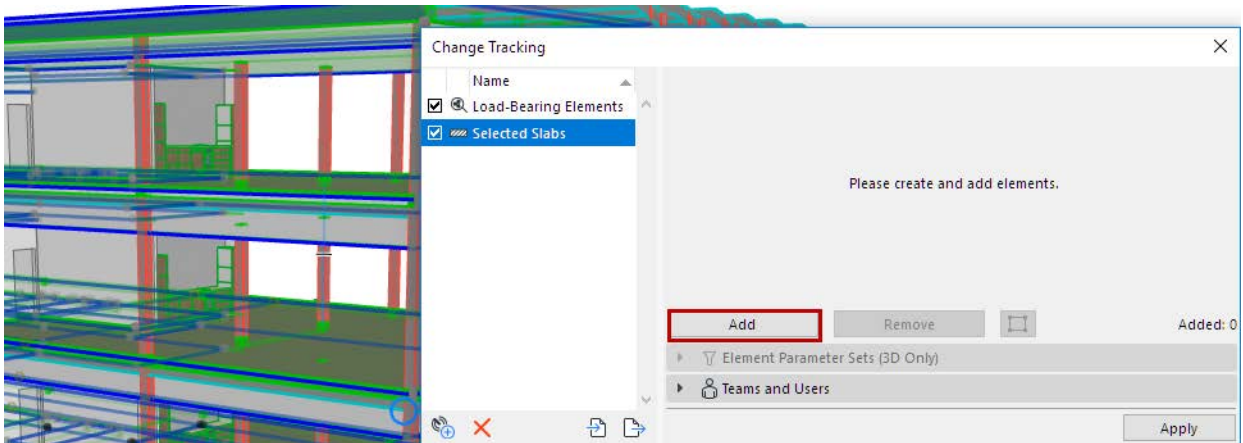
Subscribe to Changed Elements - by Selection

Use this Subscription Type to track changes in selected element instances.

1. Go to **Teamwork > Change Tracking**
2. In the appearing dialog, click the **New Subscription** icon (or right-click anywhere in the dialog box).
3. In the New Subscription dialog, choose **Selected Elements**.



4. Click OK.
5. If any model elements are currently selected: These elements are added to the Subscription list. If no model elements are selected yet, select them, then click **Add**.
Note: Selected elements that are part of a Hotlinked Module cannot be added to a Subscription.
6. In this example, the selected Slabs are added to the Subscription.



7. In the panels below, you can fine-tune this subscription by narrowing the set of tracked parameters, and/or by Teams and Users.

8. Click **Apply** to activate tracking for this subscription.

See [View and Manage Tracked Changes](#).

Related Topics

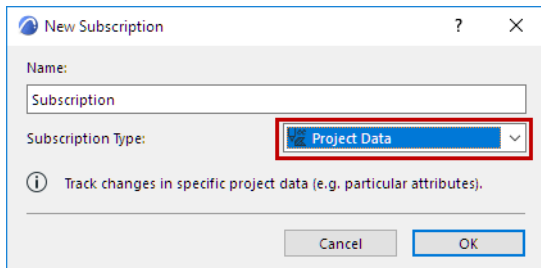
[Filter Element Changes by Parameter Type](#)

[Filter Change Notifications by Teams and Users](#)

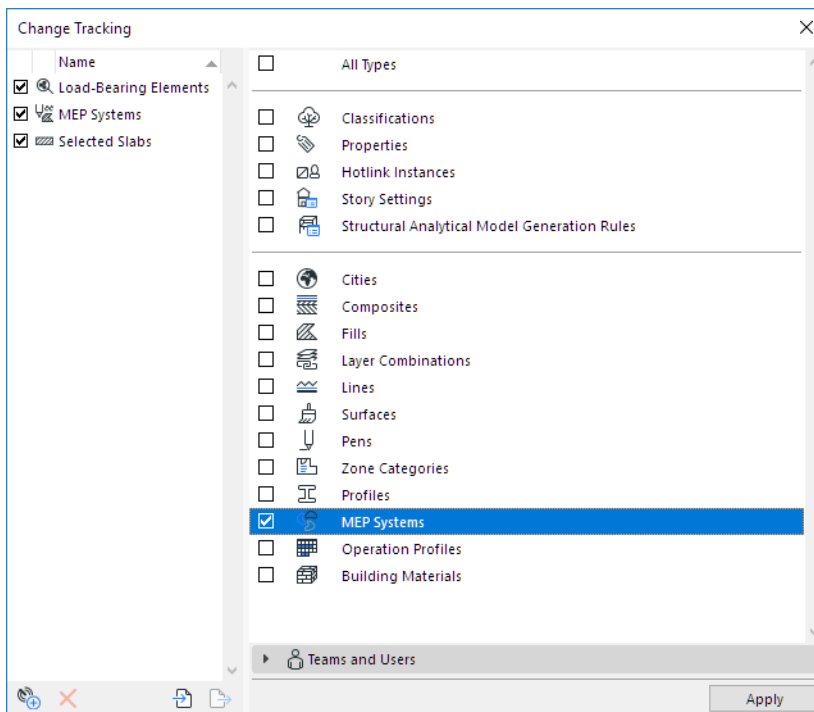
Subscribe to Changes in Project Data

Use this Subscription Type to track changes in project data (e.g. attributes or properties), rather than elements.

1. Go to **Teamwork > Change Tracking**
2. In the appearing dialog, click the **New Subscription** icon (or right-click anywhere in the dialog box).
3. In the New Subscription dialog, choose **Project Data**.



4. Click OK.
5. Review the list of project data types. Check only the data that you want to track.



6. You can fine-tune this subscription by user.
7. Click **Apply** to activate tracking for this subscription.

See [View and Manage Tracked Changes](#).

Related Topic

[Filter Change Notifications by Teams and Users](#)

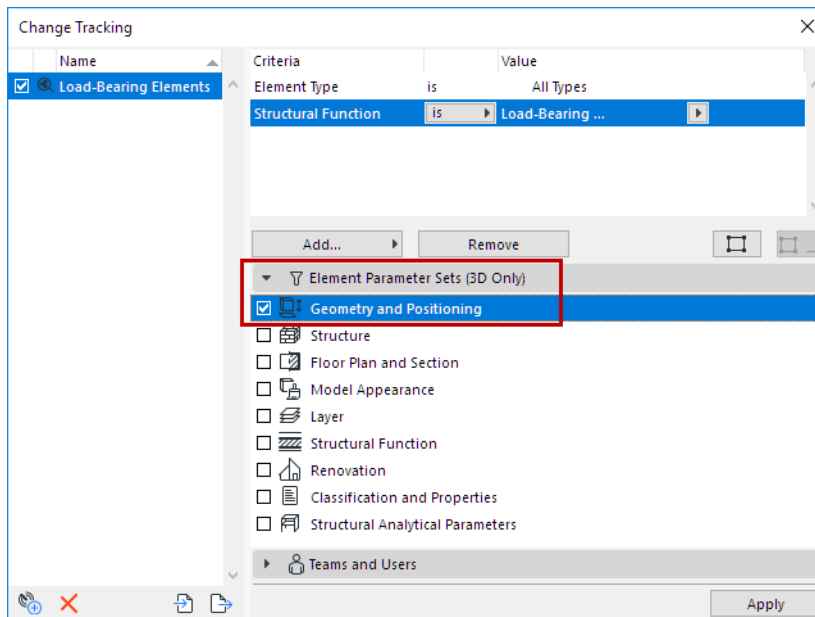
Filter Element Changes by Parameter Type

This feature is available when setting up an element-based subscription - e.g. the Subscription Type is either Criteria or Selection.

Go to the **Teamwork > Change Tracking** dialog.

On the **Element Parameter Sets (3D Only)** panel: mark the element parameters you want to track for the subscribed elements. These parameters can be tracked for 3D model elements only (not documentation elements).

Review the list of element parameters. Check only the parameter sets that you want to track with this subscription.



In this example, only “Geometry and Positioning” is checked.

Thus, in this subscription:

- Only load-bearing model elements are tracked, and
- You are notified only if the **Geometry and Positioning** parameters of those elements have changed.

For example, if a load-bearing Column’s width value is increased, then you will be notified.

If the same Column’s layer has changed, you will *not* be notified.

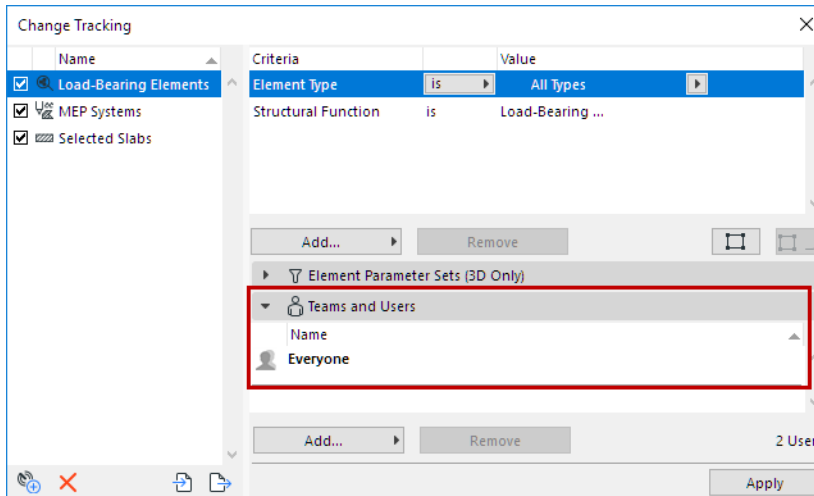
[See also Modified Element Geometry: What Counts as a Change?](#)

Filter Change Notifications by Teams and Users

Teams and Users is a panel in the **Teamwork > Change Tracking** dialog.

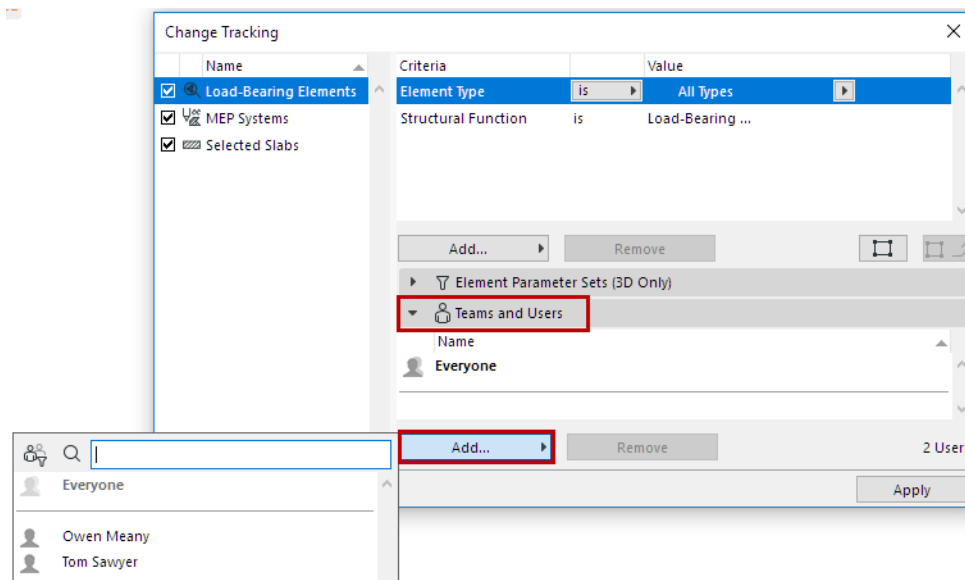
By default, each Subscription tracks activities by “Everyone” - that is, all users joined in to the project. Whenever any user sends in a change that fits your Subscription criteria, you get a notification.

Note: You do not receive notifications for your own changes.



However, you can narrow the subscription so that you are only notified if certain selected users (or Team members) make the change.

1. Open the **Teams and Users** panel.
2. Select the **Everyone** item, then click **Remove**.
3. Click the **Add** button.
4. The list of Users and Teams appears.

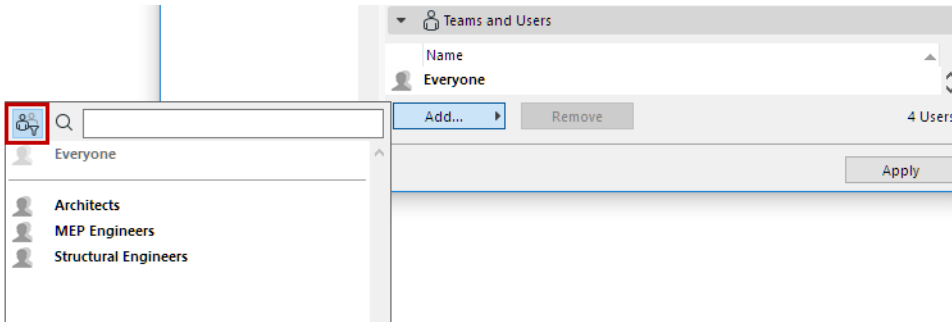


This list displays:

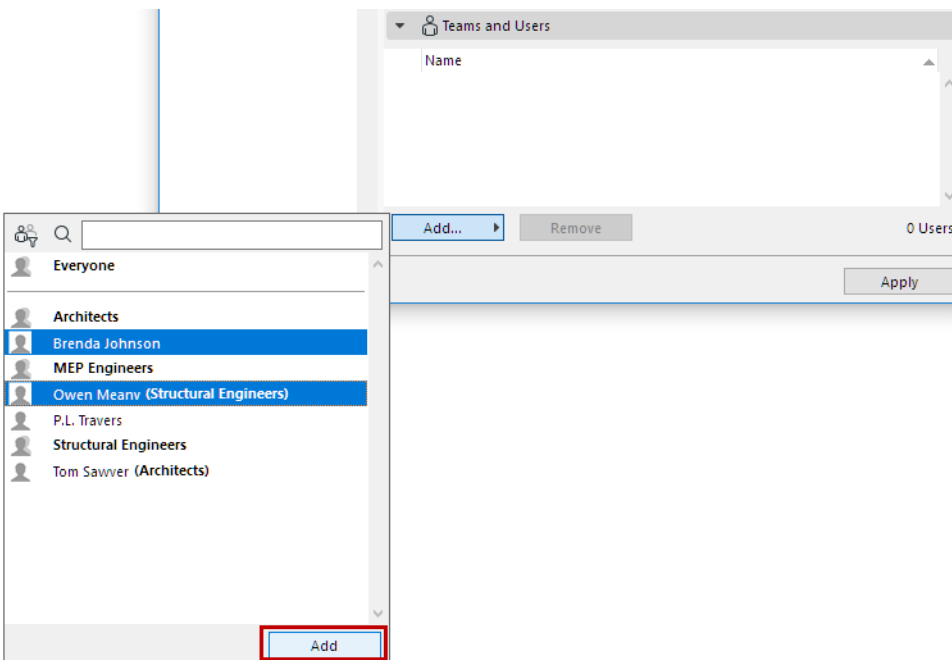
- All users who are joined into the current Teamwork Project (displaying their Team name, if any)
- Teams (if any) which have access to the current Teamwork Project.

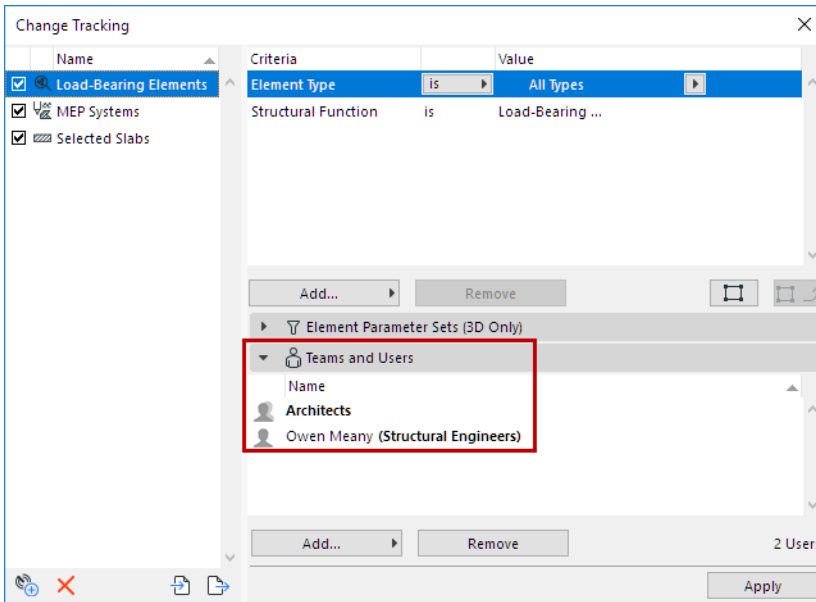
Note: “Teams” are sets of users, such as a Team of Structural Engineers, created for the “BIMcloud Project” containers on BIMcloud.

- To filter the list to display the Teams only: click the filter icon.



5. Select one or more Users or Teams, then click **Add**.

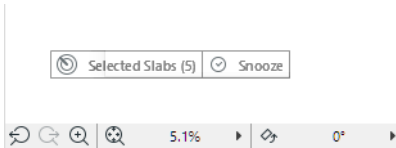




The subscription is now filtered to track only changes by the Teams/Users you have added.

View and Manage Tracked Changes

When a user sends a change to the server which triggers one of your subscriptions, you receive a notification via an on-screen pop-up.



- These notifications appear if you have checked “Show Tracked Changes Alert in Teamwork” (Options > Work Environment > More Options.) It is checked by default.
- Increase the size of the alert message by checking “Use large alert cues” in the same Work Environment dialog.

Do any of the following:

- Click the pop-up (this opens the **Tracked Changes** Palette - see below.)
- Click **Snooze** to make the pop-up disappear (it reappears after 30 minutes, or 100 steps, whichever comes first).

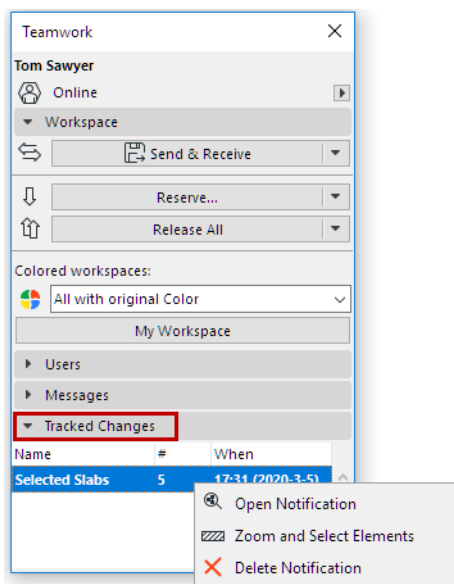
Notes:

- If an element is part of two or more subscriptions, you will receive a notification for each.
- If a particular element parameter is modified more than once, your notification contains only the latest change - not the earlier, obsolete ones.

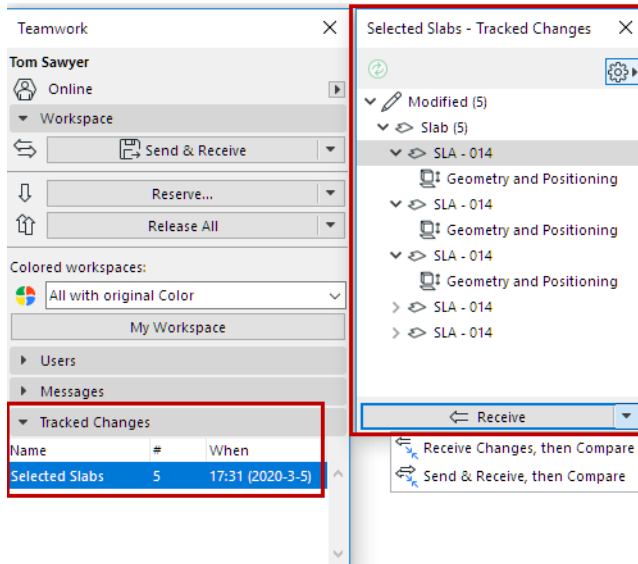
Tracked Changes Panel

The **Tracked Changes** panel of Teamwork Palette shows a list of the subscriptions which have tracked a current change.

These are changes that have been sent to the server, but not yet received by you. Once you receive the changes, the Tracked Changes list will be deleted.



For a selected subscription in this list: Use context menu commands to zoom and select, or double-click to open the **Tracked Changes** palette for a detailed view of changes.



Double-Click a Change to Open Tracked Changes Palette

The palette shows more details on the model elements or project data (per your subscription) which have been changed in the Teamwork project and sent to the BIMcloud by other users, but not yet received by you.

Use the commands at the bottom of the Tracked Changes palette to do any of the following:

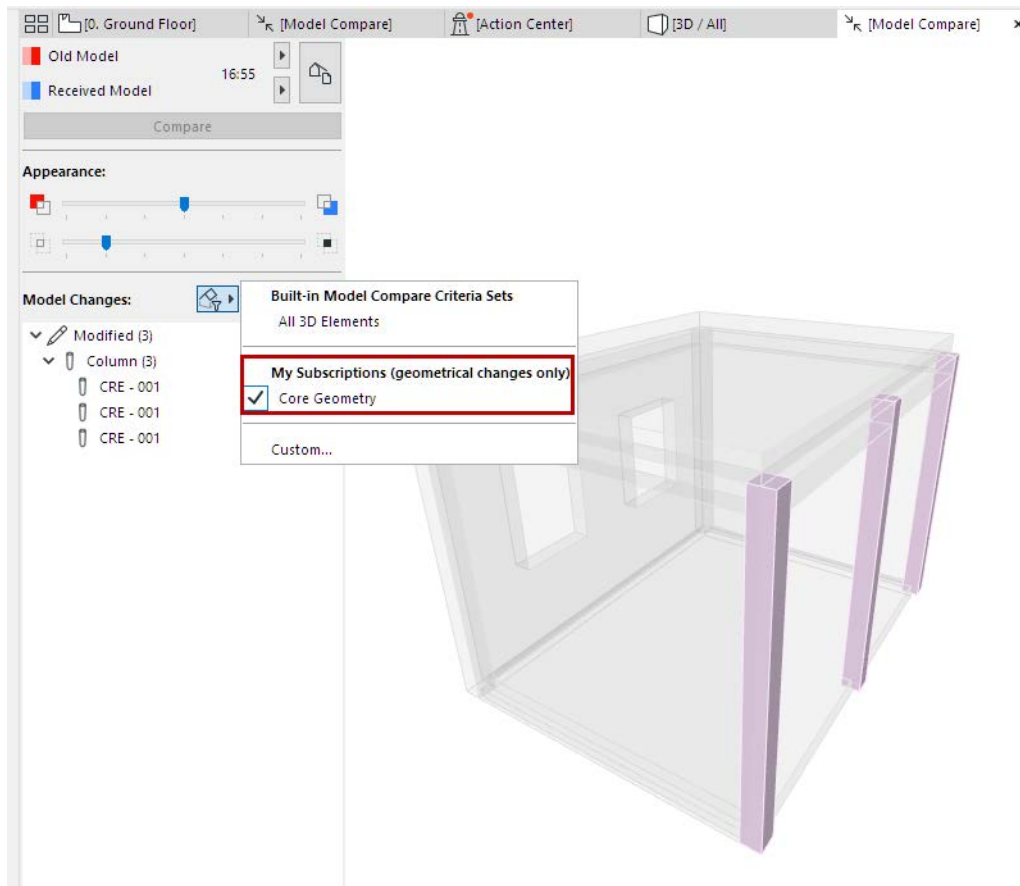
- Receive
- Receive Changes, then Compare
- Send & Receive, then Compare

Once you receive the changes, the Tracked Changes list will be deleted.

Subscriptions in the Model Compare Window

Choosing either of the Compare commands opens the Model Compare window and shows the results of the two project statuses: before and after receiving changes.

By default, the Model Compare window is filtered to show the element changes as defined in the current subscription (e.g. the changes stemming from your “Core Geometry” subscription.)



See [Model Compare: Review the Changes](#).

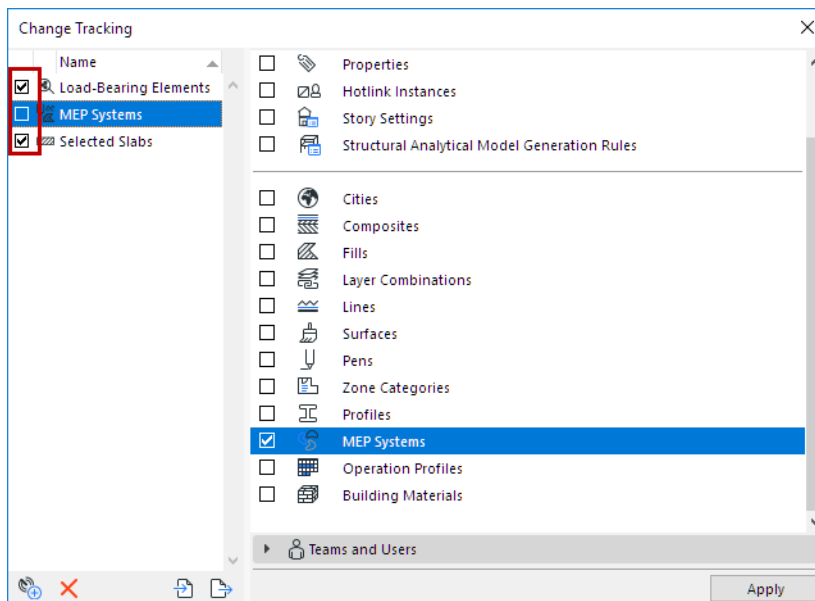
Manage Subscriptions

Manage your Subscriptions in the **Teamwork > Change Tracking** dialog.

Turn Subscription On/Off

In the Subscription list at left: Turn checkbox on or off to activate or deactivate notifications for any subscription.

- If you deactivate a subscription: You will not receive any notifications or feedback on changes. (Existing notifications listed in your Teamwork Palette will not be deleted.)
- If you reactivate a subscription: You will start receiving notifications, but you will not get any notifications on changes that occurred while the subscription was turned off.



Import/Export Subscription

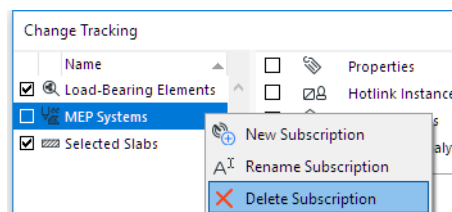
Use the import/export buttons at the bottom of the dialog to transfer the definitions of one or more selected subscriptions in XML format.

Rename Subscription

Double-click the subscription name to edit the field, or use the context menu command.

Delete Subscription

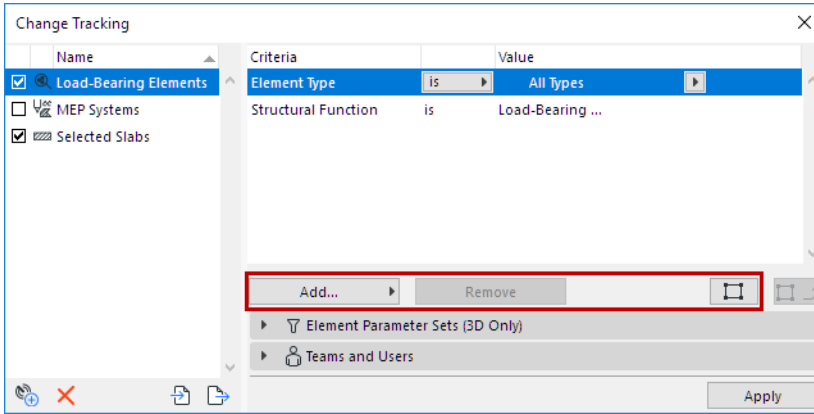
Click the red X, or use the context menu command.



For Element-based Subscriptions (by criteria or selection):

- For any selected element in the Subscription list, use the buttons to **Add** or **Remove** it at any time.

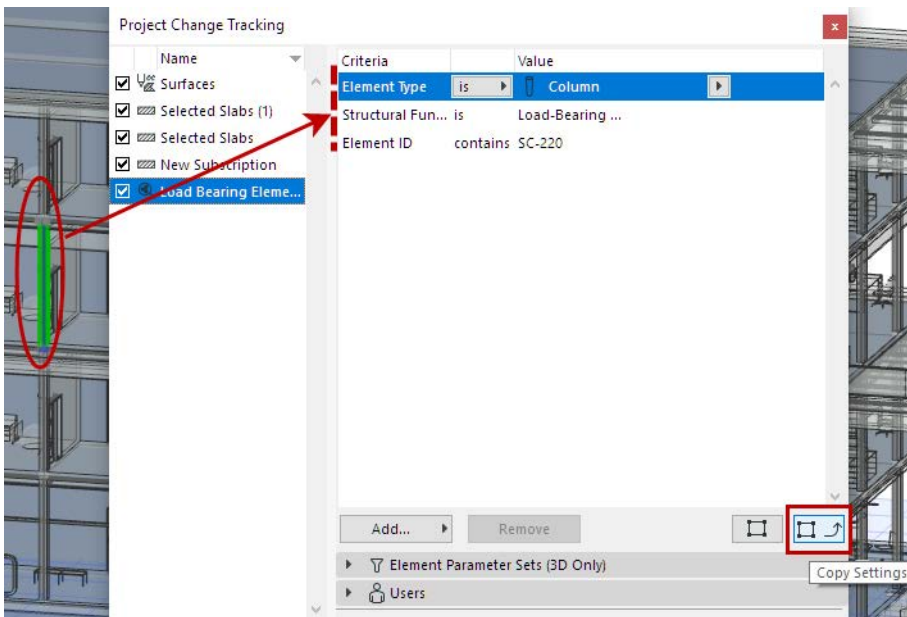
- For any selected element in the Subscription list, click the **Zoom** icon to zoom in.



Copy Settings

For criteria-based subscriptions only:

If any model elements are selected in the project, the Copy Settings becomes active. Click to fill in the values of the last selected model element into the criteria.



MEP Modeler

Use the built-in MEP Modeler to model and edit MEP (mechanical, electrical and plumbing) routes - ducts, pipes, and cable carriers. You can also place individual MEP elements using dedicated tools or the Object tool.

MEP Modeler and the MEP Library are loaded automatically with Archicad and require no extra license.

For Teamwork users

A predefined “MEP Engineer” role is available.

Specific permissions for MEP engineers:

- MEP tool elements
- MEP Preferences

Topics in this section:

[Apply MEP Work Environment](#)

[Place Route element](#)

[Edit Route graphically](#)

[Edit individual segments and nodes](#)

[Route Settings](#)

[Route Preferences](#)

[Place Terminal, Accessory, or Equipment](#)

[Select Connected MEP Elements \(Command\)](#)

[MEP Documentation](#)

[MEP-compatible Library Parts](#)

[MEP Systems](#)

[Create custom MEP object](#)

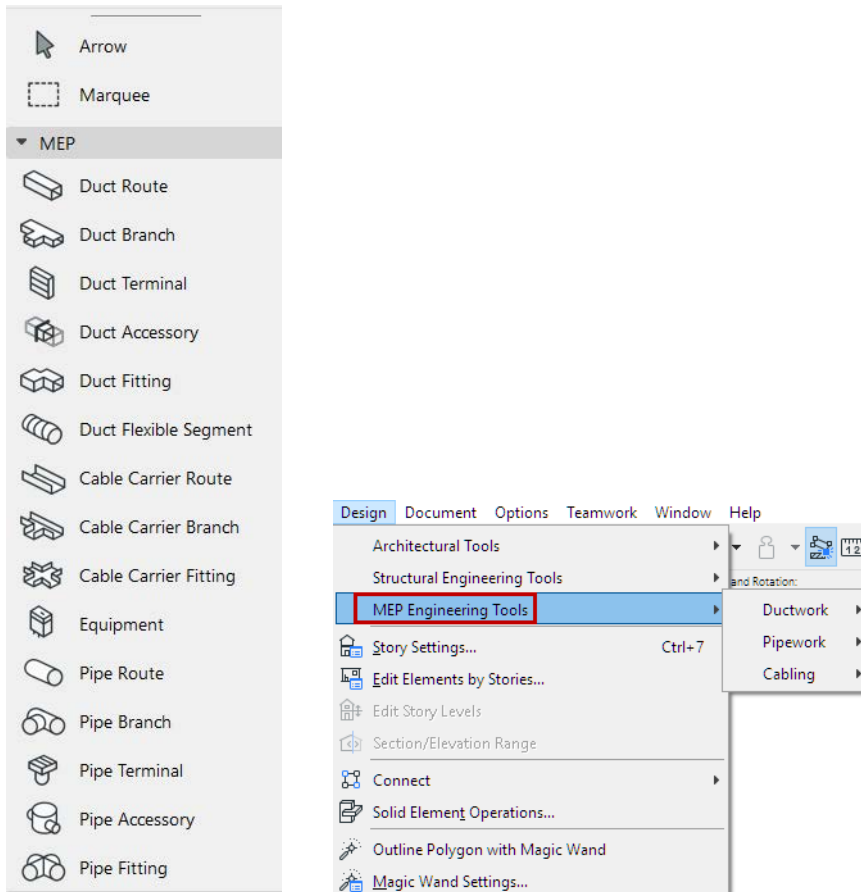
[MEP elements in IFC exchange](#)

Apply MEP Work Environment

Use **Options > Work Environment > Apply Profile > MEP Engineering Profile**.

In the MEP profile:

- The toolbox displays the MEP tools (these tools are also available at Design > MEP Engineering tools)
- The MEP Toolbar provides shortcuts to frequently used MEP commands



MEP Tools and Commands

Place Route element

1. Activate a Route tool from toolbox: Duct Route, Cable Carrier Route, or Pipe Route.
2. In Floor Plan or 3D window: Click in series to define the segments and nodes of the Route polygon.
 - Each click represents a node, which connects the adjacent segments.
 - The reference line is at the center of the Route element
 - Press Backspace to delete the last placed segment
3. To complete the Route, click a second time at the end of the final segment.



The result is a single Route element.

Elbows and transitions are automatically generated at each node.

When ending a route element on a terminal, accessory or equipment, use the up and down arrow keys to help find suitable options to connect the route. (See [Place Terminal, Accessory, or Equipment](#))

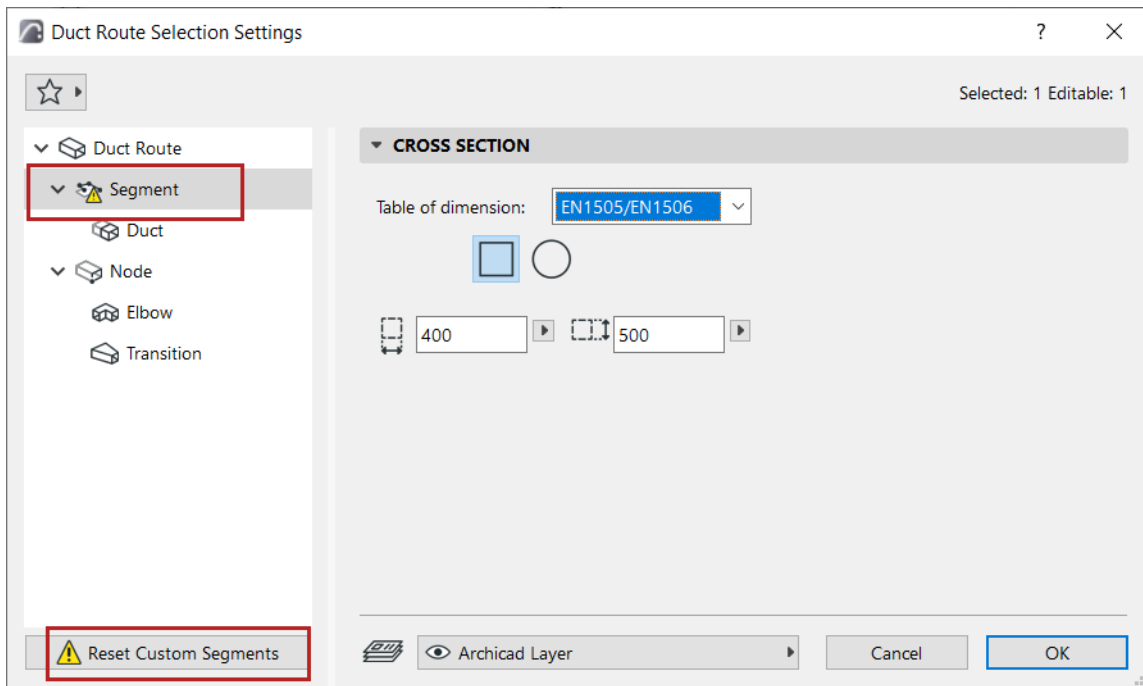
The flow of material (for example water or air) conducted by a given Route element is always constant.

Change Route parameter on the fly

Once you start placing a route element, you can change its cross-section in the Tracker.

+

Because you have changed the segment's default parameter value, it is shown in the settings dialog as "custom".



Add Branch to Route Element

You can add a branch to any placed route element. The branching node is added automatically. After a branch is created, the original route is split into multiple route elements.



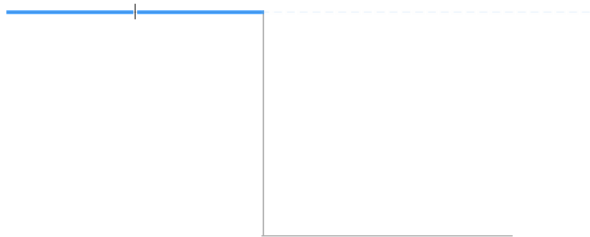
Merge Routes

Routes will merge if:

- An end node of one Route element is connected to the end node of another
- A branch is deleted (remaining Routes will be merged together)

A branch end node is moved, disconnecting the former branch.

Place Route with Magic Wand



+

Use the Magic Wand to place a route with a single click, based on an existing polyline or polygon.

1. Draw a line or polyline.
2. Choose an MEP tool (Duct, Pipe or Cable Carrier).
3. Press the Spacebar to activate the Magic Wand, then click on the polyline or polygon.

Flexible Duct

Use this tool to easily create a flexible geometry using spline-type input. Click the port of an existing route element to connect it, or double-click to end input.

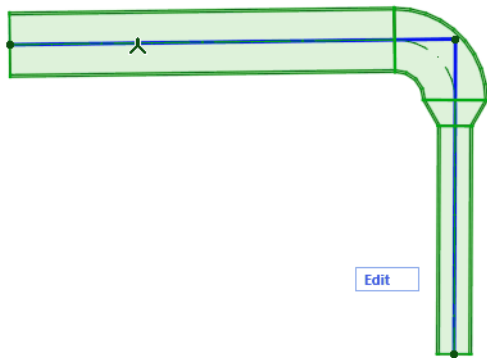
The flexible duct will not add transitions at its ends, and connections having different dimensions must be solved in the route.

For editing commands on flexible ducts, see [Edit Route graphically](#).

Edit Route graphically

For any selected Route element: click an edge, node, or reference line to access pet palette commands.

Insert new node



From the reference line:

1. Choose Insert new node.
2. Drag to create a new node and change the Route shape.

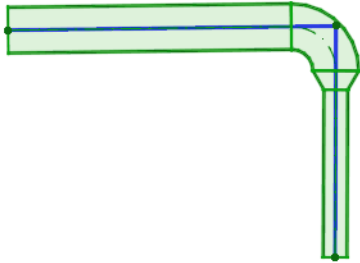
Move Node



Available for a node between two adjacent straight segments.

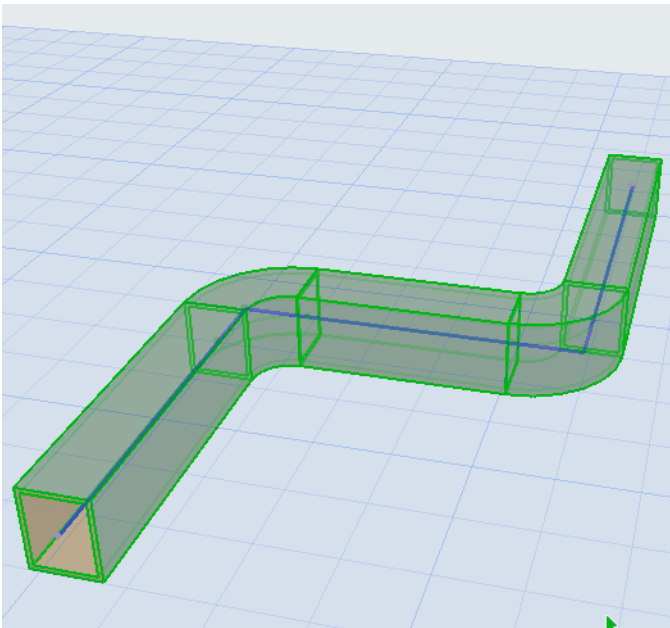
Drag the connection node, or enter values in the Tracker. This modifies the segment length.

Elevate Route (Floor Plan)



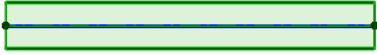
Enter the value by which you wish to elevate the Route.

Elevate Route (3D)



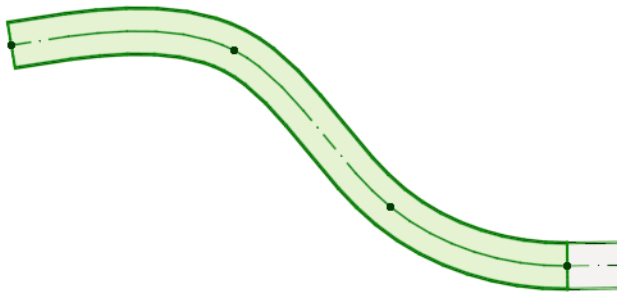
Use the Drag command to change Route elevation on the Z axis.

Continue Route



Available at the end of an unconnected straight segment.

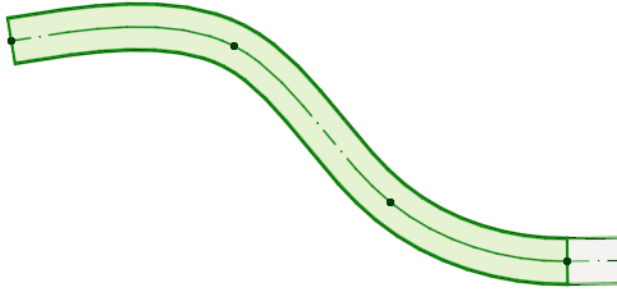
Flexible Duct: Move node



Click a node of the Flexible Duct.

Choose Move Sub-element, then drag the node.

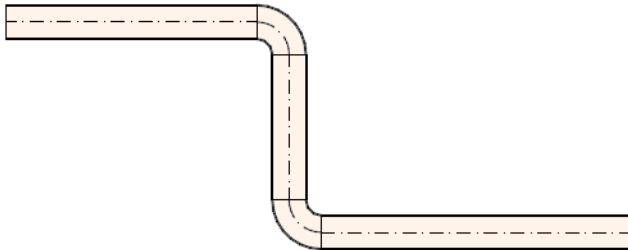
Flexible Duct: Insert new node



Click on the centerline of the Flexible Duct.

Choose Move Sub-element, then click the centerline to create a new node.

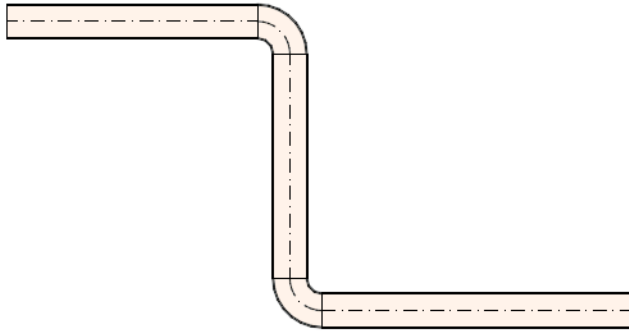
Offset Segment



1. Select the element.
2. Click the centerline/axis.
3. From the pet palette, select "Offset segment" to move it.

Note: Not available for routes connected with branches.

Split MEP Route



1. Select the route you wish to split.
2. Choose Edit > Reshape > Split command, or the hatchet icon from the toolbar.
3. Draw a temporary splitting line.
Note: In the 3D window, a red splitting plane is shown as feedback.
4. Click on either side of the splitting line/arc with the Eyeball cursor.
5. Elements on the clicked side will remain selected, while elements on the other side will be deselected.

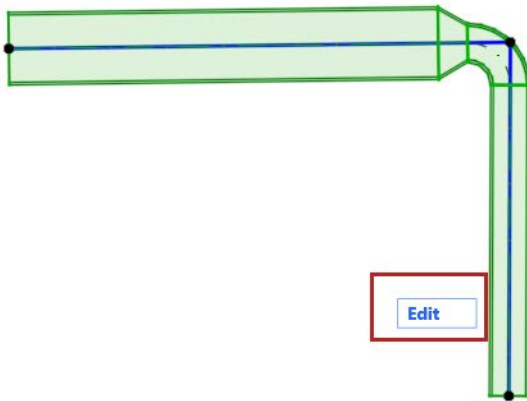
Edit individual segments and nodes

Use Edit mode to select any single segment or node of a Route element, and edit its parameters. (Outside of Edit mode, you can only select the Route element as a whole.)

Enter Edit Mode

Select a Route element, then do one of the following:

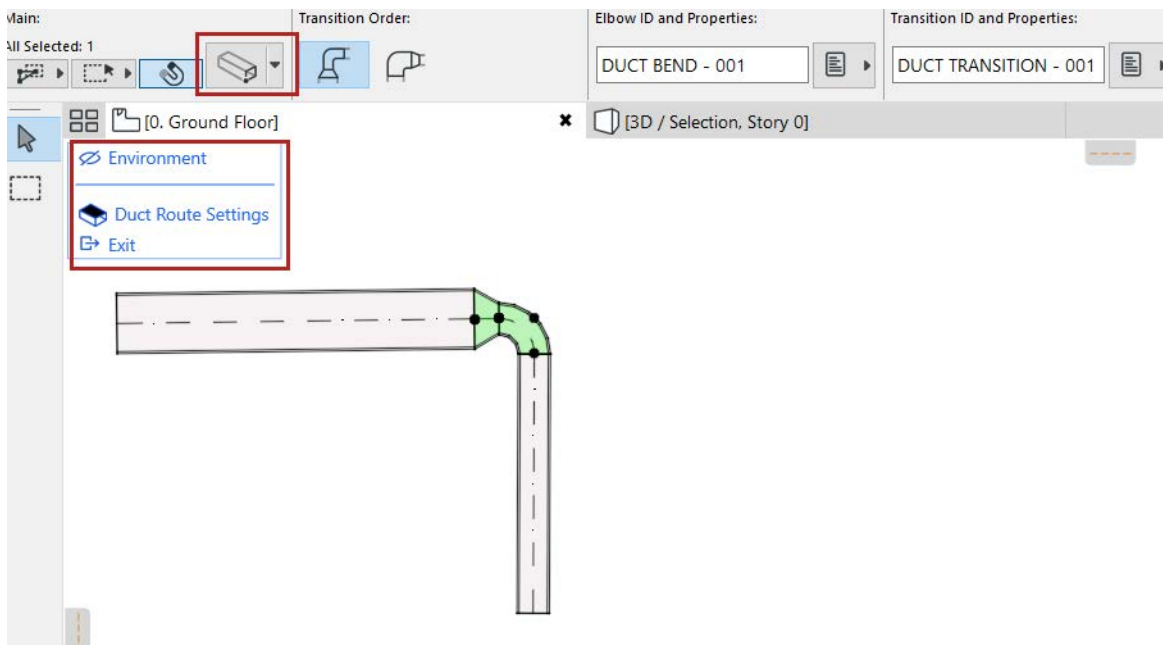
- Click the **Edit** button which appears



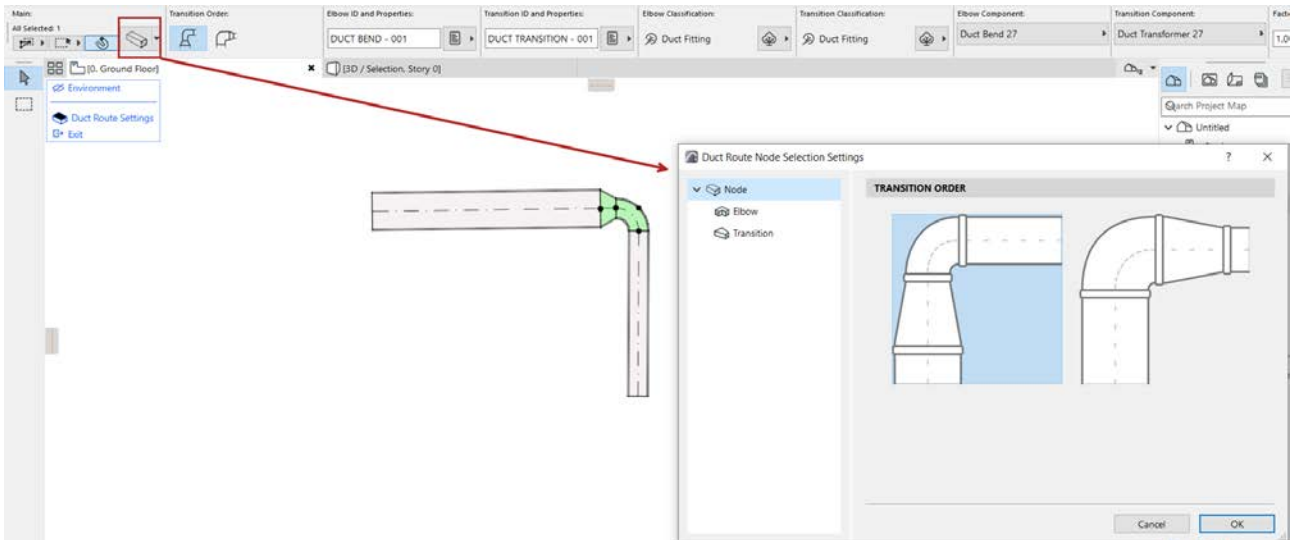
- Use **Design > Enter System Edit Mode**

Using Edit Mode

Select any sub-element (node or segment).

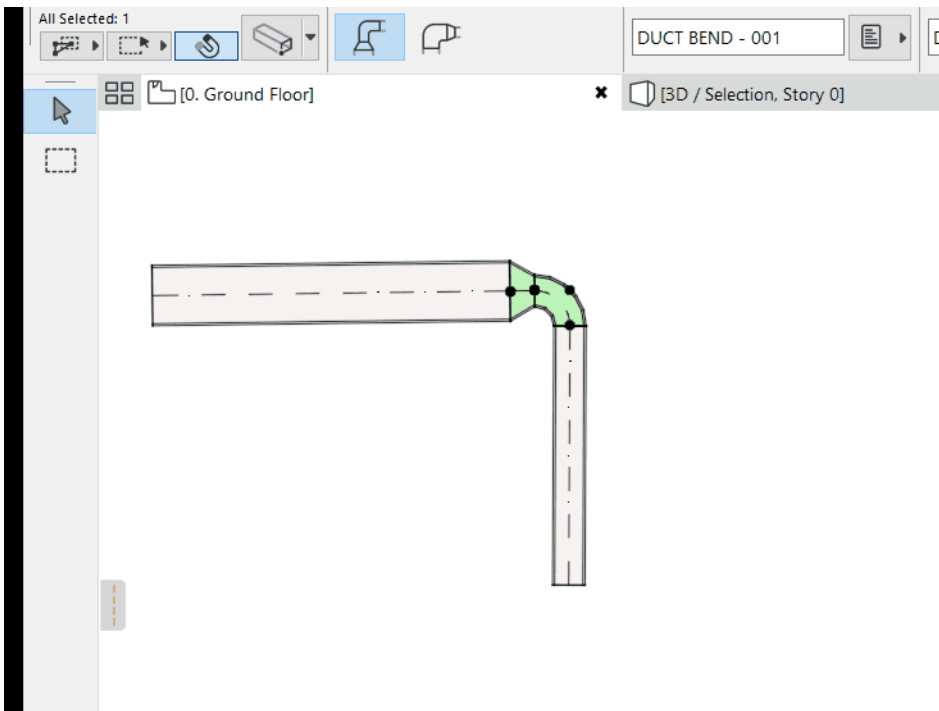


- The regular Archicad toolbox is replaced by just selection tools and a display palette.
- From the Info Box, you can open a limited Settings dialog that applies just to the selected segment or node.



- Use the Environment toggle in the display palette to show/hide the rest of the model (i.e. everything apart from the selected Route). In Edit mode, you cannot edit the rest of the model - only the sub-elements of the selected Route.
- The full Route Settings dialog is available from the display palette.

For example, edit the Transition order of a selected node:



Exit Edit Mode

Do one of the following:

- Click **Esc**
- Click **Exit Edit Mode** from the display palette.
- Use **Design > Exit Edit Mode**

Route Settings

Each MEP Route - Duct Route, Cable Carrier Route, and Pipe Route - consists of a hierarchy of the main Route element, plus its sub-elements: segments and nodes.

Each of the Route Settings dialogs reflects this hierarchy. Use the settings pages to set default options for the Route element and its sub-elements.

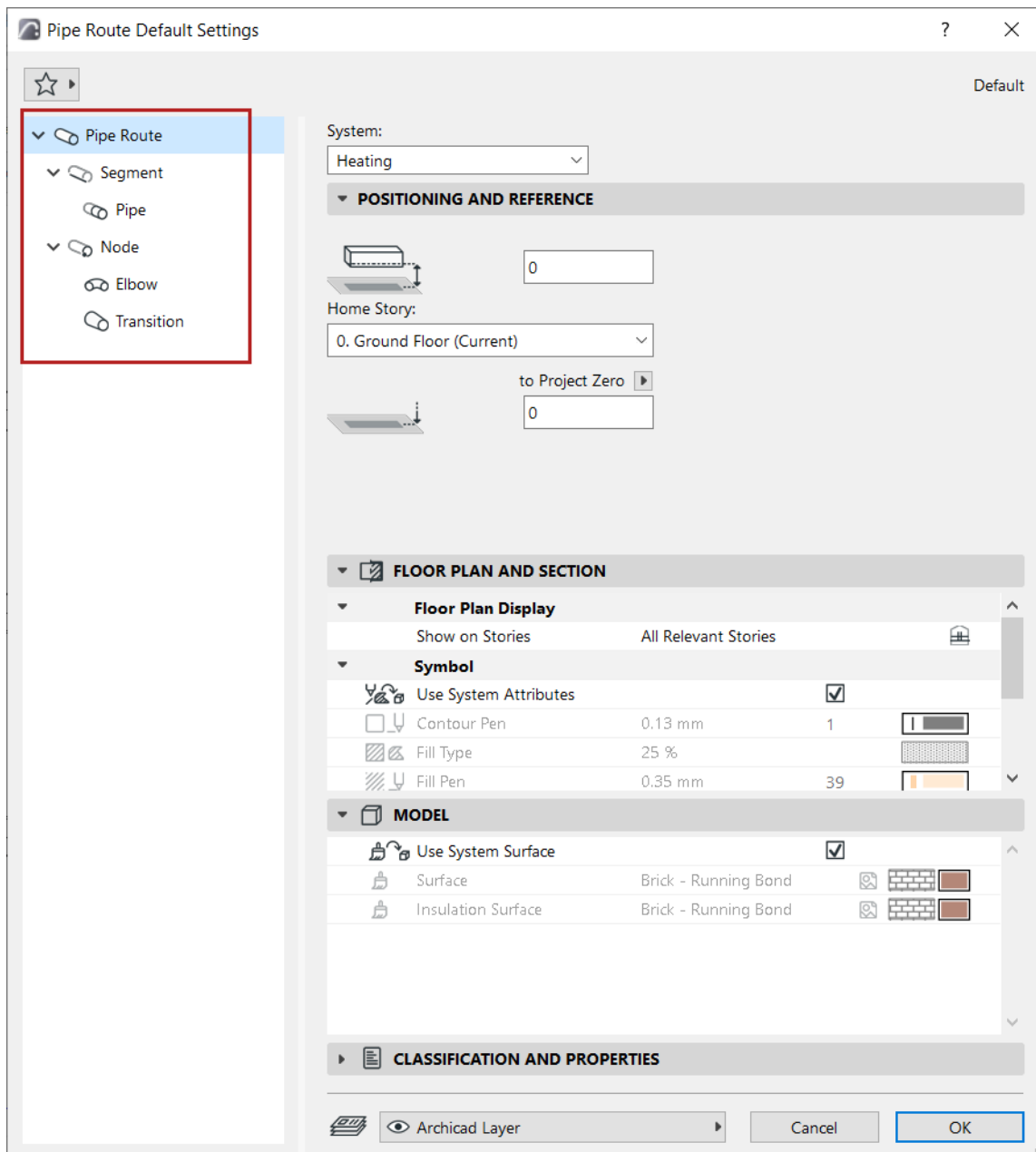
[Route Settings - main page](#)

[Display Settings: Floor Plan and Section, Model Display](#)

[Segment Settings](#)

[Node Settings](#)

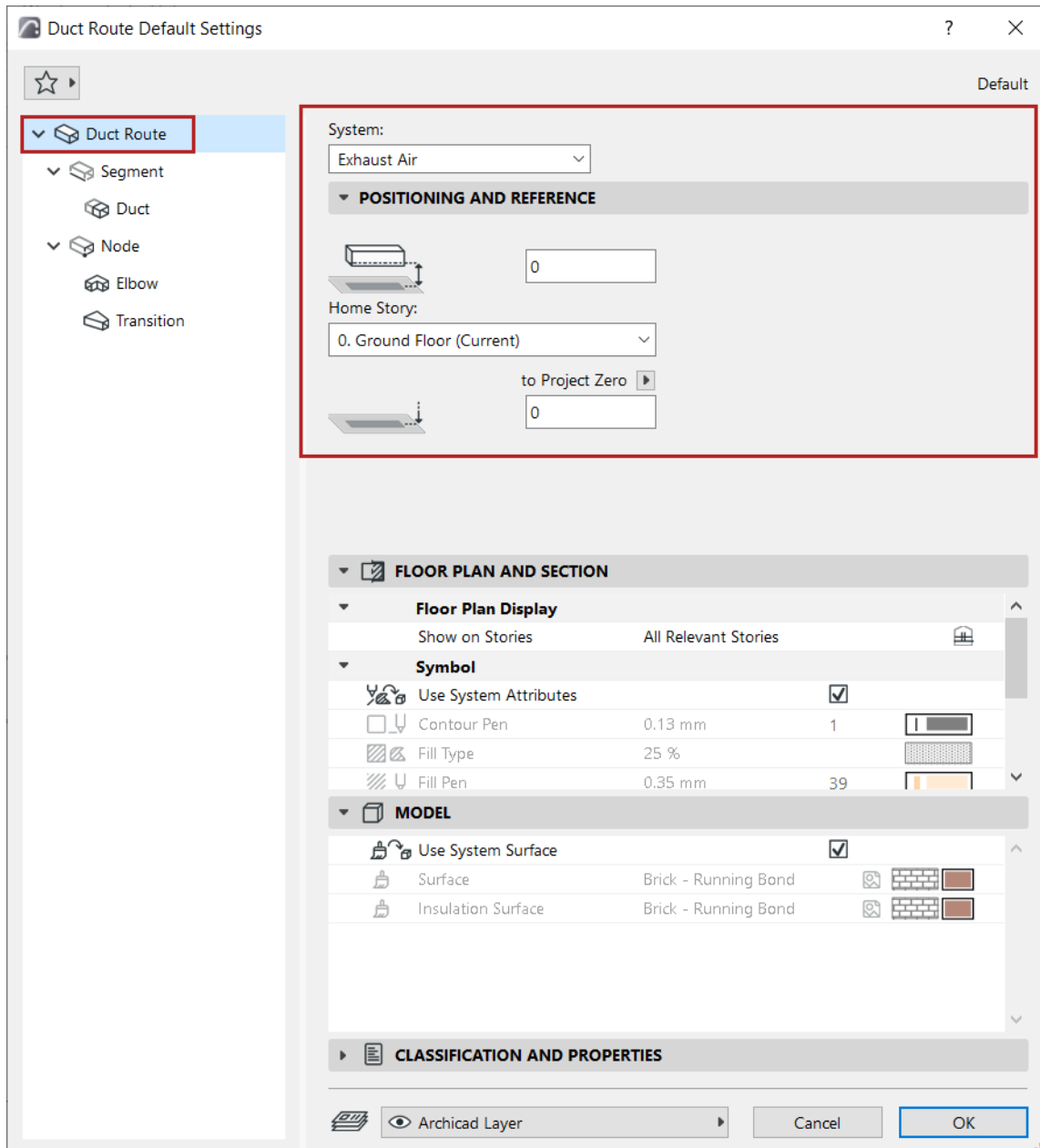
[MEP Element Settings \(Duct, Cable Carrier, Pipe, Elbow, Transition\)](#)



As with similar Archicad tools (e.g. Railing), you can customize any particular sub-element. Once a setting is customized and no longer the default, the dialog alerts you that a custom setting is in effect.

Route Settings - main page

1. Open Route Tool Default Settings
2. On the main Route settings page, choose a System. The MEP System defines the Route's appearance in 2D and 3D. [See MEP Systems.](#)

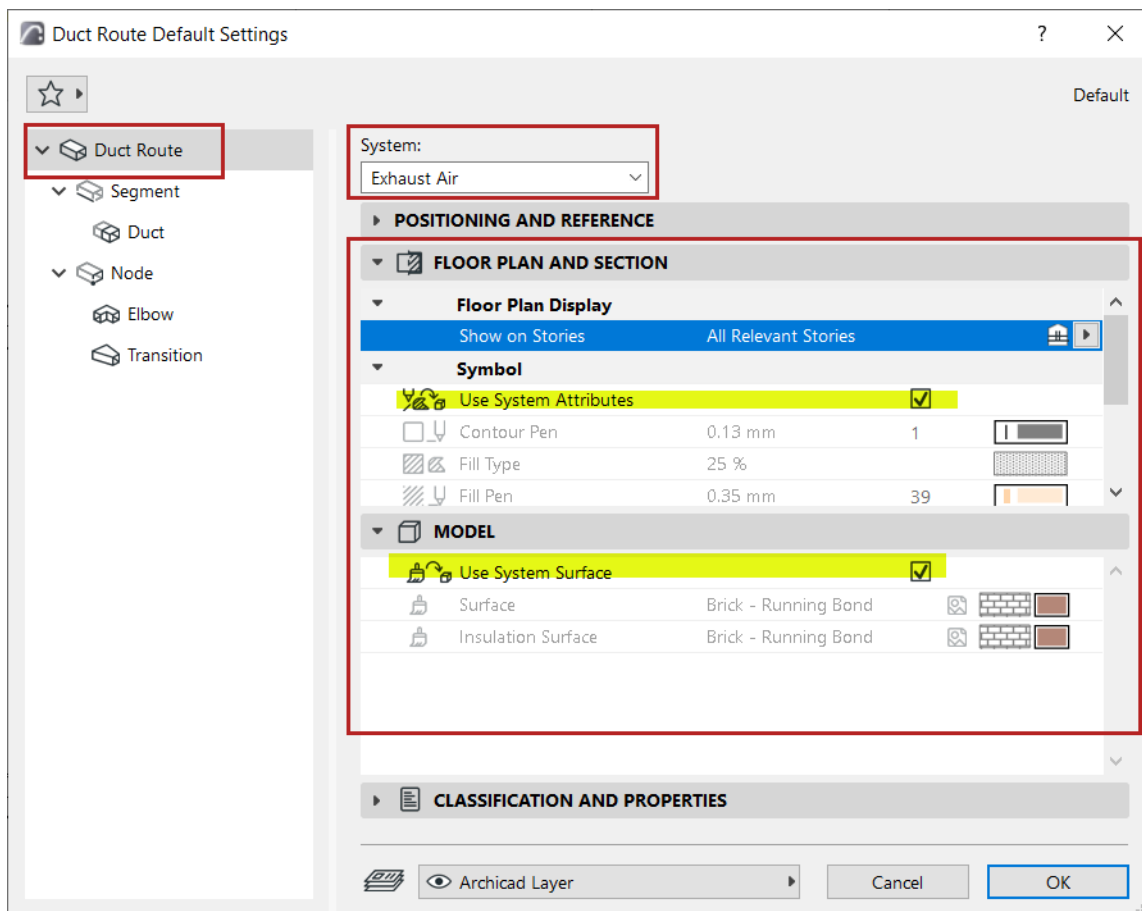


3. Set the Route element's Positioning and Reference:
 - Home story and its offset from that story level
 - Bottom offset to the chosen Reference level

Display Settings: Floor Plan and Section, Model Display

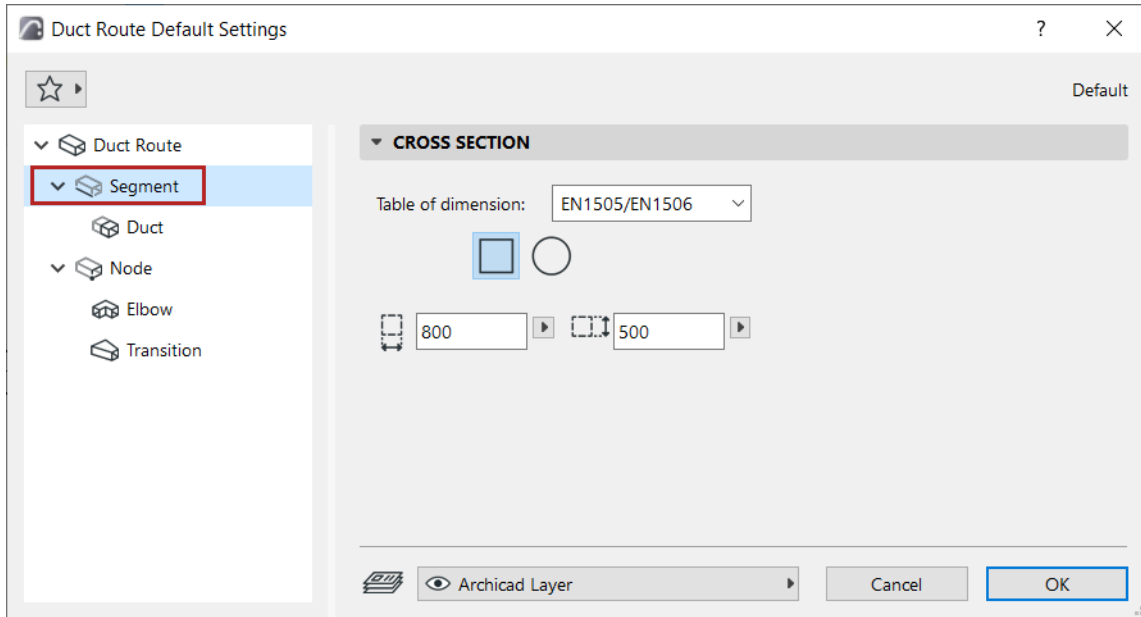
The entire Route element has a uniform appearance - you cannot customize the appearance of particular sub-elements.

- Show on Stories: Choose on which stories the MEP Route should be displayed. The default option for all MEP elements is “All Relevant Stories”.
 - Straight segments are displayed as cut on the intersected stories
 - Other (non-straight) MEP elements that intersect stories do not have cut displays - they are shown either on one story or the other.
- Fills, Lines, and Surface: By default, the Route uses the attributes of the System selected at the top of the dialog. The “Use System Attributes” box (for both Floor Plan/Section and Model) is checked by default:



- To override the MEP System attributes for this Route: Uncheck “Use System Attributes/Surfaces” to activate the local controls and override System settings.

Segment Settings



- Choose a Table of dimensions.

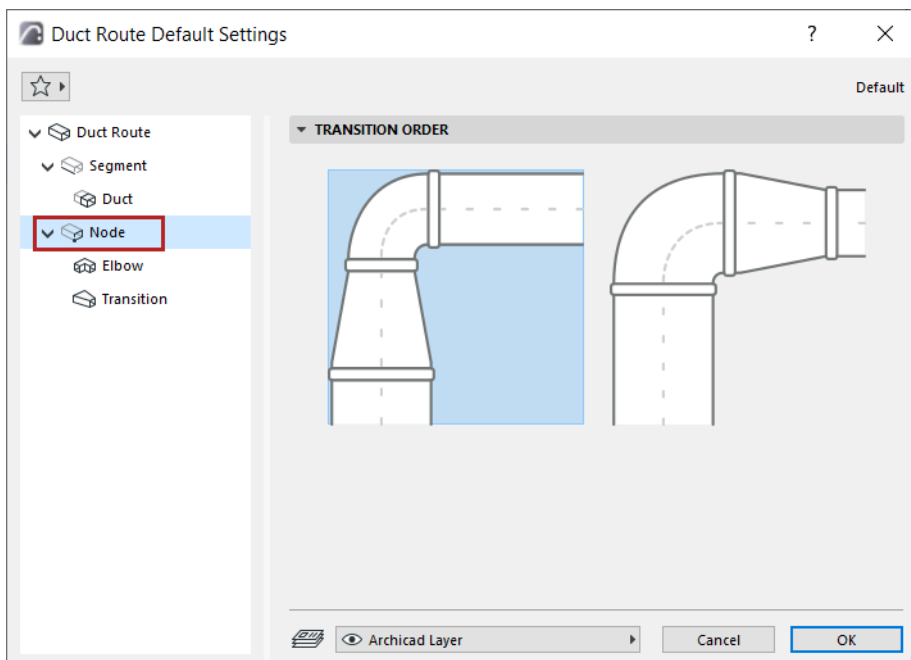
Note: Define tables at: [Route Preferences](#).

- (Duct only:) Choose a cross-section: Rectangular or Circular.
- Enter the cross-section dimensions.

Note: You can customize the cross-section of any segment during input - [see Change Route parameter on the fly](#).

Node Settings

Set the order of the Elbow-Transition elements in the node.



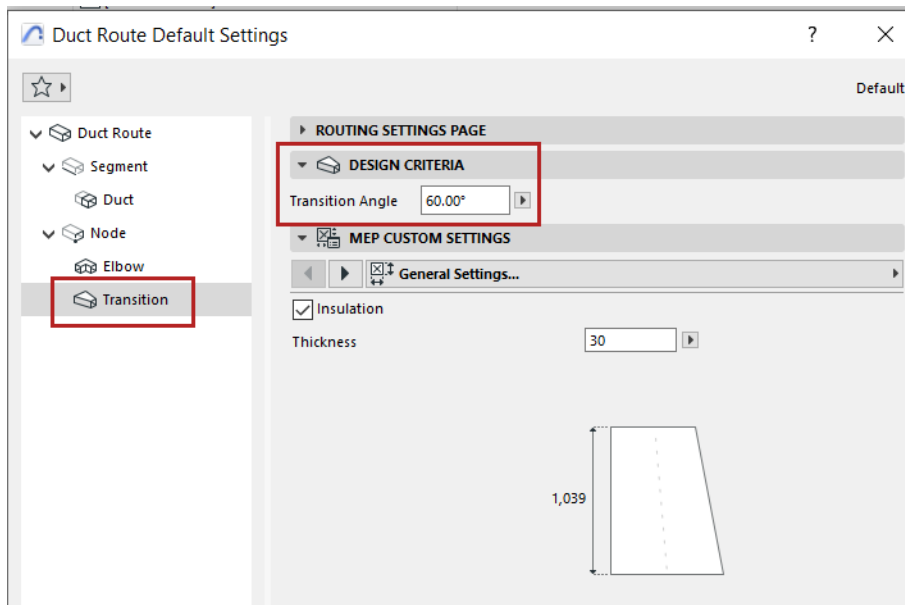
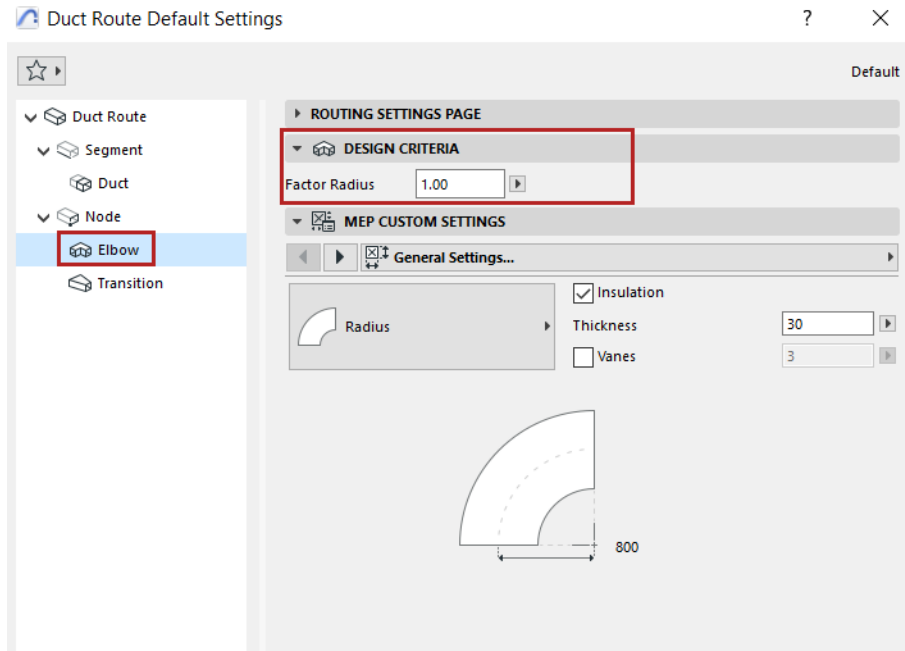
Design Criteria

Factor Radius (Elbow)

Defines elbow radius based on this value (minimum: 0.6)

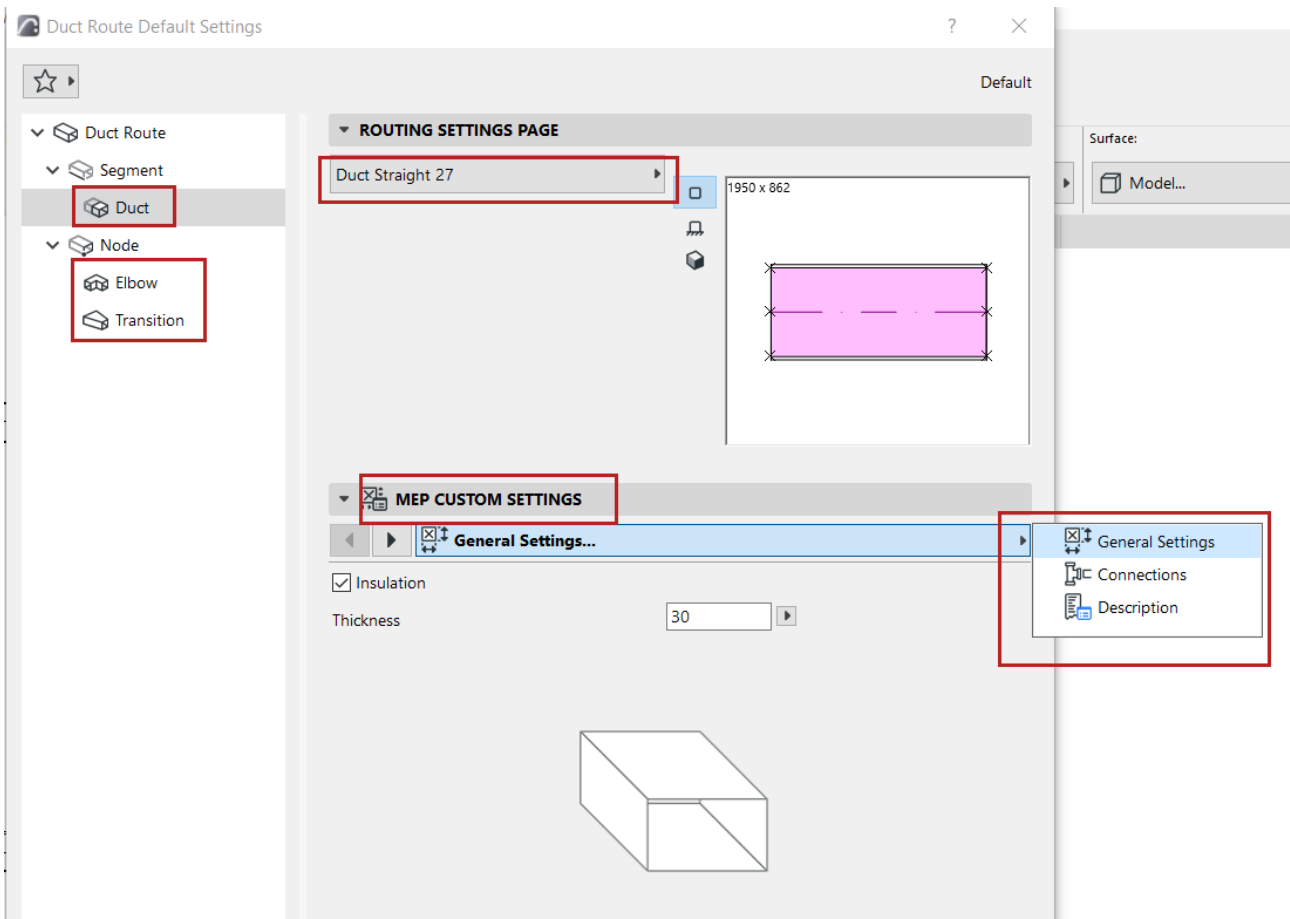
Transition Angle (Transition)

Calculates length of a transition based on this angle: the larger the delta between the two ports, the longer the transition.



MEP Element Settings (Duct, Cable Carrier, Pipe, Elbow, Transition)

- Choose the MEP element for the segments or the Node parts in this Route
- Set its Custom Settings in the panel below. (Available settings vary depending on the MEP element.)



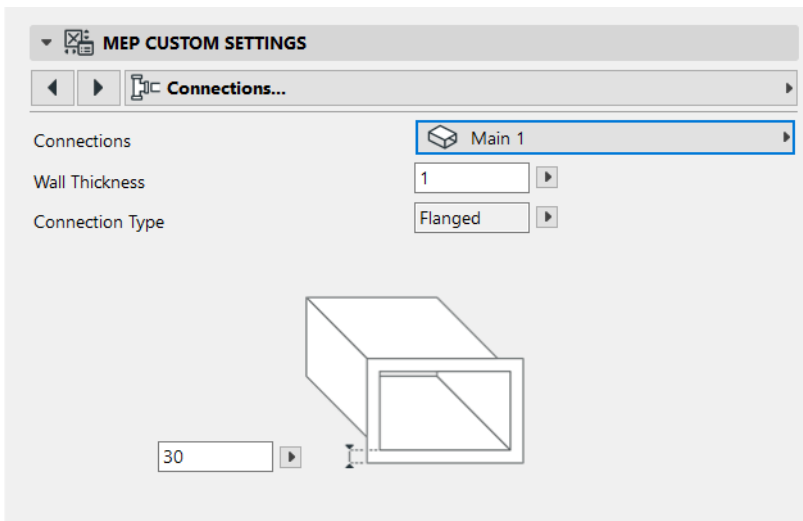
General Settings

Insulation

Check to add an insulating layer to the element's outside surface. Then enter the insulation's Thickness value.

Connections

Set parameters for the connection ports.



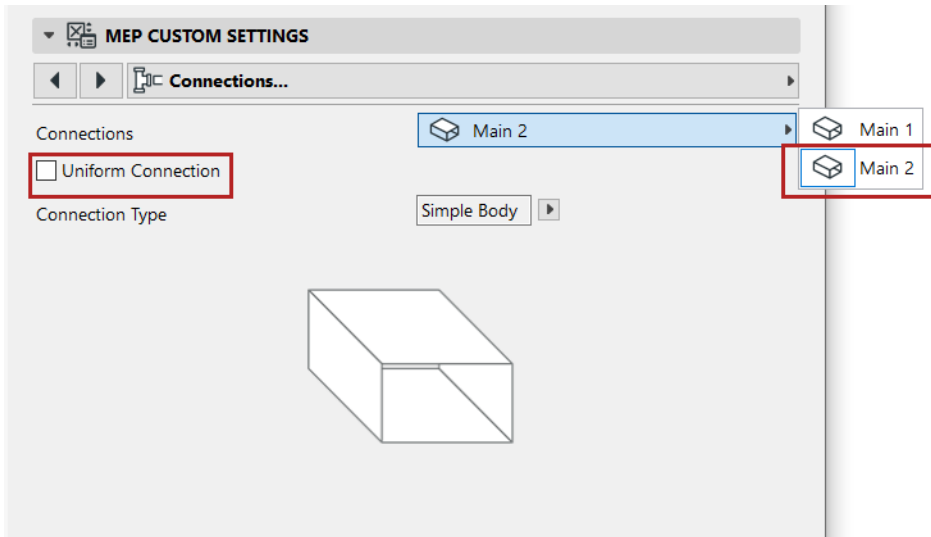
Wall Thickness

Thickness of the MEP element wall.

Connection Type

Choose the appropriate type (e.g. Flanged, Welded, Simple) - this varies depending on the Route type. Enter a width/diameter as applicable.

To use a different Connection Type for the second port:



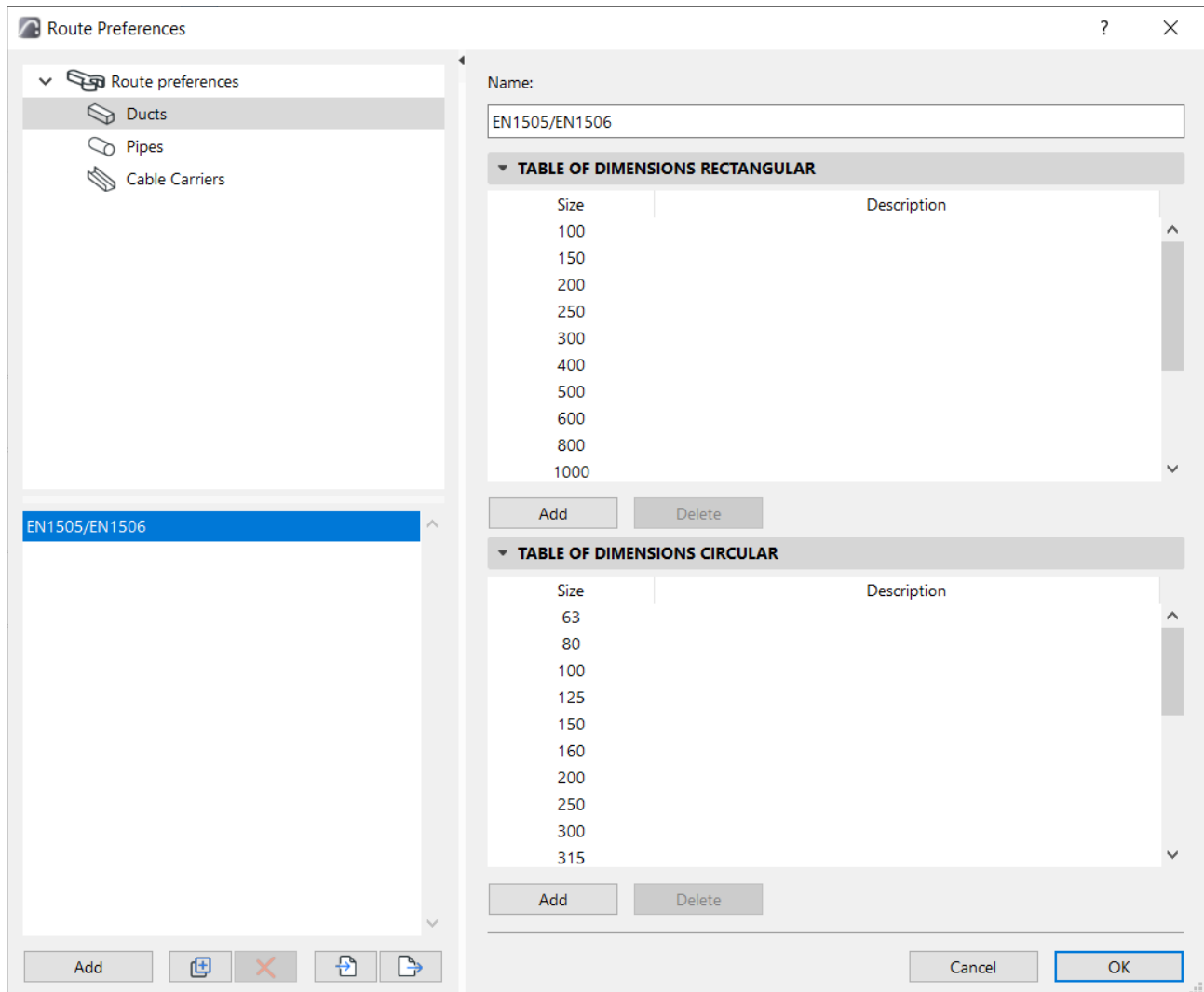
1. Use the pop-up to switch to the second port
2. Uncheck the "Uniform Connection" box
3. Edit the parameters of the second connection
 - Choose the MEP element for the elbows and transitions in this Route
 - Set their Custom Settings in the panel below.

Route Preferences

This dialog is at **Options > Project Preferences > Route Preferences**.

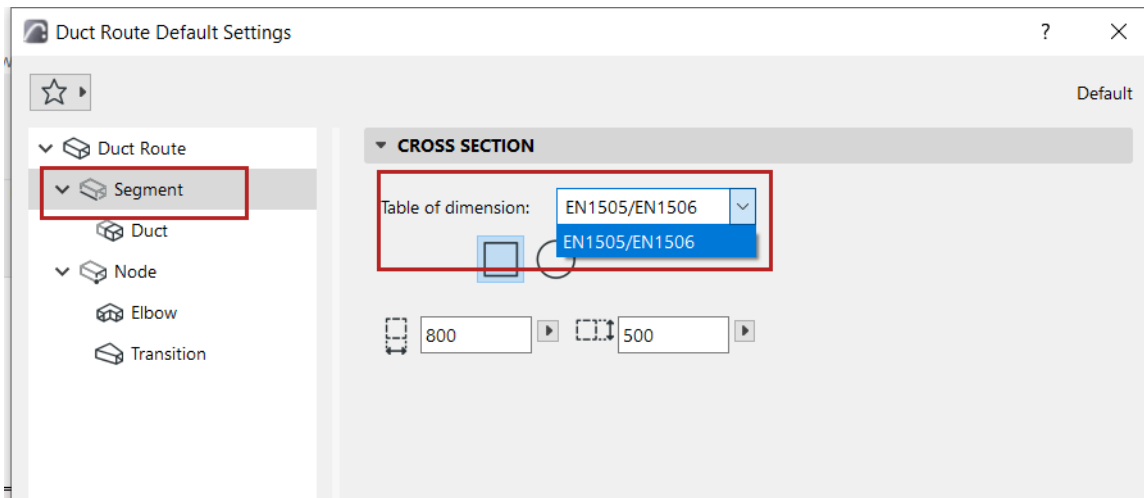
(If you don't see it, select the MEP Work Environment Profile. See [Apply MEP Work Environment](#))

Use Route Preferences to set up Tables of dimensions for the Duct, Pipe, and Cable Carrier segments in your MEP model.

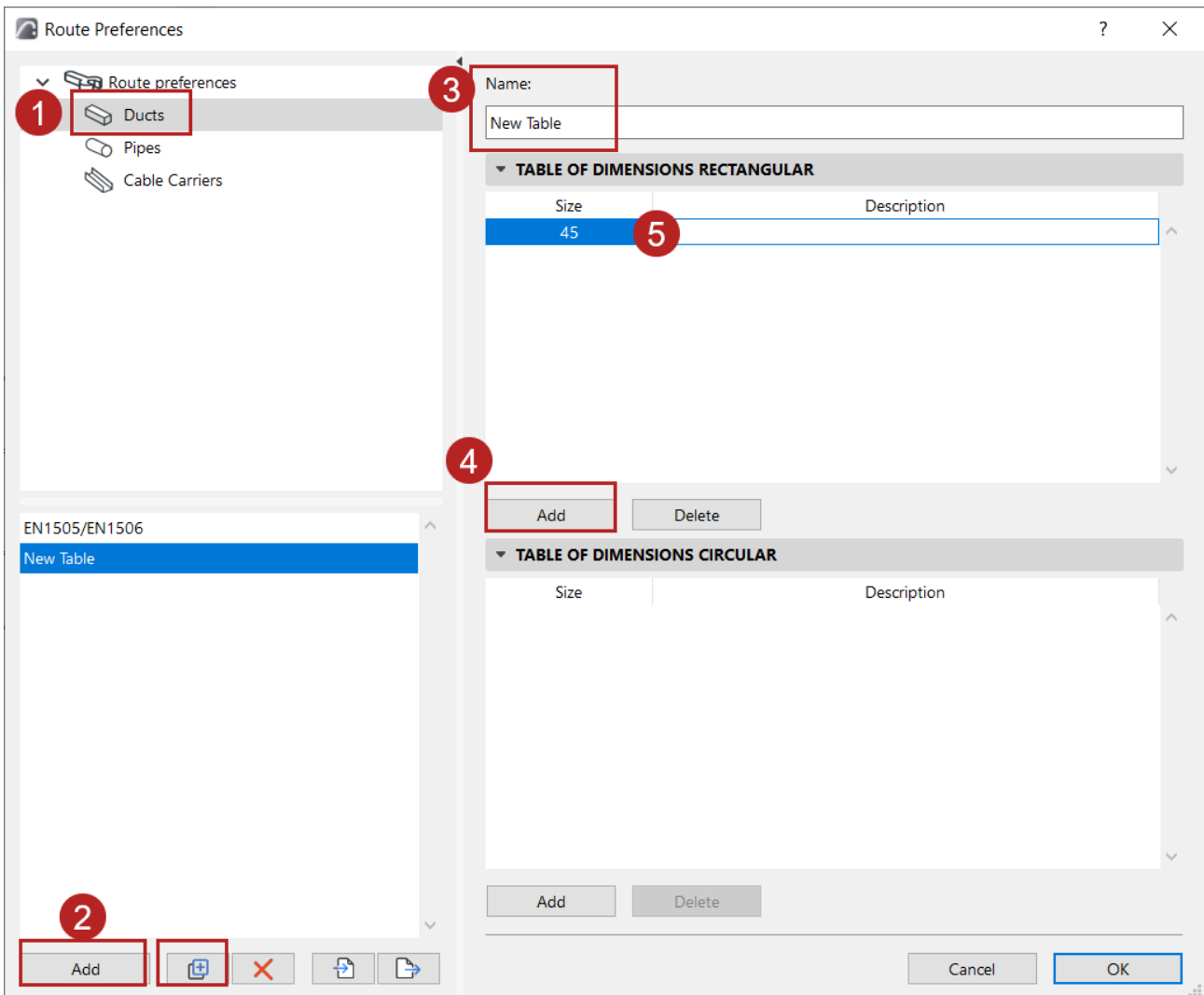


You can create as many tables of dimensions as needed for each segment type.

Then choose the table you need in the corresponding Route Segment settings.



Create new Table of Dimensions

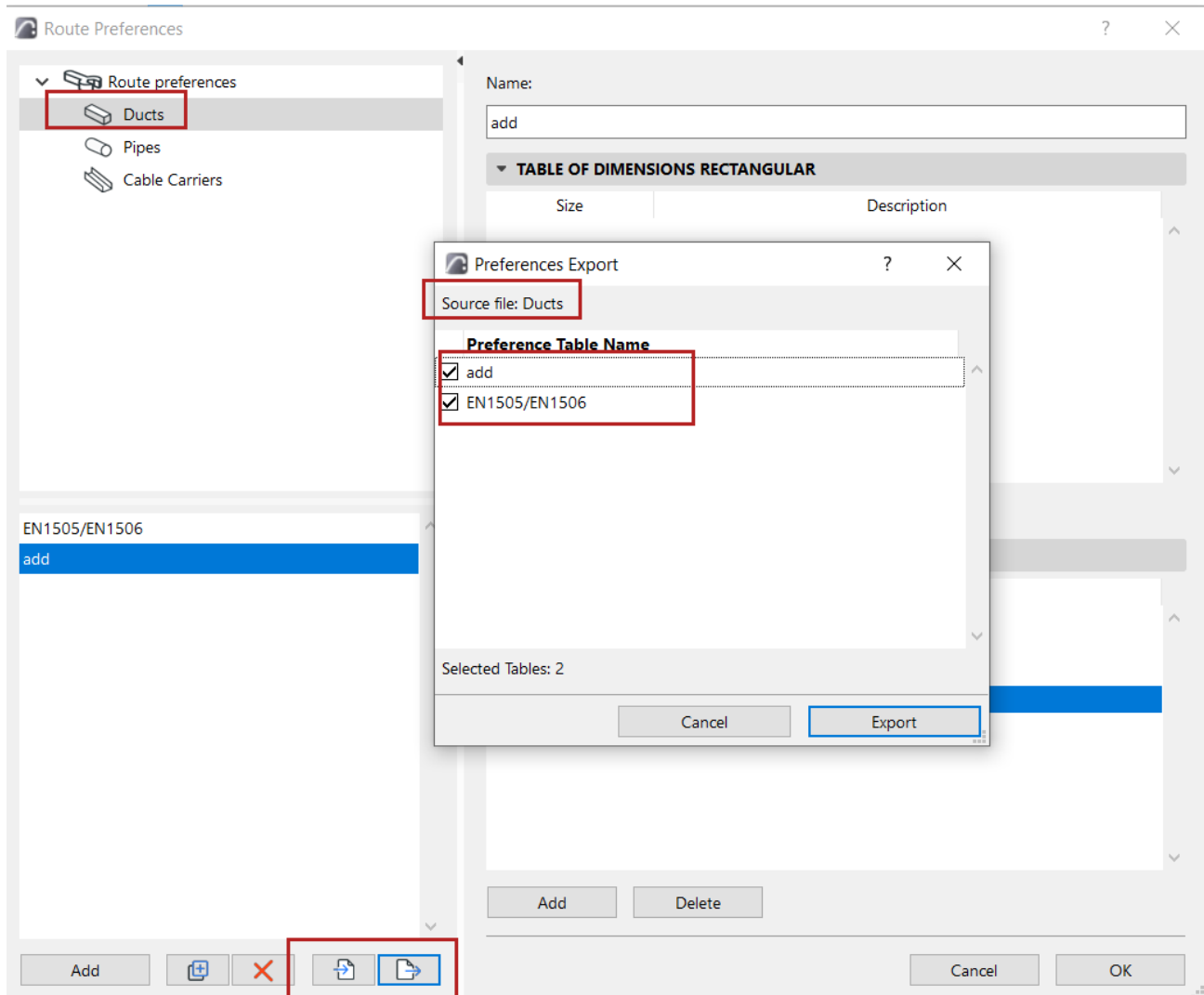


1. On the left side of Route Preferences: select the Segment type (Ducts, Pipes, or Cable Carriers).
2. Click the **Add** or **Duplicate** button.
3. On the right side, enter a **name** for the new table.

- 4. Click **Add**
- 5. Enter the dimension and an optional description. Dimensions are automatically sorted in ascending order.

Import-Export Route Preferences

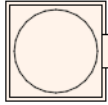
Use the Import and Export buttons to share one or more Dimensions tables of the selected Segment type, via xml files.



Place Terminal, Accessory, or Equipment

Use the Terminal, Accessory, and Equipment tools to place these MEP elements, then connect them to a route.

A route and an MEP element can only be connected if they have the same MEP System. (See [Assign MEP System](#)). When drawing the route, compatible MEP elements are highlighted upon contact.



+

Note: When ending a route element on a terminal, accessory or equipment, use the up and down arrow keys to help find suitable options to connect the route.

The route will connect correctly to the MEP element. Connection geometry includes (as applicable): shape, height/width, thickness of element wall, connection type (e.g. flanged or not): these are the same parameters you set in the Connections tab page of the element's MEP Custom Settings tab page.

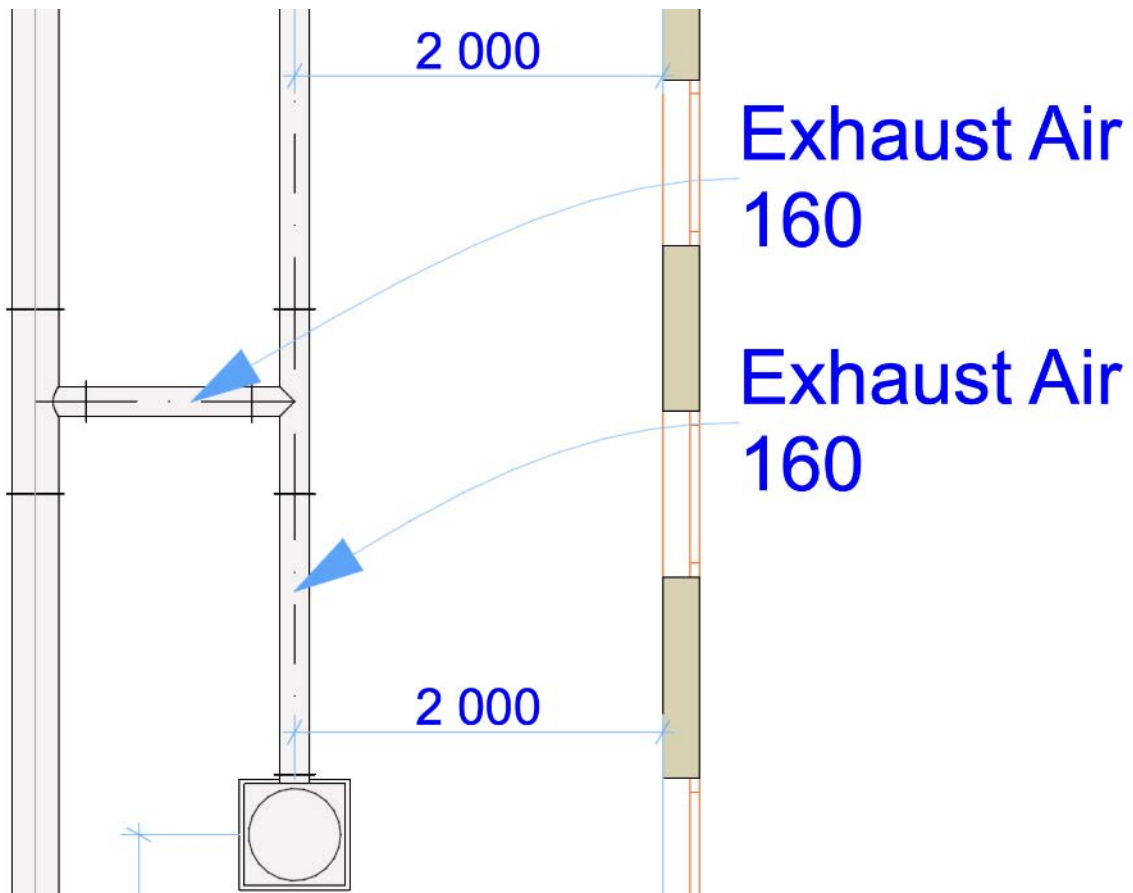
See also [MEP-compatible Library Parts](#).

Select Connected MEP Elements (Command)

1. Select element
2. Right click element and choose **Select Connected MEP Elements**.

MEP Documentation

You can label, dimension, and list parameters of MEP routes.



See also MEP Listing Parameters.

MEP-compatible Library Parts

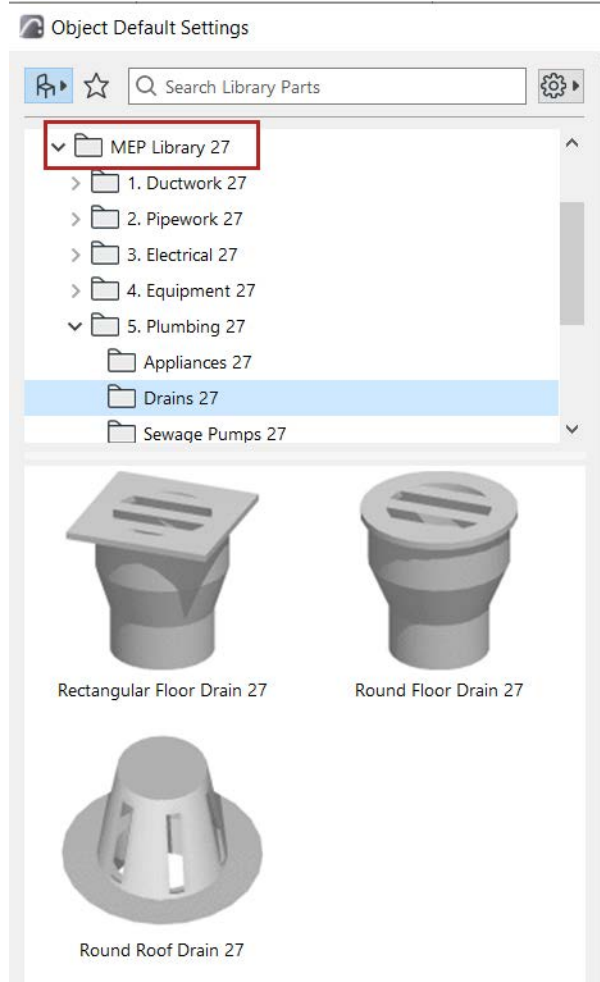
MEP-compatible Library Parts (with smart connections for use in duct, pipe and cabling systems) are found in:

- the MEP Library
- the regular Archicad Library
- in the MEP Library on BIMcomponents.

MEP Library

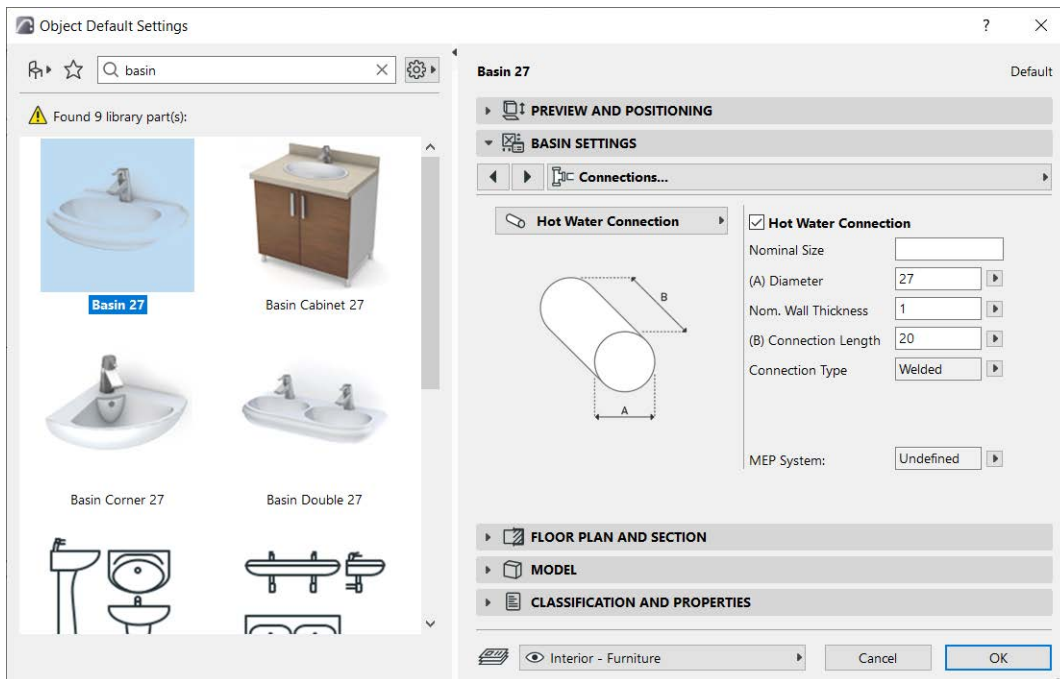
The **MEP Library** contains specially configured MEP elements that you can place using the Accessory, Equipment and Terminal tools - these can be used to start and end MEP routes.

See [Place Terminal, Accessory, or Equipment](#).



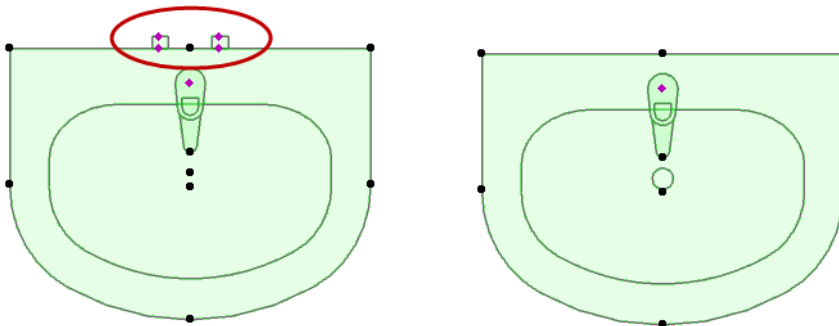
MEP-compatible Objects in the Archicad Library

Many familiar elements from the Archicad Library, such as this Basin, include MEP-compatible connection settings, so you can connect them to an MEP route:



Display MEP Connection Ports on Library Parts

1. Go to **Document > Model View > Model View Options**
2. Open the Miscellaneous Settings for Library Parts panel)
3. Enable the Model View Option for **Object MEP Connections**.

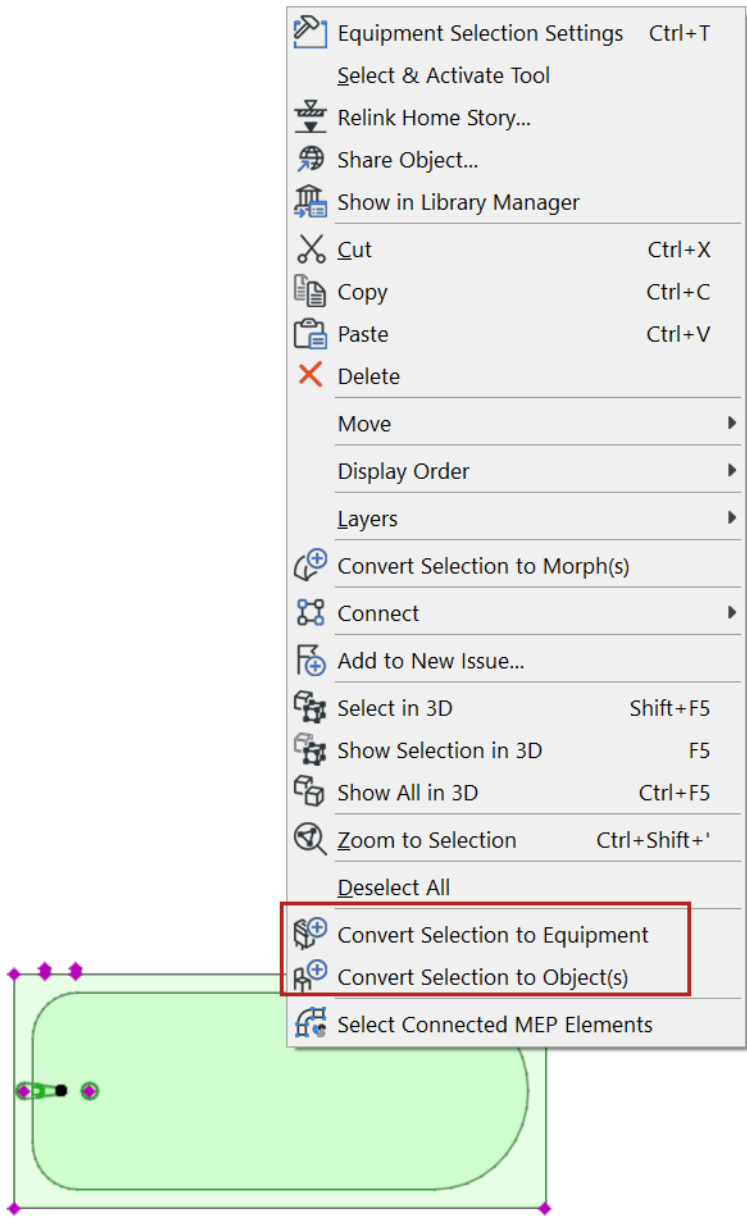


Model View Option: Show or Hide MEP Connections on Library Parts

Element Type of MEP-compatible Objects: Object or Equipment

By default, MEP-compatible objects from the Archicad Library are placed with the Object Tool and their element type is therefore "Object."

For a selected MEP object: Switch its type to Equipment (or back to Object) using the context menu commands.



MEP Systems

An **MEP System** is a saved set of the fills, lines, and surfaces used to display MEP routes and MEP Library elements.

MEP Systems are attributes of the Archicad project.

Assign MEP System

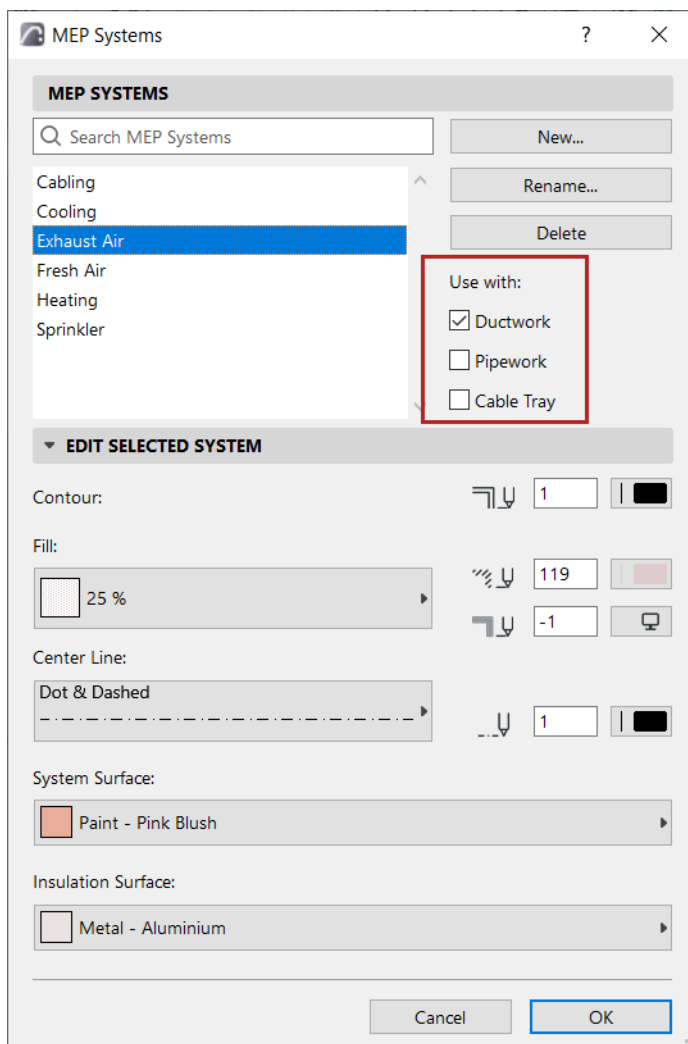
- Each route is assigned an MEP System at the top of the Route Settings dialog. See [Route Settings - main page](#).
- Assign MEP System to Terminal, Accessory, and Equipment in the [MEP-compatible Objects in the Archicad Library](#) tab page.

Define or Edit MEP System Attribute

Define or edit MEP Systems at

- **Design > MEP Systems**
- **Options > Element Attributes > MEP Systems**

(or use the MEP Systems shortcut from the MEP Engineering Toolbar).



Use or edit the predefined MEP Systems, or create new ones.

Use with: Check one or more Route types that should use this MEP System:

- Ductwork
- Pipework
- Cable Tray

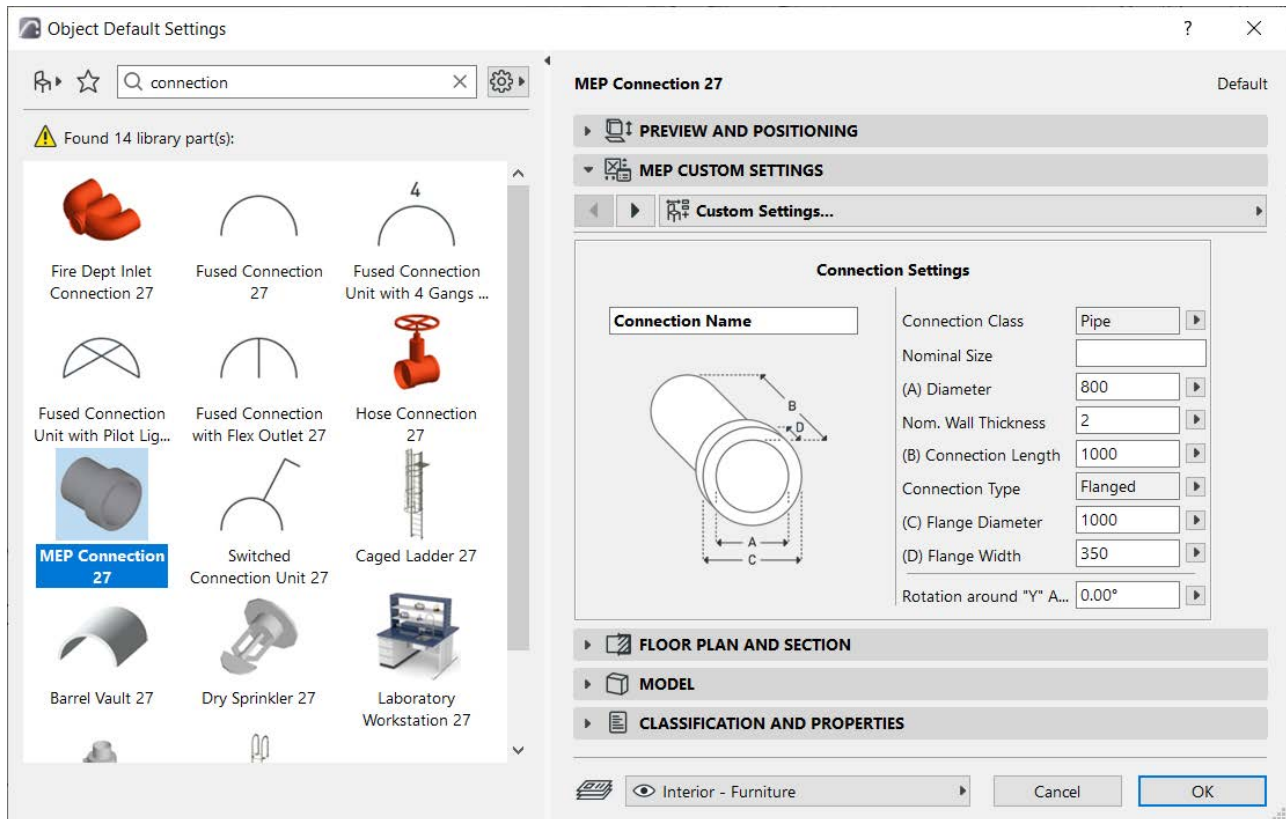
Based on this **Use with** definition, MEP Systems are available for the corresponding Route element type, at the top of its Route Settings dialog.

Edit MEP System attributes

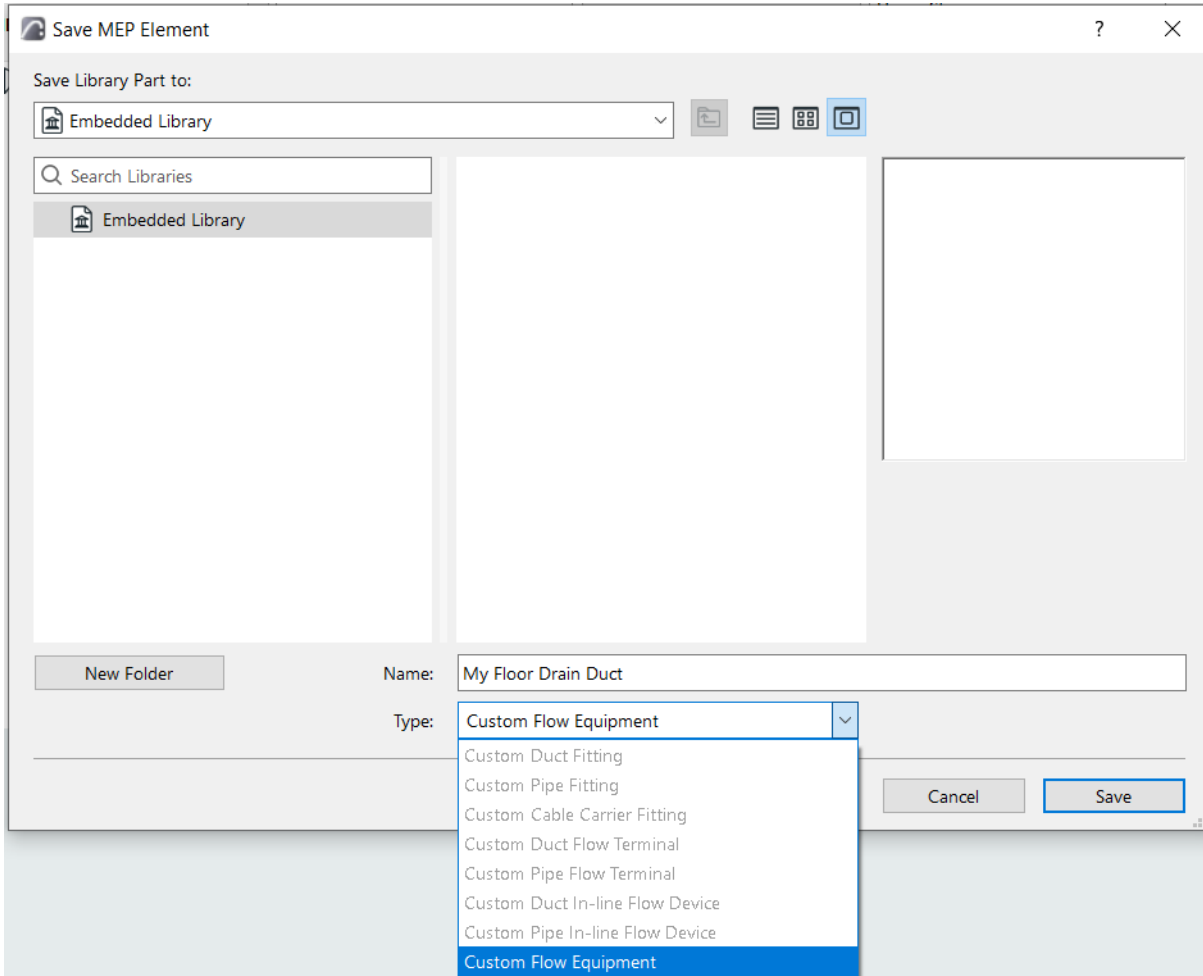
Use the controls to set fills, lines, and surfaces for the current MEP System.

Create custom MEP object

1. In the Floor Plan or 3D window, use any construction elements (e.g. Wall, Slab, Object, MEP element) to model a custom MEP part.
2. If your custom MEP element requires one or more connections, you must use the MEP Connection object as part of your element. This is available from the MEP Library:



3. From the 3D window: Use **File > Libraries and Objects > Save Selection As > MEP Part**
4. Enter a File Name and destination for the object.

5. Choose a subtype from the Type drop-down list:

Note: The available subtypes depend on the number and type of MEP Connection objects contained in the custom object. If it has only a single connection, the equipment and in-line subtypes (which require multiple collinear connections) are not available here.

6. Click **Save**. The new MEP object is accessible from the tool settings of the chosen subtype.

See also [Create Custom Library Parts and Components](#).

MEP elements in IFC exchange

At IFC import

MEP elements are converted as shown in this table.

Incoming element	Converted to Archicad as
segment	fitting
takeoff	fitting
segment + takeoff	equipment
port 1	terminal
port 2	accessory or fitting (depending on location of port)
port 3 and all other ports	fitting

At IFC export

See the *MEP tools* section at [Built-in IFC Type Mapping for Archicad](#).

Structural Analytical Model

This feature set supports collaboration between architects and structural engineers.

Overview

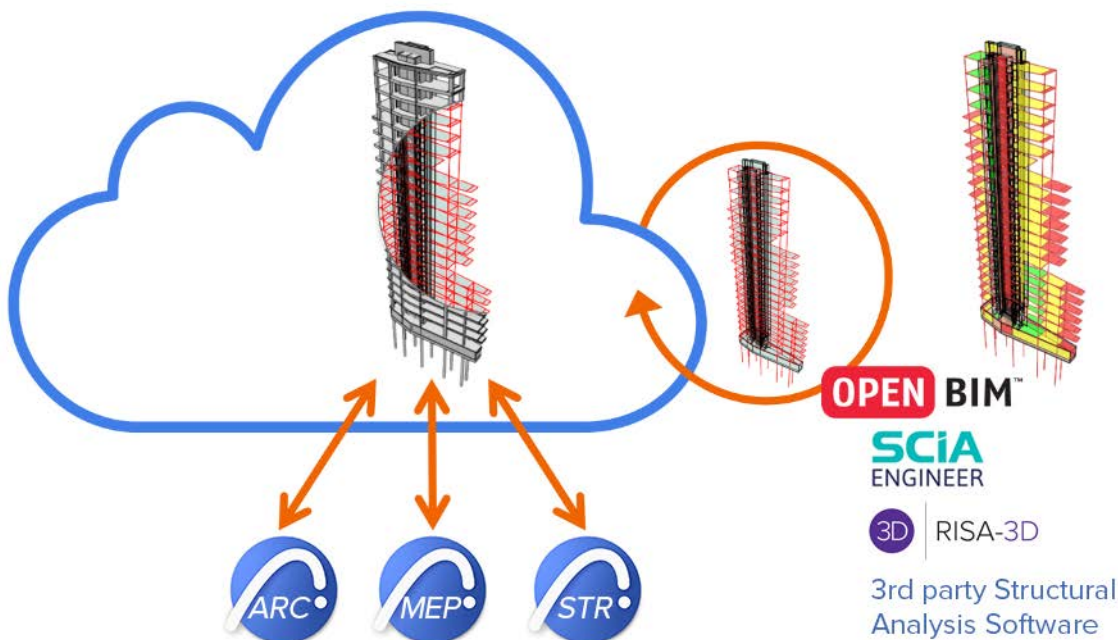
In this workflow, Archicad generates a Structural Analytical Model based on the physical (architectural) model. The Structural Analytical Model is generated in the background, simultaneously with the Physical Model. As the Physical Model changes over time, the associated Structural Analytical Model is automatically updated. The Structural Analytical Model is not a separate model; it is more like a simplified representation of the Physical Model.

The Structural Analytical Model is exported to a structural engineer, who opens the model in his/her own structural analysis program (e.g. SCIA Engineer).

In the analysis program, the engineer performs the analysis simulation and design optimization to make any necessary changes to the Structural Analytical Model. The Structural Analytical Model is then exported to Archicad, which updates the Physical Model with the changes. In this bi-directional workflow, there is no need for file conversion and no associated data loss.

Interoperability with structural analysis programs is ensured by use of the **Structural Analytical Format (SAF)**, which is explicitly optimized for such softwares and enables a much faster model exchange workflow (e.g. as compared to an IFC-based workflow).

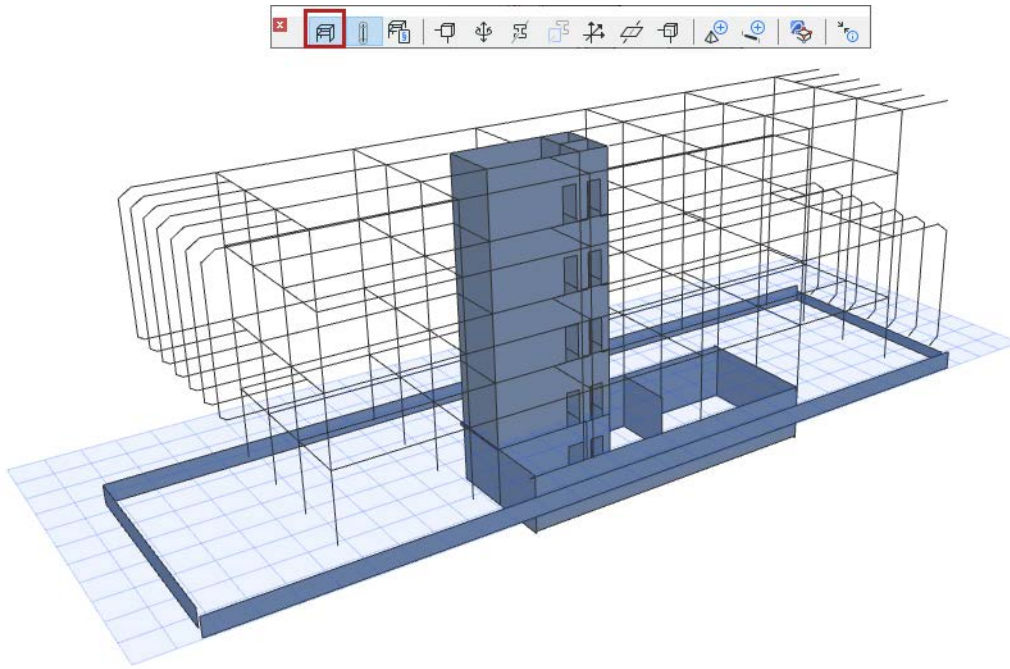
Other Archicad features - Model Compare, Issue management, Change Tracking in Teamwork, and Model Check for analytical and physical models - also support this workflow.



How do I generate a Structural Analytical Model?

Use the **Show Structural Analytical Model** toggle

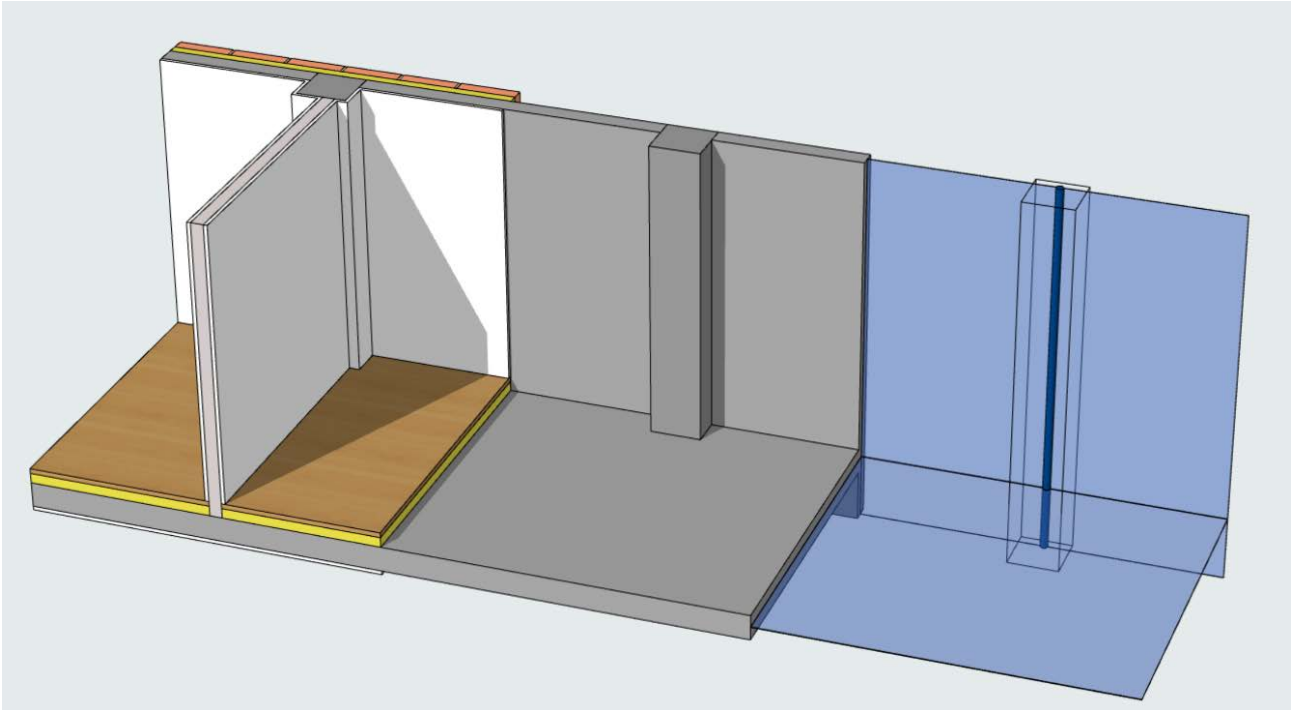
- from the **View** menu
- from the Structural Analytical Model Toolbar



Which Parts of the Archicad Model Become Part of the Structural Analytical Model?

1. Load-bearing elements
2. Contain a Core
3. Created with an eligible Archicad tool. [See *Elements of the Structural Analytical Model*.](#)
 - Beams and Columns are represented as 1D Members
 - Slabs, Walls, and Single-plane Roofs are represented as 2D Members
4. Enabled at the element level (enable checkbox for “Structural Member” in Element Settings)
5. Exists in the model as defined by Renovation Status (if it is “To Be Demolished” or “New” and hidden in the current view, it will not show up in the Structural Analytical Model)
6. Element connections (as defined by Layer Intersection Group) are reflected in the geometry of the Structural Analytical Model

7. Layer visibility is NOT considered. Even elements on a hidden layer can be part of the Structural Analytical Model.



Default Position of 1D and 2D Members

A Structural Analytical Member is generated by default in the element's center of gravity (for 1D Members) or along the center of the element's core plane (for 2D Members).

Manual adjustments to analytical Members

You can modify the generated member geometry in two ways:

- Adjust Eccentricity. For example, you can use the Eccentricity control in Slab Default Settings so that its 2D Member is always generated on the top plane of the Slab.

[See *Eccentricity*](#).

- Edit Structural Member manually

[See *Manual editing of analytical members*](#).

Automatic adjustments with Generation Rules

The "Structural Analytical Model Generation Rules" are applied automatically each time it is generated.

These rules adjust the geometry of 1D and 2D Members and certain End/Edge Release definitions. The purpose is to optimize the Structural Analytical Model to make it correct and compatible with external analysis programs.

Related Topics

[Structural Engineering Work Environment and Commands](#)

[Elements of the Structural Analytical Model](#)

[1D Members \(Column and Beam\)](#)

[2D Members \(Slab, Wall, Roof\)](#)

[Structural Supports](#)

[Structural Links](#)

[Structural Loads](#)

[Representation of Structural Analytical Model](#)

[Structural Analytical Model Generation Rules](#)

[Eccentricity](#)

[Manual editing of analytical members](#)

[End/Edge Releases](#)

[Model Check - Overview](#)

[Model Check Report](#)

[Collaboration with Structural Analysis Software](#)

Structural Engineering Work Environment and Commands

The Structural Engineering Work Environment Profile includes dedicated toolbars:

- Structural Engineering
- Structural Loads



and the Load Case palette.

Commands related to the Structural Analytical Model can be found under the existing menus, as follows:

View Menu

- Show Structural Analytical Model
- Structural Analytical Model Options
 - Structural Analytical Model Representation

Design Menu

- Structural Analytical Model Generation Rules
- Structural Analytical Elements
 - Create Structural Supports
 - Create Structural Links
- Model Check > Structural Analytical Model Quality
- Model Check > Physical Model Quality
- Model Compare

Options Menu

- Load Manager
- Load Combinations
- Project Preferences > Structural Analytical Model Units
- Project Preferences > Structural Analytical Model Representation
- Project Preferences > Structural Load Units

Elements of the Structural Analytical Model

The Structural Analytical Model represents a simplified version of the building's load-bearing elements and their supports.

Which Archicad Elements Participate in the Structural Analytical Model?

The element must meet the following conditions:

- Its Structural Function is **Load-Bearing Element**
- **Structural Member** is enabled for the element (in element settings)
- It was created using one of the **eligible Archicad tools**:
 - Column
 - Beam
 - Wall
 - Single-Plane Roof
 - Slab
- If it is a composite or Complex Profile, it must include **at least one Core skin/component**
Note: Certain element geometries are not supported. See more at [1D Members \(Column and Beam\)](#) and [2D Members \(Slab, Wall, Roof\)](#)

Components of the Structural Analytical Model

The Structural Analytical Model consists of the following:

1D Members (Column and Beam)

2D Members (Slab, Wall, Roof)

The following tools support generation of the Structural Analytical Model:

Structural Links

Structural Supports

Structural Loads

Other components are Nodes End/Edge Releases, and Local Coordinate System:

Show Nodes

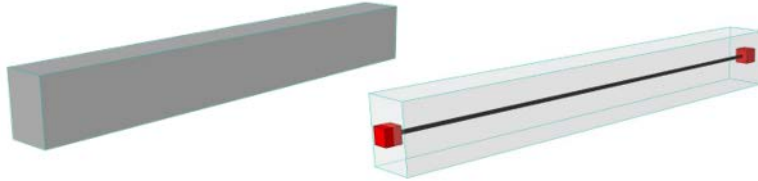
End/Edge Release Symbols

Show Local Coordinate Systems

1D Members (Column and Beam)

Beams and Columns are represented in the Structural Analytical Model as 1D Members. They have two nodes (start and end node) and one axis line connecting the nodes.

The 1D Member is generated by default at the center of gravity of the element's core (or core component).



Customize 1D Member color at [Structural Analytical Model Representation \(Preferences\)](#).

The analytical Member does not extend outside of the physical body unless it has been adjusted.

Which Columns and Beams are 1D Members of the Structural Analytical Model?

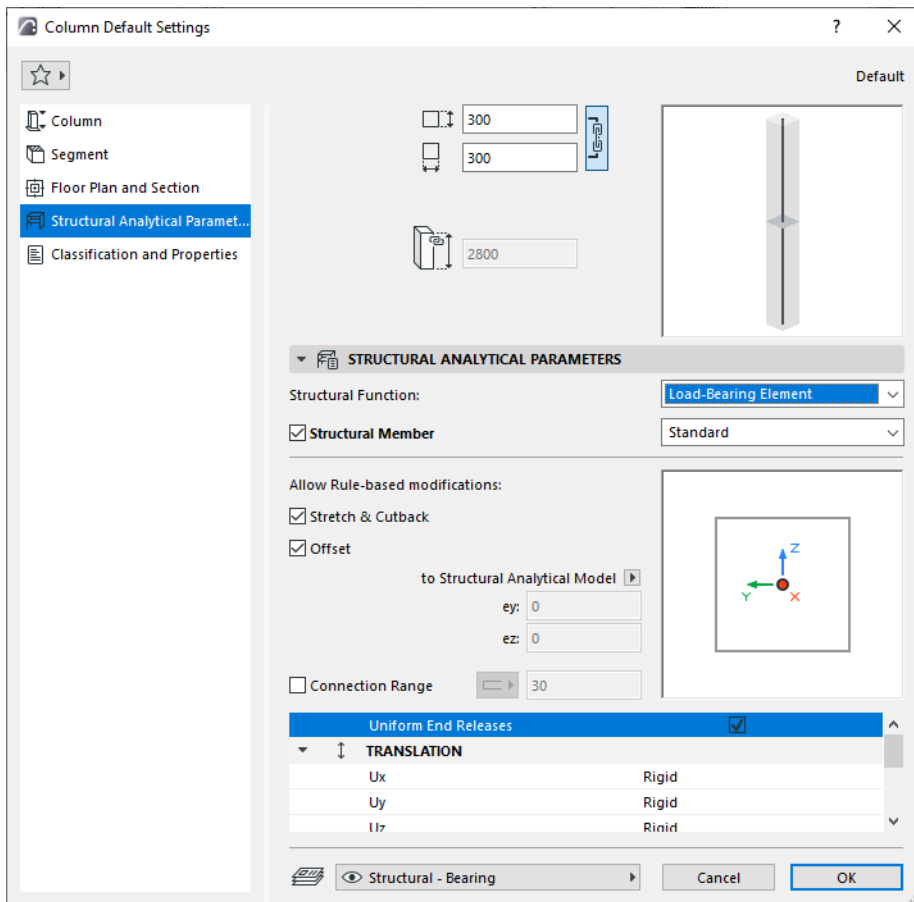
Columns and Beams will generate a 1D Member in the Structural Analytical Model if they meet these conditions:

- Load-bearing
- Contains a core
- “Structural Member” checkbox is activated in Element Settings
- Geometry is supported (see table below)

Beam-Column Geometry	Member of Structural Analytical Model?
Basic Element (Rectangular, Circular)	YES
Complex Profile - Includes a Core	YES
Complex Profile - No Core	NO
Multi-Segment and/or Tapered	YES, based on single Replacement Profile or varying profiles

Tool Settings for 1D Members

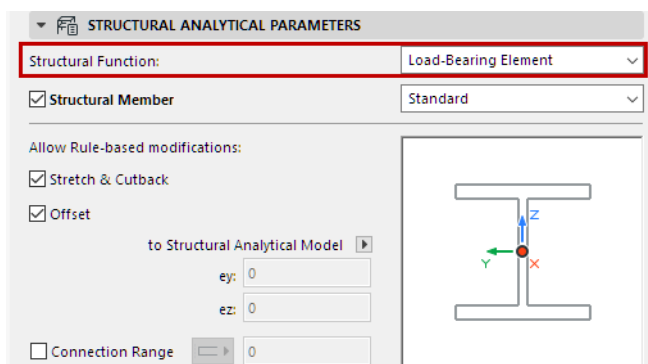
Use the Structural Analytical Parameters panel of Column or Beam settings.



Structural Function - Enable Load-Bearing

Set the Structural Function to Load-Bearing.

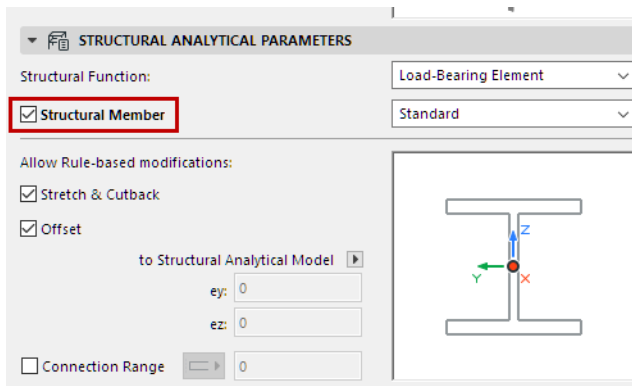
This is the same control, with the same value, as in the Structural Function control found in the Classification and Properties panel.



Structural Member - Enable for Structural Analytical Model

This box is checked by default: include this element in the Structural Analytical Model as a 1D Member.

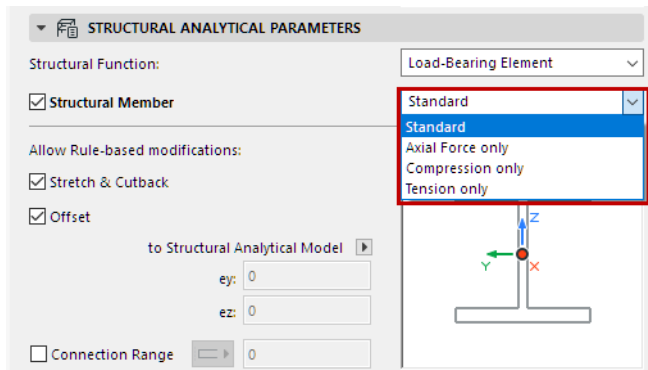
Uncheck if the element should NOT be included in the Structural Analytical Model.

**Notes:**

- Available only if Structural Function is set to Load-Bearing.
- If the element is a composite or Complex Profile, it must include at least one Core skin/component in order to be part of the Structural Analytical Model.

1D Member - Define Structural Behavior

This Structural Behavior setting defines the Member's behavior in a Finite Element Method (FEM)-based structural analysis program: specifically, which types of forces this element is designed to bear.



Choose an option to define the Member's structural behavior:

- Standard (default) - The element can transfer all forces, such as axial and shear forces.
- Axial force only
- Compression only
- Tension only

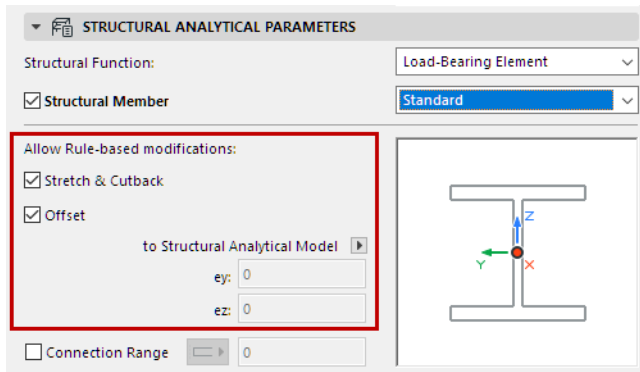
The available definitions are fixed and cannot be customized.

The Structural Behavior setting has no effect on the Physical Model.

Allow Rule-based modifications - 1D Member

Activate these checkboxes to apply the respective Adjustment Rules that are defined in the Structural Analytical Model Generation Rules dialog box.

By default, the **Stretch & Cutback** and **Offset** boxes are checked: the Stretch, Cutback and Offset adjustments defined in the Generation Rules dialog affect this 1D Member element.



If an Adjustment Rule uses the “Snap” Adjuster, it will be applied to this element only if *both* checkboxes are checked.

See [Structural Analytical Model Generation Rules](#).

To **exempt** this 1D Member from automatic adjustment rules, uncheck the box(es). The respective Adjustment Rules are not applied to this element.

If needed, you can set a custom offset for this element (see below).

Custom Offset

If the **Offset** box is unchecked:

Redefine this element’s eccentricity, either graphically or by entering a value.

See [Eccentricity](#).

Connection Range

For Beams and Columns: This option virtually extends the 1D Member’s core length along its axis.

The purpose of virtually extending the core is to enable the Beam or Column to achieve a core connection with other elements, if such a core connection does not exist. As a result of this virtual extension, Adjustment Rules (which require core connections) can go into effect.

For details, see [Connection Range \(Extend Core to Enable Connection\)](#).

Translation and Rotation

See [End/Edge Releases](#).

Connection Range (Extend Core to Enable Connection)

For Beams and Columns: This option virtually extends the 1D Member's core length along its axis.

The purpose of virtually extending the core is to enable the Beam or Column to achieve a core connection with other elements. As a result, Adjustment Rules (which require core connections) can go into effect.

This can be useful with certain prefabricated elements which contain deliberate core gaps at either end (e.g. to make room for insulation or detailing.) This Connection Range virtually extends the core to fill these gaps.

Important Notes:

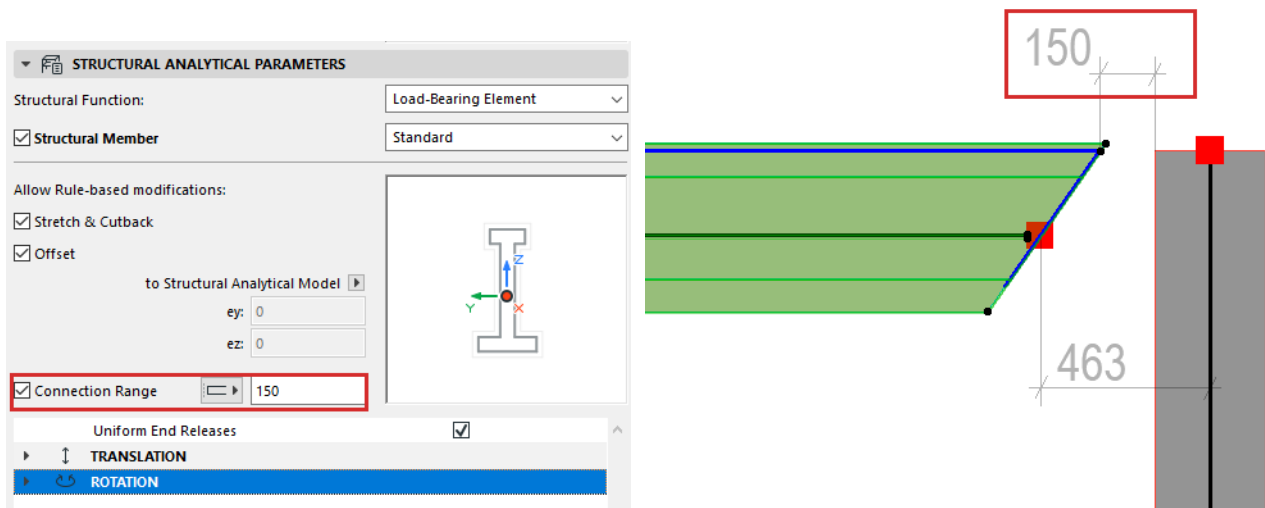
- This option extends the length of the element core only **virtually**: it has no effect whatsoever on the physical model element, and does not extend the 1D Member.
- The extra Connection Range is considered when you run a Model Check on the Structural Analytical Model. If the Connection Range enables the structural connection, then the Model Check will consider the elements to be continuous for the purposes of the Structural Analytical Model.
- You can add an extra Connection Range to the element only if the Stretch & Cutback and/or Offset boxes are also checked in Column or Beam Settings.

In Beam or Column Settings:

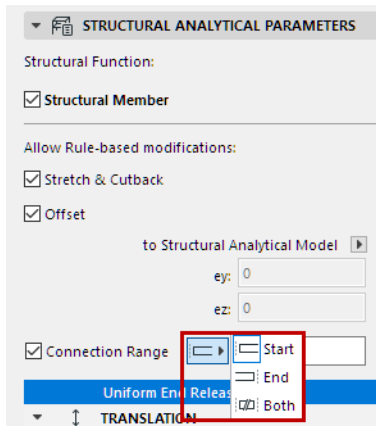
1. Check the Connection Range option.
2. Enter the length of the extension.

The value must be at least as long than the shortest distance between the element cores to be connected.

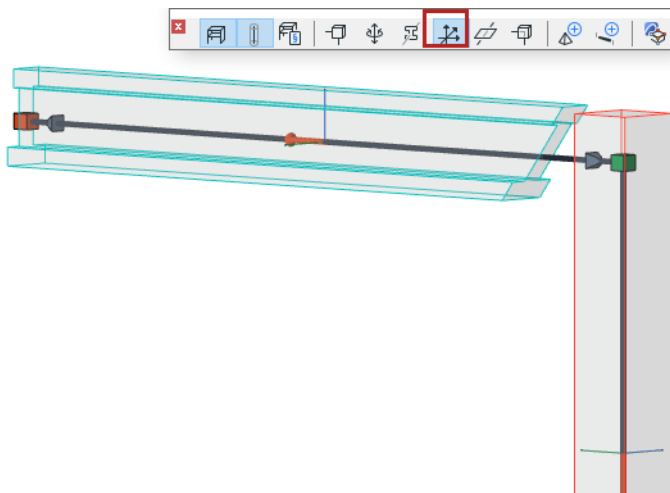
In this example, we want to extend the Beam Connection Range must be at least 150.



3. Choose "Start", "End", or "Both" to define which end should be extended. The "Both" option adds the extended length **at both ends**.

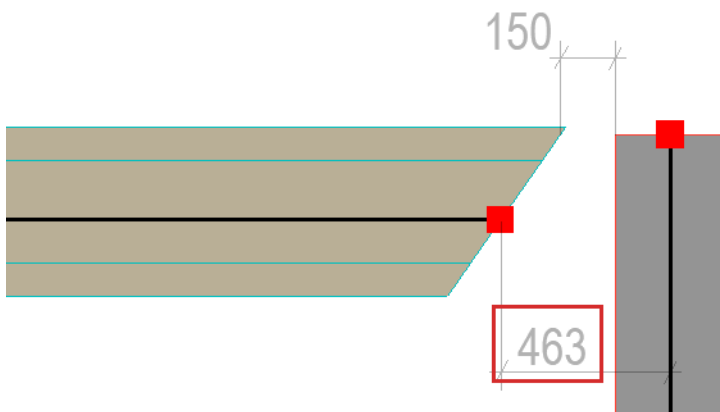


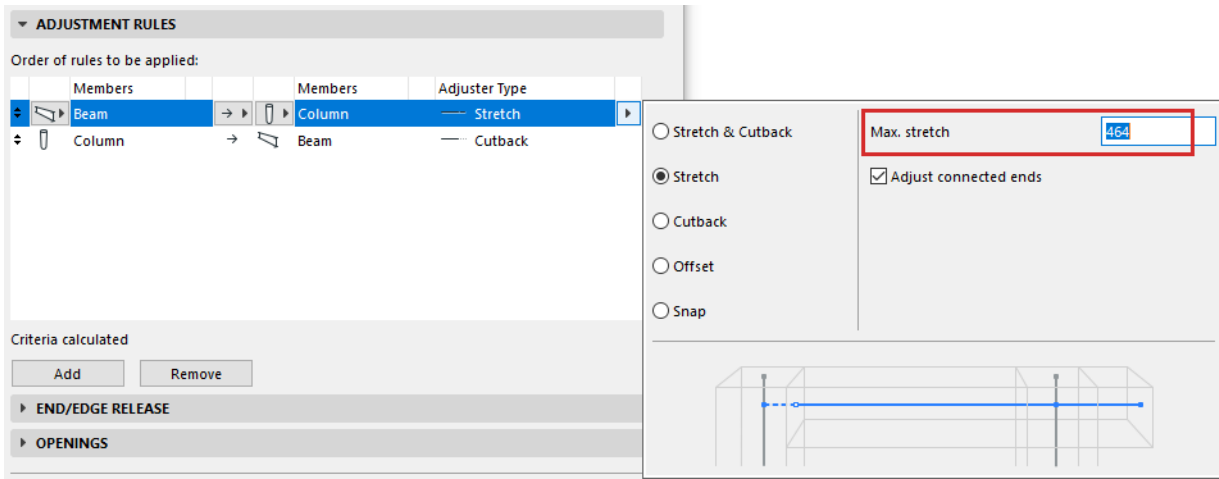
Note: To see which end is which, turn on “Show Local Coordinate Systems” to see the element’s direction arrow.



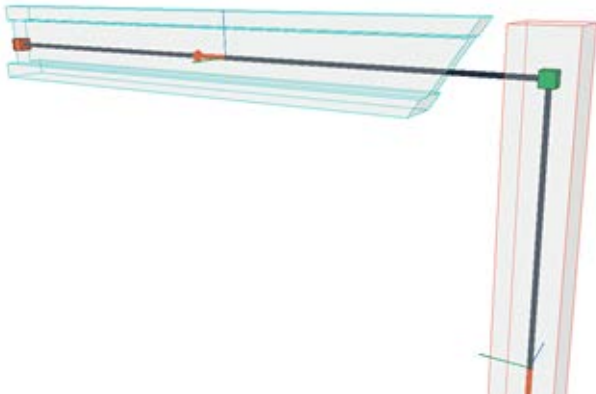
In Adjustment Rules:

The maximum allowed stretch must exceed the distance between the structural axis to be connected. This “Max stretch” value is applied to the either or both ends of the Connection Range.





Thanks to the Beam's Connection Range setting, the Stretch Adjuster works as desired, and the 1D Members are connected.



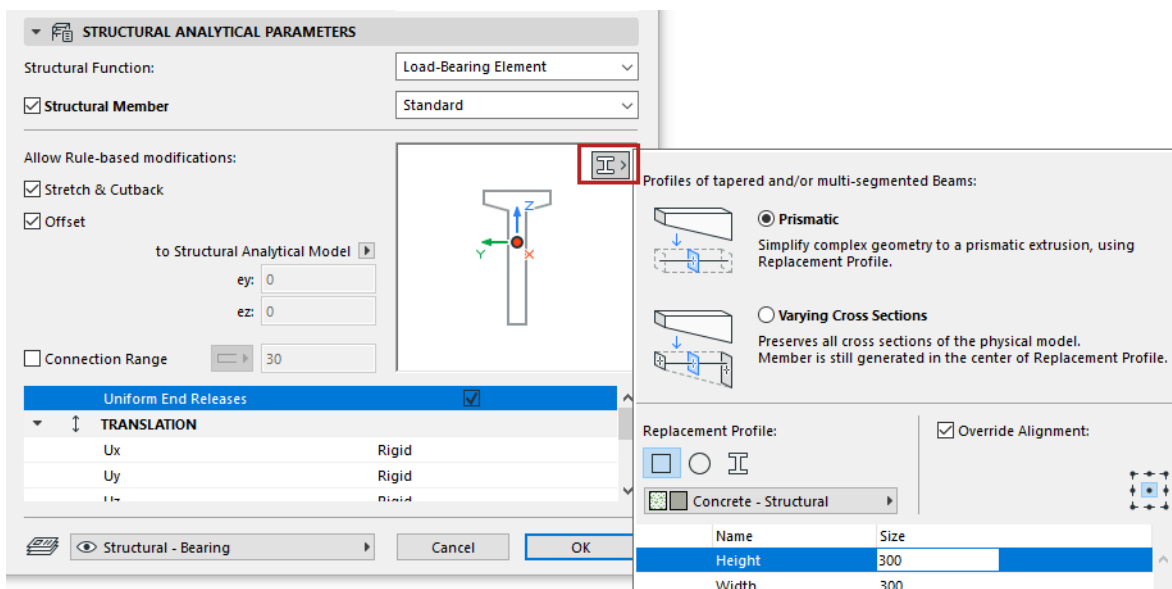
Beam/Column Profiles in SAF export

Some Column or Beam elements can use multiple cross-sections and/or several Building Materials:

- Multi-segment Beams or Columns
- Tapered Beams or Columns
- A single Complex Profile with different Profile Modifiers at each end

Such a Column or Beam generates a single 1D Member in the Structural Analytical Model. However, you have two options to handle the cross section export to SAF: use a single Replacement Profile, or export all the cross sections.

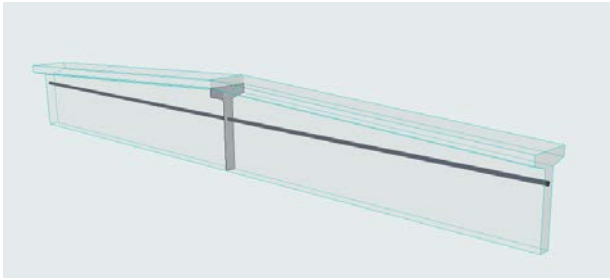
1. Go to the **Structural Analytical Parameters** panel of Beam or Column Settings.
2. If the element has a varied cross-section: the Replacement Profile pop-up is available in the preview window.



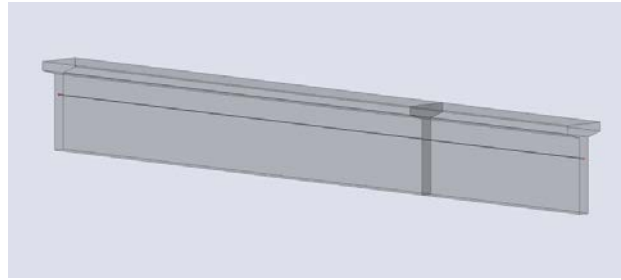
3. Choose how to export the cross-section profile(s):
 - **Prismatic:** export a single Replacement Profile to represents the 1D Member cross section. (Recommended for early-stage calculations of schematic design)
 - **Varying cross sections:** Export all the cross sections of the 1D element, preserving all details of the physical model. (Recommended for project stages where precise calculations are needed).



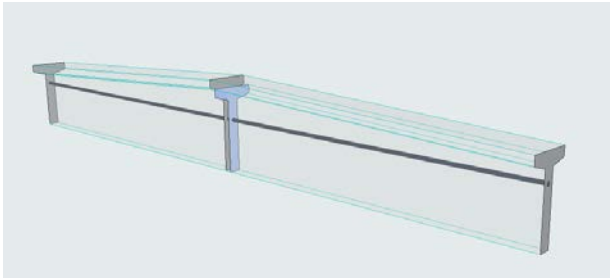
Original physical model in Archicad



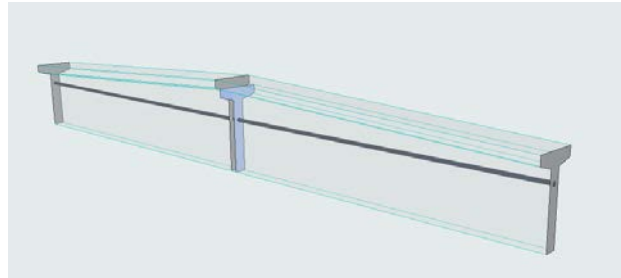
Prismatic (Export using single replacement profile)



Prismatic export in analysis software



Varying cross sections (Export all cross-sections)



Varying cross sections in analysis software

4. Choose a structure for the Replacement Profile core: Rectangular, Circular, or Complex Profile

For a Complex Profile structure:

- Choose a Profile and set editable parameters (if any). The list includes:
 - Profiles available for Column or Beam (as applicable)
 - Profiles that include at least one core component

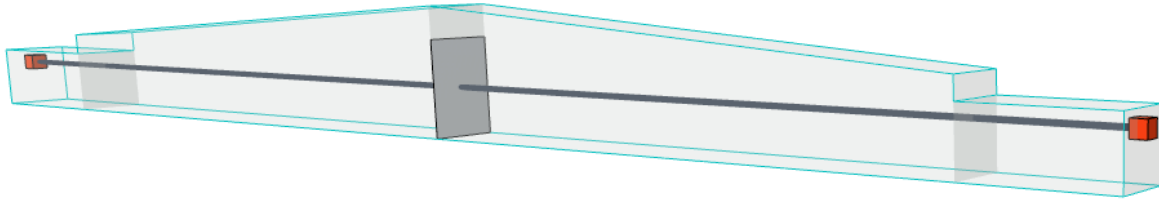
For a Rectangular or Circular structure:

- Choose Building Material and dimensions

Override Alignment of Replacement Profile

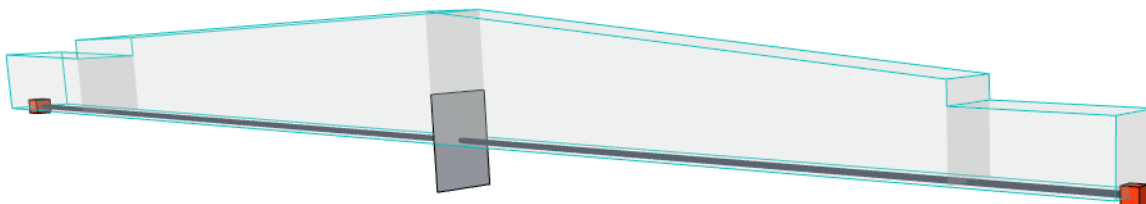
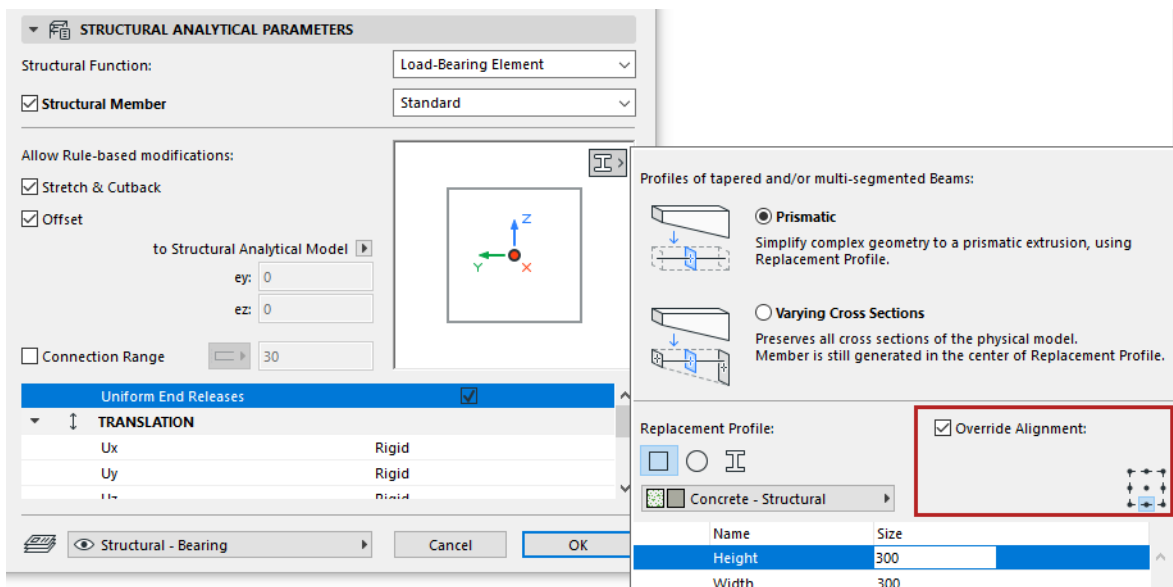
Available with **Prismatic** option.

By default, the Replacement Profile origin is inserted at the Member's reference line.



Default position of Replacement Profile

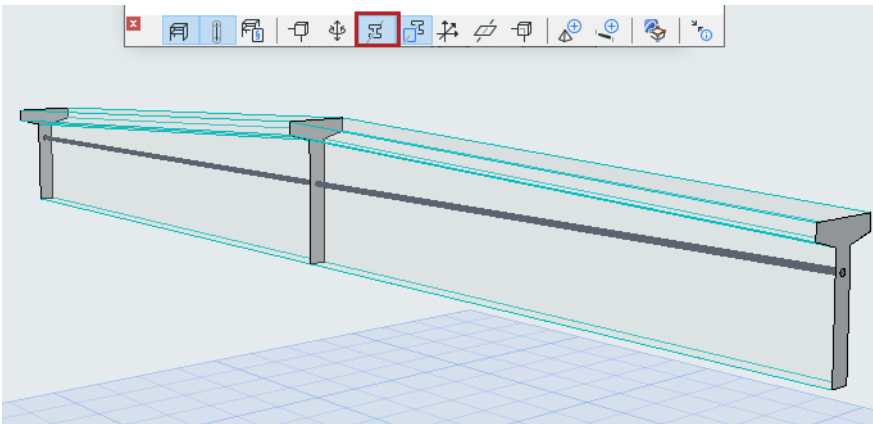
If needed, realign the Profile Origin to better reflect the architectural model. Check the **Override Alignment** box, then choose a different anchor point:



Override position of Replacement Profile

Show Profiles of Load-Bearing Cores

Turn on this option (from the Structural Analytical Model toolbar).

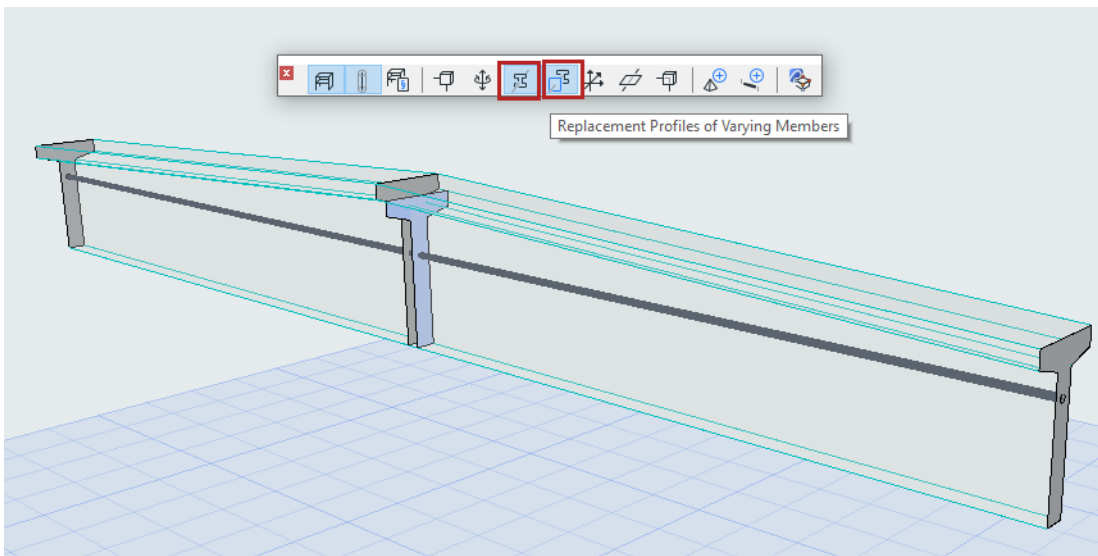


Replacement Profile for Varying Cross Sections: Position of 1D Member

For varying cross sections, the Replacement Profile is used only to position the 1D Member, which is generated in its center. To see this Replacement Profile, turn on the toggle.

(The Show Profiles toggle must also be on.)

Use this display option to understand how the 1D Member is positioned in a varying cross-section.



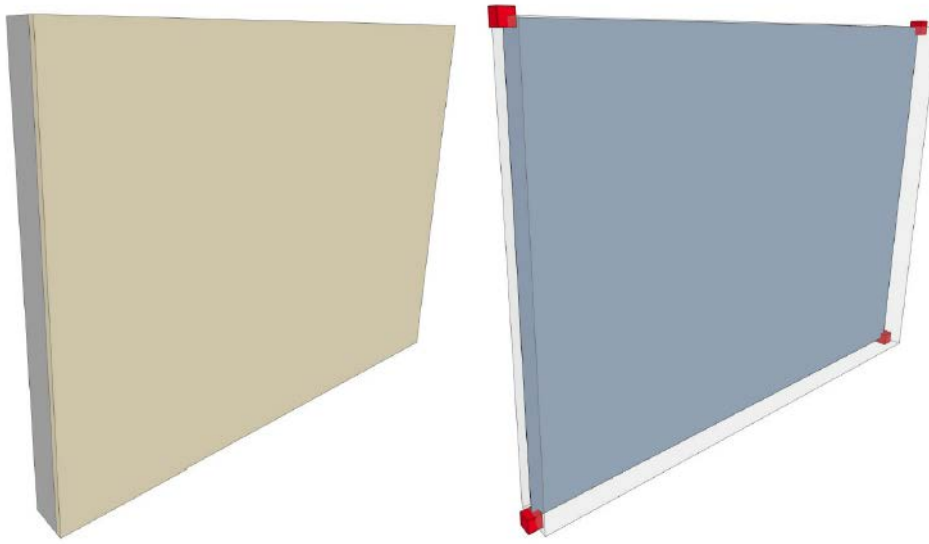
2D Members (Slab, Wall, Roof)

Slabs, Walls, and Single-plane Roofs are represented as 2D Members in the Structural Analytical Model.

2D Members are plane surfaces generated along the **center** of the element's **core plane**. The 2D Member has at least four nodes and one surface area connecting them.

The analytical Member does not extend outside of the physical body unless it has been adjusted.

Customize 2D Member color at [Structural Analytical Model Representation \(Preferences\)](#).



Which Elements Are 2D Members of the Structural Analytical Model?

Slabs, Walls, and Roofs will generate a 2D Member in the Structural Analytical Model if they meet these conditions:

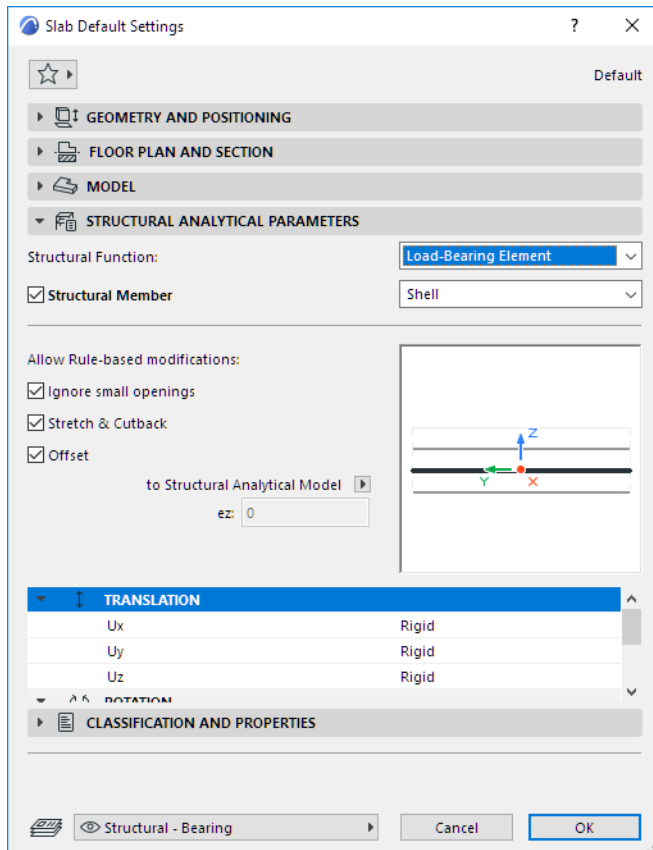
- Load-bearing
- Contains a core
- “Structural Member” checkbox is activated in Element Settings
- Geometry is supported (see table below)

Element Type	Geometry	Member of Structural Analytical Model?
Slab	all	YES
Wall	Straight	YES
	Curved	YES
	Slanted (straight)	YES
	Double Slanted	NO
	Trapezoid	NO
	Slanted, Double Slanted (curved)	NO
	Complex Profile	YES
	Polygon Wall	NO

Element Type	Geometry	Member of Structural Analytical Model?
Roof	Single-Plane	YES
	Multi-Plane	NO

Tool Settings for 2D Members

Use the Structural Analytical Parameters panel of Slab, Wall or Roof settings.



Structural Function - Enable Load-Bearing

Set the Structural Function to Load-Bearing. This is the same control, with the same value, as in the Structural Function control found in the Classification and Properties panel.

Structural Member - Enable for Structural Analytical Model

This box is checked by default: include this element in the Structural Analytical Model as a 2D Member.

Notes:

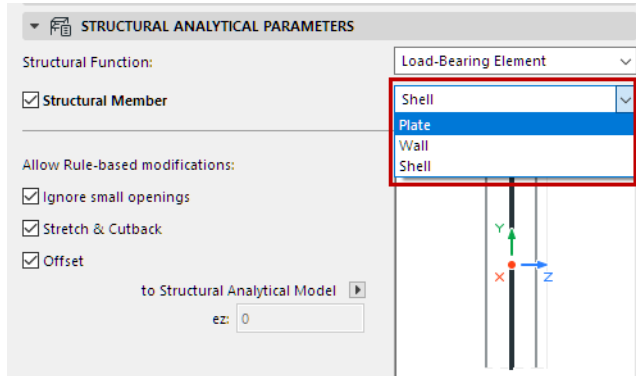
- Available only if Structural Function is set to Load-Bearing
- If the element is a composite or Complex Profile, it must include at least one Core skin/component in order to be part of the Structural Analytical Model

Uncheck if the element should NOT be included in the Structural Analytical Model. (For example, a thin moisture-blocking membrane on an underground story.)

2D Member - Structural Behavior

The Structural Behavior setting defines the Member's behavior in a FEM-based structural analysis program: specifically, which types of forces this element is designed to bear.

Choose an option to define the Member's structural behavior:



- Shell: Takes all loads
- Plate: Element is able to take loads perpendicular to the plane.
- Wall: Element is able to take loads along its plane.

The available definitions are fixed and cannot be customized.

The right choice here depends on the receiving structural analytical software. Some softwares will filter out incoming elements based on this Structural Behavior setting. If you find that exported 2D Members are not being imported to the receiving software, you may have to adjust their Structural Behavior definition in Archicad.

The definition can be entirely independent of the tool used to create the element: a vertical Wall's behavior can be set to "Plate" or "Shell"; a horizontal Slab's behavior can be set to "Wall".

The Structural Behavior setting has no effect on the Physical Model.

Allow Rule-based modifications - 2D Member

By default, all checkboxes are on:

- Ignore small openings
- Stretch & Cutback
- Offset

This means that corresponding adjusters and opening filter, defined in the Generation Rules dialog, will affect this 2D Member element.

[See Structural Analytical Model Generation Rules.](#)

To **exempt** this 2D Member from adjustments: uncheck the box(es).

Custom Offset

If the **Offset** box is unchecked:

Redefine this element's eccentricity along the z axis (perpendicular to the plane of the 2D Member), either graphically or by entering a value.

[See Eccentricity.](#)

Translation and Rotation

[See End/Edge Releases.](#)

Structural Supports

The Structural Support tool is a schematic element that indicates support of physical elements - e.g. by the soil stratum, other elements, or tethers.

Supports are created on 1D or 2D Members as applicable: at nodes (Point Support), along linear elements or edges (Line Support), or in the 2D plane (Surface Support).

Structural Supports are placed only on Members of the Structural Analytical Model - you cannot use Structural Supports with physical model elements in Archicad.

Once Structural Supports are placed, you can customize their options in the Info box or Tool settings:

- Define supports according to the global or local coordinate system of its host element
- Set translation and (where applicable) rotation stiffness (x, y, z, Rx, Ry and Rz)

To customize Structural Support colors, [Structural Analytical Model Representation \(Preferences\)](#).

To show or hide these elements in 3D: use View > Elements in 3D View > Filter and Cut Elements in 3D

Notes:

- Support elements are specific to their host element. If you make a copy of the host element only, the new copy will not have analytical support. To copy both, you must select both: element and its supports.
- Support elements inherit the Coordinate System settings (global vs. local) of their host elements. This information is exported accurately through SAF format. However, please note that the local coordinates in the receiving program may be interpreted differently than in Archicad.
- Support elements attached to certain holes or openings have specific limitations. See [Structural Supports at Openings and Holes](#).

Related Topics

[Place Structural Support](#)

[Support Elements: Symbol Options](#)

[Structural Supports at Openings and Holes](#)

Place Structural Support

There are two ways to place Structural Supports:

Structural Support Tool

“Create Structural Supports” Command

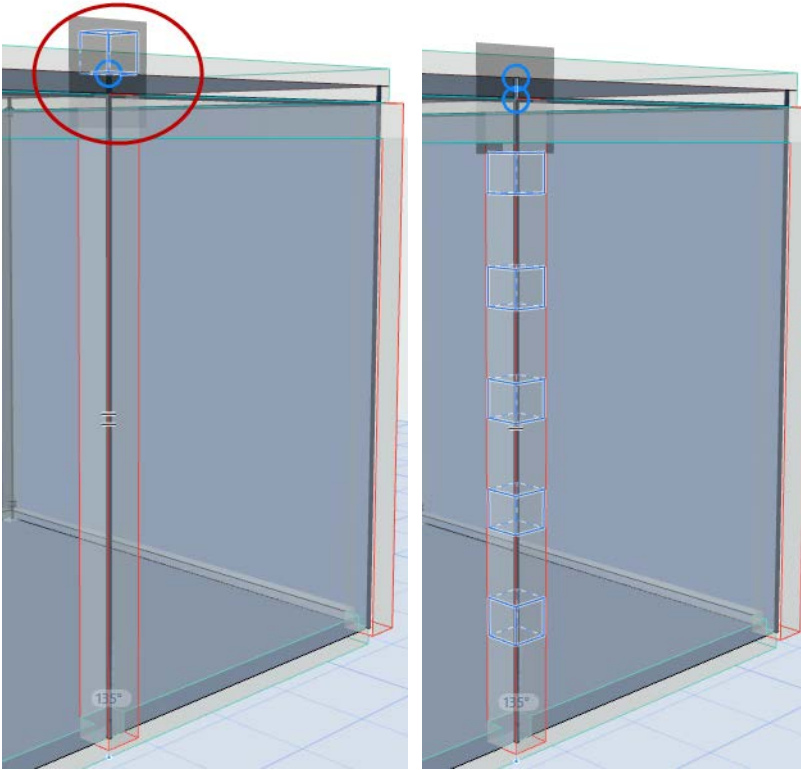
Structural Support Tool

1. Activate the Structural Support tool

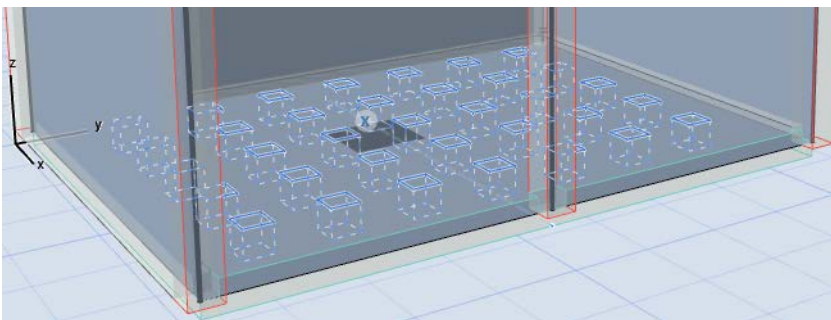
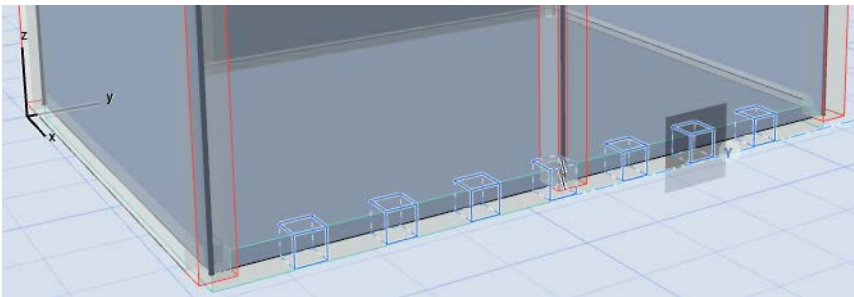


- from the toolbox, or
 - Design > Structural Engineering Tools > Structural Support
2. On Floor Plan or 3D window: hover over the Structural Analytical Member to which you want to add the support.

3. Depending on the Member type (1D or 2D), the graphical feedback indicates the available Support types: Point, Line, and/or Surface.

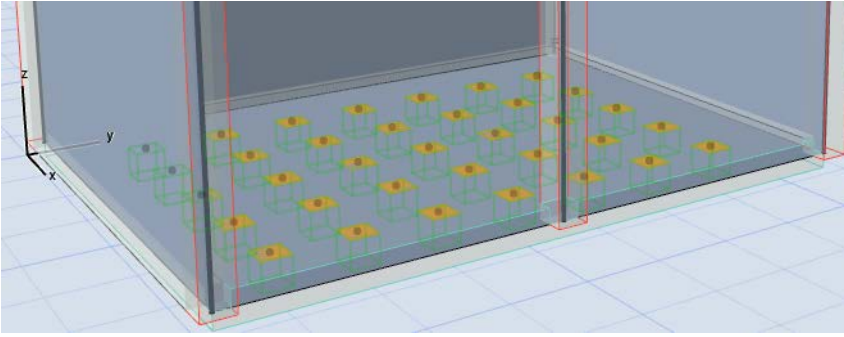


Graphical Preview: Point and Line Support for 1D Member

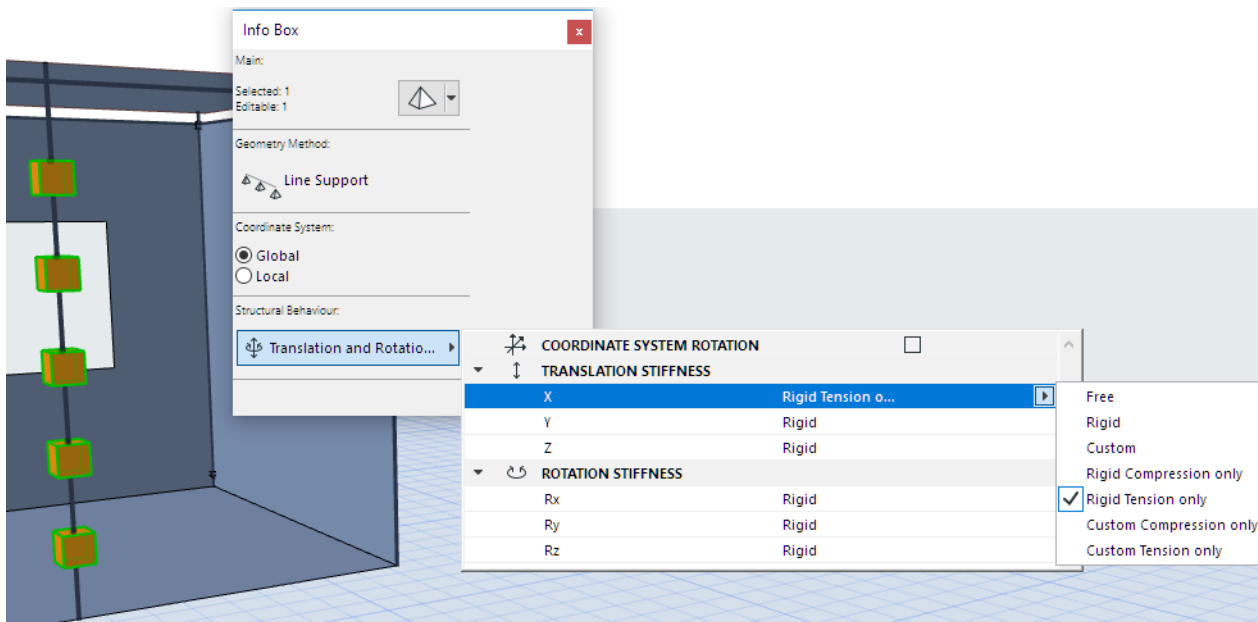


Graphical Preview: Line and Surface Support for 2D Member

4. When you see the Support geometry you need, click to place.



5. Use Support tool settings or the Info Box to set additional options as needed.



Translation/Rotation Stiffness of Line Support Element

Default Settings of Structural Support Tool

When you place a new Structural Support, the element Default Settings reflect the options and working units used by Point-type Supports (some of which are not relevant for other Support geometries). Even so, just set the values needed for your specific Support geometry (e.g. Line or Surface Support).

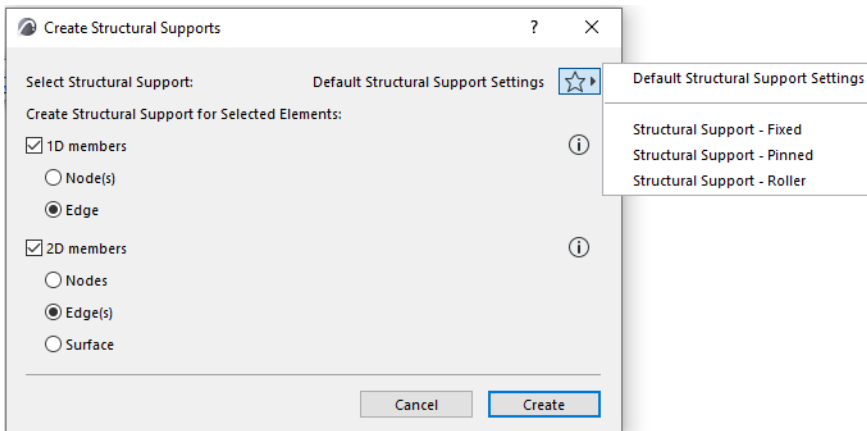
After placement, its Selection Settings will now show the correct and relevant options and working units.

“Create Structural Supports” Command

Use the **Create Structural Supports** command to generate supports on multiple elements. The function works on selected elements, or (if nothing is selected) the entire Structural Analytical Model.

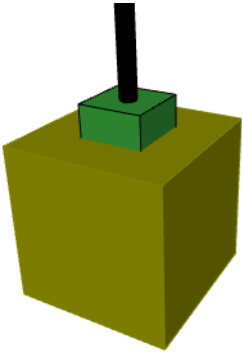
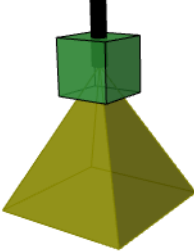
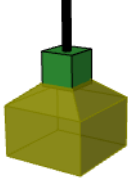


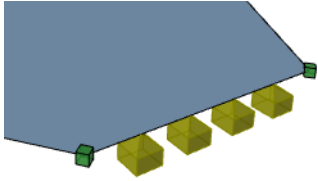
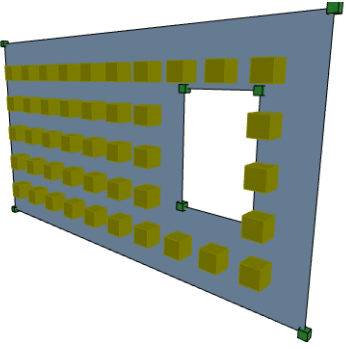
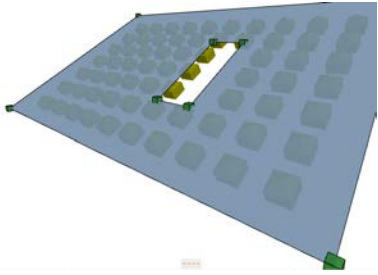
1. Select the elements that need supports.
2. Activate **Design > Structural Analytical Elements > Create Structural Supports**
3. In the appearing dialog, set options for the placement of supports for 1D and/or 2D Members (or choose Favorite settings of the Structural Support tool).

Depending on the selected elements, some options may be greyed out.



4. Click **Create**. The Structural Supports appear on the selected Members.

Support Elements: Symbol Options

	Rigid	Hinge	Custom																																																						
	<table border="1"> <thead> <tr> <th colspan="2">TRANSLATION STIFFNESS</th> </tr> </thead> <tbody> <tr> <td>X</td> <td>Rigid</td> </tr> <tr> <td>Y</td> <td>Rigid</td> </tr> <tr> <td>Z</td> <td>Rigid</td> </tr> </tbody> </table> <table border="1"> <thead> <tr> <th colspan="2">ROTATION STIFFNESS</th> </tr> </thead> <tbody> <tr> <td>Rx</td> <td>Rigid</td> </tr> <tr> <td>Ry</td> <td>Rigid</td> </tr> <tr> <td>Rz</td> <td>Rigid</td> </tr> </tbody> </table>	TRANSLATION STIFFNESS		X	Rigid	Y	Rigid	Z	Rigid	ROTATION STIFFNESS		Rx	Rigid	Ry	Rigid	Rz	Rigid	<table border="1"> <thead> <tr> <th colspan="2">TRANSLATION STIFFNESS</th> </tr> </thead> <tbody> <tr> <td>X</td> <td>Rigid</td> </tr> <tr> <td>Y</td> <td>Rigid</td> </tr> <tr> <td>Z</td> <td>Rigid</td> </tr> </tbody> </table> <table border="1"> <thead> <tr> <th colspan="2">ROTATION STIFFNESS</th> </tr> </thead> <tbody> <tr> <td>Rx</td> <td>Free</td> </tr> <tr> <td>Ry</td> <td>Free</td> </tr> <tr> <td>Rz</td> <td>Free</td> </tr> </tbody> </table>	TRANSLATION STIFFNESS		X	Rigid	Y	Rigid	Z	Rigid	ROTATION STIFFNESS		Rx	Free	Ry	Free	Rz	Free	<table border="1"> <thead> <tr> <th colspan="2">TRANSLATION STIFFNESS</th> </tr> </thead> <tbody> <tr> <td>X</td> <td>Rigid Compression only</td> </tr> <tr> <td>Y</td> <td>Custom Tension only</td> </tr> <tr> <td>Y value</td> <td>1.00 [MN/m]</td> </tr> <tr> <td>Z</td> <td>Custom</td> </tr> <tr> <td>Z value</td> <td>0.50 [MN/m]</td> </tr> </tbody> </table> <table border="1"> <thead> <tr> <th colspan="2">ROTATION STIFFNESS</th> </tr> </thead> <tbody> <tr> <td>Rx</td> <td>Rigid</td> </tr> <tr> <td>Ry</td> <td>Custom</td> </tr> <tr> <td>Ry value</td> <td>1.00 [MNm/rad]</td> </tr> <tr> <td>Rz</td> <td>Free</td> </tr> </tbody> </table>	TRANSLATION STIFFNESS		X	Rigid Compression only	Y	Custom Tension only	Y value	1.00 [MN/m]	Z	Custom	Z value	0.50 [MN/m]	ROTATION STIFFNESS		Rx	Rigid	Ry	Custom	Ry value	1.00 [MNm/rad]	Rz	Free
TRANSLATION STIFFNESS																																																									
X	Rigid																																																								
Y	Rigid																																																								
Z	Rigid																																																								
ROTATION STIFFNESS																																																									
Rx	Rigid																																																								
Ry	Rigid																																																								
Rz	Rigid																																																								
TRANSLATION STIFFNESS																																																									
X	Rigid																																																								
Y	Rigid																																																								
Z	Rigid																																																								
ROTATION STIFFNESS																																																									
Rx	Free																																																								
Ry	Free																																																								
Rz	Free																																																								
TRANSLATION STIFFNESS																																																									
X	Rigid Compression only																																																								
Y	Custom Tension only																																																								
Y value	1.00 [MN/m]																																																								
Z	Custom																																																								
Z value	0.50 [MN/m]																																																								
ROTATION STIFFNESS																																																									
Rx	Rigid																																																								
Ry	Custom																																																								
Ry value	1.00 [MNm/rad]																																																								
Rz	Free																																																								
Point Support																																																									
Line Support																																																									
Surface Support		Not applicable																																																							

Structural Supports at Openings and Holes

Support elements attached to certain holes or openings have specific limitations.

Support placed at	Allowed Support Types	Support is associated to
Opening element	Point Support at nodes only	2D Member
Door-Window-Skylight hole	Point Support at nodes only	2D Member
Solid Element Operation hole	Point Support at nodes only	2D Member
Slab or Roof hole	Point Support at nodes AND Line Support along hole edges	Hole

Structural Links

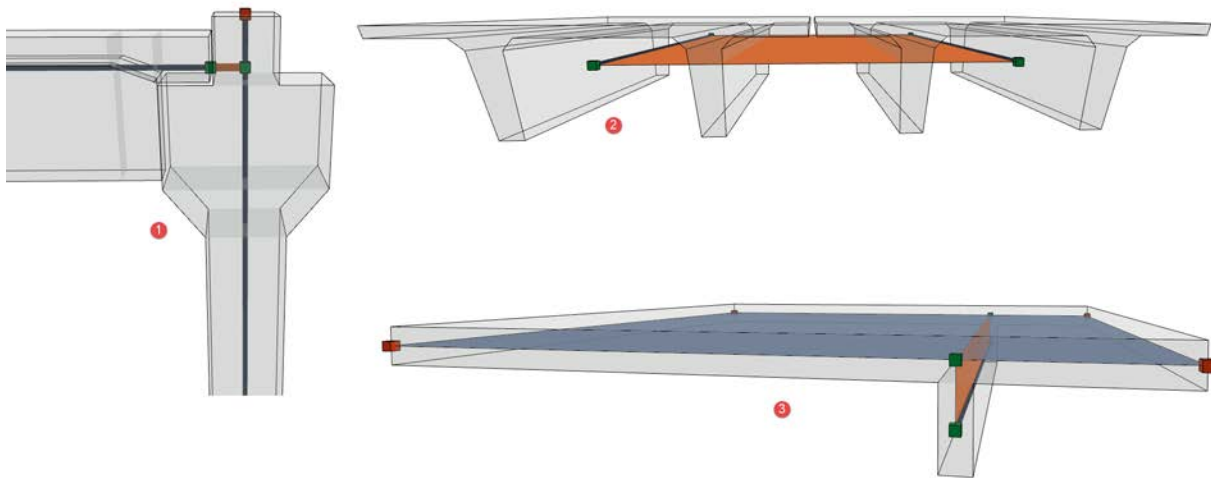
Instead of connecting Structural Analytical Members using Adjustment Rules, it can make more sense to keep the elements separate. (For example, if adjusting 2D Members to each other would move one of the planes outside of its element.)

In such a case, you can use Structural Links instead, to indicate how applied loads are transferred between the Structural Members.

- The Structural Link can be a 1D Link or a Surface Link, depending on the situation.

Note also:

- A 1D Link with zero length can be placed at a point - for example, to link two bracings while keeping them as independent elements.
- A Surface Link having a length dimension but zero width can be placed between two edges.
- They are placed only on 1D and 2D Members of the Structural Analytical Model - you cannot use Structural Links with physical model elements in Archicad.
- A Structural Link necessarily has two “parent” Members. If either Member is not part of the current Structural Analytical Model view (due to Renovation Status), the Link element also disappears.



1: Prefab Reinforced Concrete elements with cantilever connection; 2: Precast panels modeled with Beams; 3: High rib (Offset adjustment not allowed)

There are two ways to create a Structural Link:

- Use the Structural Link tool
- Use the Create Links dialog

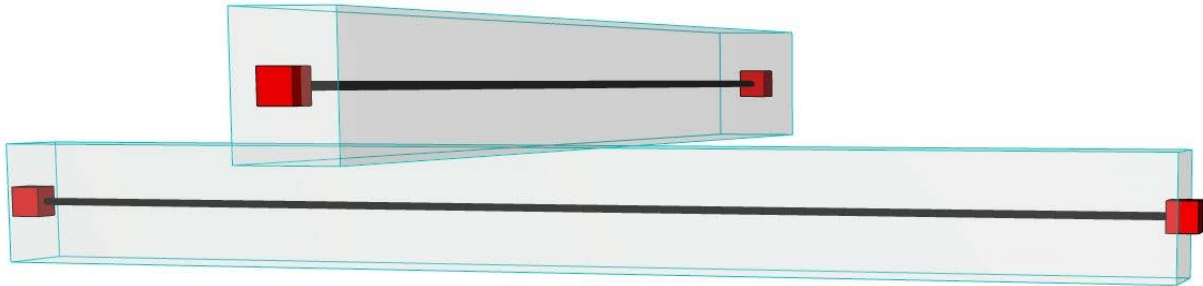
The resulting Structural Links are associated to the connecting Members: if you move the element, the link moves with it.

Customize Structural Link colors at [Structural Analytical Model Representation \(Preferences\)](#).

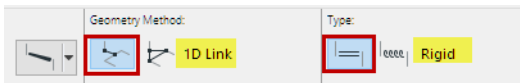
To show or hide these elements in 3D: use View > Elements in 3D View > Filter and Cut Elements in 3D

Example: Use Structural Link Tool

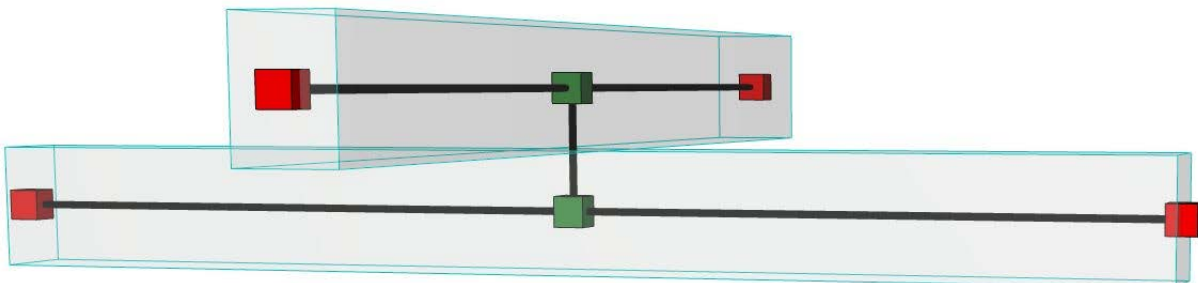
In this example, the user wants to transfer the forces from the upper Beam to the lower one. The two Beams should be “connected” through an analytical link.



1. Activate the Structural Link tool
 - from the Toolbox
 - from **Design > Structural Engineering Tools > Structural Link**
2. From the Info box, choose **1D Link** as the Geometry method.
3. Choose **Rigid** as the link type.



4. Click twice - once on each 1D Member - to create the link between them.



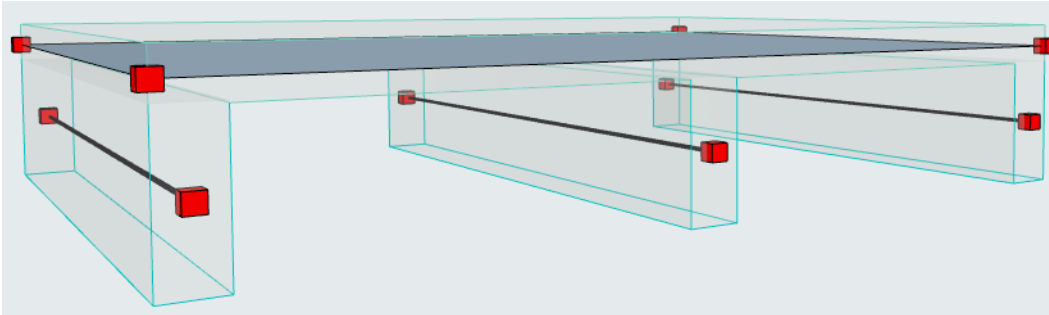
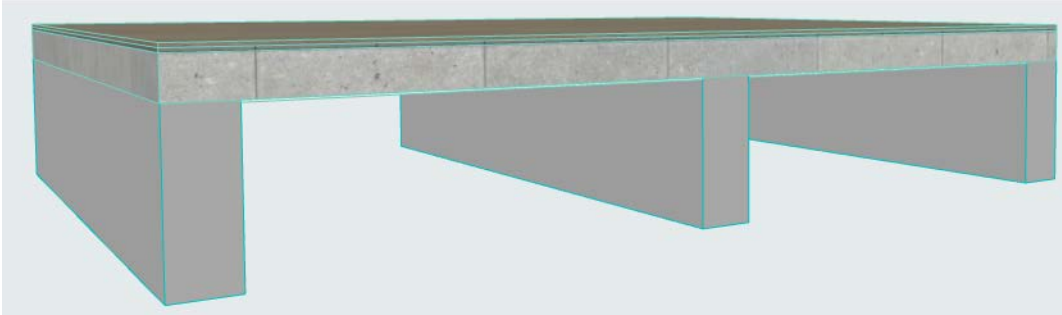
Notes:

- If you move linked elements, the link remains connected to the originally clicked point.
- Use **space + click** to automatically create a Structural Link that is the shortest distance between the two Members. If you later move these linked elements, the associated Structural Link will be adjusted as needed to be the shortest link possible.

Example: “Create Structural Links” Command

Use the **Create Structural Links** function to create several link types between different elements. The function works on selected elements, or (if nothing is selected) the entire Structural Analytical Model.

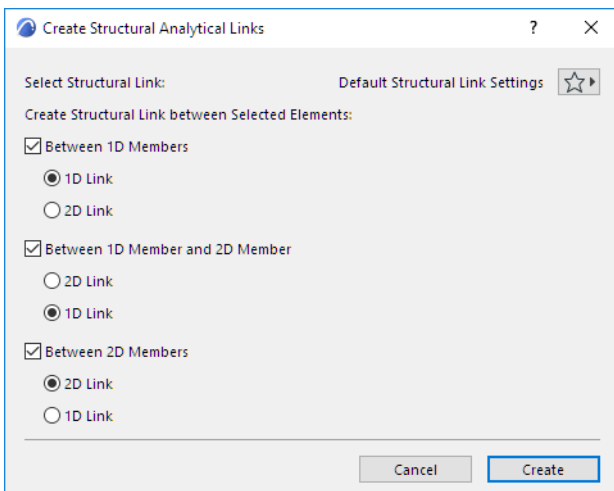
Important: This function affects only elements that are connected at their cores.



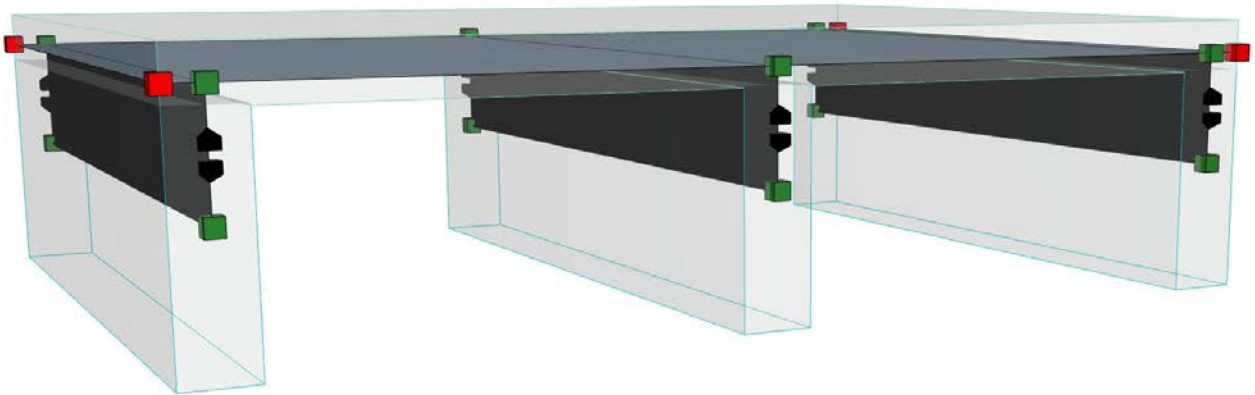
In this example, the Slab is supported by Beams of significant height. It is not structurally realistic to use the offset adjuster in Generation Rules. The Beams should retain their structural axis at their default location at the center of gravity.

We will add Structural Links to indicate load transfer from the Slab to the Beams.

1. Select the elements to be linked.
2. Activate **Design > Structural Analytical Elements > Create Structural Links**.
3. In the appearing dialog, set options:
 - In this case the link should be **Between 1D Member and 2D Member** (Beam and Slab), and the link should be along the full Beam length (**2D Link**).



4. Click **Create**. The 2D Structural Links appear:



2D Structural Links Between 1D (Beam) and 2D (Slab) Members

As expected, the structural axes remain at their original position (center of gravity), but new nodes now appear on the Slab plane, indicating where the structural links are connected to the Slab.

When the Structural Analytical Model is exported, the structural analysis software will interpret that the loads imposed on the Slab will be transferred through the links to the Beams.

Display Surface Link as Transparent

[See *Transparent 2D Members*.](#)

Structural Loads

Loads represent the stresses and tensions which act upon the load-bearing structure, and which must be considered throughout building's life cycle.

These are represented in Archicad's Structural Analytical Model via Load elements, and supported in the SAF-based data exchange workflow. During roundtrips, all Load-related information is retained in the BIM model.

Note: Loads are visible only when the Structural Analytical Model is on. To turn on the Structural Analytical model, use the toggle from the View menu or the Structural Analytical Model Toolbar

You can also place new Load elements (Point, Line, Surface) in Archicad, and define or edit data for Load Cases, Groups and Combinations.

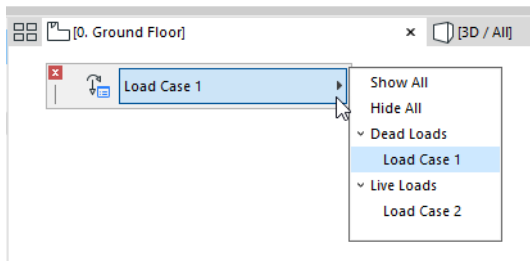
Loads are associative to the Member they are placed on. Any change in the Member geometry or position (including the effects of Generation Rules) also affects the Load element.

Load Cases and Load Groups

A Load Case is a “compatible Load arrangement, set of deformations and imperfections considered simultaneously with fixed variable actions and permanent actions for a particular verification.” (BS EN 1990:2002+A1:2005 - Eurocode. Basis of structural design)

In Archicad:

- When you create a Load element, its Load Case is set in Load Tool Settings.
- The Load Case determines whether a Load element is visible or not (similar to element visibility depending on its Layer status).
 - Set the visible Load Case(s) using the Load Case palette, visible by default in the Structural work environment.



- The visible Load Case can be saved for each view, in View Settings.
- To define and manage the project's Load Cases, go to **Options > Load Manager**.
- Load Cases listed in the Load Manager are organized into **Load Groups**, a category used by many structural analysis programs recognize.

Load Combination

A Load Combination contains the data of multiple Loads which act on a structure simultaneously. Many analysis programs use Load Combination data to calculate the effect of these multiple Loads, using various factors and multipliers.

Load Combinations in Archicad are used for compatibility with Structural Analysis programs. Within Archicad, they function only to store and manage data.

Related Topics

[Load Manager \(for Load Cases and Load Groups\)](#)

[Load Case Settings](#)

[Load Group Settings](#)

[Load Combination Dialog Box](#)

[Place Loads](#)

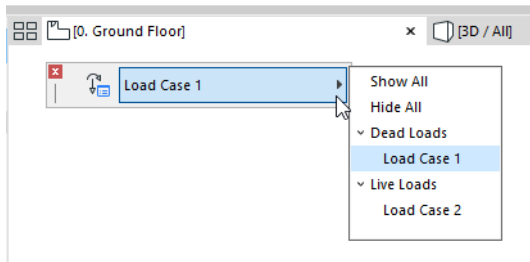
[Create/Update Live Loads](#)

[Load Tool Settings](#)

[Representation of Loads](#)

[Loads in SAF Import/Export](#)

Load Case Palette



Load Case Palette

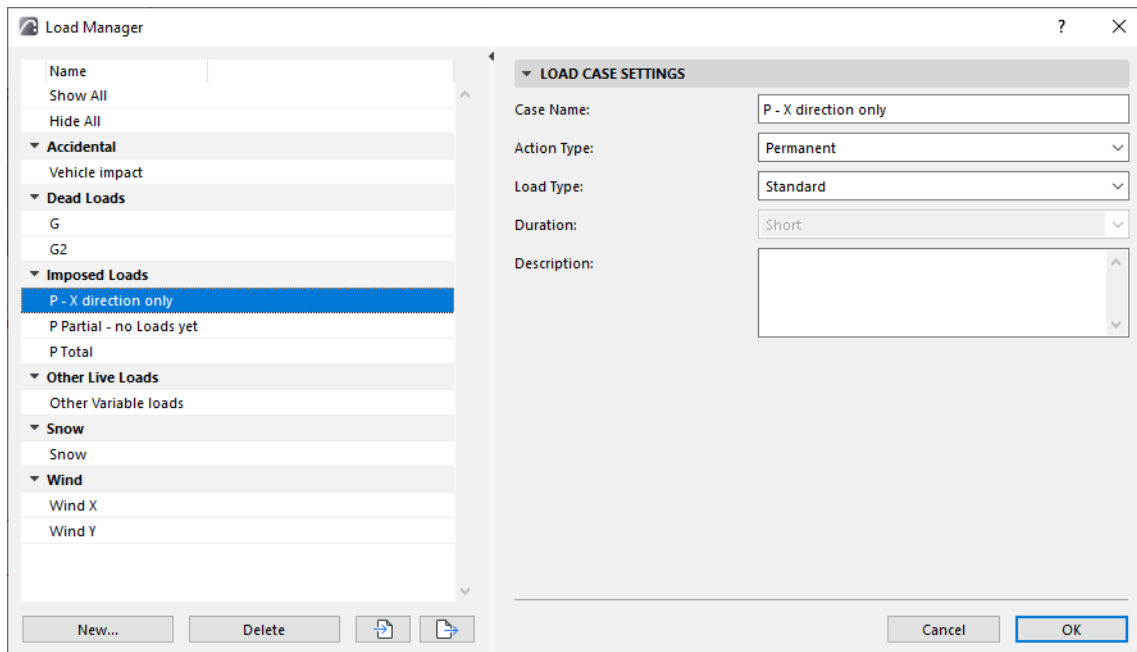
- Load Case Palette appears by default in “Structural” work environments
- Set the visible Load Case easily
- Show or hide any or all Load Cases at once

Load Manager (for Load Cases and Load Groups)



Load Manager lists all of the project's **Load Cases**, organized into **Load Groups**.

From Load Manager, you can also import and export Load Cases and Load Groups.



Load Manager, with Selected Load Case

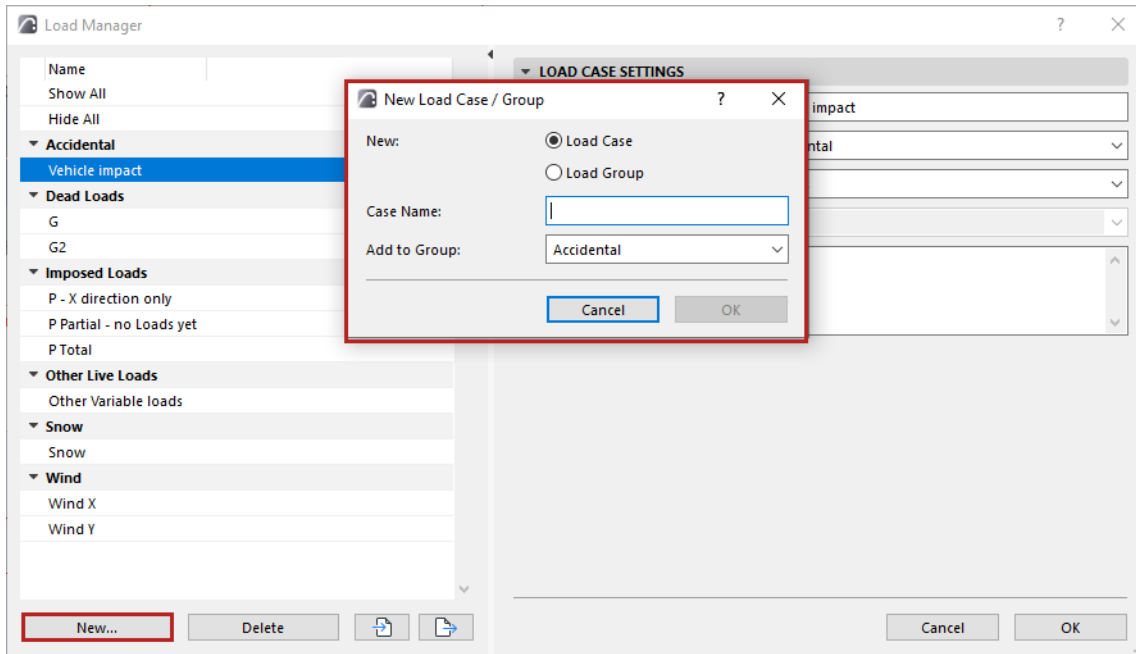
Open Load Manager

Do one of the following:

- Options > Load Manager
- Structural Engineering toolbar
- Use the **Edit Loads** button in the Load Combination dialog

Create Load Group or Load Case

1. Click New
2. From the dialog, select Load Case or Load Group.



For a new Load Case:

1. Add to Group: Choose a Load Group for the new Load Case, then click OK
2. The new Load Case is listed on the left side of the dialog.
3. Select the Load Case to access its settings on the right.

For a new Load Group:

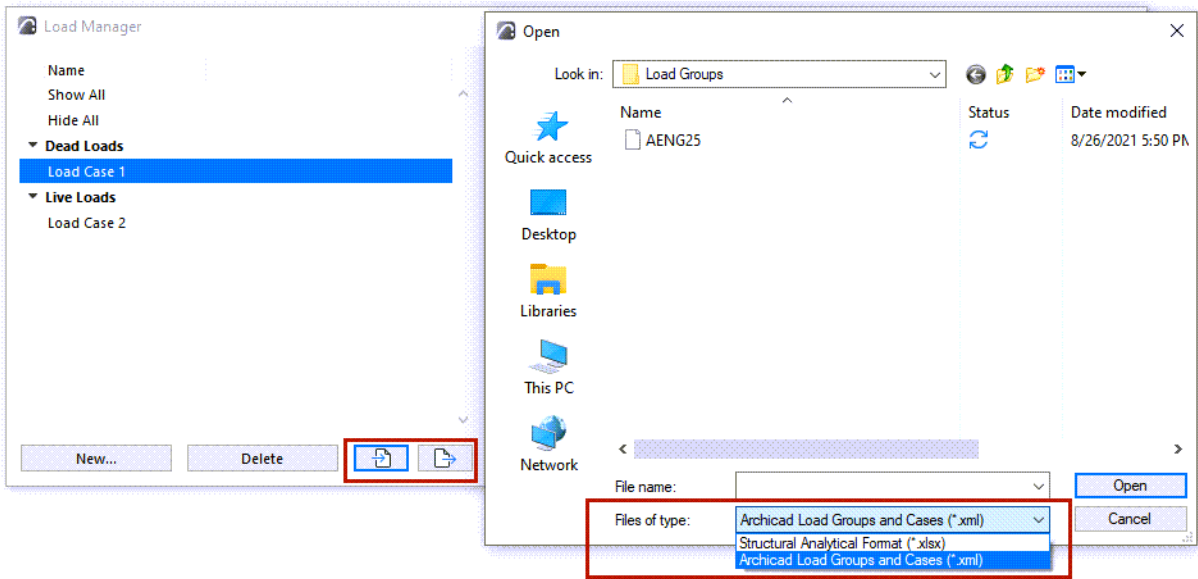
1. Add a Group Name, then click OK
2. The new Load Group is listed on the left side of the dialog.
3. Select the Load Group to access its settings on the right.

Click OK in the Load Manager dialog to save your changes.

Import and Export Load Cases and Load Groups

In Load Manager, use the Import-Export buttons to transfer Load Group and Load Case data.

- Archicad imports these data in either XML or SAF format.
- Archicad exports these data only in XML format



Related Topics

[Load Case Settings](#)

[Load Group Settings](#)

Load Case Settings

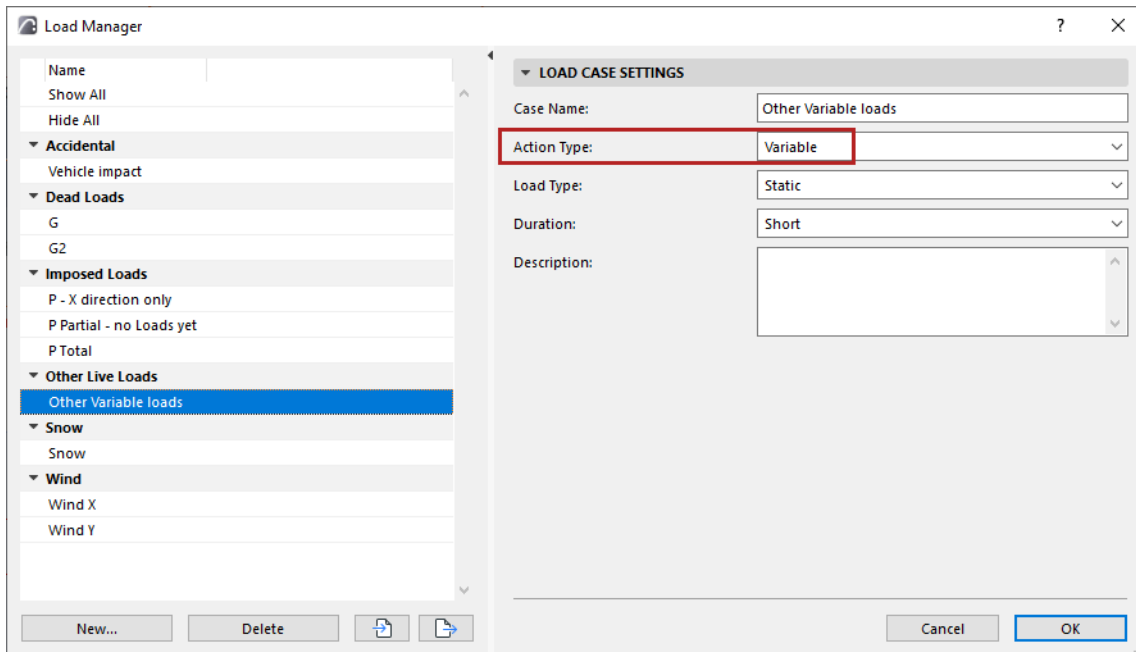
In Load Manager, select a Load Case to access its settings on the right.

Note: The selected Load Case becomes the visible Load Case for the current view.

Case Name

Each Load Case must have a unique name.

Action Type



Permanent

The Load is constantly present over the life of the building. Load size does not vary significantly over time.

For example: self-load of load-bearing element.

Variable

Load size varies significantly over time

For example: meteorological Loads (like snow)

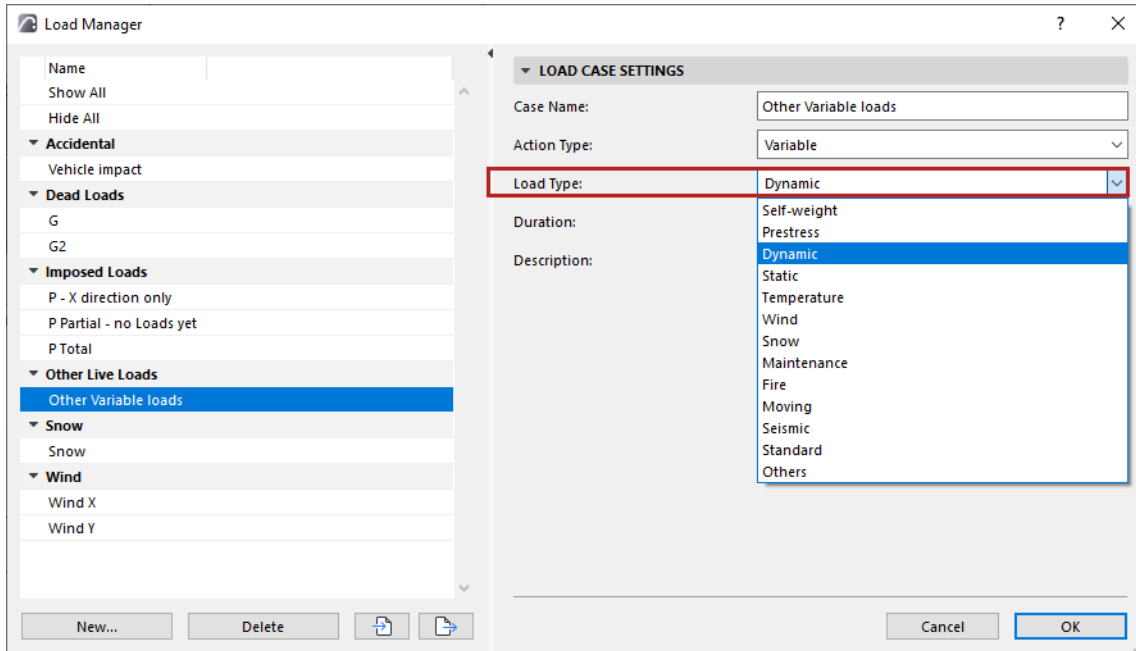
Accidental

Short-term Load with significant effect

Unlikely to occur but cannot be excluded with complete certainty

For example: Collision, fire, explosion

Load Type for a Load Case



Choose a Load Type from the list to characterize the Load.

Concepts vary depending on standard definitions.

Duration

For Loads whose Load Type is Variable: Expresses duration of Load effect

In SAF: if Action Type = Variable, then the Duration value is required

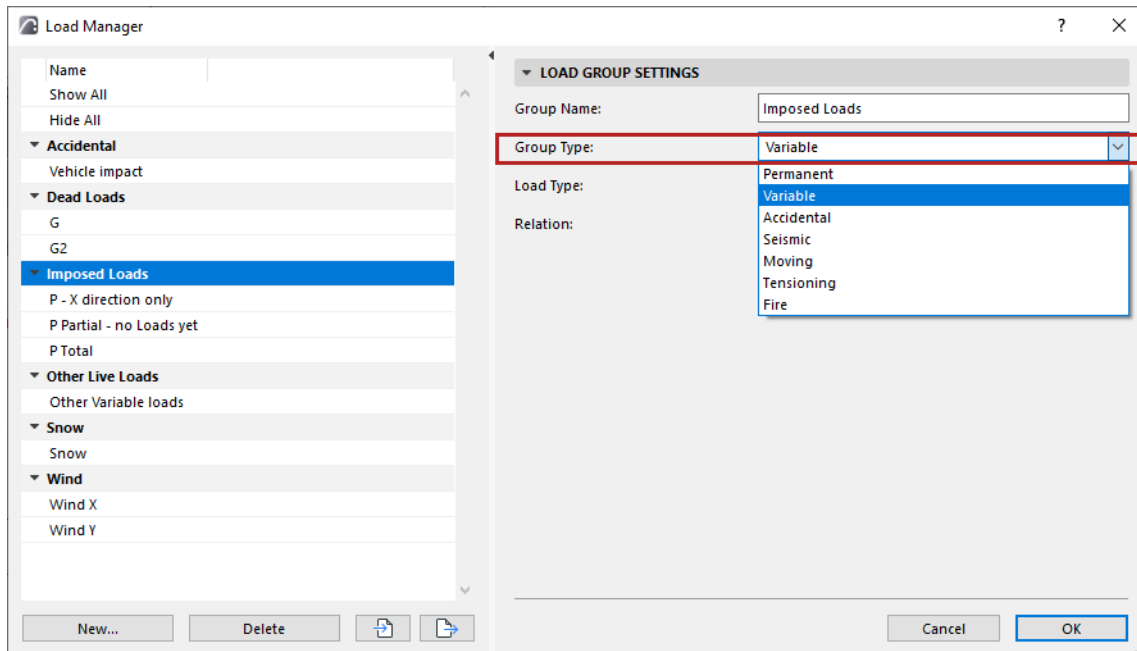
Load Group Settings

1. Open Load Manager.
2. Select a Load Group from the list on the left.
3. Adjust its Load Group Settings on the right.

Group Name

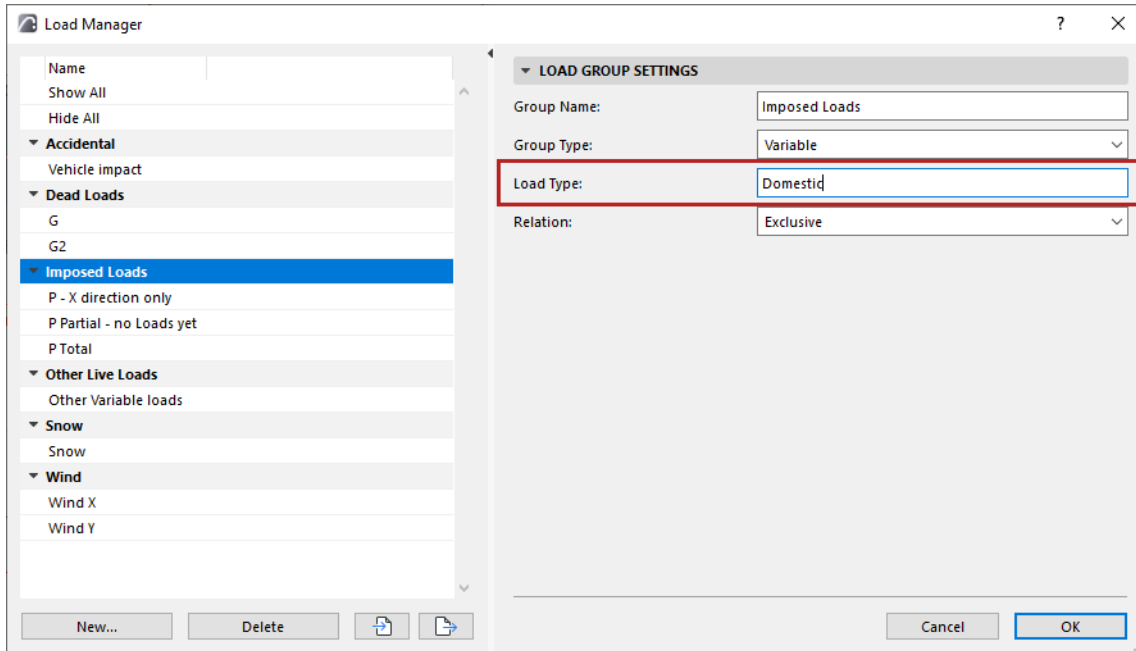
Each Load Group must have a unique name.

Group Type



- Permanent
- Variable
- Accidental
- Seismic
- Moving
- Tensioning
- Fire

Load Type for a Load Group



Optionally, enter text to describe the Load Type.

Relation

- Exclusive
- Standard
- Together

Load Combination Dialog Box

A Load Combination contains the data of multiple Loads which act on a structure simultaneously. Many analysis programs use Load Combination data to calculate the effect of these multiple Loads, using various factors and multipliers.

Load Combinations in Archicad are used for compatibility with these analysis programs. Within Archicad, they function only to store and manage data (not for calculation purposes).

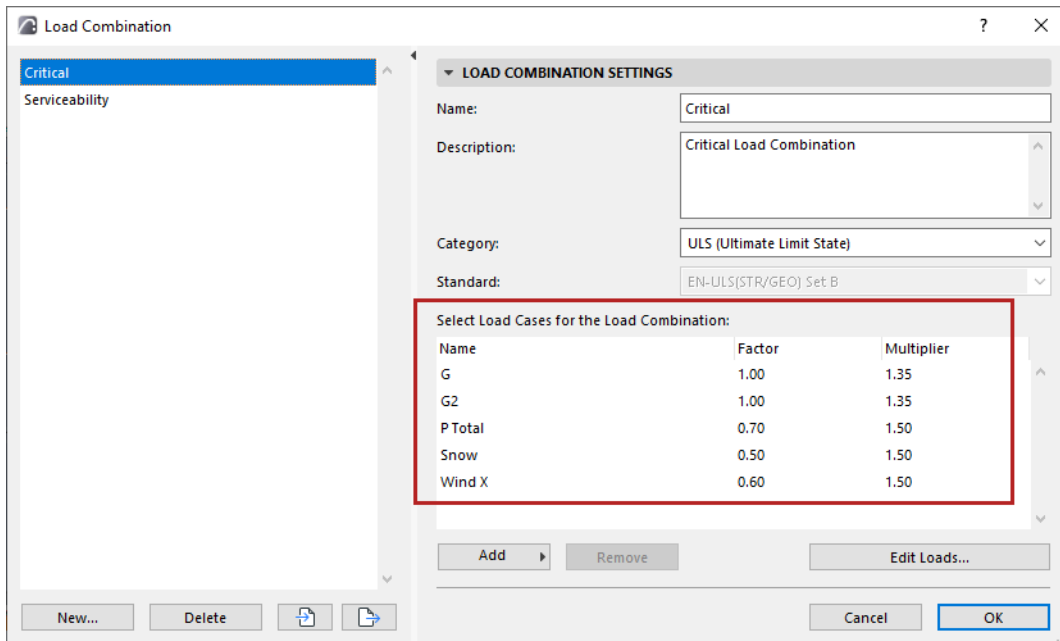


Open the Load Combination dialog by doing one of the following:

- Options > Load Combinations
- Structural Loads toolbar

The project's Load Combinations are listed on the left.

Each selected Load Combination's settings are shown on the right, and its Load Cases are listed with Factor and Multiplier data.



Load Combination Settings

Name

Each Load Combination must have a unique name.

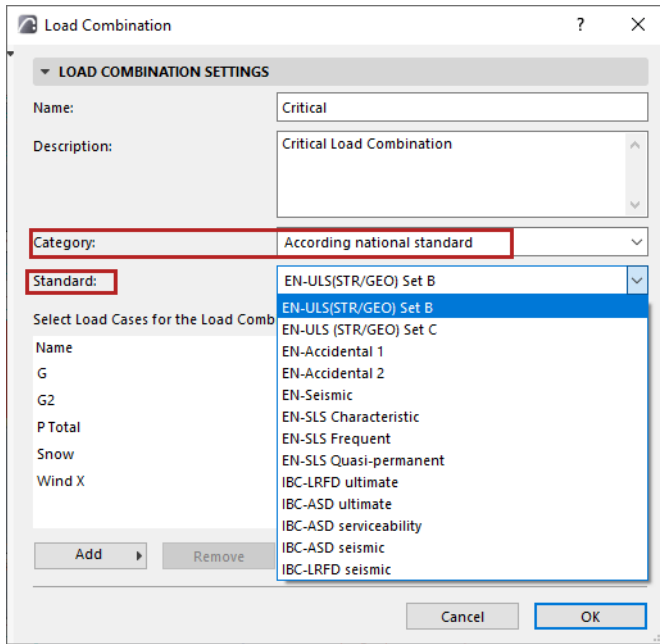
Description

Optionally, enter any further information here to guide the user

Category

Choose from the pop-up:

- a **Limit State** standard, or
- According to **National Standard**. In this case: choose from the **Standard** pop-up list.

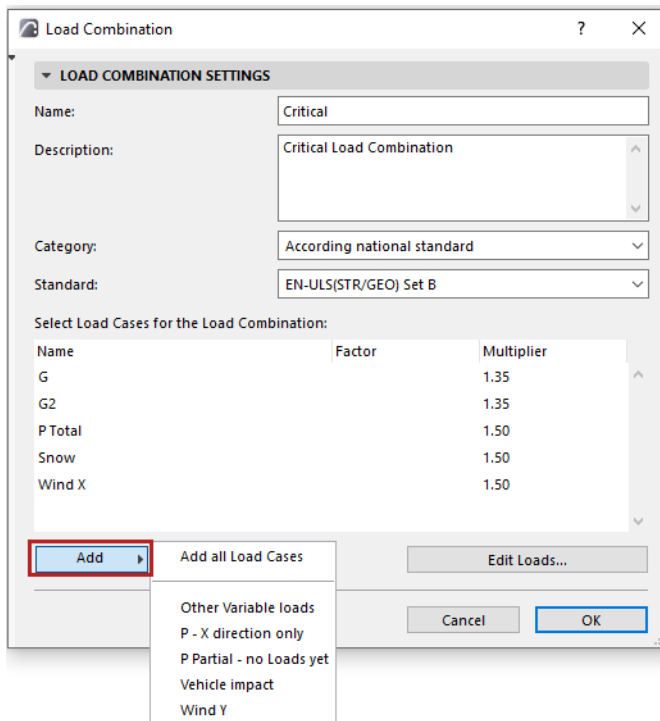


Add Load Case

Any Load Case in the project can be included in a Load Combination (regardless of its Load Group)

At the bottom of the Load Combination dialog, click **Add**:

- Choose a Load Case to add it to this Load Combination
- or **Add all Load Cases**

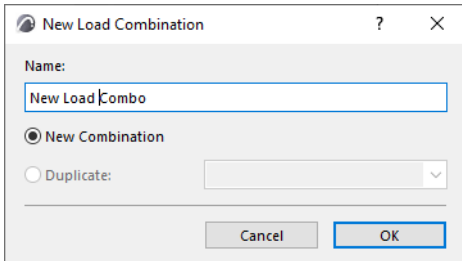


Edit Loads...

Click **Edit Loads** to open the Load Manager.

Create New or Duplicate Load Combination

1. Click **New** at the bottom of the Load Combinations dialog box.
2. The New Load Combination dialog appears: choose either New or Duplicate.
3. Give the new/duplicate Load Combination a **Name**, then click **OK**.



4. Adjust the Load Combination Settings, then click **OK** to close the Load Combinations dialog.

Import and Export Load Combinations

In the Load Combinations dialog, use the Import-Export buttons to transfer Load Combination data.

- Archicad imports these data in either XML or SAF format.
- Archicad exports these data only in XML format

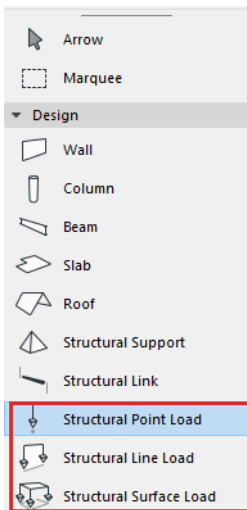
Place Loads

Loads can be placed on the Structural Member of a load-bearing element, in Floor Plan or 3D view. Therefore, the Structural Analytical Model must be enabled.

To place a Load, use one of the three Load tools:

- Structural Point Load
- Structural Line Load
- Structural Surface Load

To activate a tool, select it in the toolbox.



Related Topics

[Place Point Load](#)

[Place Line Load](#)

[Place Surface Load](#)

Place Point Load

Point Loads can be input on:

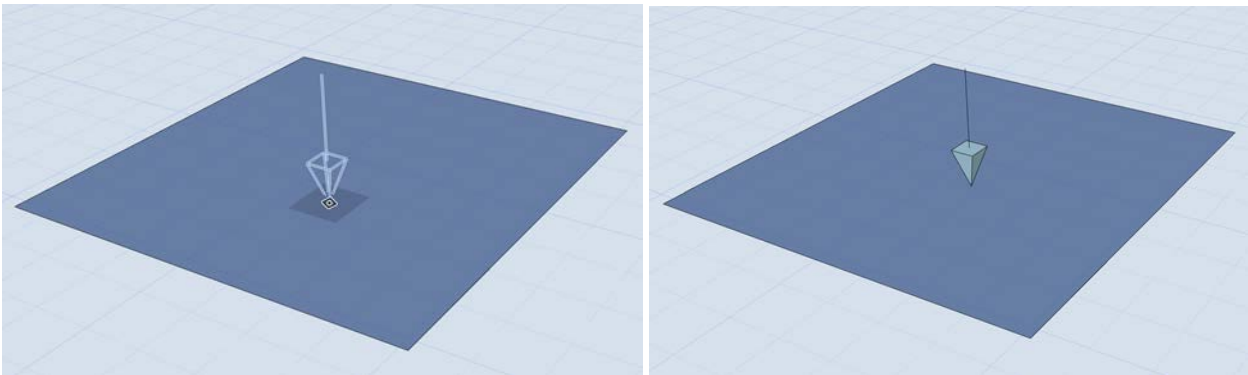
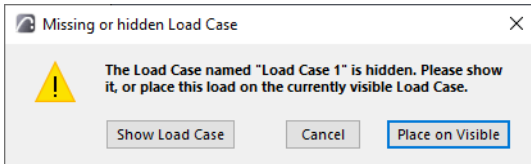
- 1D Members (node or edge)
- 2D Members (node, edge or surface)

1. Activate the Structural Point Load tool.
2. In Tool Settings or Info Box: Select a Load Case ([see Load Tool Settings](#)).
3. While you place the Load element, the potential host element (Structural Analytical Member) is highlighted. You also get a preview of what the Load will look like after placement.

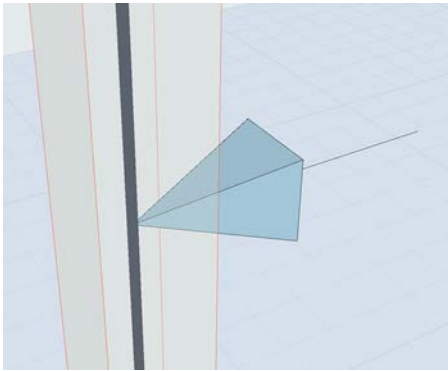
Note: If you are hovering over multiple host elements, use tab to cycle among them until the right one is highlighted.

4. Click to place.

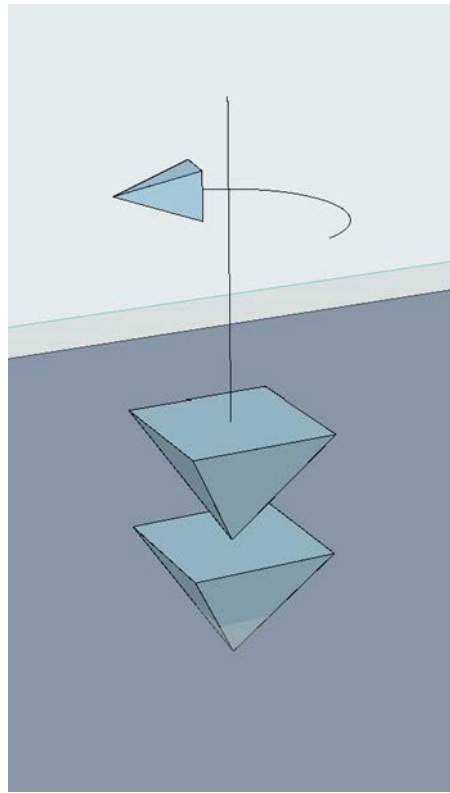
Note: Loads are visible only if they belong to the currently visible Load Case. If you try to place a Load element that does not belong to the currently visible Load Case, you get a notification.



Placing a Point Load



Point Force

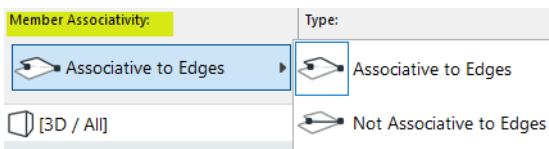


Point Moment

Place Line Load

Line Loads can be input on 1D or 2D Members.

1. Activate the Line Load tool.
2. In Tool Settings or Info Box: Select a Load Case for your new Load element ([see Load Tool Settings](#)).
3. In the Info Box: Choose whether the Load should be associative to the edges or not. (This is relevant if you are placing the load along a 2D Member edge. See [Line Load: Edge Associativity](#).)

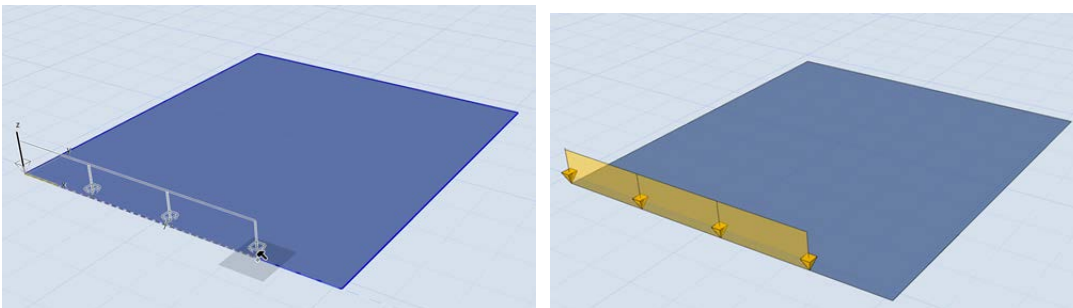


4. As you move the cursor, the potential host element (Structural Analytical Member) is highlighted. You also get a preview of what the Load will look like after placement.

If you are hovering over multiple host elements, use Tab to cycle among them until the right one is highlighted.

5. Click on a Member to specify the start point.
6. Drag the cursor, then click to place the Line Load.

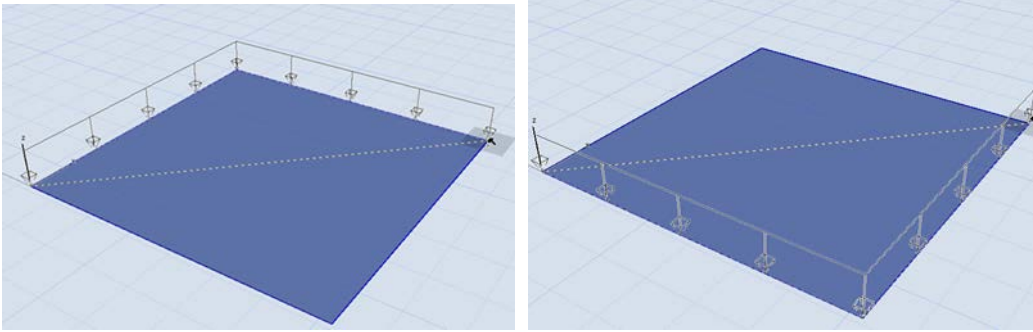
Note: Loads are visible only if they belong to the currently visible Load Case. If you try to place a Load element that does not belong to the currently visible Load Case, you get a notification.



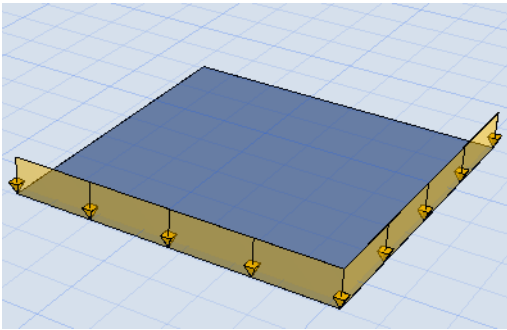
Placing a Line Load

Place Chain of Line Loads on 2D Member Polygon

1. Make sure the Line Load is set to “Associative to Edges.”
2. Click one corner of the polygon, then hover over an opposite corner.
3. Click Tab to see previews of alternative Line Load paths, then click to place.



Associative Line Load around Slab edge: click Tab to see alternatives



Line Loads on All Edges (Shortcut)

Use the Magic Wand (space + click) to create Line Loads on all 2D Member edges.

Line Load: Edge Associativity

Before placing a Line Load, choose its Edge associativity setting in the Info Box. This is relevant for Line Loads on the edge of a 2D Member. (It does not affect Line Loads that do not coincide with a Member edge.)

When you edit the Member edge:

- Associative Line Loads will follow any changes in edge geometry.
- Non-associative Line Loads will *not* follow changes in edge geometry (but when the Member is dragged, the Line Loads move too).

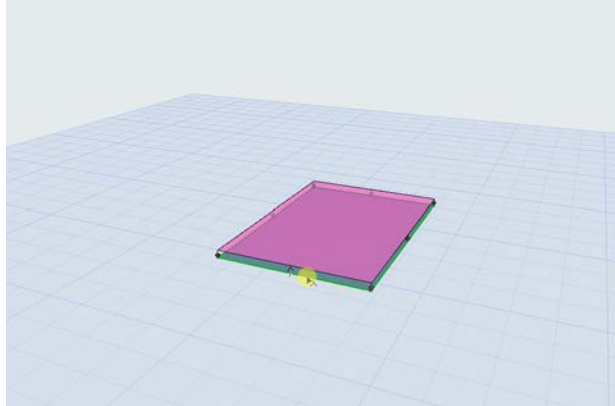
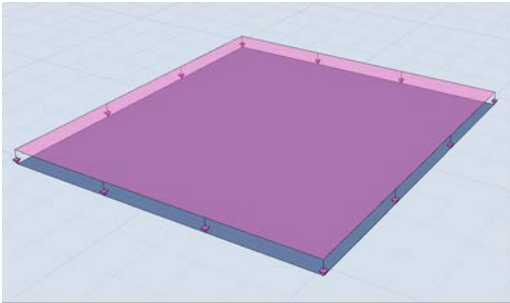
If you place a Line Load on the edge of an *Opening* element (e.g. within a Slab), the Load cannot be associative.

In contrast, a Line Load on the edge of a *hole* can be either associative or non-associative.

Place Surface Load

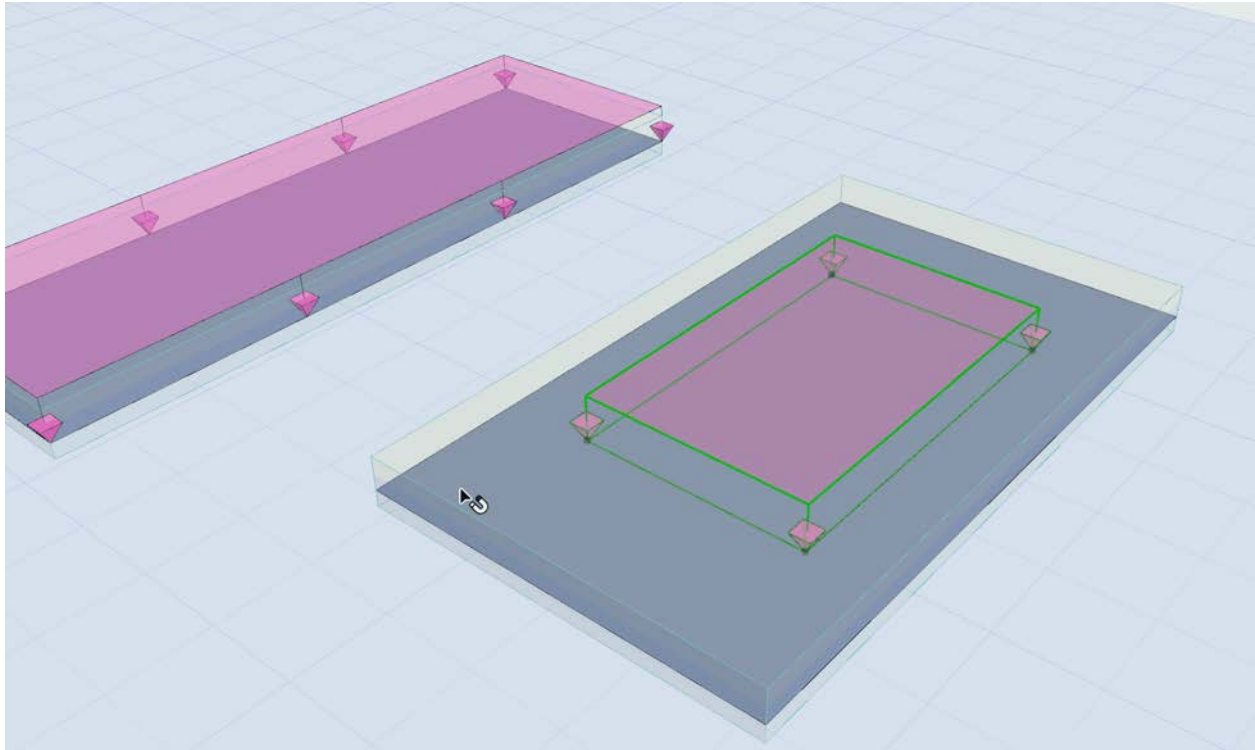
Surface Loads can be placed over the whole 2D Member surface, or any part of it.

1. Activate the Structural Surface Load tool
2. In Tool Settings or Info Box: Select a Load Case for your new Load element ([see Load Tool Settings](#))
3. Choose a geometry method:
 - **Entire Member:** A single click places a Surface Load over the entire 2D Member. This Surface Load is associative to the Member edges.



Associative Surface Load (“Entire Member” geometry method)

- **Rectangle:** Create a rectangle Load by defining two corners with two clicks, over any part of the 2D Member. (For example, to represent the load of MEP machinery.) This Surface Load is *not* associative to Member edges. It moves with the Member during a drag, but will not follow any changes in the edge geometry.



Rectangle Surface Load: Non-associative

Note: Loads are visible only if they belong to the currently visible Load Case. If you try to place a Load element that does not belong to the currently visible Load Case, you get a notification.

You can also generate live loads automatically on zones.

See [Create/Update Live Loads](#).

Create/Update Live Loads

With this feature, structural engineers can automatically create live loads on zones.

How it works

- Map your zones (via their Classification) to the appropriate Load Category
- Load Categories determine the live load values. Archicad comes with default Load Category definitions, which you can customize as needed.
- Use **Design > Structural Analytical Elements > Create/Update Loads**

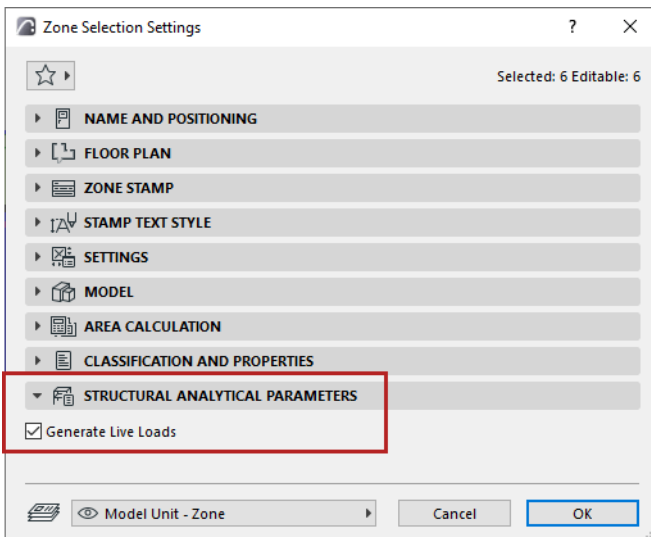
Which zones create live loads?

Live loads are created for zones which meet the following conditions:

- “Generate Live Loads” checkbox is activated (Zone Settings)
- the zone has a Classification (Zone Settings)
- Zone Classification is mapped to a Load Category (Load Mapping dialog)

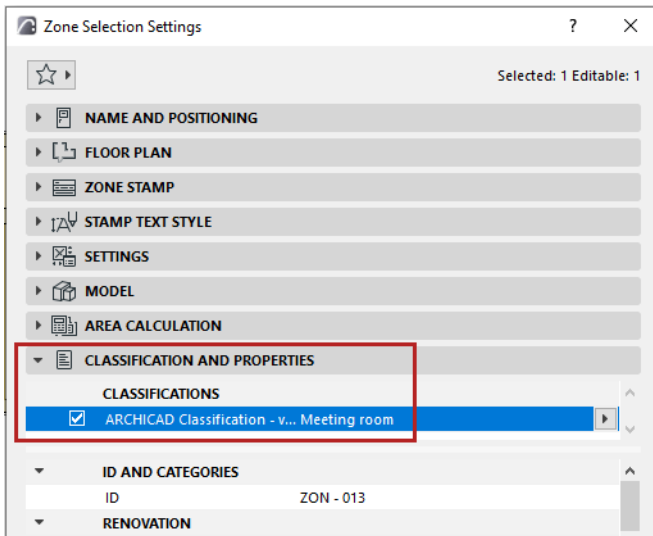
Activate checkbox: Generate Live Loads

Activate this in Zone Settings or the Info Box (Structural Analytical Parameters panel):



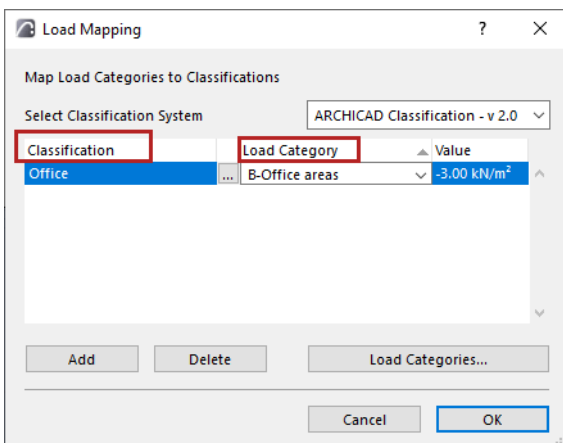
Classify zone

Classify a zone in Zone Settings or Info Box (Classification and Properties panel):

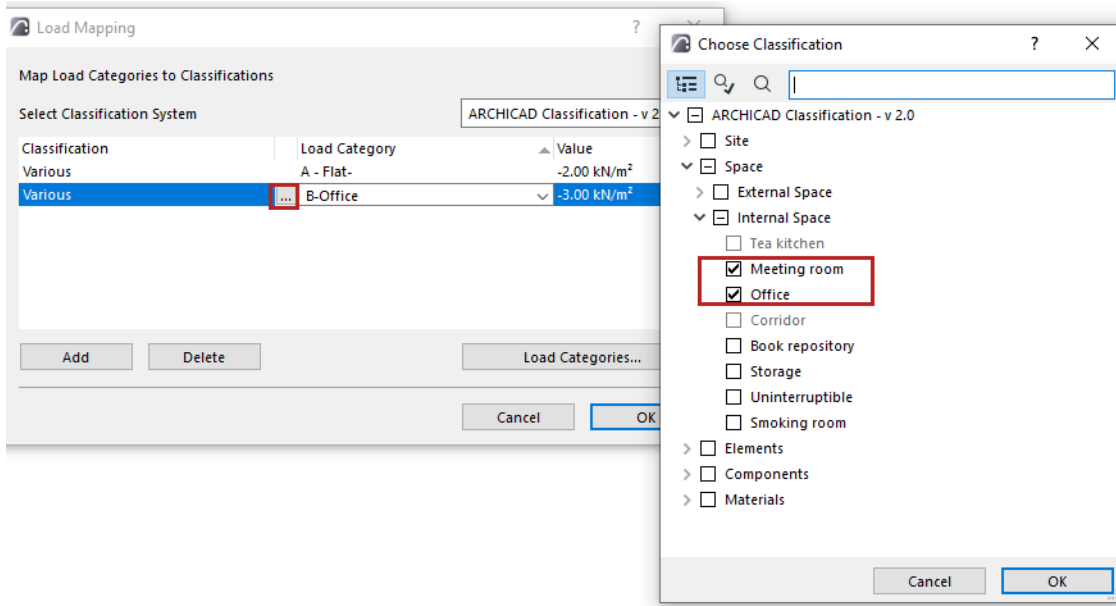


Set up Load Mapping

1. Open **Options > Load Mapping**
2. Use this table to map Classifications (left column) to Load Categories.



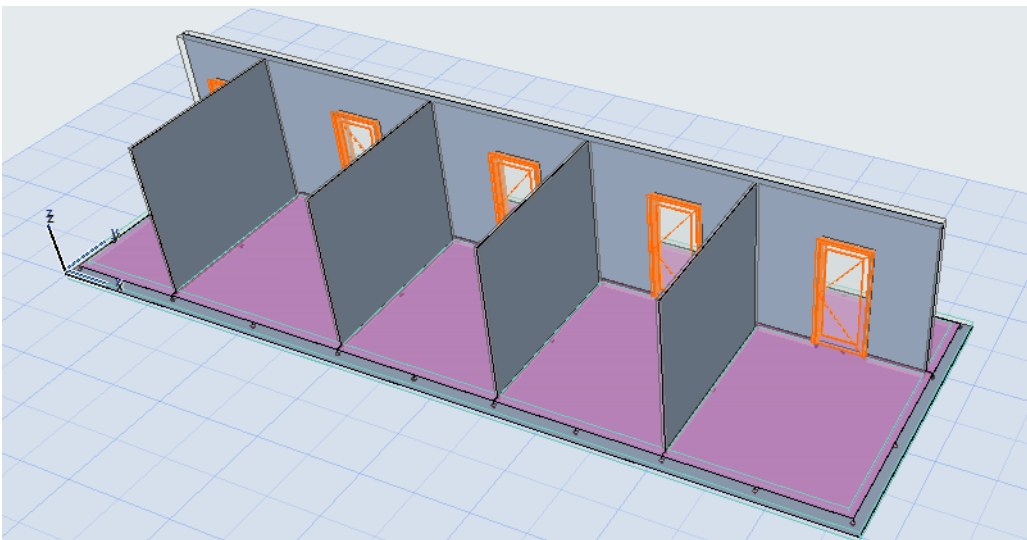
- Click Add to create as many mapping rules as needed
- This dialog displays the Load Category value (To edit this value, use the Load Categories dialog)
- You can map one or more zone classifications to any single Load Category: here, the “Meeting room” and “Office” classifications are mapped to a single Load Category.



Create Live Loads Command

1. Make sure the Structural Analytical Model is on: use the **Show Structural Analytical Model** toggle
 - from the **View** menu
 - from the Structural Analytical Model Toolbar
2. Use **Design > Structural Analytical Elements > Create/Update Loads**
3. Check the Live Loads box.
4. Choose a Load Case for the live load.
5. Click OK.

Check the resulting surface loads in 3D.



Update Live Loads

When the model changes, you can update live loads the same way:

Use **Design > Structural Analytical Elements > Create/Update Loads**

New live loads are created, updated or deleted as needed, to reflect changes in the model (e.g. zone geometry, classification) or in load mapping.

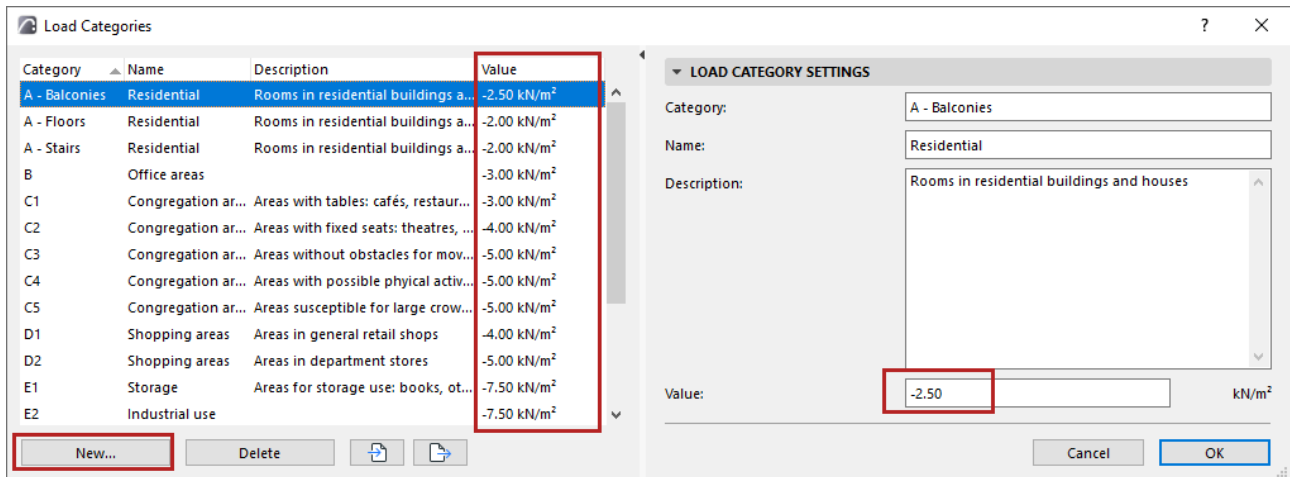
Related topics

[Set up Load Categories](#)

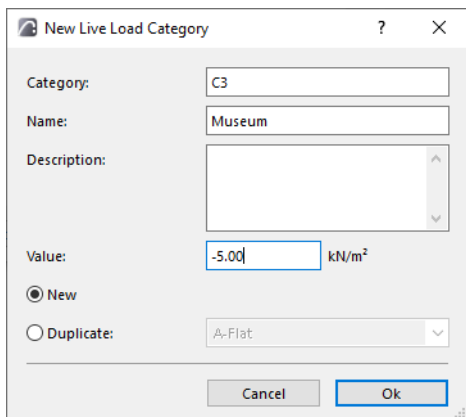
Set up Load Categories

Use Load Categories in the workflow that automatically generates live loads on zones, based on their classification. Load Categories determine the live load values.

1. Open the dialog from **Options > Load Categories**
2. For each Category, review and set live load values as needed.
 - The unit for the value (e.g. kN/m²) comes from the Surface Force setting at **Project Preferences > Structural Load Units**



3. Click **New** to create Load Categories. You can also duplicate and edit an existing one.



- Fill in Category, Name, Description, and Value.
- The “Category” text must be unique (you can’t repeat it in the list)

Import/export Load Category values

Use the buttons at the bottom of the Load Categories dialog to export or import Load Categories via an XML file.

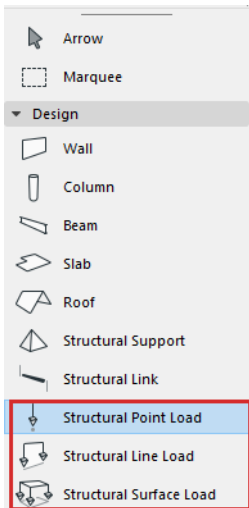
Related topics

[Create/Update Live Loads](#)

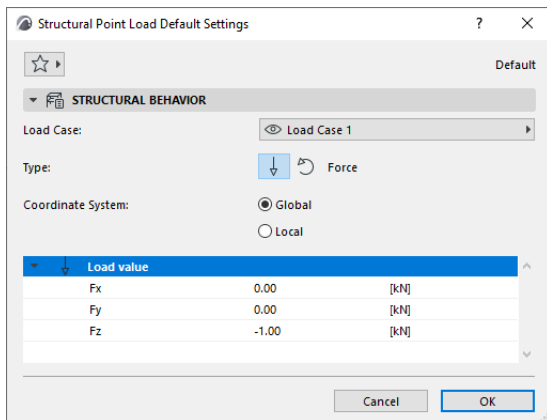
Load Tool Settings

There are three Load tools in Archicad:

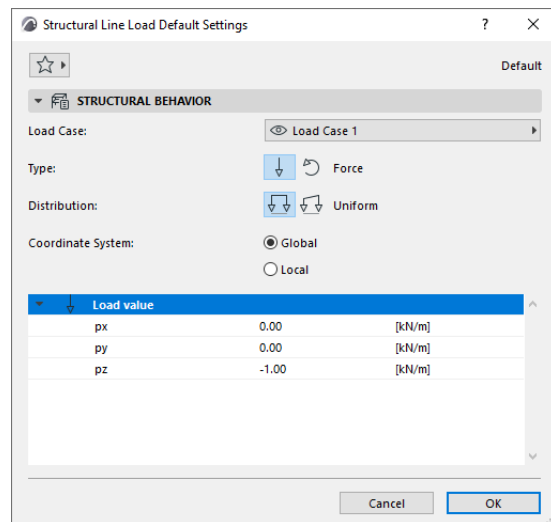
- Structural Point Load
- Structural Line Load
- Structural Surface Load



To open its Settings, double-click the tool in the toolbox.



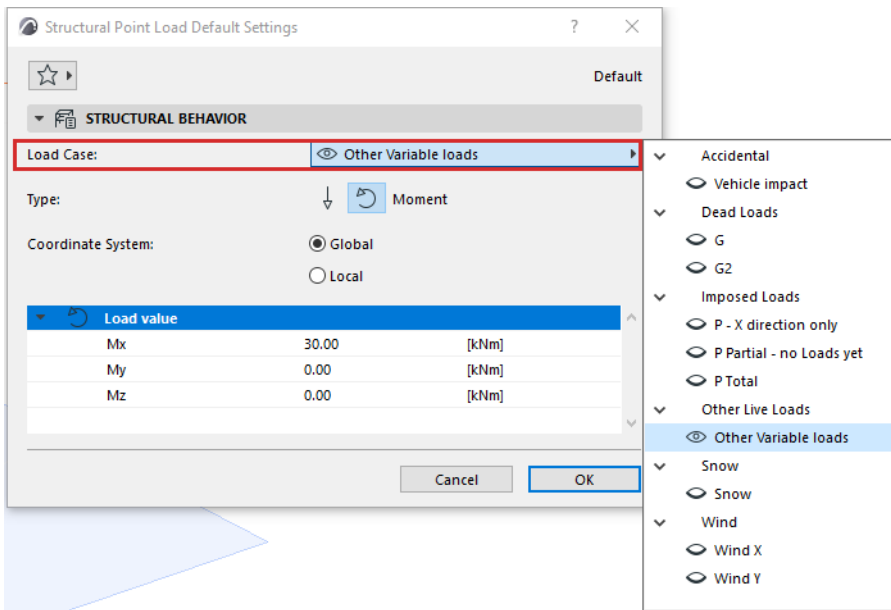
Point Load Settings



Line Load Settings

Load Case

Choose a Load Case for this Load element.



The Load Case determines whether a Load element is visible or not (similar to element visibility depending on its Layer status).

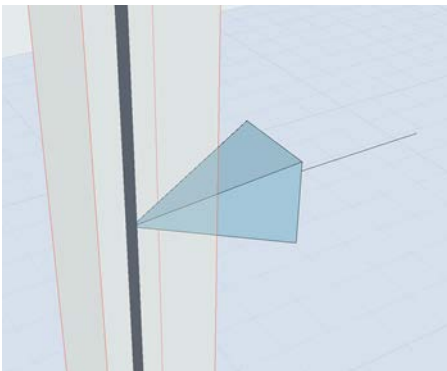
Notes:

- Set the visible Load Case(s) using the Load Case palette, displayed by default in the Structural work environment.
- The visible Load Case can be saved for each view, in View Settings.

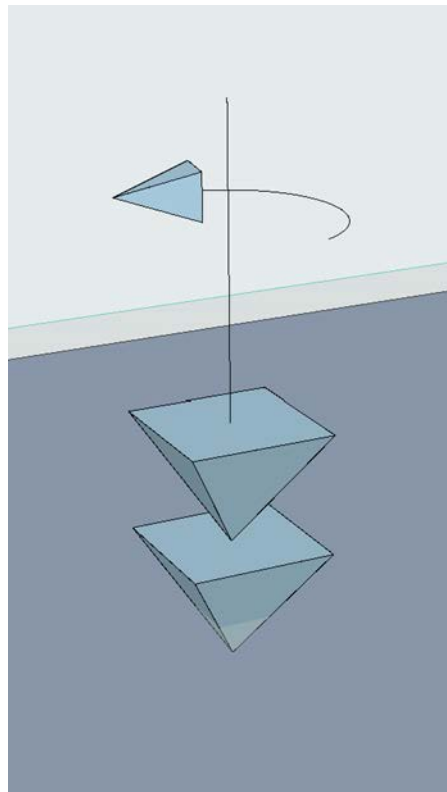
Type

Choose Force or Moment to describe the Load's behavior.

Moment is indicated with a double arrowhead and a curved arrow.



Point Force



Point Moment

Distribution (Line Loads only)

Describes Load distribution along its length.

- Uniform: Load is distributed uniformly along the edge. Load values are identical at both ends.
- Trapezoid: Load values are different at the two ends. Load intensity varies linearly between the two points.

Coordinate System

Defines how to interpret the vector direction of the Load Value.

- Global (Coordinate System): the project's global coordinate system
- Local (Coordinate System): the direction defined for the Load's parent element

Load Value

For each Load, specify its intensity in each direction

Point Load Value	Description
Point Force	
Fx, Fy, Fz	The magnitude of Force applied in the X, Y and Z direction.
Point Moment	
Mx, My, Mz	The magnitude of Moment applied about the X, Y and Z direction.
Line Load Value	Description
Line Force	

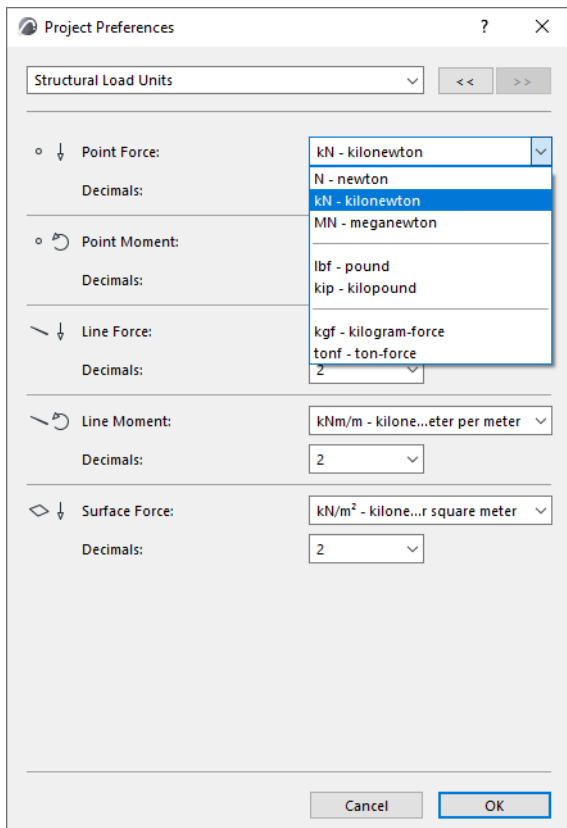
Line Load Value	Description
px, py, pz	The magnitude of Force applied in the X, Y or Z direction at all points along Line Load
px1, py1, pz1	The magnitude of Force applied in the X, Y or Z direction at the startpoint of the Line Load
px2, py2, pz2	The magnitude of Force applied in the X, Y or Z direction at the endpoint of the Line Load

Line Moment	
Mx, My, Mz	The magnitude of Moment applied in the X, Y or Z direction at all points along the Line Load
Mx1, My1, Mz1	The magnitude of Moment applied in the X, Y or Z direction at the startpoint of the Line Load
Mx2, My2, Mz2	The magnitude of Moment applied in the X, Y or Z direction at the endpoint of the Line Load

Surface Force Value	Description
px, py, pz	Magnitude of Force applied in the X, Y or Z direction of the Surface Load.

Structural Loads Units (Project Preferences)

For each Load tool and type, set its units at **Options > Project Preferences > Structural Load Units**:



Representation of Loads

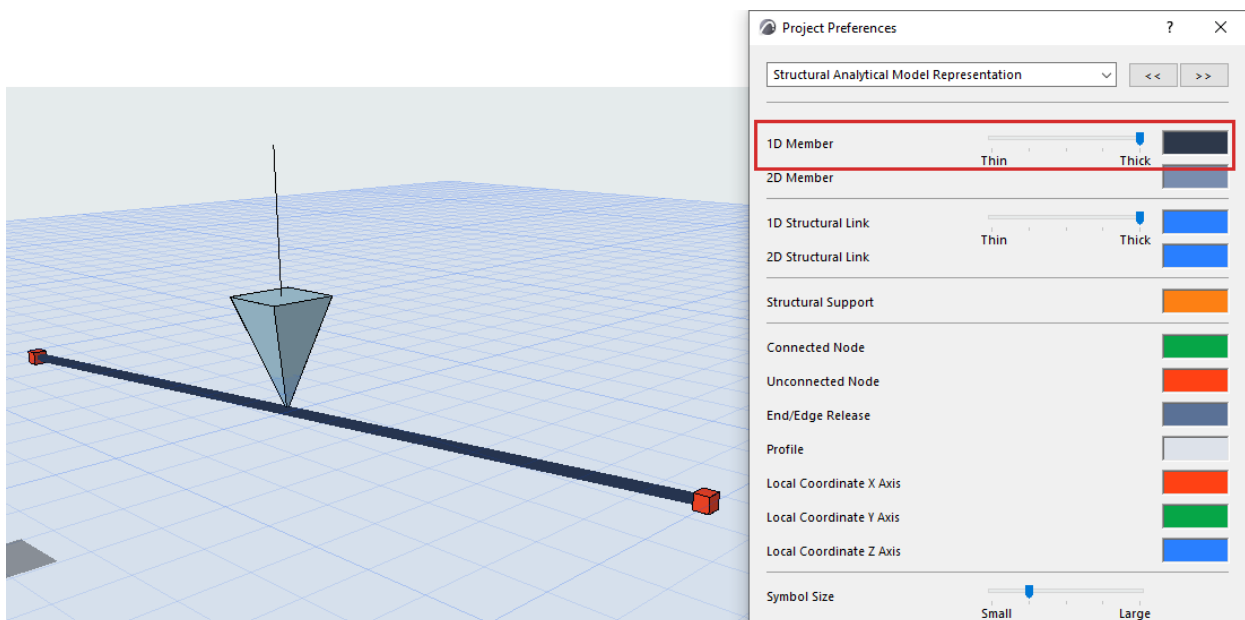
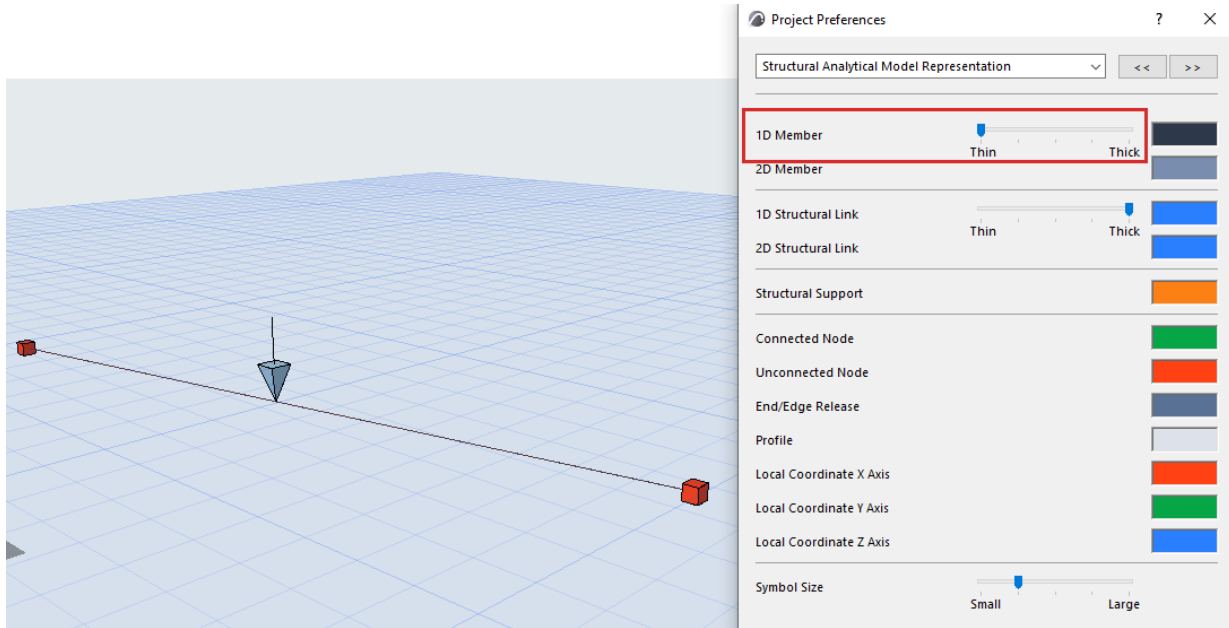
Scaling of Load Sizes (Loads on 1D Members)

To optimize Load display, the displayed Load symbol is automatically resized based on 1D Member thickness.

1D Member thickness is set at: **Project Preferences > Structural Analytical Model Representation**

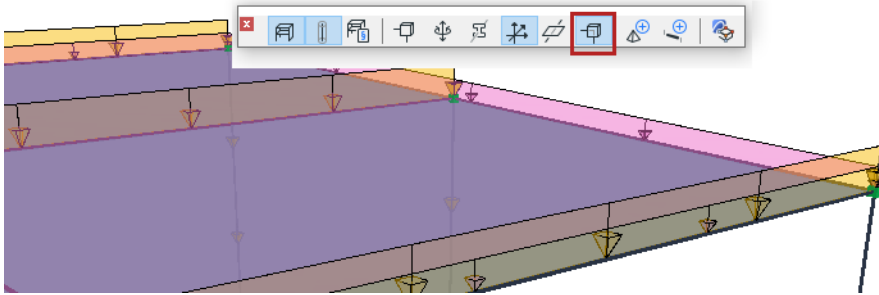
For example, a 1D Member with smaller thickness will display smaller Load elements.

Tip: Set the 1D Member thickness to suit your needs in the project, but be aware that this is a project-level setting, and affects the representation of Structural Analytical Model elements for all Teamwork users.



Show Point Load Arrows as Transparent

Use the “Transparent Symbols” toggle of the Structural Engineering toolbar. These affect the display of Point Load arrowheads in 3D views.



Minimum and Maximum Load Size representation

To improve visibility in the project, geometric limits have been introduced in the representation of Loads. After reaching certain intensity limits (either the minimum or the maximum value) the size of Loads will not change.

The default limit values are as follows:

Load Tool	Minimum Intensity	Maximum Intensity
Point	1 kN	1501 kN
Line	1 kN/m	76 kN/m
Surface	1 kN/m ²	26 kN/m ²

Note: If needed, these min/max values can be changed from the registry.

Loads in SAF Import/Export

SAF Import

Recognized Load Elements, Cases, Groups, Combinations are automatically imported from the SAF file during the process.

The following table lists the Load entities recognized by Archicad:

Load Entity Name	SAF Data	Available in ARCHCAD?
Point Force	StructuralPointAction	YES
Point Moment	StructuralPointMoment	YES
Line Force	StructuralCurveAction	YES
Line Moment	StructuralCurveMoment	YES
Surface Force	StructuralSurfaceAction	YES
Thermal Load on surface elements	StructuralSurfaceActionThermal	NO
Thermal Load on line elements	StructuralCurveActionThermal	NO
Free Point Force	StructuralPointActionFree	NO
Free Line Force	StructuralCurveActionFree	NO
Free Surface Force	StructuralSurfaceActionFree	NO
Load Panels	StructuralSurfaceActionDistribution	NO

SAF Export

Load-related data are exported by default.

[See *Miscellaneous Settings - Export*.](#)

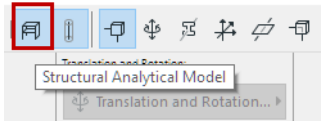
[See also *Load Cases in Model Compare*.](#)

Representation of Structural Analytical Model

Show/Hide Structural Analytical Model

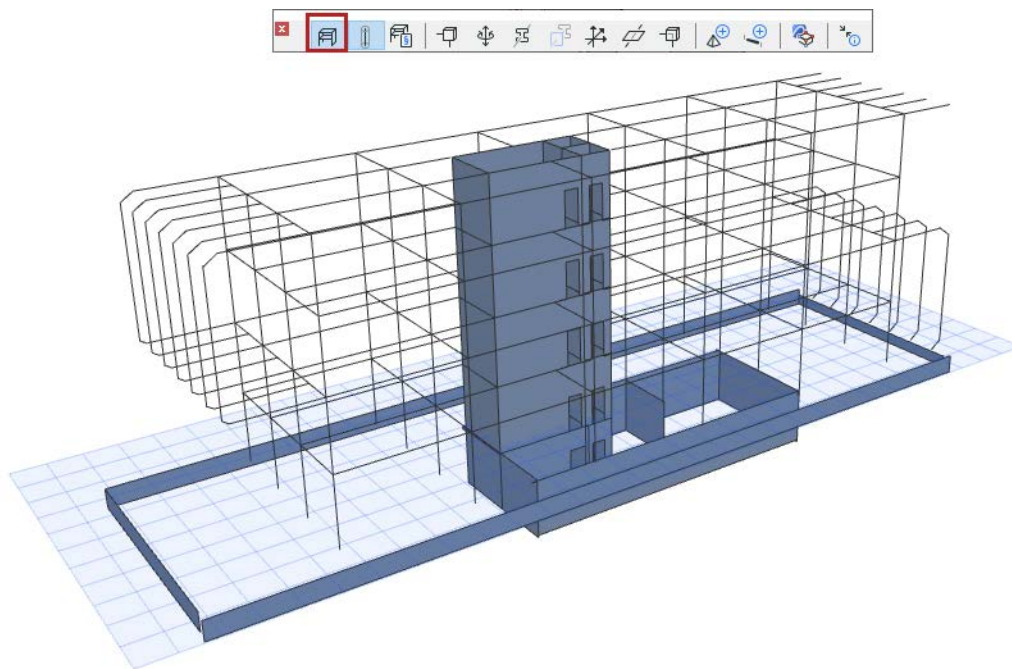
The Structural Analytical Model can be displayed in Floor Plan, Section windows, and 3D views.

Use the **Structural Analytical Model** toggle



- in the Structural Analytical Model Toolbar
- in the **View** menu

The Show/Hide status is specific to each view (if it is on in one view, it can be off in another view).

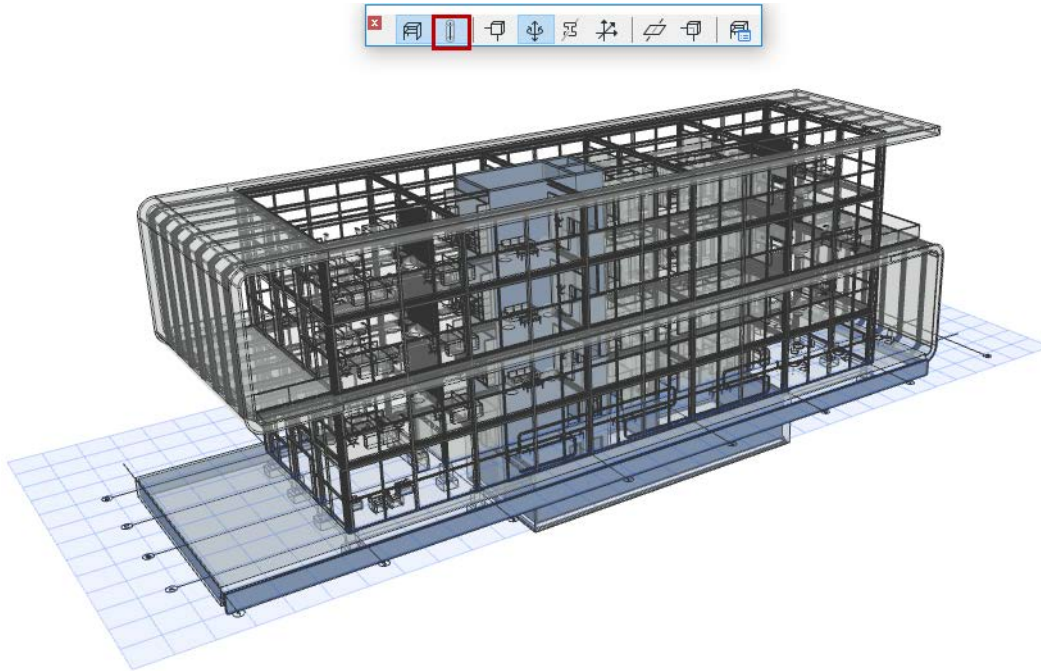


Important: Archicad elements are part of the Structural Analytical Model only if they have certain geometric and property settings. See [Elements of the Structural Analytical Model](#).

Show Transparent Physical Model

If the Structural Analytical Model toggle is on:

Use the Transparent Physical Model toggle to also display the Physical Model in transparent form, in the current view:

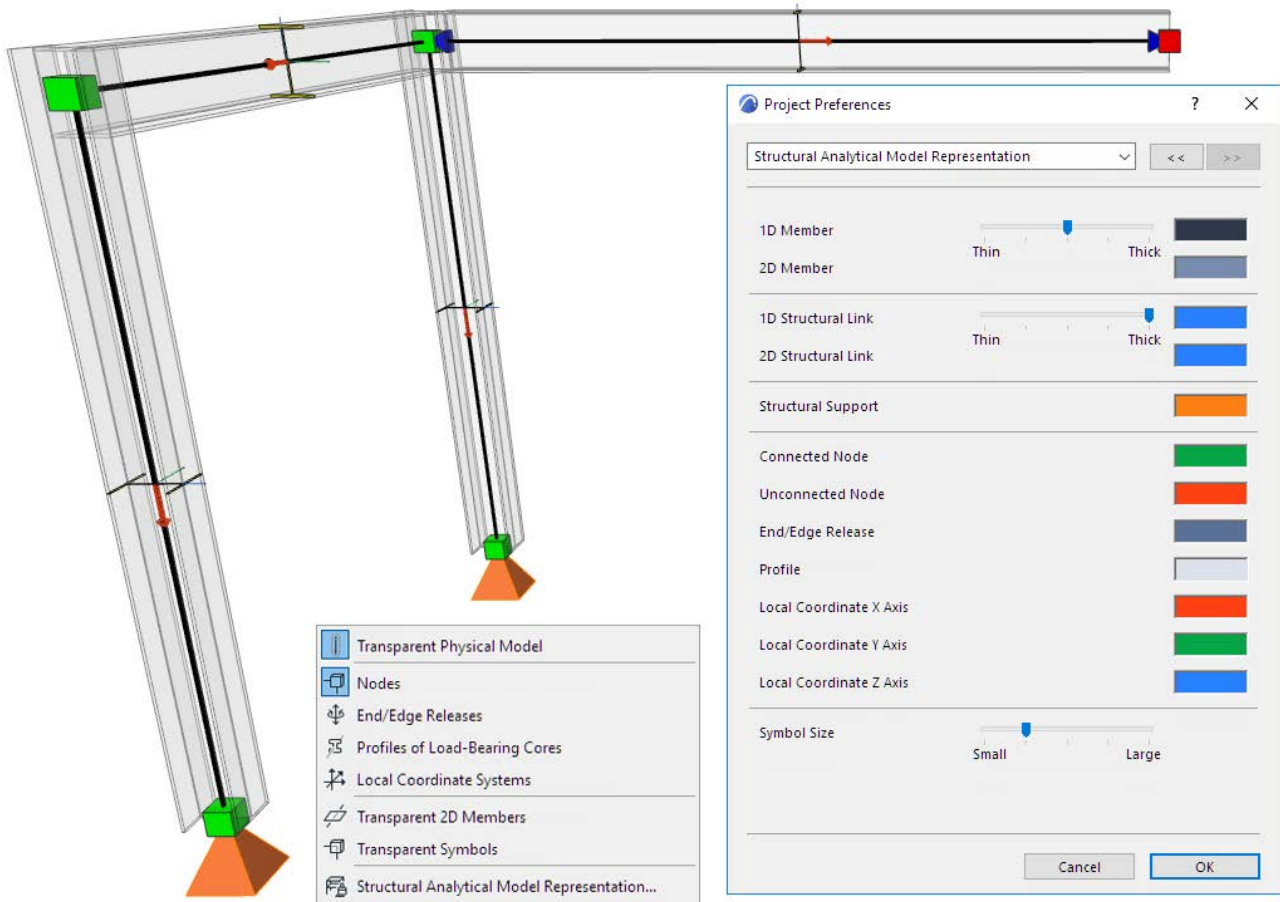


Show Components of Structural Analytical Model

Use the following additional toggle controls to show or hide Structural Analytical Model components. The controls are available from:

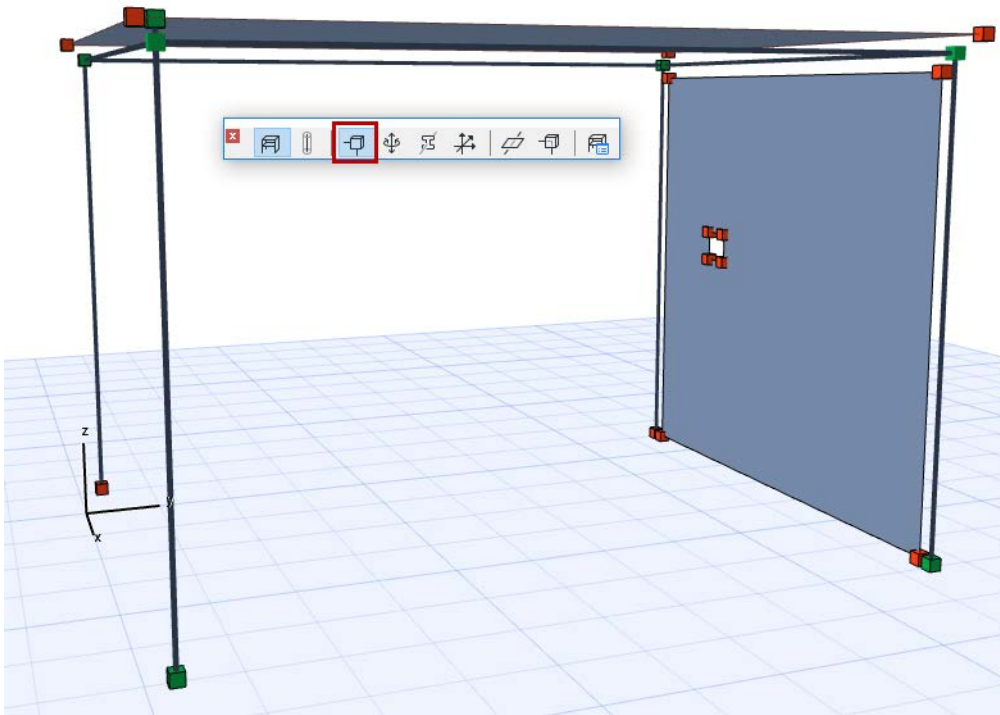
- Structural Analytical Model Toolbar
- View > Structural Analytical Model Options
- View > Elements in 3D View > Filter and Cut Elements in 3D (for Structural Support and Link elements)

Customize component colors at [Structural Analytical Model Representation \(Preferences\)](#)



Show Nodes

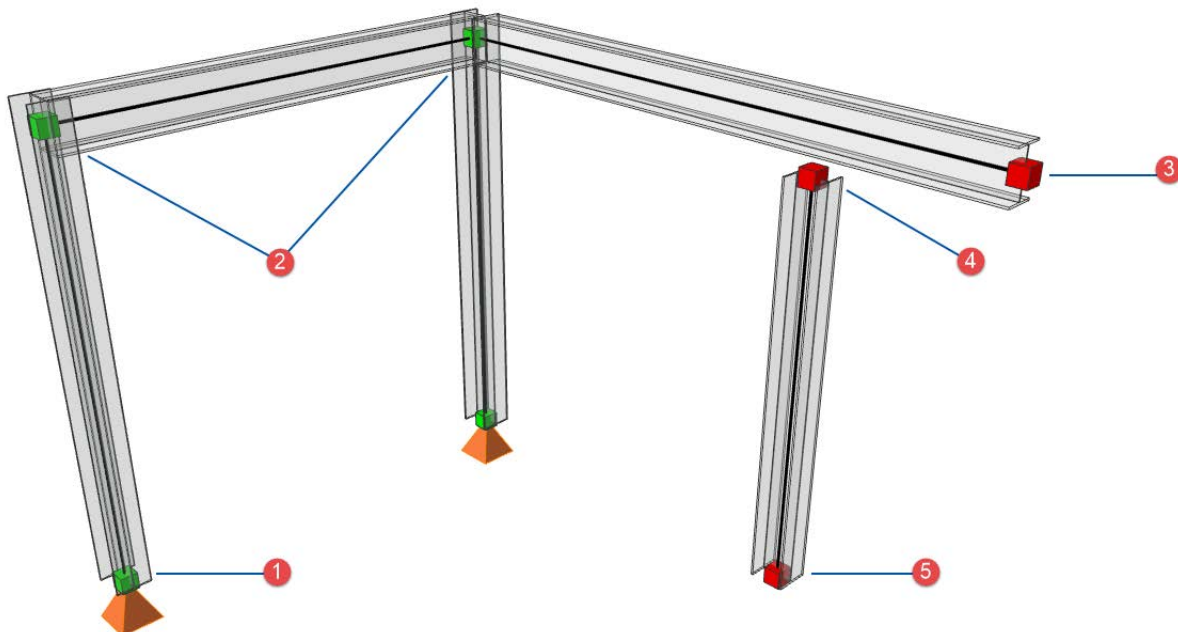
Display nodes at ends and corners of the 1D and 2D Members.



Node Connections: Color Feedback

By default:

- A green node represents a correct connection point or intersection between joined Analytical Members
- A red node represents a point or end that is either free (no connection needed), or unconnected/unsupported:

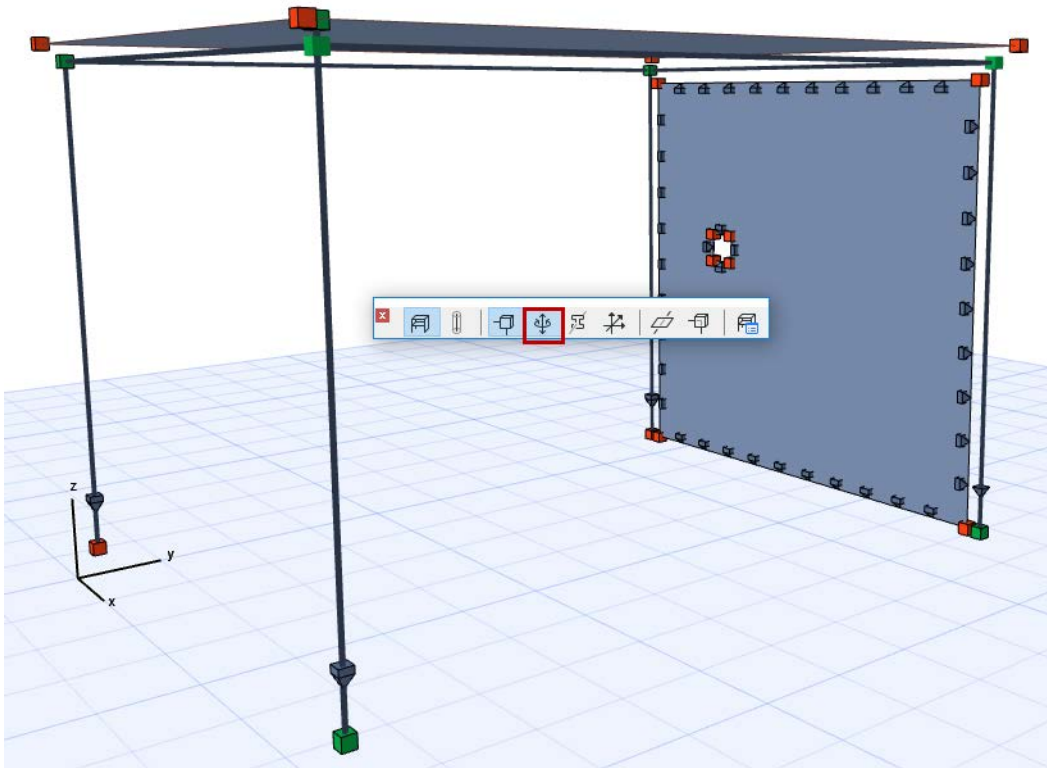


1: Supported, 2: Connected, 3: Free end (Cantilever), 4: Unconnected, 5: Unsupported

Show End/Edge Releases

Use this toggle to display their symbols in the Structural Analytical Model.

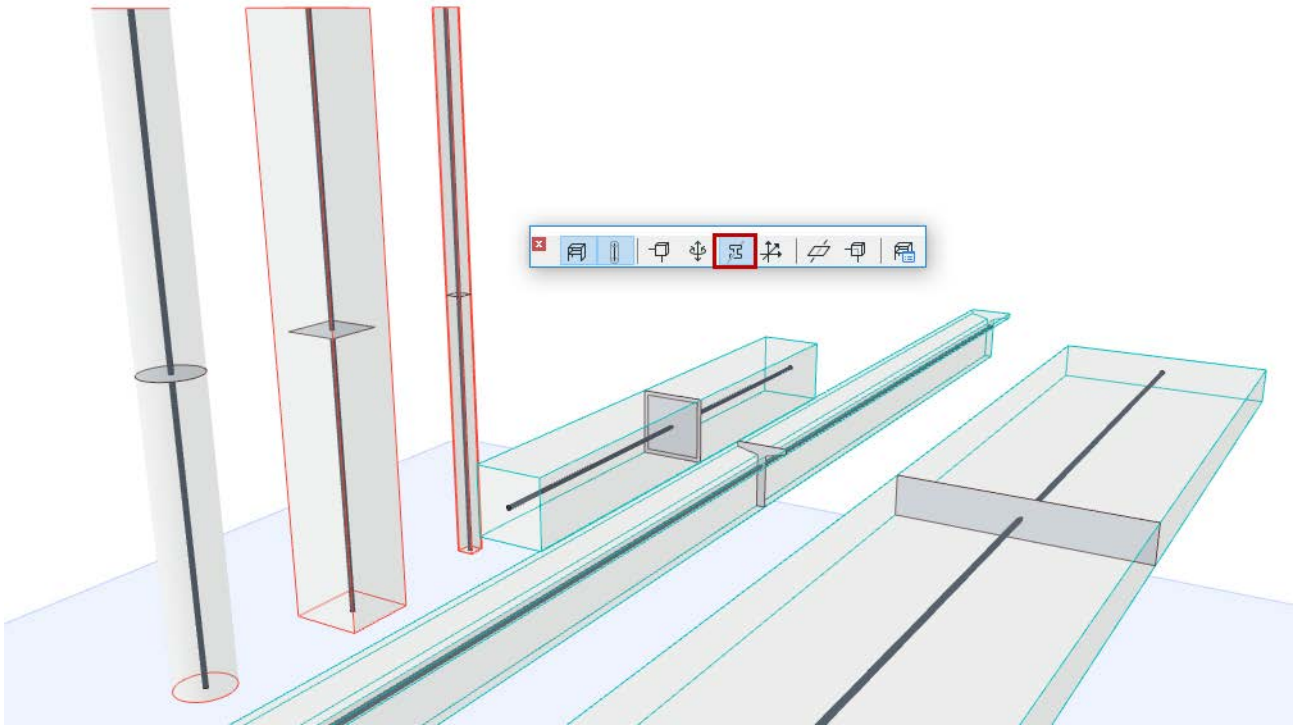
See [End/Edge Release Symbols](#).



Show Profiles of Load-Bearing Cores

Only 1D Members include a cross-section profile. Use this toggle to display their symbols in the Structural Analytical Model.

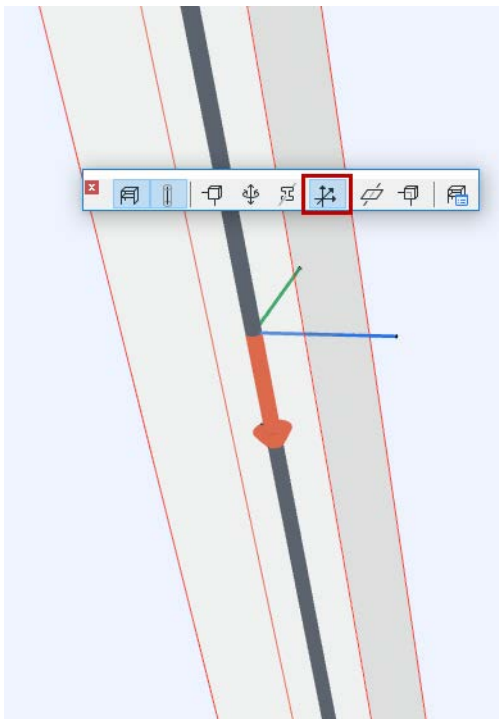
To make this symbol transparent: Use [Transparent Symbols](#).



Show Local Coordinate Systems

Display the element's local coordinate system (in the 3D window only).

- Local X: thick red line
- Local Y: thin green
- Local Z: thin blue



Local Coordinate System of Structural Analytical Member

Transparent 2D Members

This toggle has the following effect:

- Show 2D Members as Transparent
- Show Surface-type Structural Links as Transparent

Transparent Symbols

This toggle turns the following symbols transparent:

- Node
- End Release
- Spring-type Structural Link
- Support symbols
- Profiles of Load-Bearing Cores
- Point Load arrowheads in 3D view

Structural Analytical Model Generation Rules

Generation Rules are applied to the Structural Analytical Model to optimize it for export to structural analysis softwares.

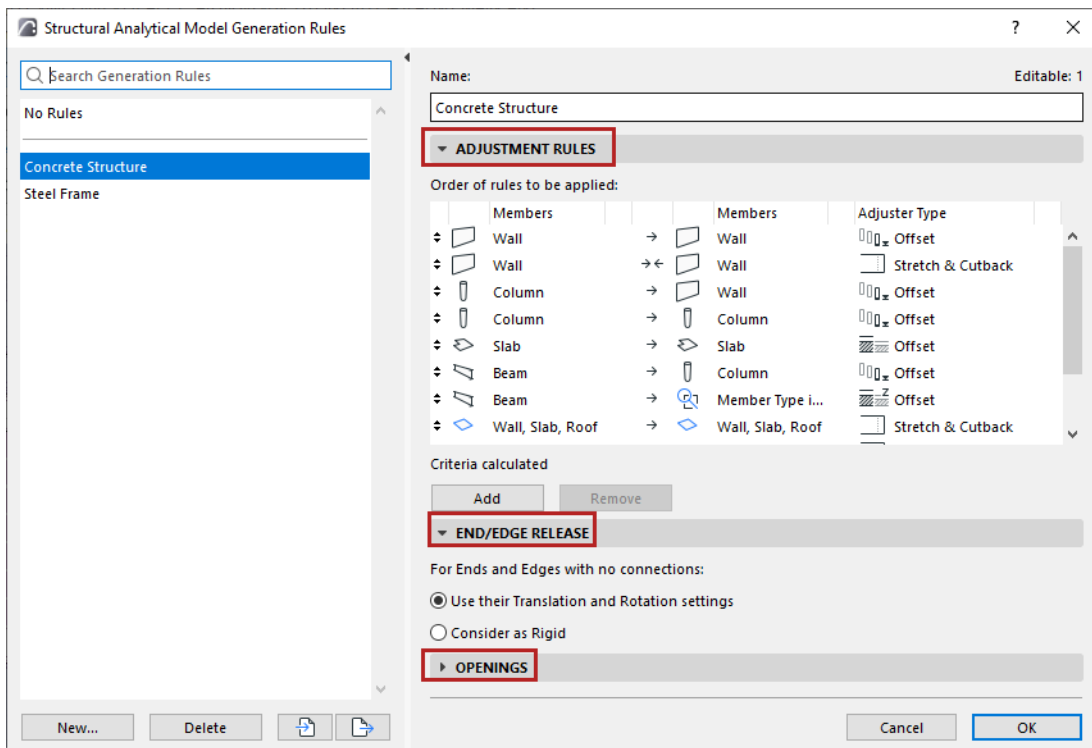
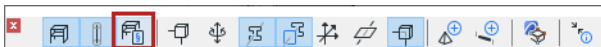
The current Generation Rules automatically go into effect whenever you turn on the Structural Analytical Model - you do not have to activate the rules by command.

Each set of Generation Rules includes:

- **Adjustment Rules:** These rules adjust (stretch, cutback, offset) 1D and 2D Members to achieve continuity among the nodes. Continuity is required for structural analysis, because a disconnected model results in errors.
- **End/Edge Releases:** Define a general rule for handling End/Edge releases that have no connections. (This can be useful if the receiving analysis software requires all such releases to be “Rigid”.)
- **Opening filtering:** Define rules to ensure that very small openings and very narrow 2D Member parts are ignored in the Structural Analysis model.

Open this dialog from:

- **Design > Structural Analytical Model Generation Rules**
- Structural Analytical Model toolbar



The default template includes several predefined sets of Generation Rules (plus a No Rules option). Try out the defaults.

If you need to edit the Rules, click Add to make a copy of the selected Rule set, then fine-tune them.

Details of Generation Rule settings are described in the sections below:

Adjustment Rules

End/Edge Release (Generation Rules)

Filter Openings for Structural Analytical Model

Adjustment Rules

This is a panel of the Structural Analytical Model Generation Rules dialog.

The Goal: A Continuous Structural Analytical Model

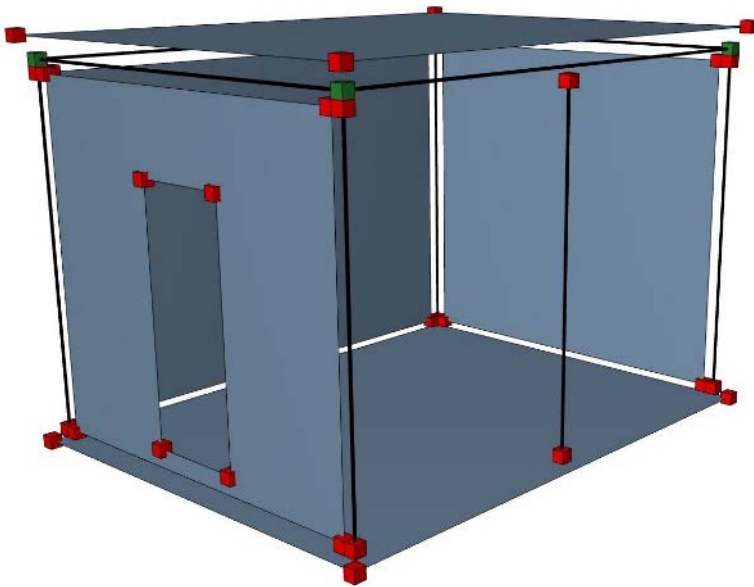
In order to perform a structural analysis correctly, the exported model must be **continuous**: the nodes should be connected. This continuity is required for structural analysis, because a disconnected model results in errors.

The first image below shows the initial Structural Analytical Model: though the physical elements are connected to each other at their core, nevertheless the red (disconnected) nodes of 1D and 2D Members indicate several places where they ought to be connected.

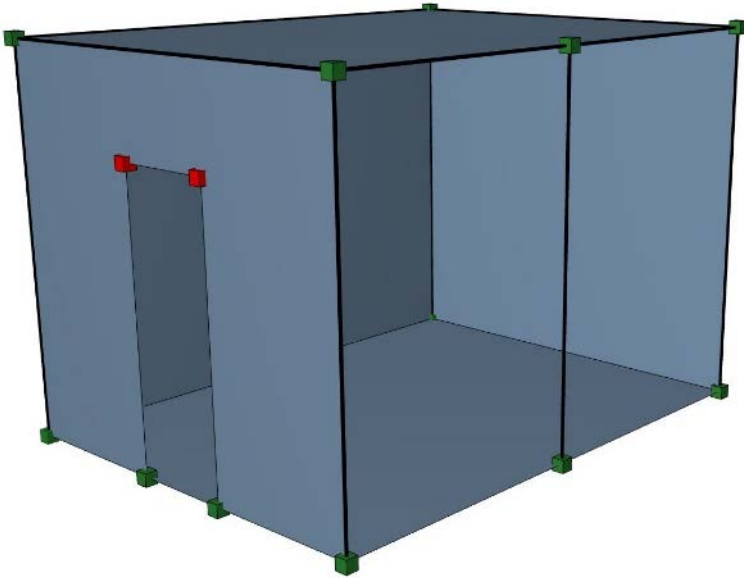
One solution is to create and/or apply Adjustment Rules from the Generation Rules dialog. These rules adjust the specified Member types to each other via Stretch, Cutback, Offset and/or Snap operations.

Important: Adjustments affect the Structural Analytical Model only. They do NOT affect the geometry of Physical Model elements!

As a result, the Structural Analytical Members are automatically connected wherever these rules apply.



Before Adjustment Rules: Disconnected Members



After Adjustment Rules: Connected Members

Conditions for Adjustment

An Adjustment Rule applies to Members only if:

- The Members fit the element criteria defined in the Adjustment Rule
- The adjustment (e.g. stretching distance) is within the tolerances and conditions set in the Adjustment Rule
- The two elements in the adjustment have a core connection (load-bearing cores meet at one or more points or adjoining surfaces) (see also [Important Notes](#), below.)
- Adjustments are enabled for this Member in Element Settings.

By default, adjustments are enabled for each Member. However, you can exempt any Member at the element level (to leave the Member unchanged, or to adjust it manually).

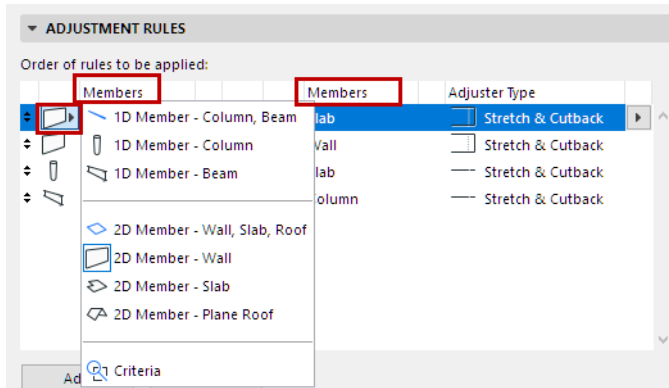
See [Allow Rule-based modifications - 1D Member](#) or [Allow Rule-based modifications - 2D Member](#).

Important Notes

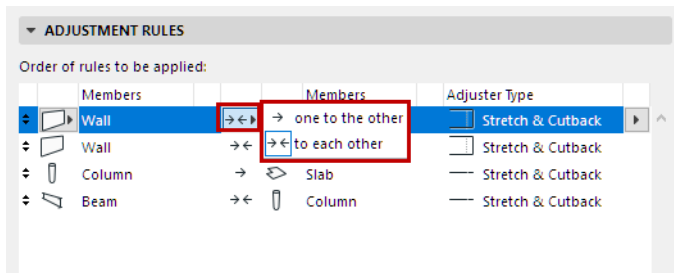
- Certain Archicad functions can help to identify and achieve core connections where they are missing:
 - See the “Lack of Core Connection” parameter at [Physical Model Quality Check](#)
 - See the [Connection Range \(Extend Core to Enable Connection\)](#) checkbox, in the Element Settings of 1D Members
- The current Generation Rules (including Adjustment Rules) go into effect whenever you turn on the Structural Analytical Model - you do not have to activate the rules by command.

Set Up Adjustment Rules

1. Define the two Groups of Members: Elements by type or criteria (see [Members in Adjustment Rule](#) below)

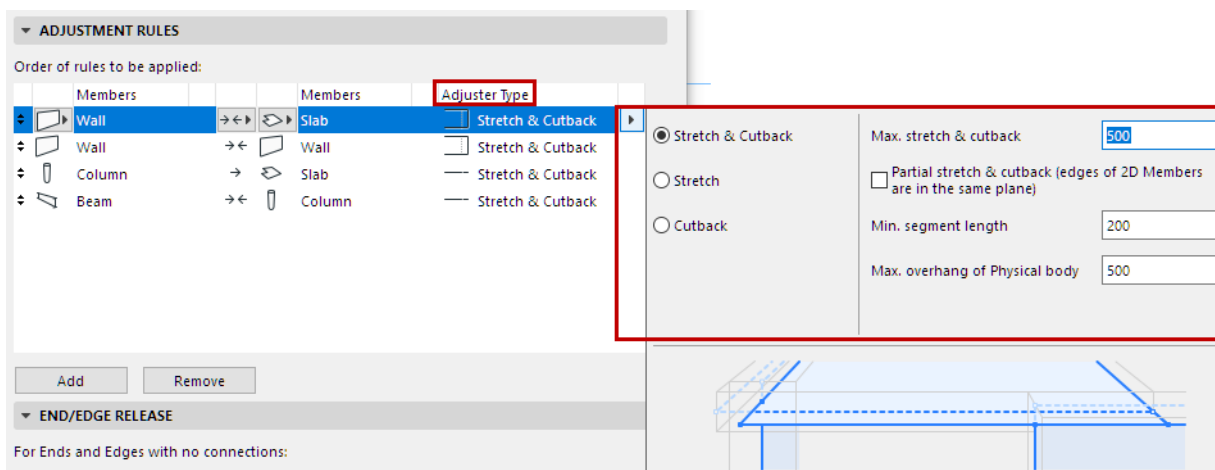


2. Define the Rule Method (see [Adjustment Rule Methods: One to the Other, or To Each Other](#) below)



3. Set the Adjuster Type.
4. Set additional options depending on the specific adjuster

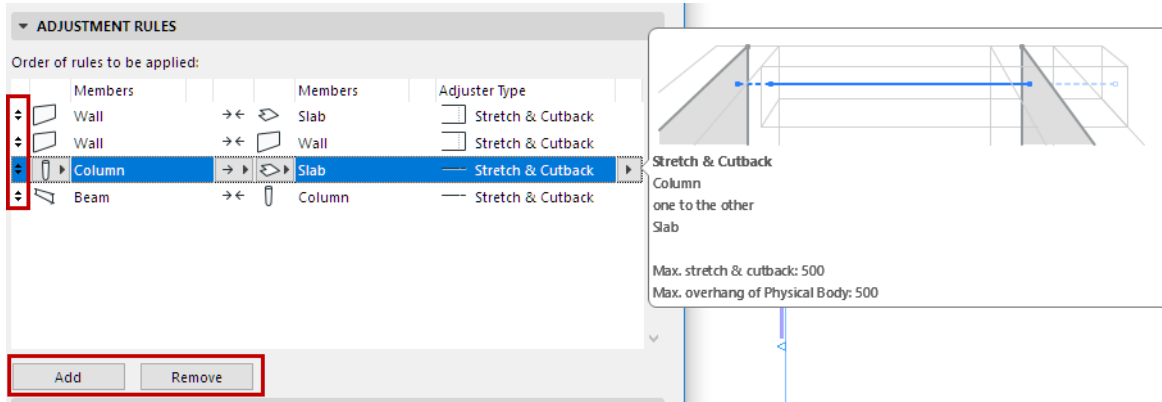
See [Adjuster Types and Options for details](#).



Notes on Adjustment Rules

- Rules are applied in the order they are listed, and they are all carried out. You can drag rules to change their order.
- Generally, the Offset rules come at the top of the list: they are used not to create connections per se, but to line up the elements so that the rest of the rules (Stretch, Cutback) can work as needed.

- Hover over any Adjustment Rule to see an info pop-up, showing its parameters and a preview.



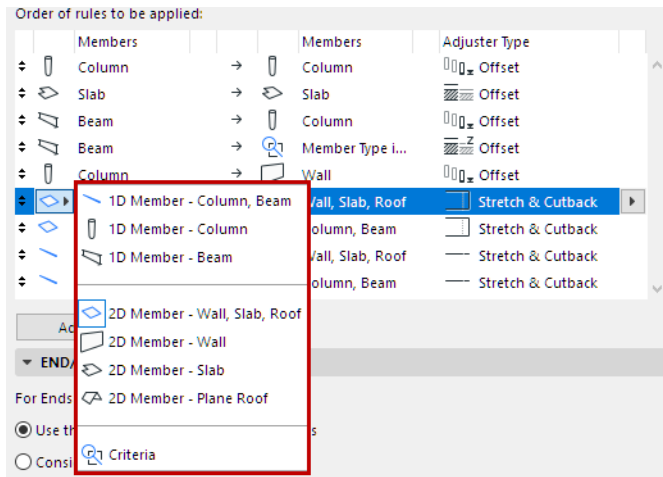
- Use the buttons at the bottom of the Adjustment Rules panel to Add or Remove rules.

Important: If you remove a Rule, the associated adjustment is no longer applied, and the Structural Analytical Model reverts to its original position!

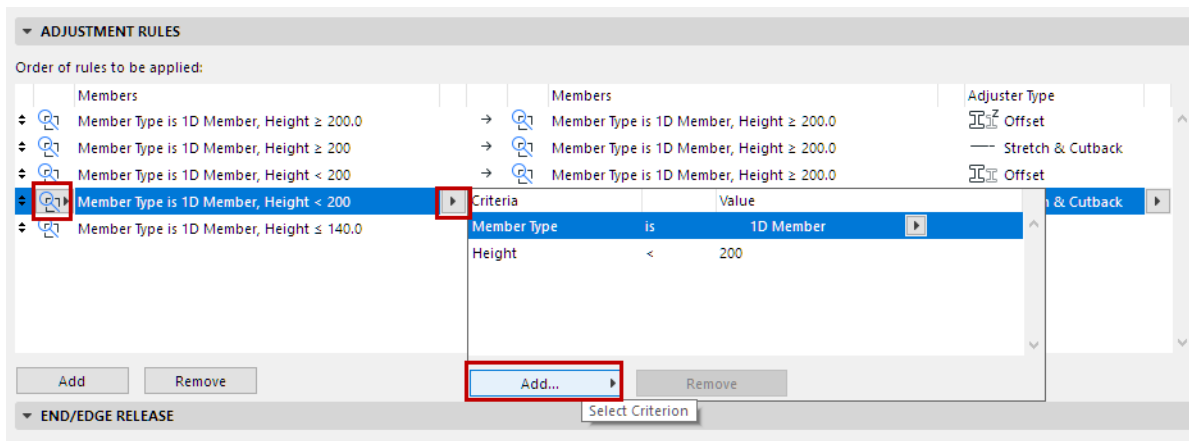
Members in Adjustment Rule

Members of an Adjustment Rule can be

- Any element type in the Structural Analytical Model
- All 1D or all 2D Members

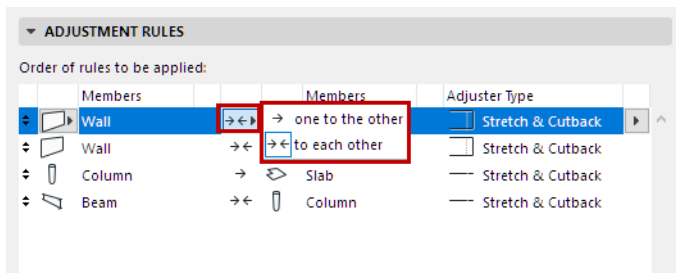


- Criteria-based groups of Structural Analytical Members



Adjustment Rule Methods: One to the Other, or To Each Other

This control sets how the adjustment should affect the two Member groups in the Adjustment Rule.



- **One to the other:** → The Members in the left column are modified; the Members in the right column remain unchanged. Use this method to Stretch and/or Cutback, or Offset one group of elements to each other. The Offset and Snap adjusters always use this unidirectional method.
- **To each other:** → ← Members in both categories are affected. Use this method to Stretch and/or Cutback two groups of elements to each other.

See the following sections for details and examples:

[Adjuster Types and Options](#)

[Examples of Adjustment Rules](#)

Adjuster Types and Options

The following table summarizes the available adjuster types and methods. If all the conditions for adjustment are met, the adjustment takes place: the Member in red is adjusted in relation to the black Member; the Member in black remains unadjusted.

Adjustment Method	Member Types	Stretch	Cutback	Stretch & Cutback	Offset	Snap
One to the other →	1D > 1D	+	+	+	+	+
	1D > 2D	+	+	+	+	
	2D > 1D	+	+	+	+	
	2D > 2D	+	+	+	+	
To each other → ←	1D <> 1D	+	+	+		
	2D <> 1D	+	+	+		
	2D <> 2D	+	+	+		

Depending on the specific adjuster, you can set additional options and tolerances. These are also explained below.

Topics in this section:

[Stretch 1D Member](#)

[Stretch 2D Member](#)

[Cutback 1D Member](#)

[Cutback 2D Member](#)

[Stretch & Cutback](#)

[Adjust Connected Ends](#)

[Offset 1D to 1D](#)

[Offset 1D to 2D](#)

[Offset 2D Member](#)

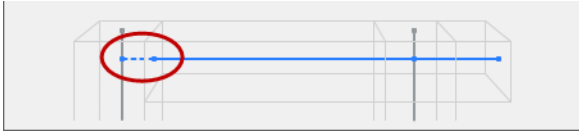
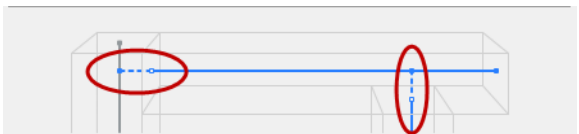

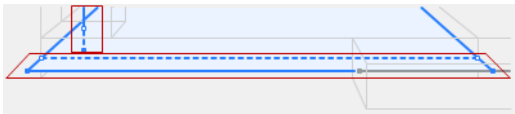
[Offset Options In Case of Multiple Targets](#)

[Snap \(1D to 1D\)](#)

Stretch 1D Member

Stretch the 1D Member along its length (local X axis), so that it can connect to another nearby Member. For each stretch adjuster, you can set a maximum allowed stretch distance ("Max. stretch"). This distance is measured between unadjusted 1D Member Nodes and/or unadjusted 2D Member planes.

Note: The defined tolerances are length-type values: the units specified for Length in **Preferences > Working Units** apply.


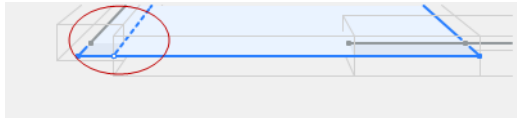

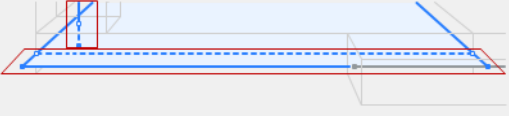
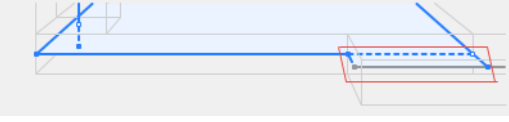
STRETCH 1D MEMBER TO 1D MEMBER	
Stretch 1D to 1D (one to the other) →	
Stretch 1D to 1D (to each other) → ←	
STRETCH 1D MEMBER TO 2D MEMBER	
Stretch 1D to 2D (one to the other) →	
Stretch 1D to 2D (to each other) → ←	 <p><i>1D Column is stretched down to reach the 2D Slab; Slab edge is stretched to reach the side Beam</i></p>


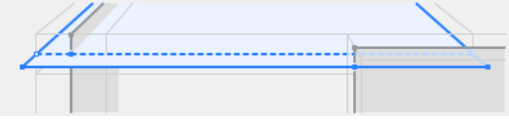

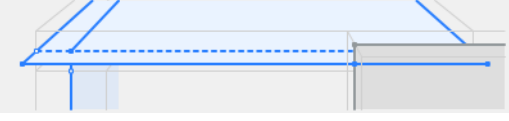
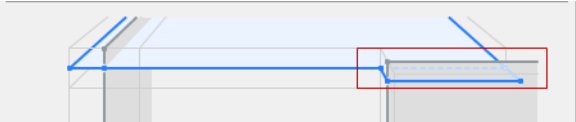
See also [Adjust Connected Ends](#).

Stretch 2D Member

Stretch part or all of one edge of a 2D Member polygon, so that it can connect to another nearby Member. For each stretch adjuster, you can set a maximum allowed stretch distance (“Max. stretch”). This distance is measured between unadjusted 1D Member Nodes and/or unadjusted 2D Member planes.

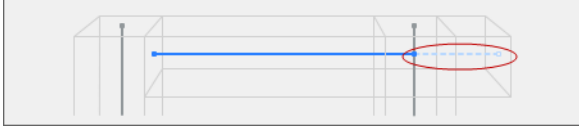
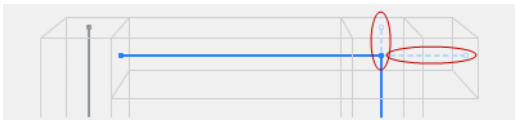
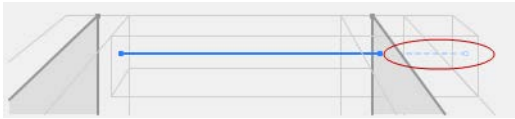
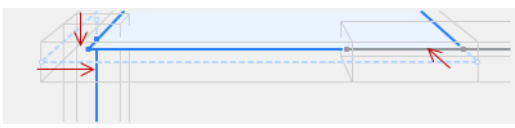
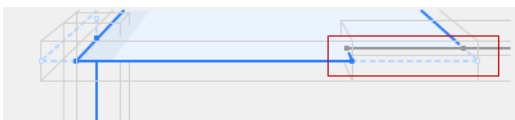
Note: The defined tolerances are length-type values: the units specified for Length in **Preferences > Working Units** apply.

STRETCH 2D MEMBER TO 1D MEMBER	
Stretch 2D to 1D (one to the other) 	
Stretch 2D to 1D (to each other) 	 <p><i>1D Column is stretched down to reach the Slab; 2D Slab edge is stretched to reach the side Beam</i></p>
Partial stretch	<p>Check this to stretch only part of the 2D Member edge. This applies if the 1D target is in the same plane as the 2D Member that is being stretched.</p> 
Min. segment length	<p>If Partial Stretch is enabled: An edge part will be stretched only if it is at least this long.</p>

STRETCH 2D MEMBER TO 2D MEMBER	
Stretch 2D to 2D (one to the other) 	
Stretch 2D to 2D (to each other) 	
Partial stretch	<p>Check this to stretch only part of the 2D Member edge. This applies if the 2D target is in the same plane as the 2D Member that is being stretched.</p> 
Min. segment length	<p>If Partial Stretch is enabled: An edge part will be stretched only if it is at least this long.</p>

Cutback 1D Member


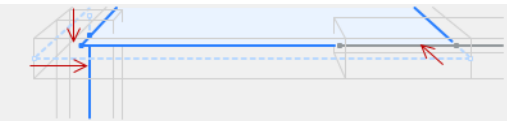
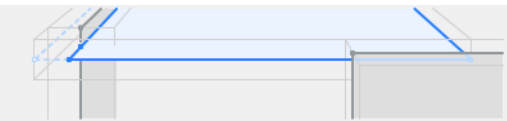
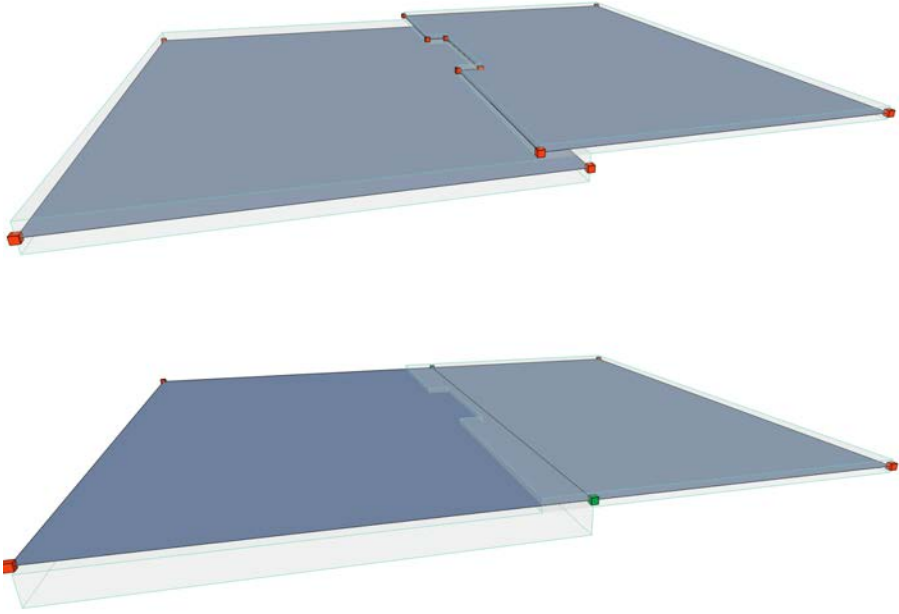
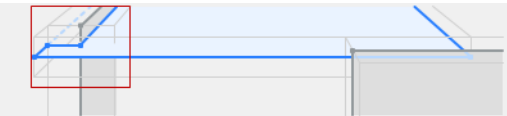
Cut back 1D Member along its length (Local X axis), so that it can connect as needed to another nearby Member.

CUTBACK 1D MEMBER TO 1D MEMBER	
Cutback 1D to 1D (one to the other) →	
Cutback 1D to 1D (to each other) → ←	
CUTBACK 1D MEMBER TO 2D MEMBER	
Cutback 1D to 2D (one to the other) →	
Cutback 1D to 2D (to each other) → ←	 <p><i>Column is cut back to the Slab</i> <i>Slab edge is cut back to the Beams</i></p>
Partial cutback	Check this to cut back only part of the 2D Member edge. 
Min. segment length	If Partial Cutback is enabled: An edge part will be cut back only if it is at least this long.

See also [Adjust Connected Ends](#).

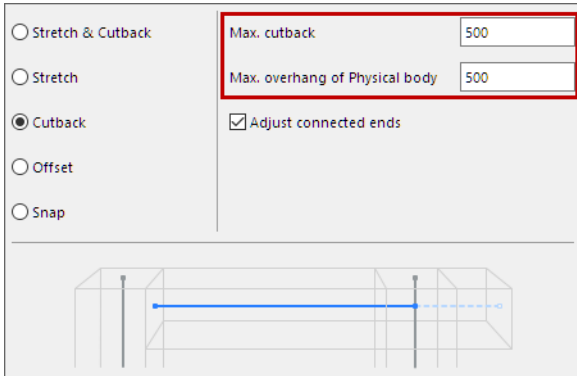
Cutback 2D Member

Cuts back all or part of the edge of a 2D Member plane, so that it can connect as needed to another Member.

CUTBACK 2D MEMBER	
<p>Cutback 2D to 1D (one to the other)</p> <p>→</p>	
<p>Cutback 2D to 1D (to each other)</p> <p>→ ←</p>	
<p>Cutback 2D to 2D (one to the other)</p> <p>→</p>	
<p>Cutback 2D to 2D (to each other)</p> <p>→ ←</p>	<p>If two 2D members overlap, this Cutback adjuster also eliminates the overlap:</p> <div style="text-align: center;">  </div>
<p>Partial cutback</p>	<p>Check this to cut back only part of the 2D Member edge. This applies if the 2D Member edge and its target are in the same plane.</p> <div style="text-align: center;">  </div>

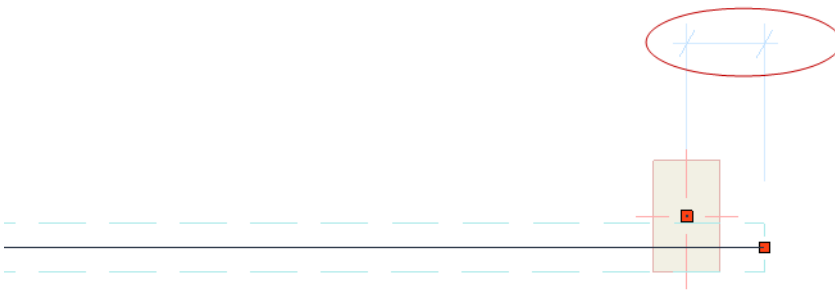
CUTBACK 2D MEMBER	
Min. segment length	If Partial cutback is enabled: An edge part will be cut back only if it is at least this long.

Maximum Cutback/Maximum Overhang



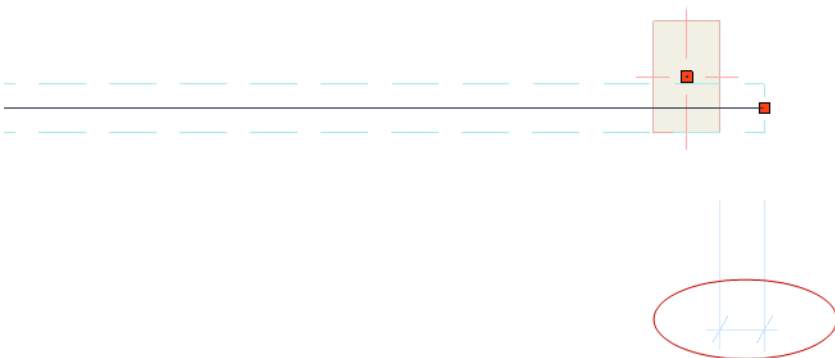
Maximum cutback

For each Cutback (or Stretch & Cutback) adjuster, you can set a maximum allowed cutback (“Max. cutback”). This distance is measured between unadjusted 1D Member Nodes and/or unadjusted 2D Member planes, in the structural analysis model.



Maximum overhang of Physical body

In contrast to the maximum cutback value (which refers to the structural member), the Maximum overhang refers to the *physical element's* protrusion. Set this limit so that longer extensions in the physical model (e.g. consoles) are not cut back.



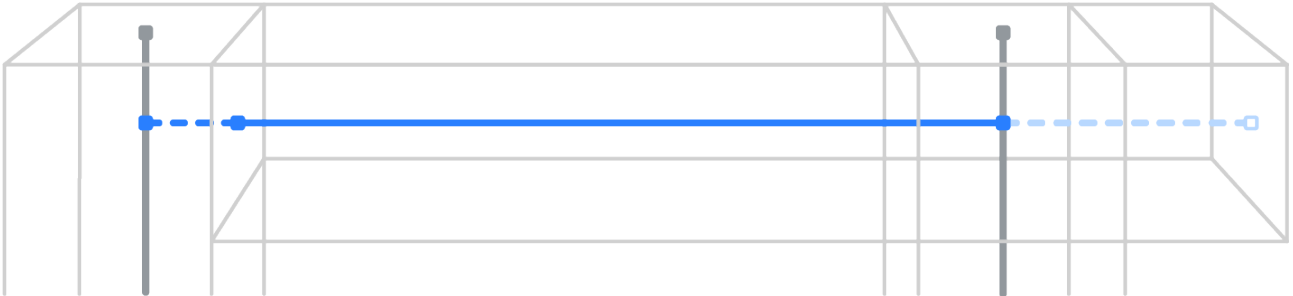
Note: The defined tolerances are length-type values: the units specified for Length in **Preferences > Working Units** apply.

Stretch & Cutback

Use this to apply both of these adjusters to a single 1D or 2D Member.

The Stretch & Cutback combination has the same effect as using the two adjusters separately.

For example, a 1D Member may need to be stretched at one end, and cut back at the other end.



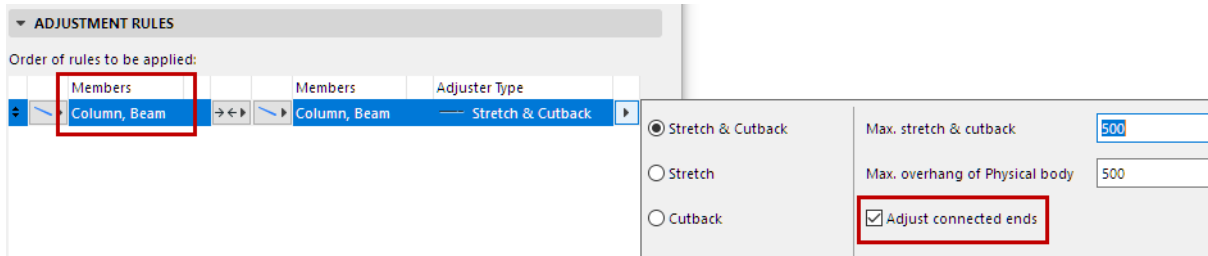
A 2D Member can be stretched on one edge and cut back on another.



Adjust Connected Ends

Available when adjusting a 1D Member, using Stretch, Cutback or Snap.

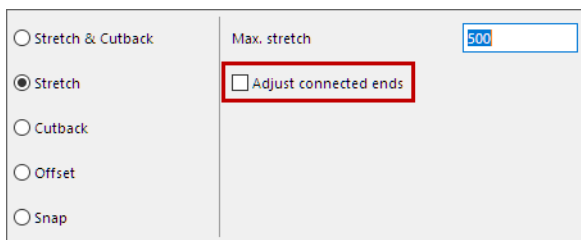
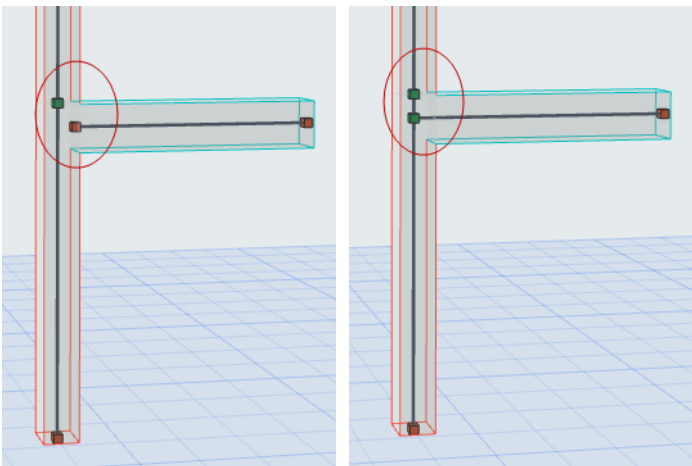
If the current adjustment rule results in a new node: this option enables further adjustment(s) which take the newly created connection node(s) into account.



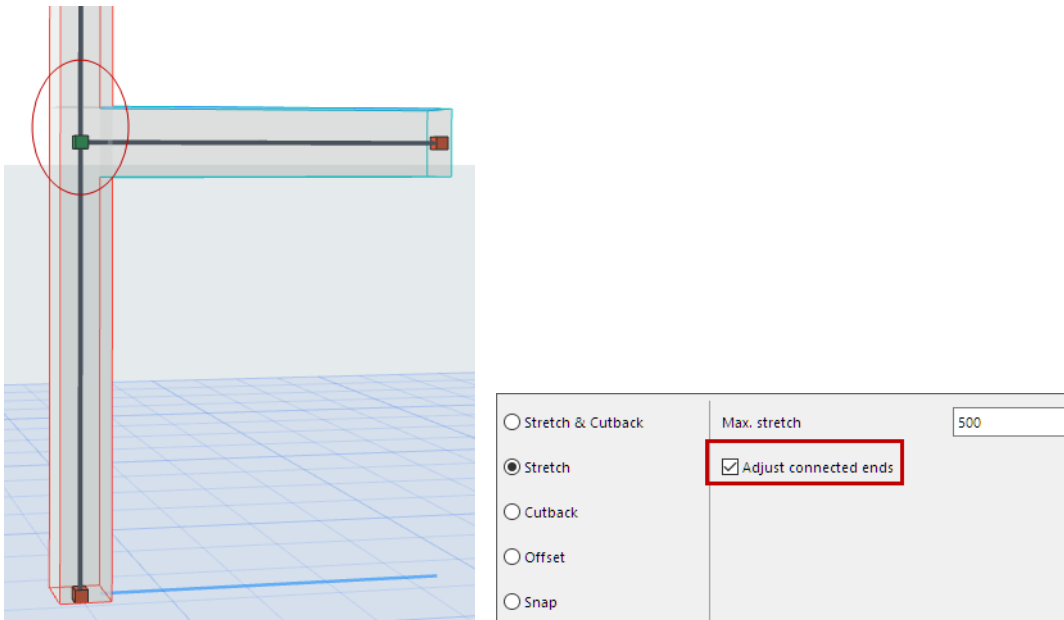
- Additional adjustments will be carried out on connected 1D Member(s) at each new node.
- These additional adjustments occur along the 1D Members' local x axis.
- The 1D Members' End Release values are based on this new connection node
- The 1D Members' Startpoint/Endpoint values in SAF are based on this new connection node

For example:

A Beam is stretched to meet the Column, creating a new node. The "Adjust Connected Nodes" option is OFF.


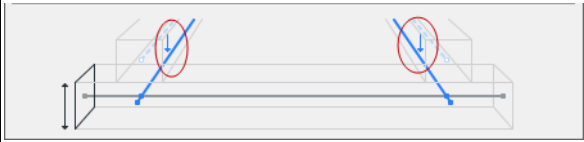


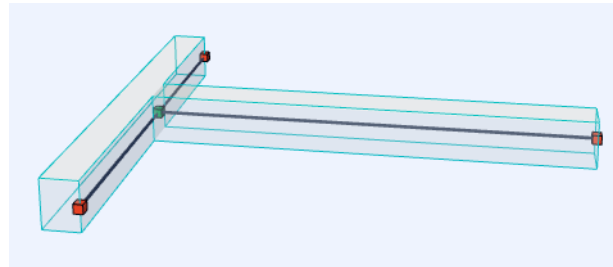
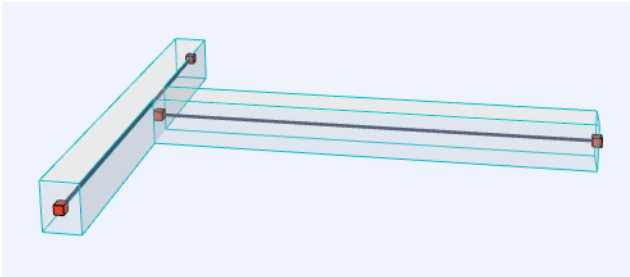
If you turn “Adjust Connected Ends” ON: the new connection node is adjusted further, to coincide with the existing node.



Offset 1D to 1D

Offset always occurs in one direction (one to the other): the adjusting Members will be offset to their targets; the targeted Members will remain unchanged.

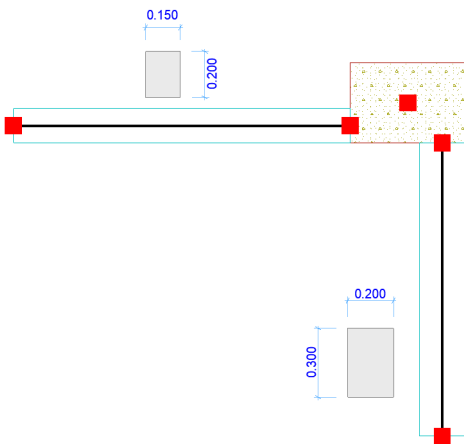
OFFSET 1D MEMBER	
Offset 1D to 1D 	



Connecting Beams: Before and After Offset Adjustment

1D to 1D Offset Options

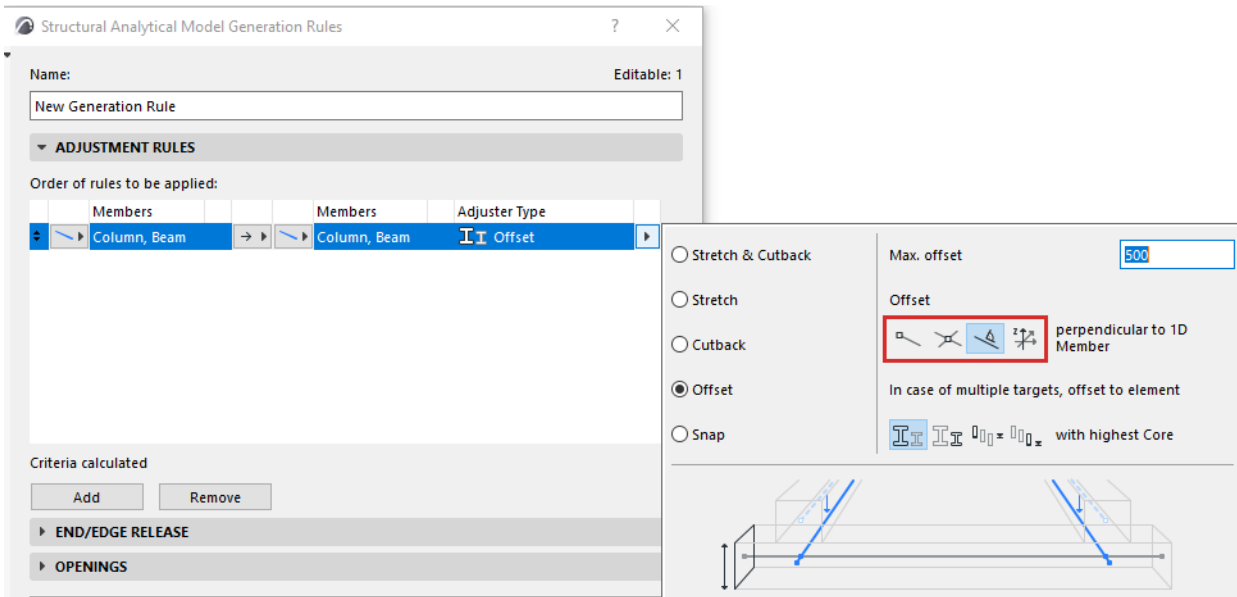
Let's use an example of 2 beams connected to a column. Each beam has a different cross section and is placed with different eccentricity to the column.



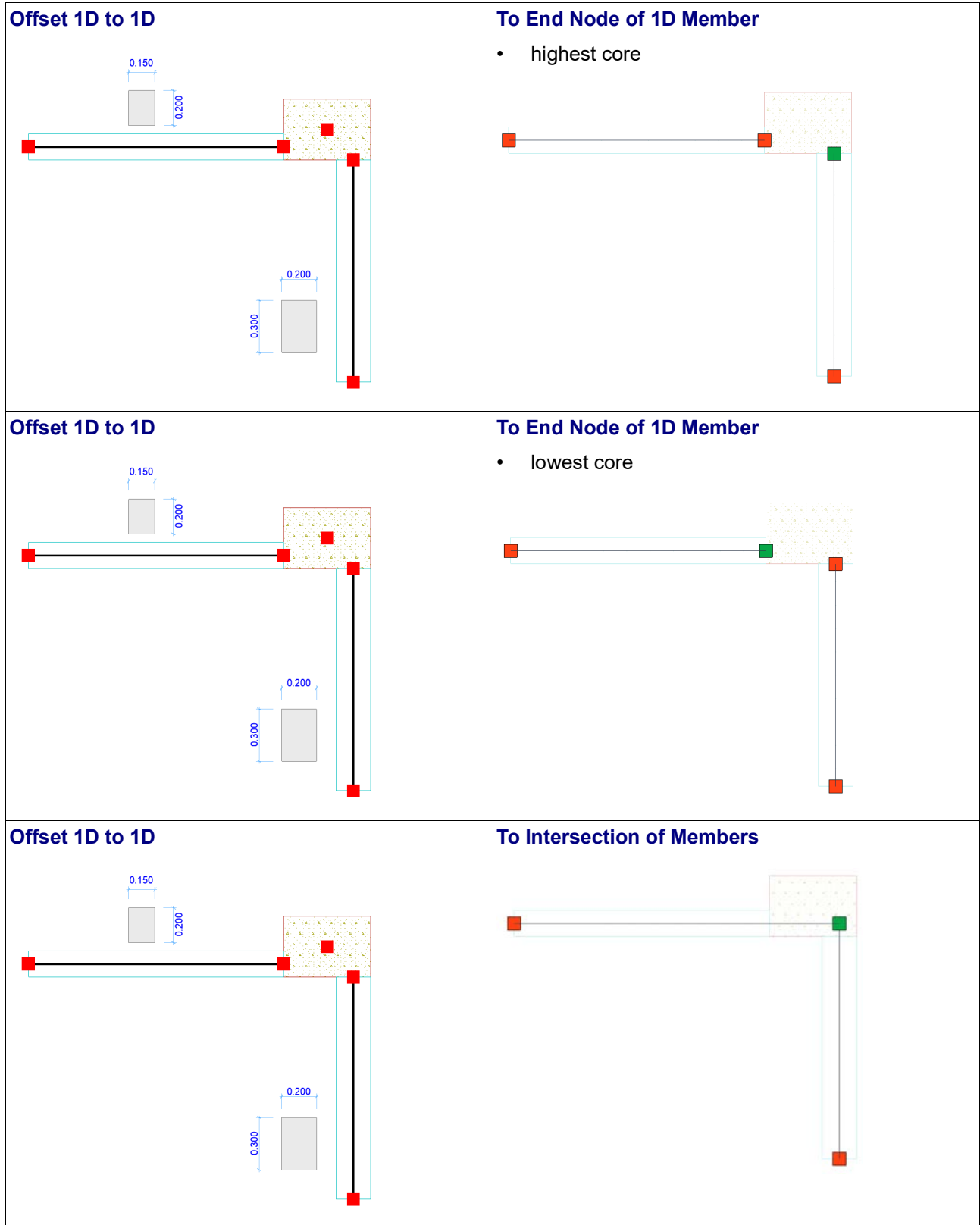
Click the arrow of the Offset Adjuster Type to choose options for 1D Offset.

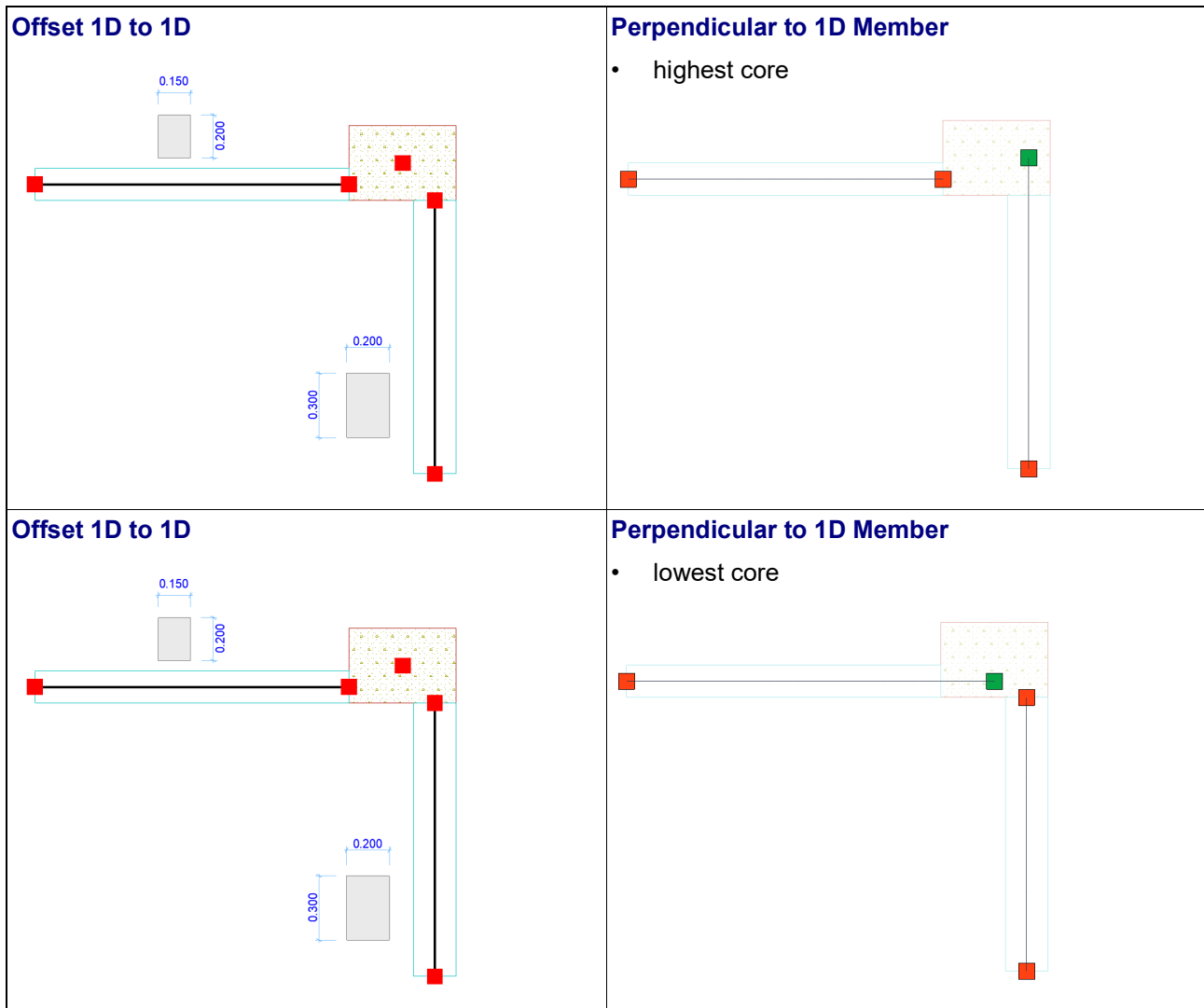
Set the maximum allowed offset distance.

Choose one of the four offset logic options:



Each option is illustrated in the table below.





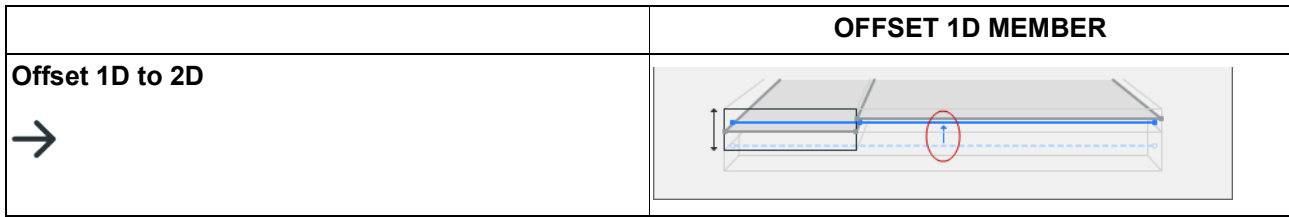
Global Z direction only

Check this if the **1D Member** should only be offset in the global Z direction. Offsets in other directions will not be applied.

See also [Offset Options In Case of Multiple Targets](#).

Offset 1D to 2D

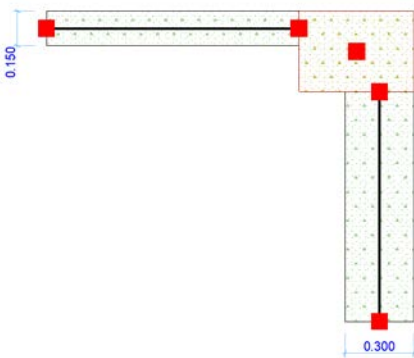
Offset always occurs in one direction (one to the other): the adjusting Members will be offset to their targets; the targeted Members will remain unchanged.



This adjuster offsets the 1D Member to the nearest edge of the target 2D Member (not the nearest point) - provided that the target edge is still within the “Max. offset” range

1D to 2D Offset Options

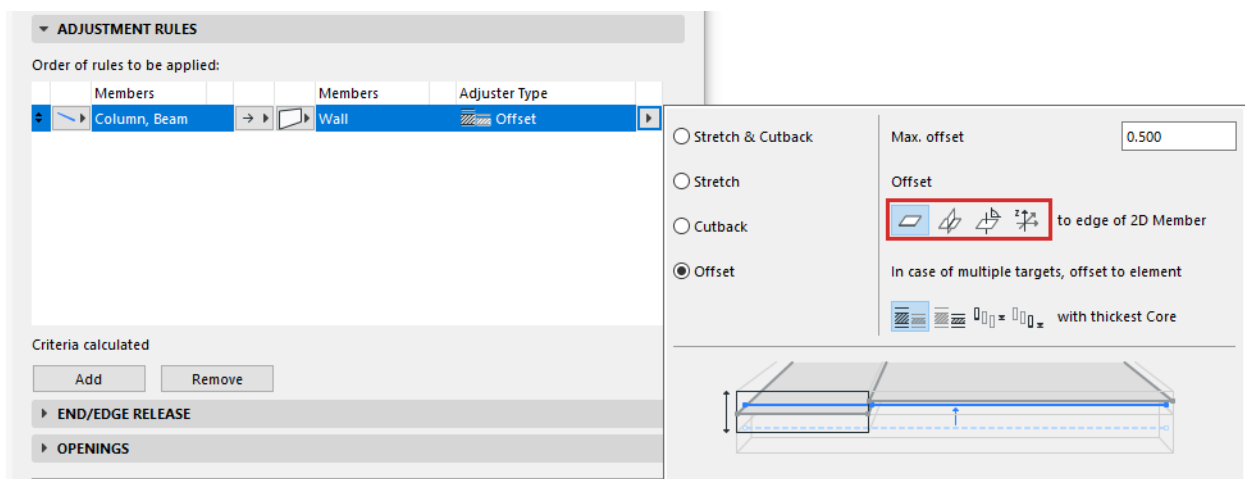
Let’s use an example of 2 walls connected to a column. Each wall has a different core thickness and is placed with external face aligned with the column.



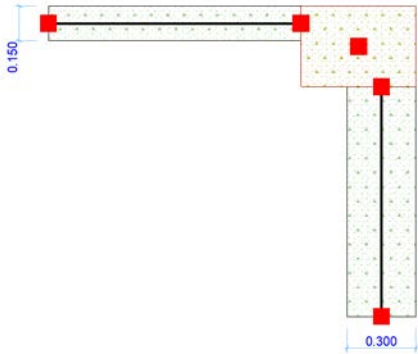
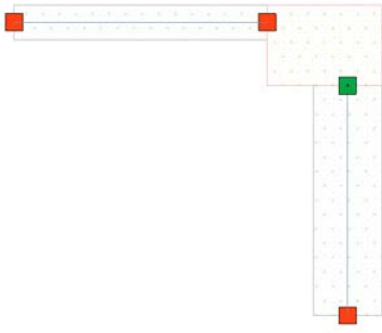
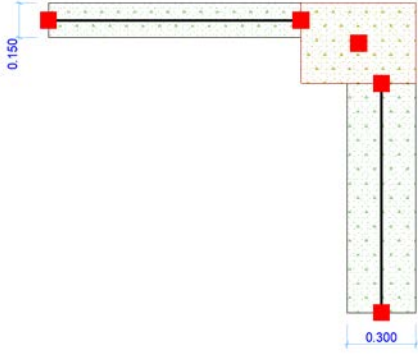
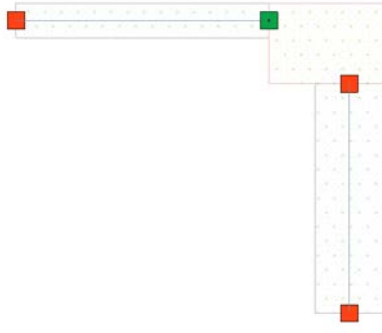
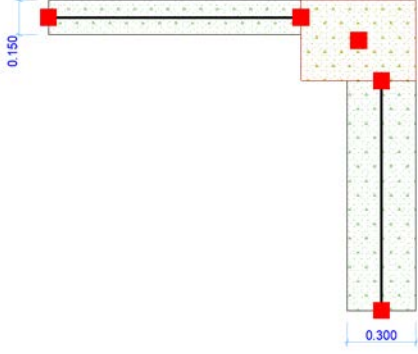
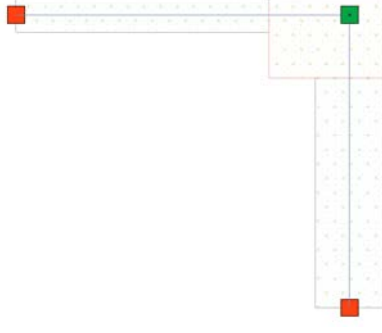
Click the arrow of the Offset Adjuster Type to choose options for 1D Offset to 2D.

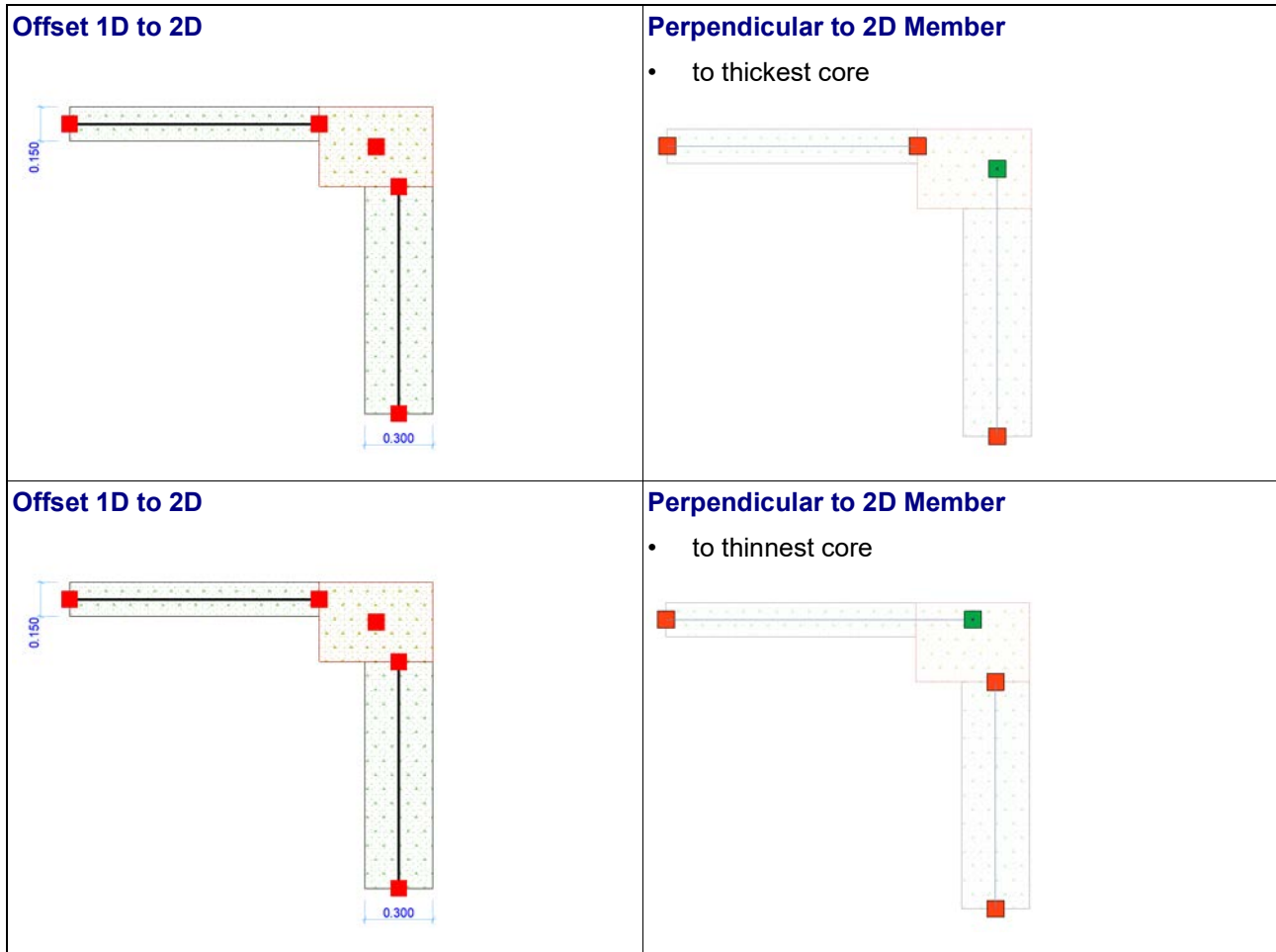
Set the maximum allowed offset distance.

Choose one of the four offset logic options.



Each option is illustrated in the table below.

<p>Offset 1D to 2D</p> 	<p>To Edge of 2D Member</p> <ul style="list-style-type: none"> • thickest core 
<p>Offset 1D to 2D</p> 	<p>To Edge of 2D Member</p> <ul style="list-style-type: none"> • to thinnest core 
<p>Offset 1D to 2D</p> 	<p>To Intersection of 2D Members</p> 



Note: If you also activate “Offset only in Global Z Direction”, the “Find edge” option is limited to the Z direction.

Global Z direction only

Check this if the **1D Member** should only be offset in the global Z direction. Offsets in other directions will not be applied.

See also [Offset Options In Case of Multiple Targets](#).

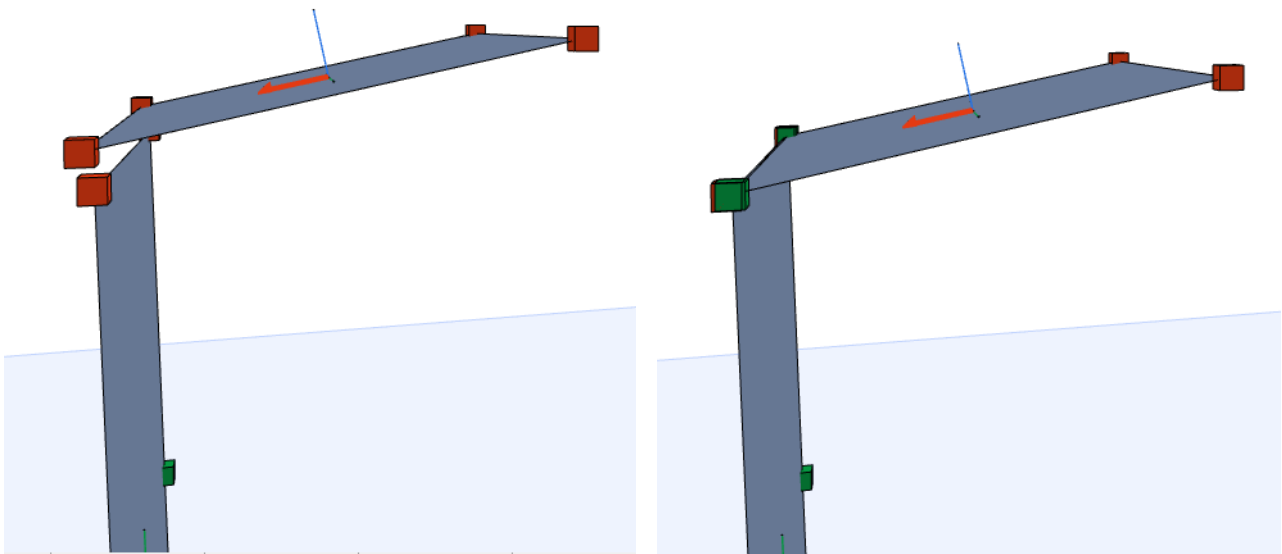
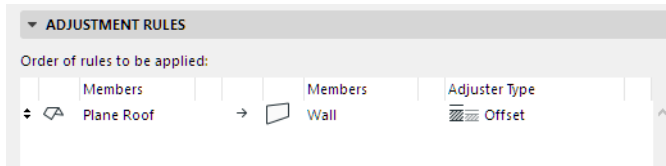
Offset 2D Member

Offset always occurs in one direction (one to the other): the adjusting Members will be offset to their targets; the targeted Members will remain unchanged.

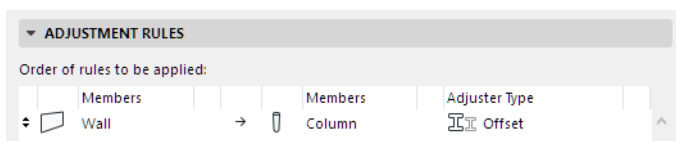
A 2D Member is offset perpendicular to its own plane. This is the local Z direction.

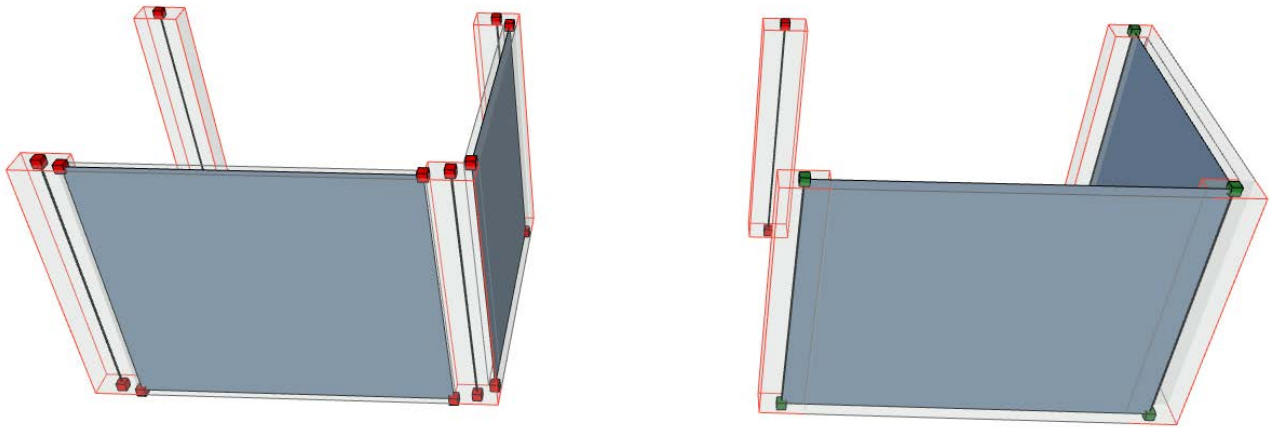
- For each Offset adjuster, you can set a maximum allowed offset distance (“Max. offset”). This distance is measured between unadjusted 1D Members and/or unadjusted 2D Member planes.

In this example, the offset adjuster enables the Roof’s 2D Member plane to connect to the 2D Member of the Wall below:



Roof Plane: Before and After Offset Adjustment





Offset 2D Wall Members to 1D Column

OFFSET 2D MEMBER	
<p>Offset 2D to 1D</p> <p>→</p>	
<p>Offset 2D to 2D</p> <p>→</p>	
<p>Offset only parallel 2D Members</p> <div style="border: 1px solid #ccc; padding: 5px;"> <p> <input type="radio"/> Stretch & Cutback Max. offset <input type="text" value="500"/> </p> <p> <input type="radio"/> Stretch </p> <p> <input type="radio"/> Cutback </p> <p> <input checked="" type="radio"/> Offset only parallel 2D Members </p> <p> In case of multiple targets, offset to element </p> <p> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> with thickest Core </p> </div>	<p>Restrict the 2D-2D offset adjustment to take place only if the 2D Members are parallel</p>

See also [Offset Options In Case of Multiple Targets.](#)

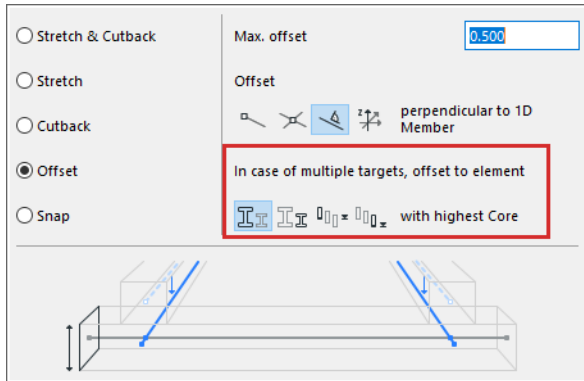
Offset Options In Case of Multiple Targets

Offset always occurs in one direction (one to the other): the adjusting Members will be offset to their targets; the targeted Members will remain unchanged. But what if there are several possible target elements?

In this case, use the “Multiple targets” controls (shown in the images below) to define the preferred group of target elements based on their geometry.

In the dialog box, click on the icons to see a text and illustration for each possibility.

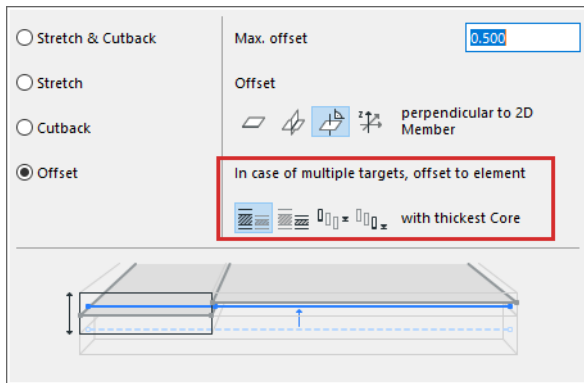
Target Options When Offsetting to a 1D Member



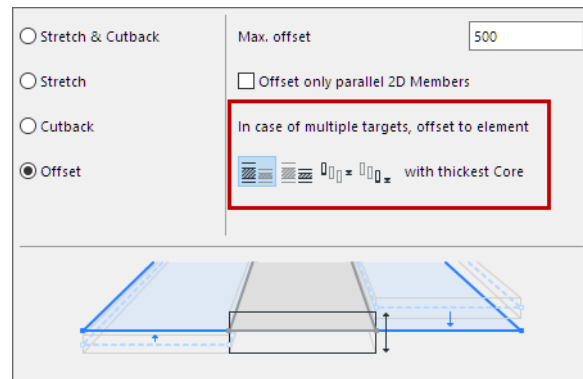
In case of multiple targets, offset to element with

- Highest core
Offset to the target 1D Member(s) whose core *Height dimension is larger*
- Lowest core
Offset to the target 1D Member(s) whose core *Height dimension is smaller*
- Highest position
Offset to the target 1D Member(s) that have the *highest elevation value*
- Lowest position
Offset to the target 1D Member(s) that have the *lowest elevation value*

Target Options When Offsetting to a 2D Member



Offsetting 1D to 2D Member

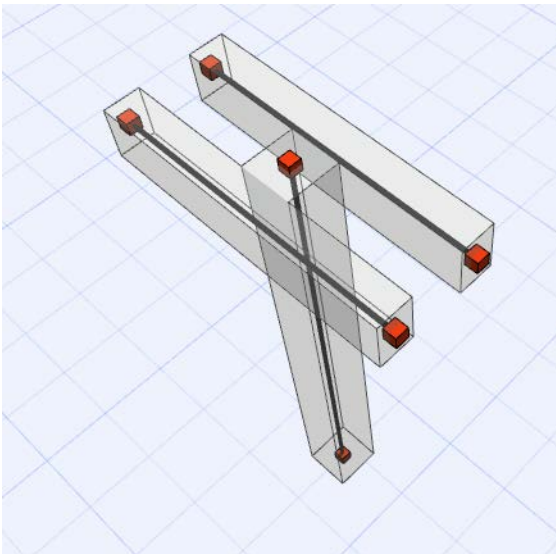


Offsetting 2D to 2D Member

In case of multiple targets, offset to element with

- Thickest core
Offset to the target 2D Member(s) with the *largest core thickness* value
- Thinnest core
Offset to the target 2D Member(s) with the *smallest core thickness* value
- Highest position
Offset to the target 2D Member(s) that have the *highest elevation value*
- Lowest position
Offset to the target 2D Member(s) that have the *lowest elevation value*

If there are still multiple possible targets:



The target is chosen in the following order of priority:

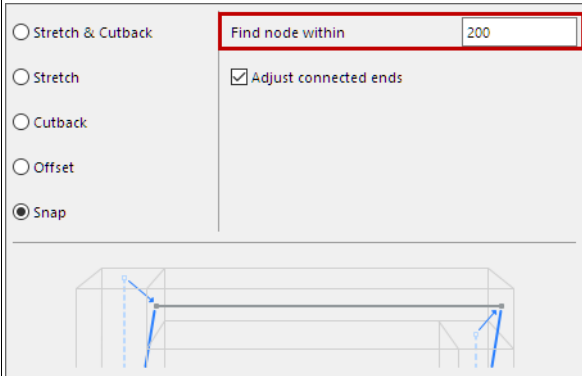
1. Closest by Eccentricity (shorter move distance)
2. Target with stronger Building Material
3. The element that was placed earlier

Snap (1D to 1D)

The Snap adjustment is available only to adjust a 1D Member to another 1D member, “one to the other” only. Snap connects a 1D Member to another nearby 1D Member node, if the Members are not in the same plane. The Snap adjuster combines stretch, offset and rotation. This can be useful, for example, in connecting diagonal bracing to supporting columns.

See also [Example 3: Snap](#).

See also [Adjust Connected Ends](#).

SNAP 1D MEMBER TO 1D MEMBER	
→	
“one to the other” only	
Find Node within	<p>Find the target node within this range. (Nodes located farther away will be ignored.)</p> 

Examples of Adjustment Rules

See the following examples:

Example 1: Offset Beams to Slabs

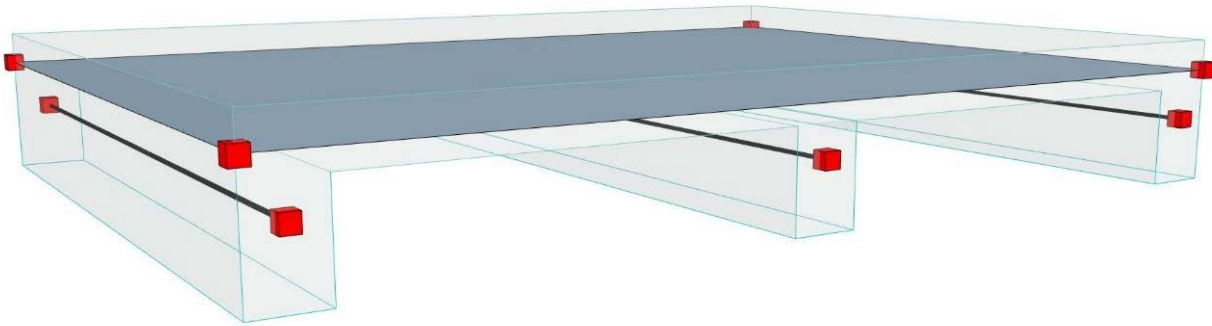
Example 2: Stretch Beams to Columns

Example 3: Snap

Example 4: Cutback

Example 5: Stretch & Cutback

Example 1: Offset Beams to Slabs



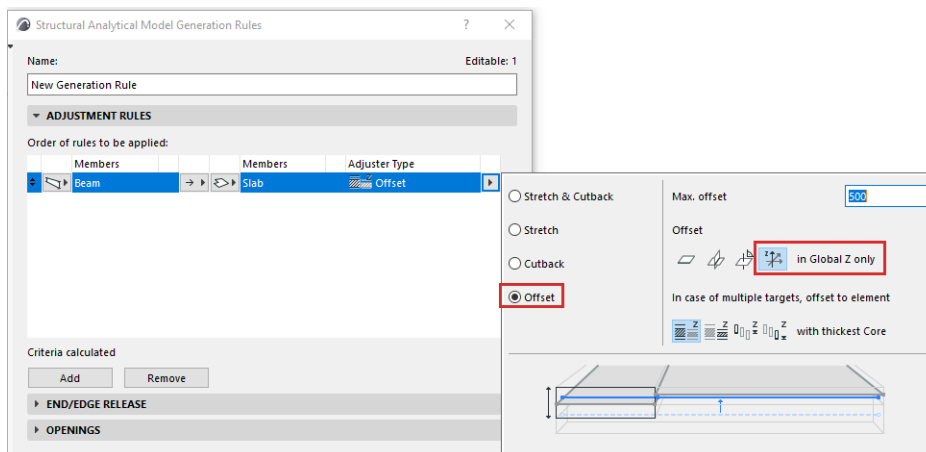
Turning the Structural Analytical Model on, we see that the Beams' 1D Members (axes) are not connected to the Slab: there is no continuity, which is necessary for structural analysis.

The solution is to generate a rule that offsets the Beam axes to the Slab.

- Members Group 1: Beam
- Members Group 2: Slab
- Method: one to the other (adjust Beams to Slabs)
- Adjuster type: Offset
- Additional options, including the maximum allowed offset.

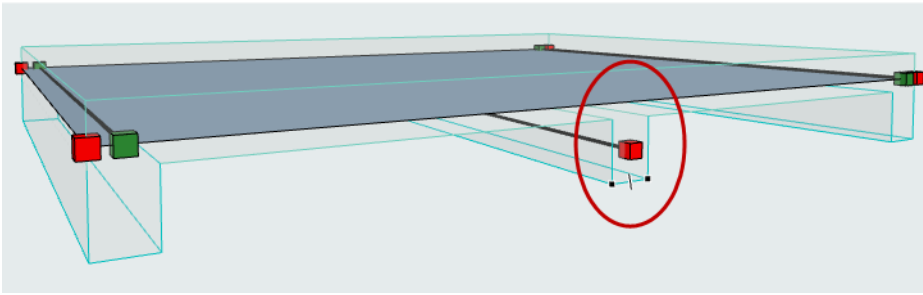
In this case, let's use the global Z direction.

Note: The defined tolerances are length-type values: the units specified for Length in **Preferences > Working Units** apply.

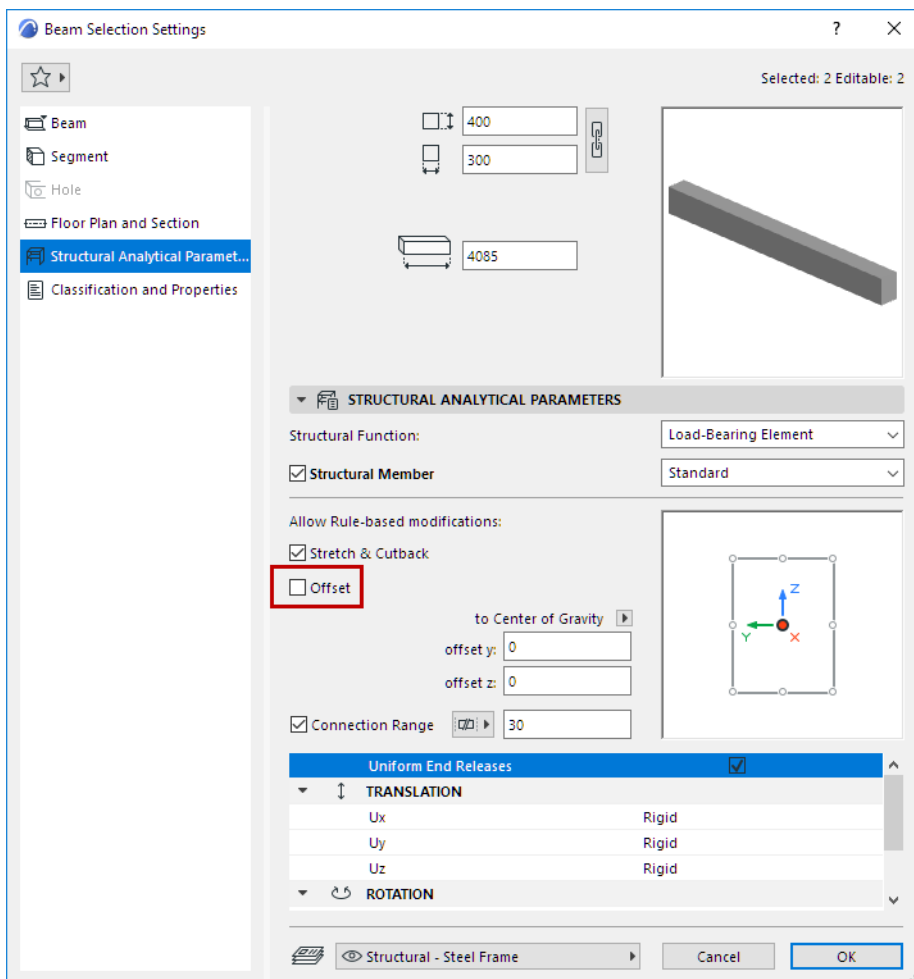


Click OK to apply the rule. As expected, the Beam axes are offset to the plane of the Slab (straight up, in the Z direction).

But one of the Beams' axes (the center Beam) stayed in place. This seems to be an error.



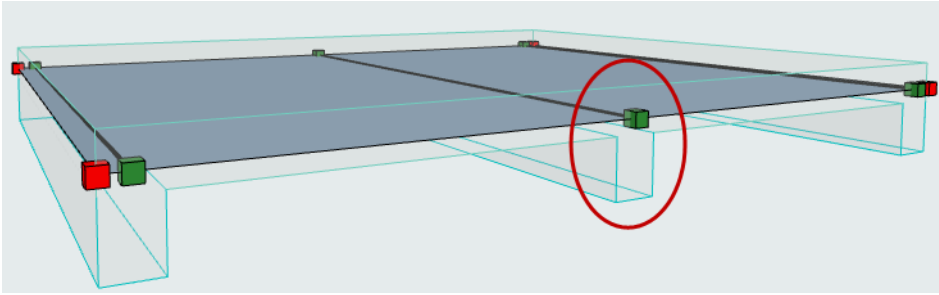
Select the center Beam and check its Settings. Here, it is clear that this Beam did not participate in the Offset rule, because the element-level Offset setting, in Beam Settings - Structural Analytical Parameters panel, is unchecked.



To fix the problem:

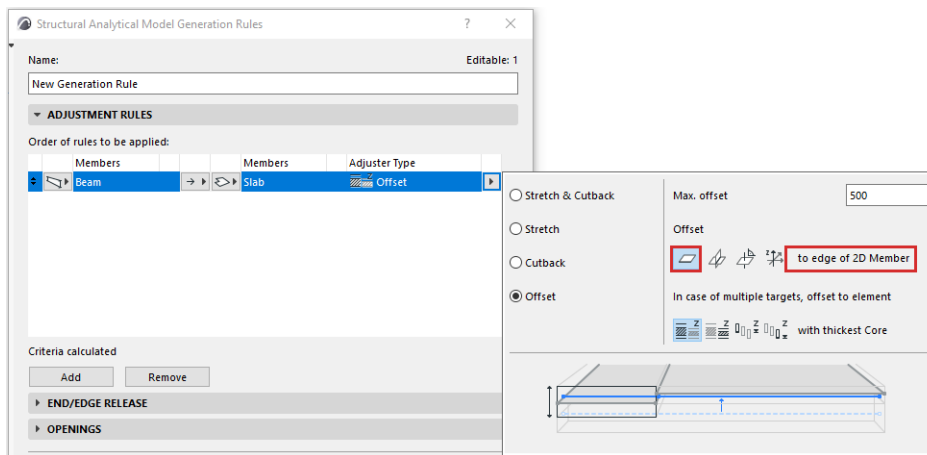
- Activate the **Offset** checkbox in Beam Settings.
- Click OK to close the Settings dialog.

Now the model automatically applies the defined Generation Rules. The Beam axis is now offset, according to the rule:



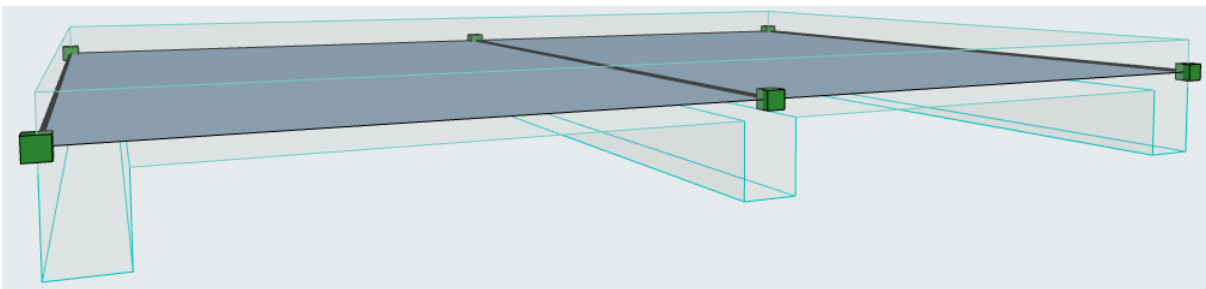
The current rule offsets Beams to Slabs in the Z direction. This model can be considered structurally correct. You may prefer a different option: Beam axes should be offset to the edge of the Slab.

Adjust this option in Generation Rules:



Click OK to apply the rule.

As expected, the axes of the two edge Beams are offset to the edge of the Slab plane (instead of straight up, as in the previous example, in which only "Global Z offset" was enabled).



If the element could be offset to several target elements, define which element should be the target.

The preview drawing helps you understand each option:

ADJUSTMENT RULES

Order of rules to be applied:

Members	Members	Adjuster Type
Beam	Slab	Offset

Criteria calculated

Add Remove

- END/EDGE RELEASE
- OPENINGS

Stretch & Cutback
 Stretch
 Cutback
 Offset

Max. offset:

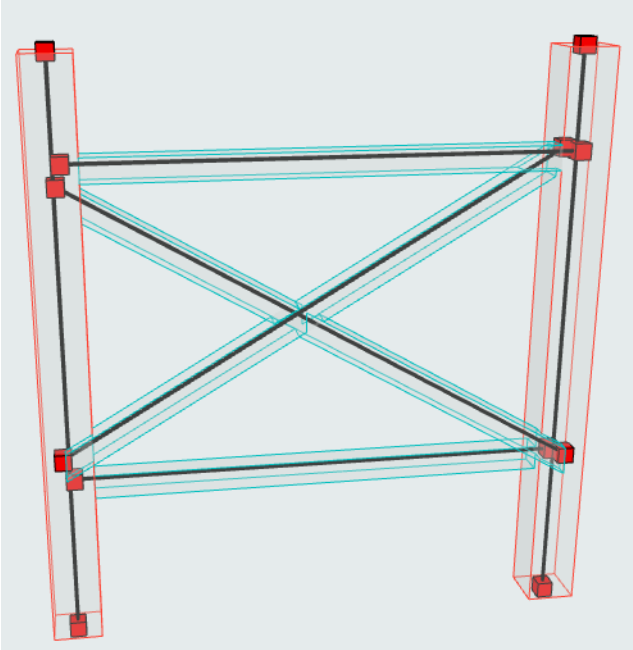
Offset: to edge of 2D Member

In case of multiple targets, offset to element with thickest Core

[See Offset Options In Case of Multiple Targets.](#)

Example 2: Stretch Beams to Columns

To illustrate this adjuster type, let's use a common steel structure composed of Columns, Beams and diagonal bracing:

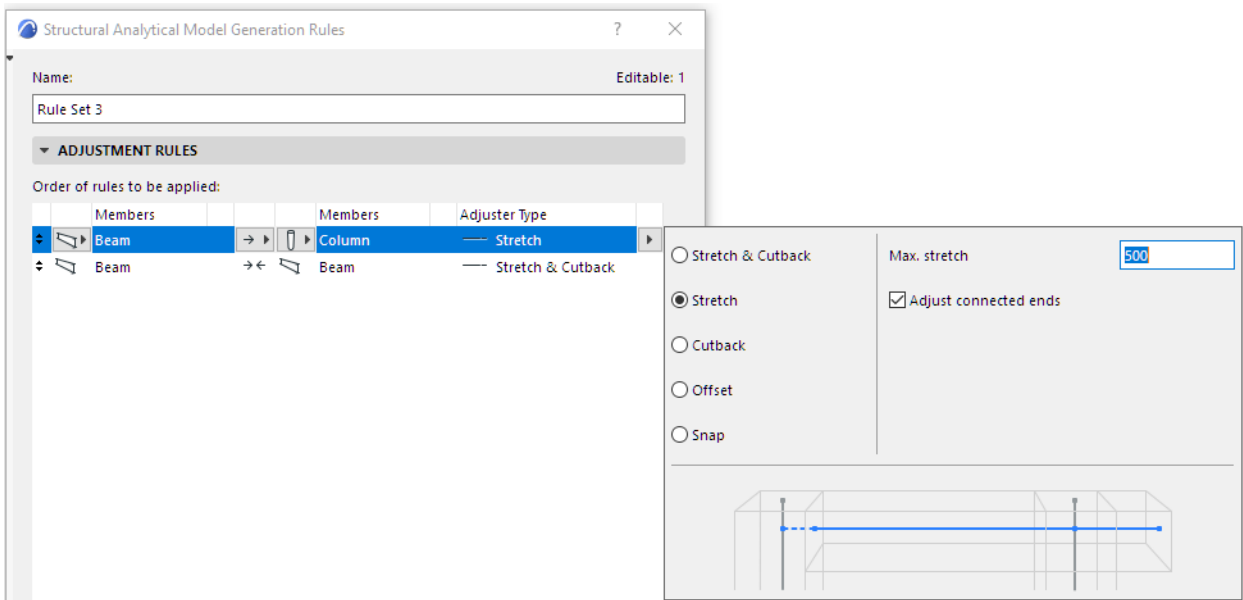


We see that the Beams' 1D Members are not connected to the Columns: there is no continuity, which is necessary for structural analysis.

The solution is to generate a rule that stretches Beam axes to the Columns.

- Members Group 1: Beam
- Members Group 2: Column
- Method: one to the other (stretch the Beam to the Column, but leave the Column as is.)
- Adjuster type: Stretch
- Additional options for stretch: the maximum stretch allowed.

Note: The defined tolerances are length-type values: the units specified for Length in Preferences > Working Units apply.



The Beams now meet the Columns, but the bracing is still not right.

See the next section to continue: [Example 3: Snap](#).

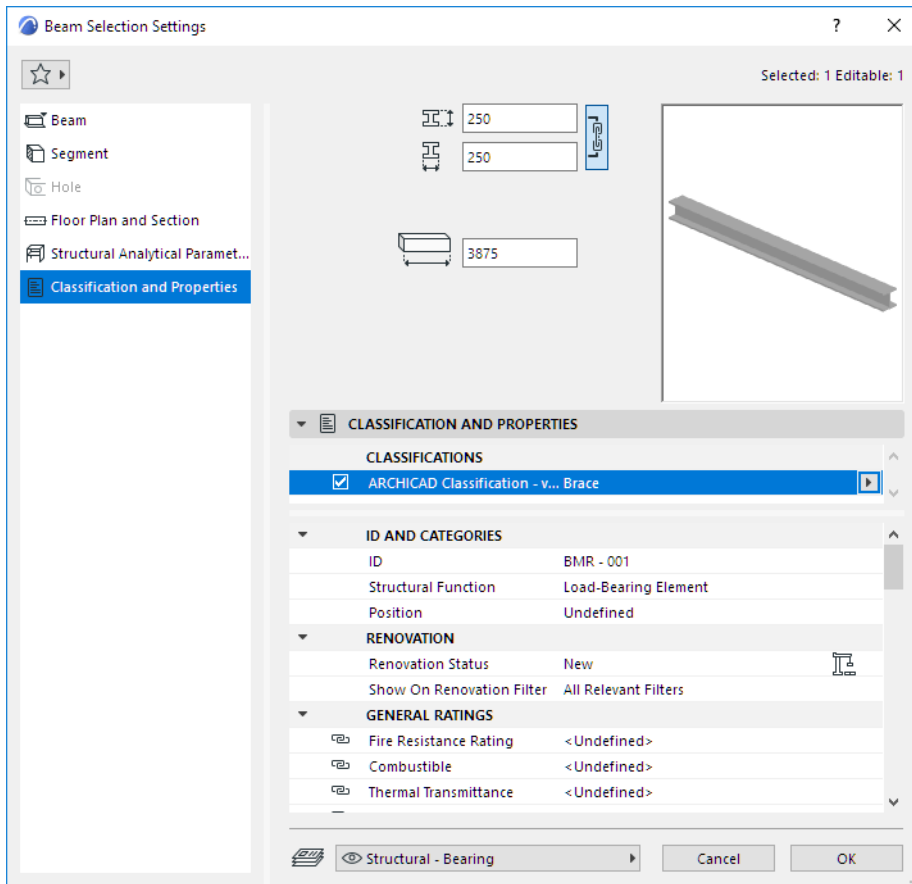
Example 3: Snap

Let's continuing working on the previous example. The diagonal bracing still does not meet the continuity requirements, which in a later stage might lead to some issues in the structural analysis software.

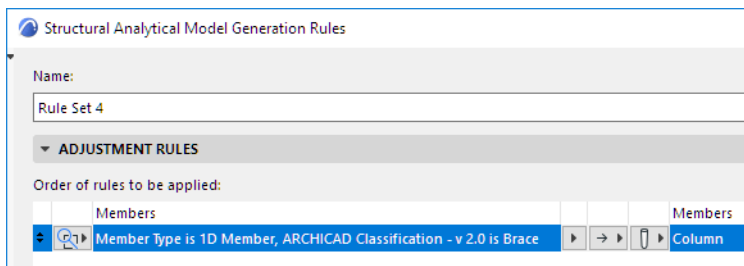
To adjust these, the user should use the snap adjuster.

This allows the structural axis to rotate in order to connect its end nodes to the Beam-Column connection generated with the stretch adjuster.

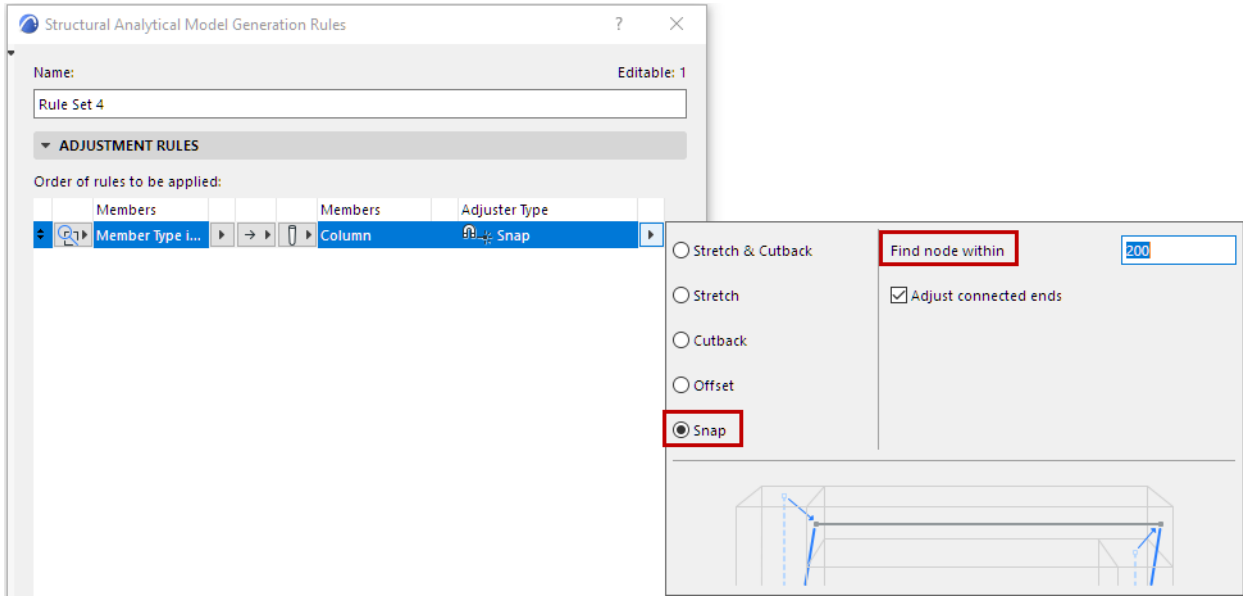
Notice that this adjuster should be defined to work only for the Beams that are being used to model bracing elements. In this project, such Beams are classified as "Brace".



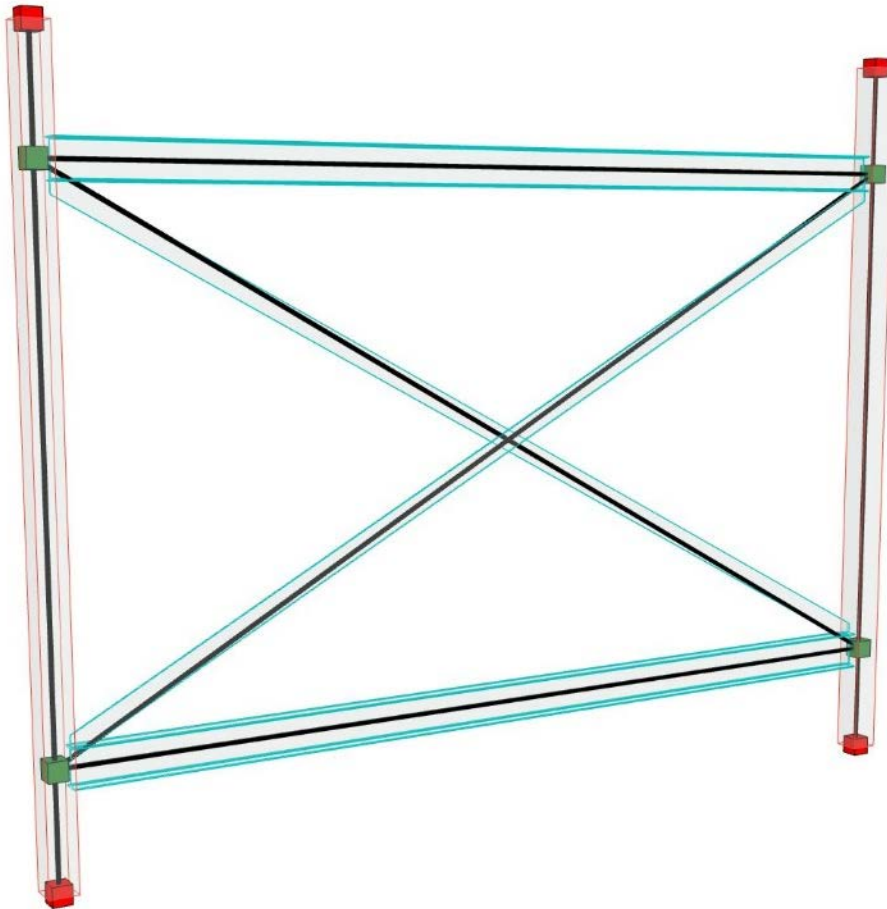
Define the Snap rule to adjust only the 1D Members which are classified as "Brace":



This adjuster type allows you to define the maximum distance of the targeted snap node.

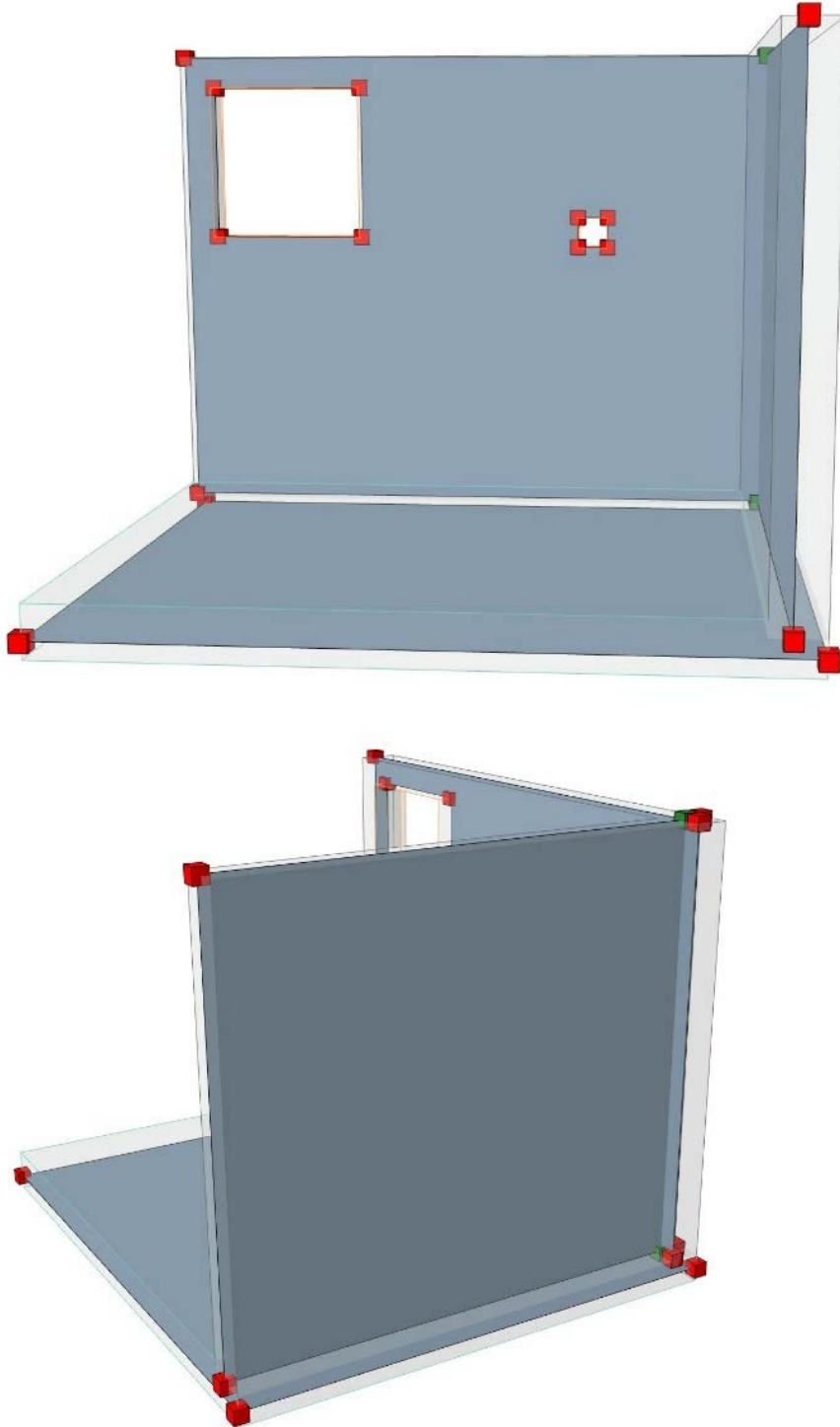


The diagonal bracing will snap to the Columns.



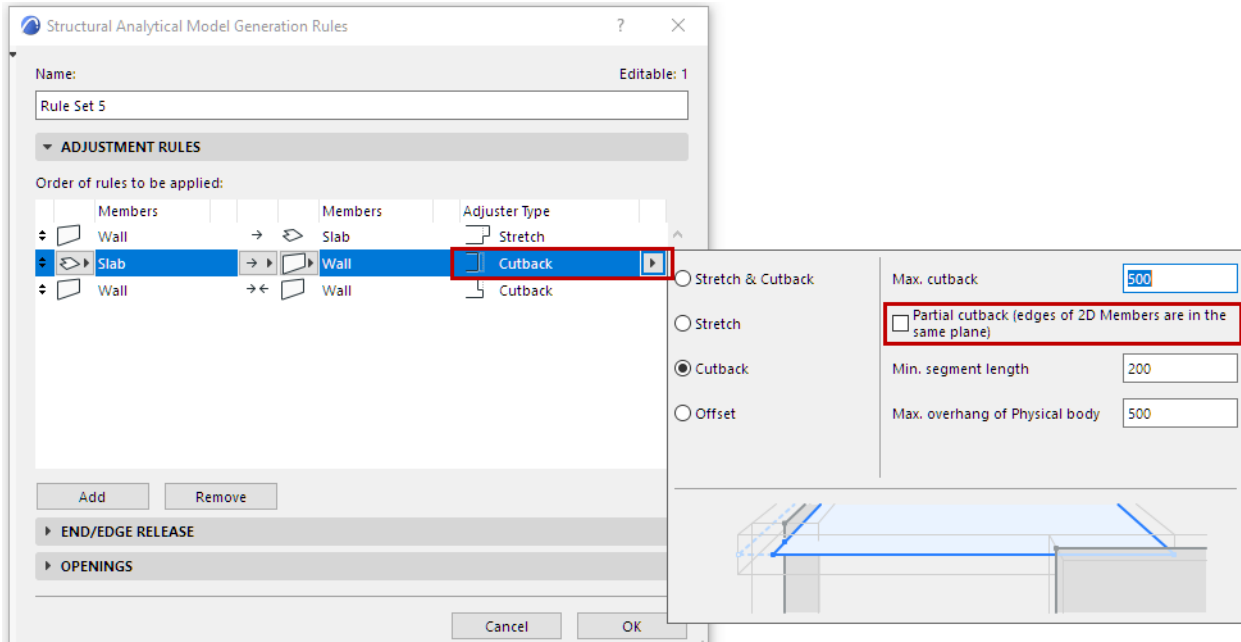
Example 4: Cutback

This adjuster is the opposite of Stretch: it cuts back the Structural Analytical Members which overlap. Turning on the visibility of the Structural Analytical Model, it is possible to detect the following issues:

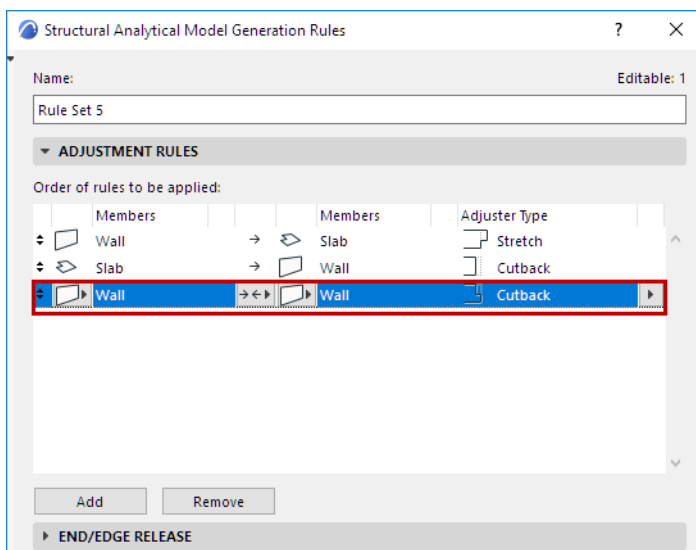


- Gap between Walls and Slab
- Slab edge is not aligned with the Walls;

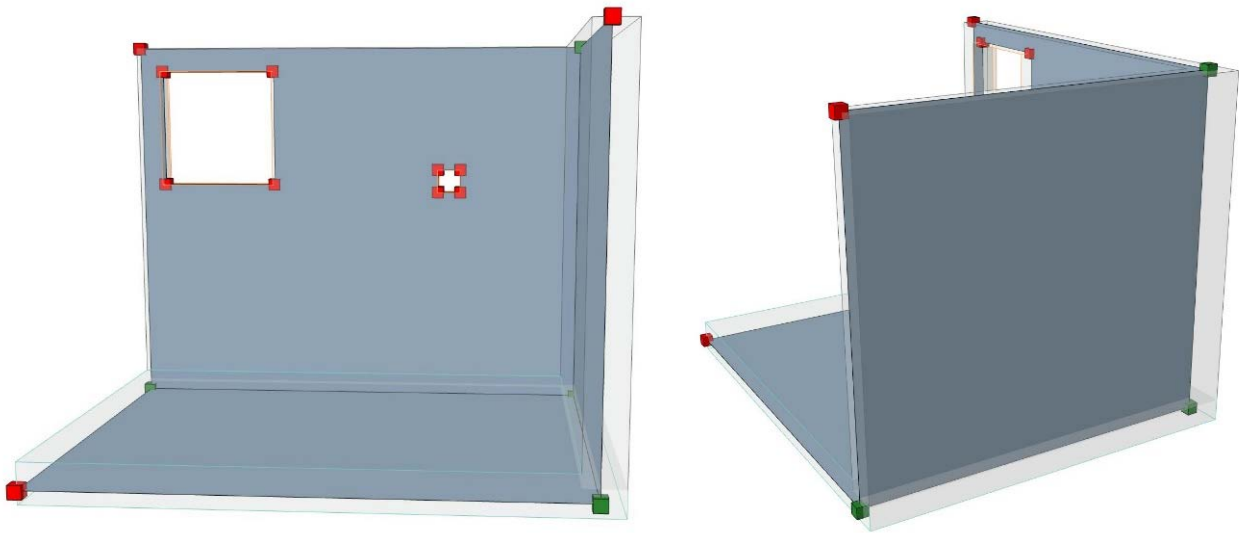
- The Walls overlap each other at the corner.
1. To solve the first issue, stretch Walls to Slabs using the Stretch adjuster.
 2. Next, cut back the excess Slab to align it to Wall edges, using the Cutback adjuster
 - Parameters: In this example, we are using a full edge cut, since the entire Slab runs along the full Wall length. Thus, uncheck the “Partial cutback” option.



3. To solve the third issue, we will cut back both Wall edges that intersect and overlap in the corner. Here, Walls act as both modifiers and targets, and the operator is “to each other”, since both Walls must be analyzed and cut.

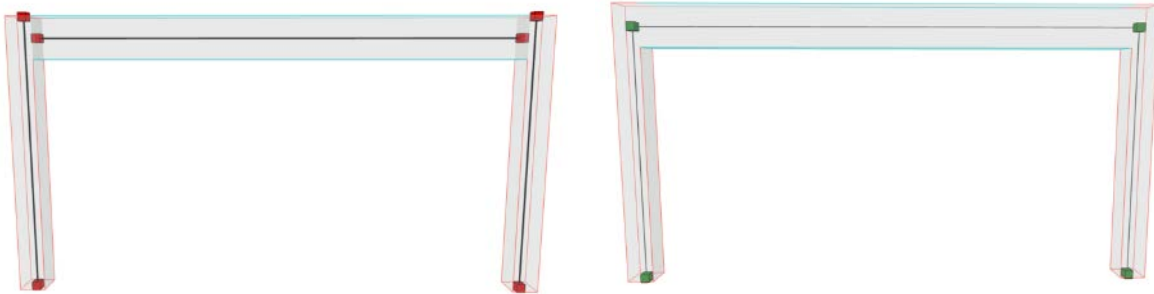
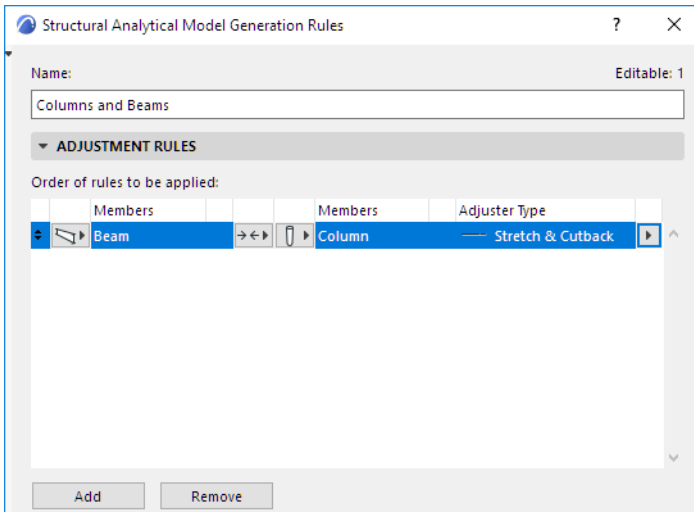


The adjusted structural analytical model should appear as shown below.



Example 5: Stretch & Cutback

The “Stretch & Cutback” adjuster performs both these adjustments in one step. This adjuster can be used in one direction or both directions.

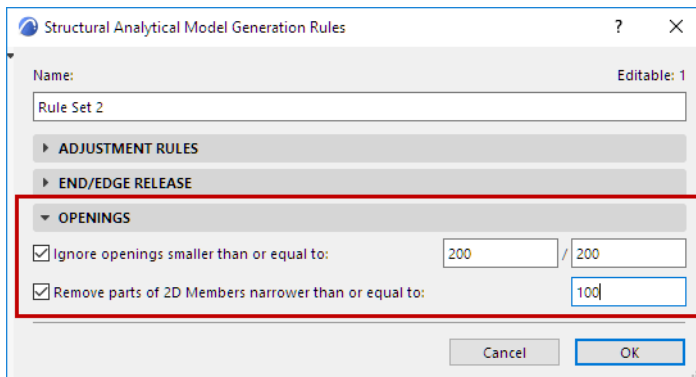


Stretch & Cutback Adjustment to Beams and Columns

Filter Openings for Structural Analytical Model

These settings filter 2D Members of the Structural Analytical Model to account for small holes and narrow strips which are unimportant structurally, and can be ignored in the exported model.

Use the **Openings** Panel of the Structural Analytical Model Generation Rules dialog box.



Ignore openings smaller than or equal to

Check this to define whether **small openings in 2D Members** should be ignored in the Structure Analytical Model.

In this context, “openings” can be any of the following:

- In the 2D Member of a Slab or Roof: Opening, Skylight, inner polygon of Slab, hole as a result of Solid Element Operation
- In the 2D Member of a Wall: Opening, Door, Window, hole as a result of Solid Element Operation

In the Generation Rules dialog, enter two dimensions to define a “small” opening.

Any opening of these dimensions or smaller will be ignored in the Structural Analytical model, and will not be represented.

To exempt any particular 2D Member from this rule: you can disable the “Ignore small openings” filter in element Settings.

[See *Allow Rule-based modifications - 2D Member*.](#)

Remove 2D Member parts narrower than or equal to

Check this to remove very **narrow parts of a 2D Member** from the 2D Member surface.

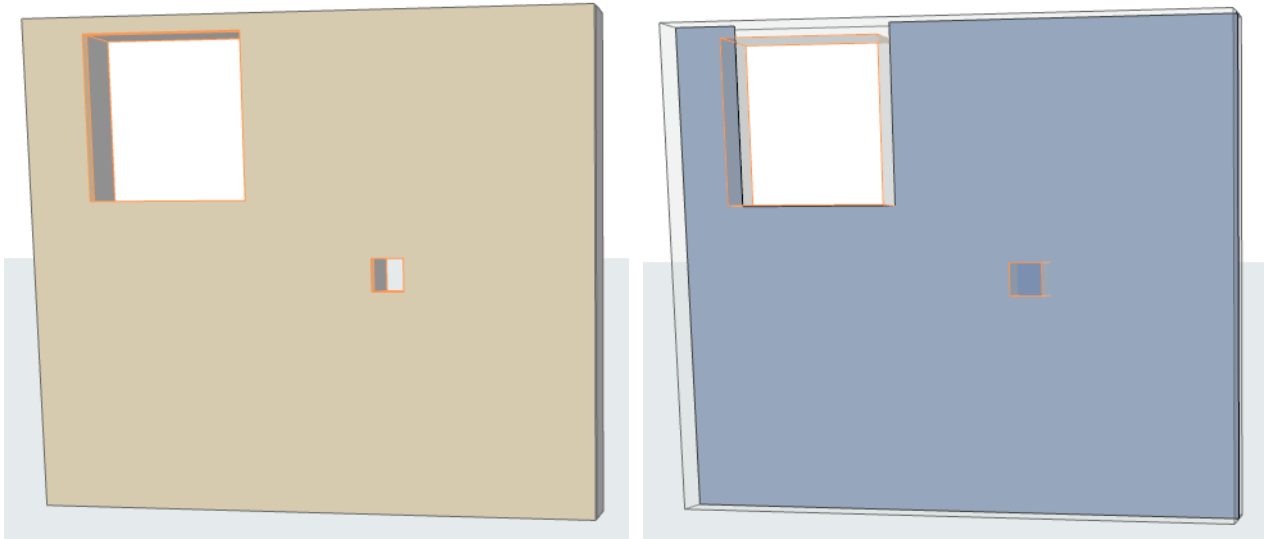
For example, the narrow strip left over between two openings, or the Wall’s remaining part after the window opening has been cut out).

This “Remove narrow parts” filter cannot be disabled at the element level.

Example

Let’s assume that a Wall contains two openings, one for a window (1000x1000mm) and another for a ventilation duct (200x200mm). For structural analysis purposes, the opening for the ventilation duct does not have a significant impact on the structural behavior of the Wall: thus, the structural analysis software should ignore this opening.

After applying these Rules, we see the result: the smaller opening is not considered in the Structural Analytical model, and the narrow Wall part (adjoining the larger opening) is also excluded.



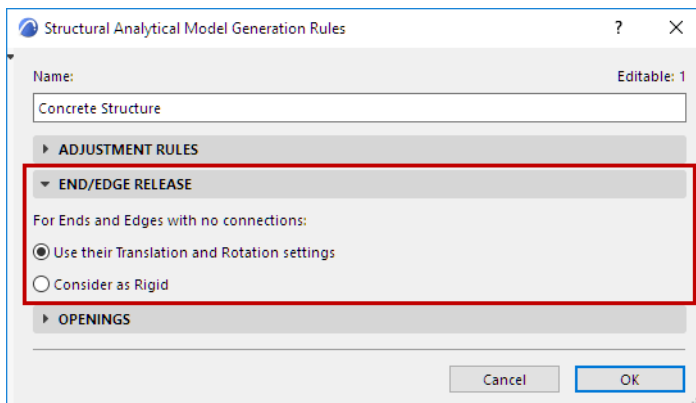
Effect of Opening Filter on 2D Member

End/Edge Release (Generation Rules)

This is a panel of the Generation Rules dialog box.

This control defines how the structural analysis software should interpret Release settings values of Ends or Edges *which have no connections*.

Some structural softwares automatically consider all such disconnected ends/edges to be “rigid”. If you are using such a software, you might have to change this setting in this panel. See details below.



For Ends and Edges with no connections:

Use their Translation and Rotation settings (default setting)

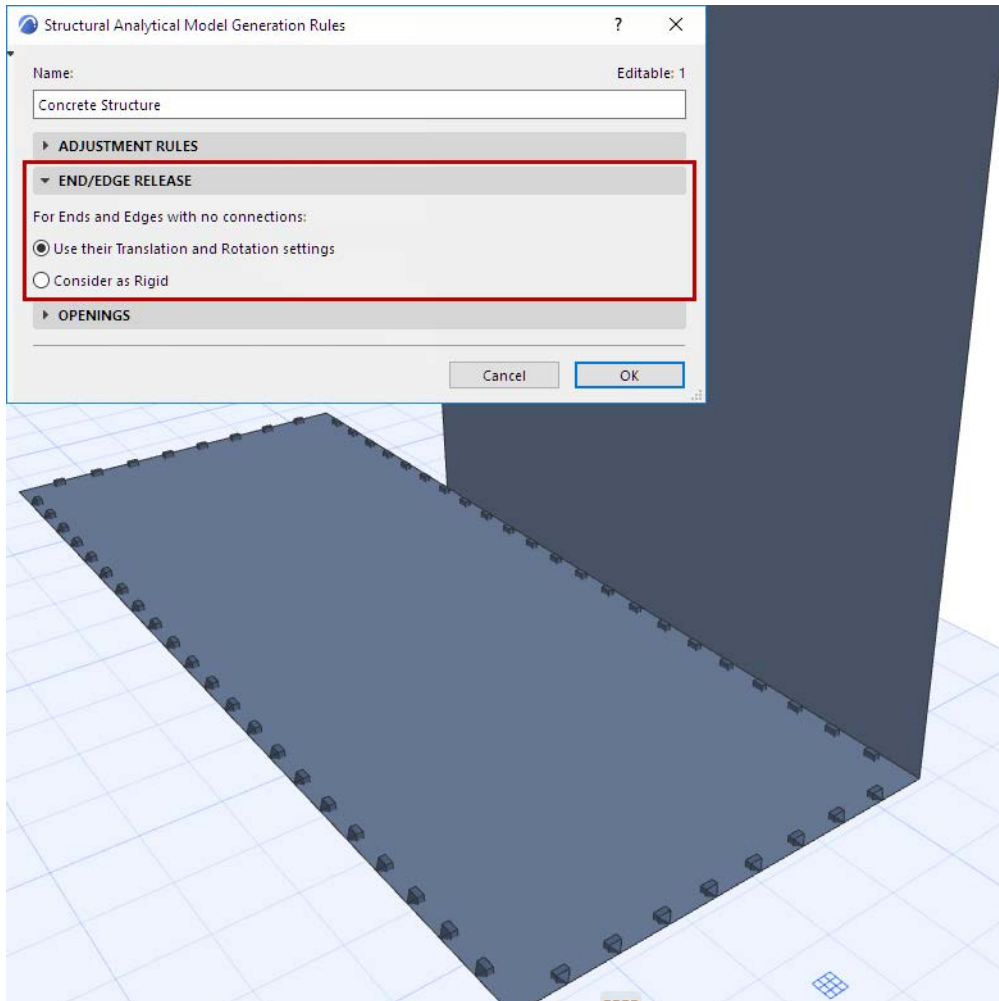
By default: the program looks at the End/Edge Release settings at the element level.

If you have customized any Translation or Rotation value, these are taken into account when the Structural Analytical Model is exported in SAF format.

[See *End/Edge Releases*.](#)

In the exported SAF file, the values at RelConnectsSurfaceEdge are “Free” or “Custom”, reflecting the element-level End/Edge Release settings.

In the model, Release symbols are shown on all edges with custom values, as usual.



Custom Edge Releases

Consider as Rigid (Affects *Unconnected* Ends/Edge Segments)

Certain receiving softwares consider all unconnected Ends, Edges and Edge segment to be Rigid.

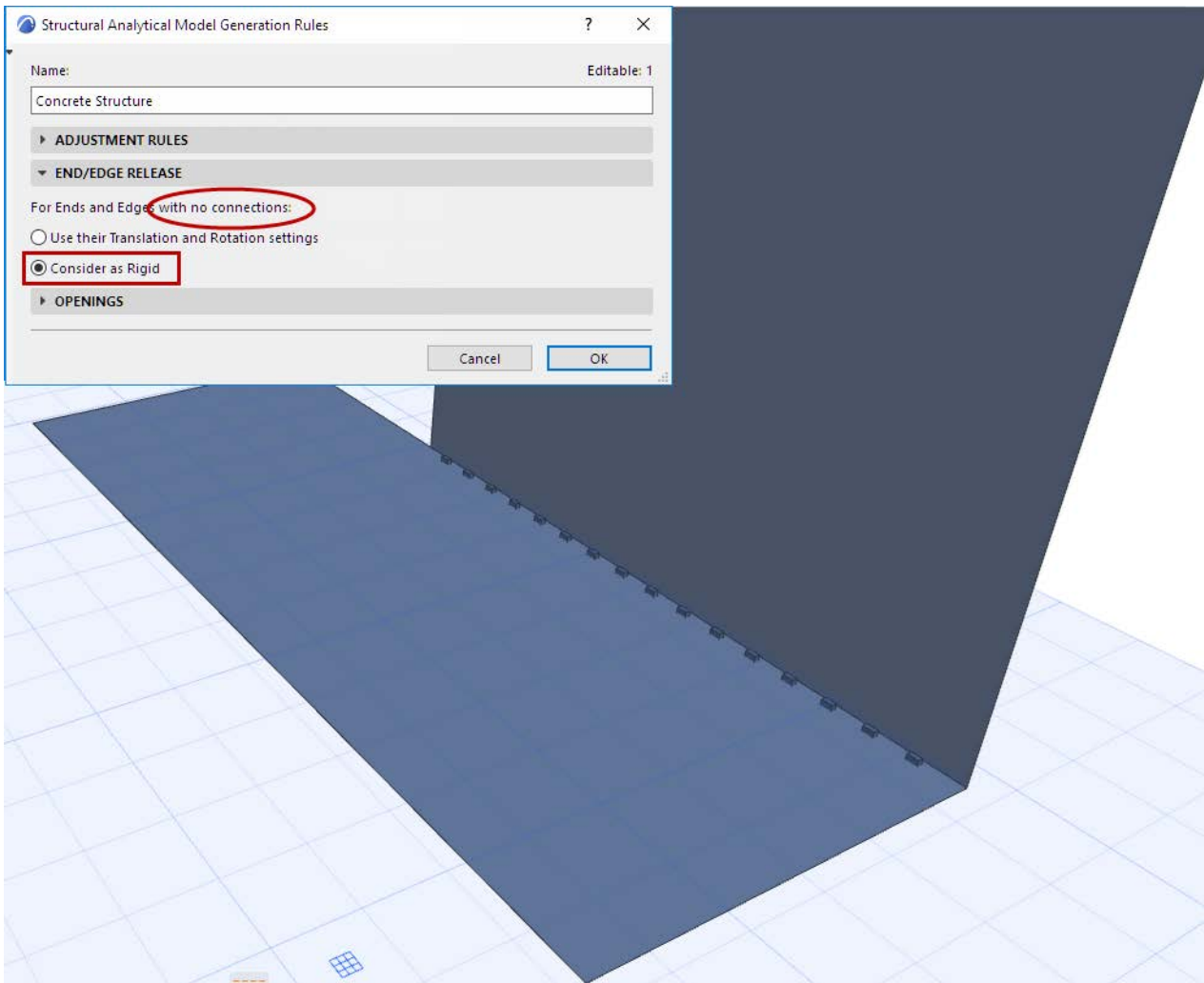
You may wish to export your model from Archicad so that it is compatible with such a software.

In the End/Edge Release panel: Choose **Consider as Rigid**.

As a result:

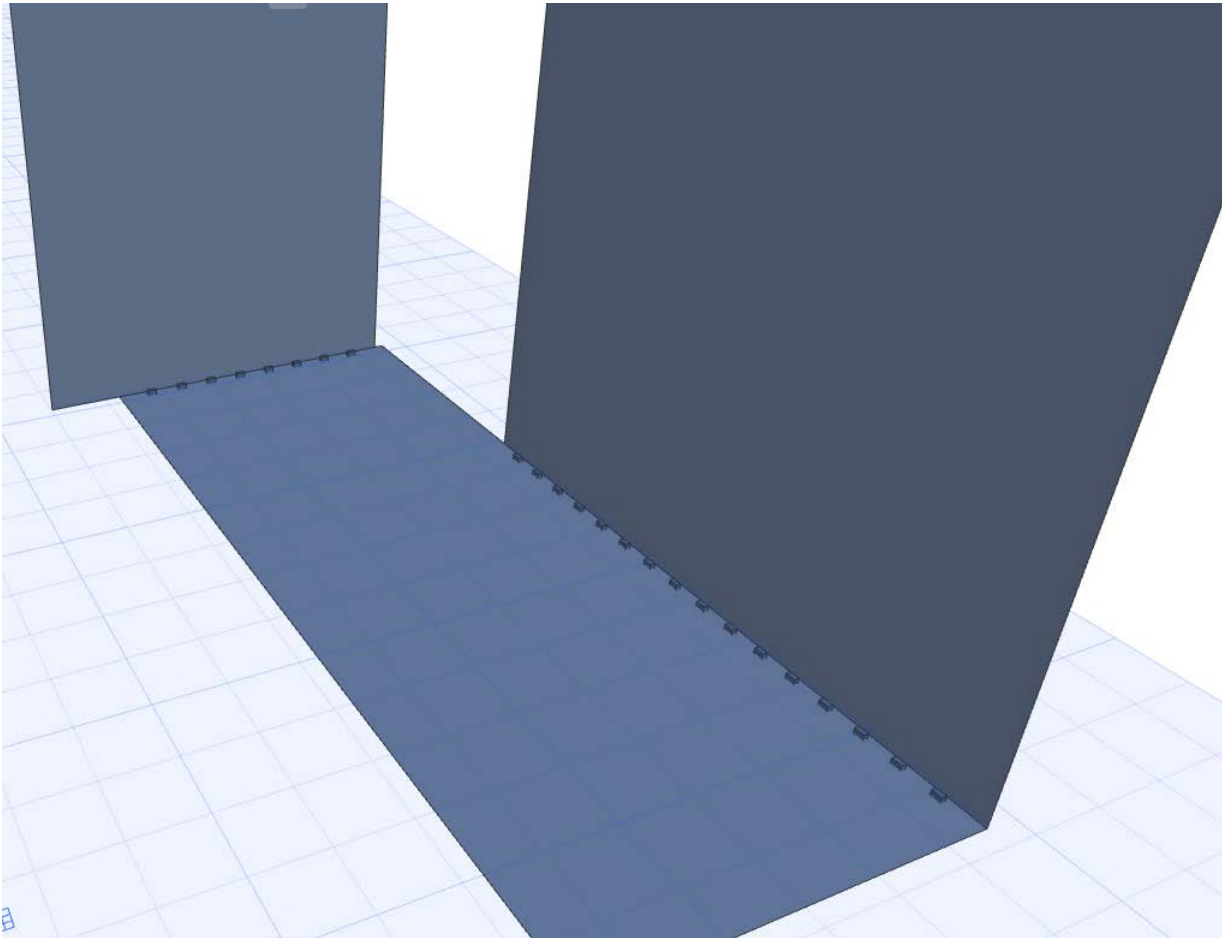
- All *unconnected* Ends, Edges and Edge segments are exported as "Rigid", regardless of the release values set at the element level.
- The SAF file values at RelConnectsSurfaceEdge are "Rigid", even if the element-level End/Edge Release settings have been set to different values

- Customized Edge Release values will be generated, and the Release symbols shown in the model, only where Members are *connected* to another Member.



Custom Edge: Shown Along Connected Segment Only

If you later add additional Structural Analytical Members to the model, the *connected* edge (segments) will be exported with their customized release values, and will be displayed where applicable.



Eccentricity

1D Members are generated at the **center of gravity** of the **element's core**

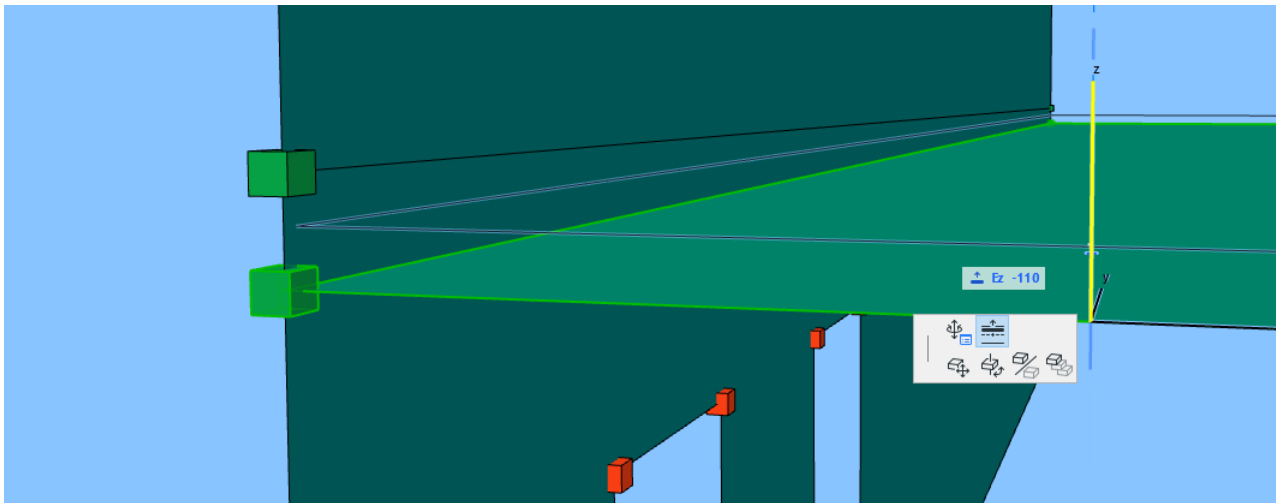
2D Members generated along the **center** of the element's **core plane**.

Any deviation of Member position from this default is called "Eccentricity".

Eccentricity values have no effect on the Physical Model, but are significant for the Structural Analytical Model.

Adjust Eccentricity (Graphically)

1. Select the Member.
2. Activate the Eccentricity command from the pet palette. (Not needed or available for 1D Members if the Physical Model is not currently visible).
3. Drag the Member to its new location (enter Tracker values as needed).



Notes:

- For 2D Members, Eccentricity/Offset occurs perpendicular to the Member plane (regardless of any customized edge angles).
- For any selected Member, the eccentricity value is shown in the Element Info Tag (as well as Settings and Info Box).



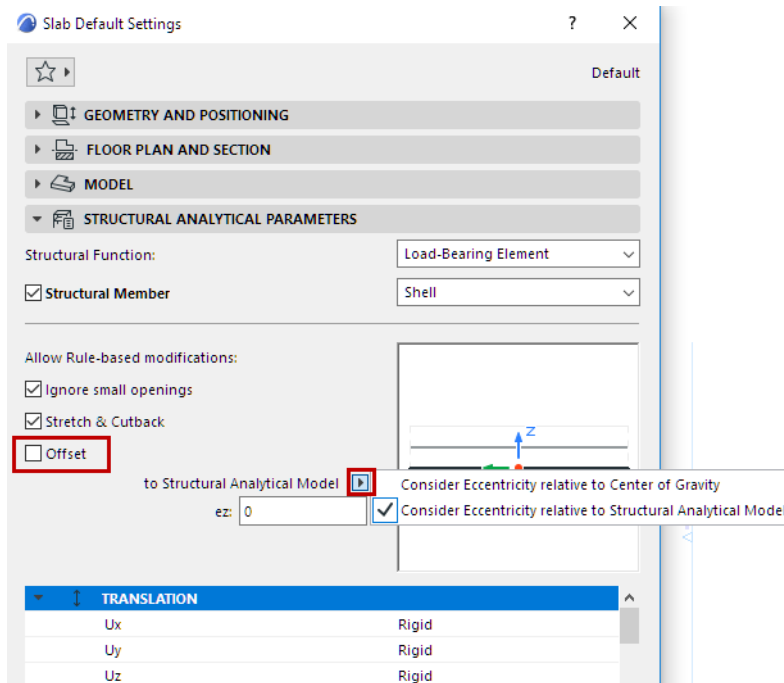
- Once you have dragged a Member out of its original position, the global Adjustment Rules no longer apply to it; the "Offset" checkbox in its Settings dialog is no longer activated. To reapply the global Adjustment Rules, go to its Settings dialog and reactivate the Offset checkbox.

See [Allow Rule-based modifications - 1D Member](#) or [Allow Rule-based modifications - 2D Member](#).

Adjust Eccentricity (Element Settings)

1. Select the 1D or 2D Member.
2. Open the Settings dialog (Structural Analytical Parameters panel).
3. Uncheck the **Offset** option, to enable editing. (Unchecking this box means that the global Adjustment Rules are not applied to this element, and you are adjusting it manually.)
4. Choose a method for measuring Eccentricity:
 - relative to the element's Center of Gravity, or
 - relative to the element's Structural Analytical Model

The two approaches have the same result, but the values are defined using opposite signs.



5. Define the eccentricity values by doing either of the following:
 - Click in the preview window to enter the new location at any bounding box node
 - Enter offset values manually

Multiple Eccentricity Values of Imported Elements

Structural Analytical Format version 1.0.6 or later can handle varying and multiple eccentricity values at each end of the 1D Member.

Archicad does not support multiple eccentricity values at each end. When you import such a model, Archicad converts the incoming elements by assigning an average eccentricity value, where applicable.

In these cases, a warning alerts you to the conversion.

Related topics

[Adjustment Rules](#)

[Manual editing of analytical members](#)

Manual editing of analytical members

Use manual editing commands to quickly adjust the geometry of Member nodes and edges to achieve a continuous model.

Switch on the Structural Analytical Model in Archicad. Select a 1D or 2D Member, then use one of these commands from the pet palette:

Stretch 1D Member

Move node of 2D Member

Offset 2D Member edge

Insert new node in 2D Member

General info

- You can only manually edit one Member at a time (no multi-select).
- Commands are available on Floor plans, Sections, Elevations and 3D views - as long as they make sense in the given view.
- If a Member is manually edited, it no longer acts as an adjuster. However, the manually edited Member can still act as a target of the Adjustment Rules.
- The manually edited Member moves (or rotates or mirrors) together with the physical model element.

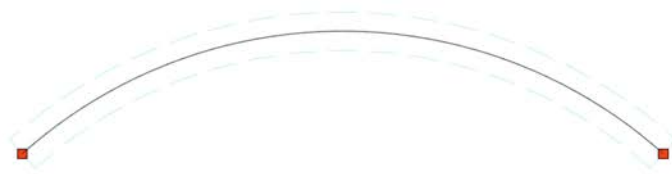
Stretch 1D Member

1. Select the 1D Member and click a node.
 2. From the pet palette, choose the applicable Stretch command:
- Stretch Member length



Straight beam - manual stretch

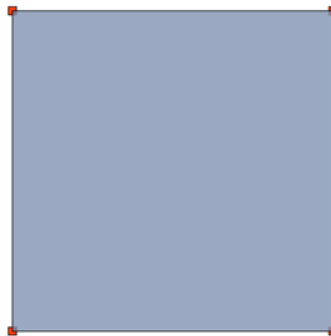
- Angular Stretch Member length (curved Beam): Adjust end of 1D Member while keeping it on the same curve.



Curved beam - manual stretch

Move node of 2D Member

1. Select the 2D Member and click the node. (Not available for straight Wall on Floor Plan)
2. From the pet palette, choose **Move Member's node**.

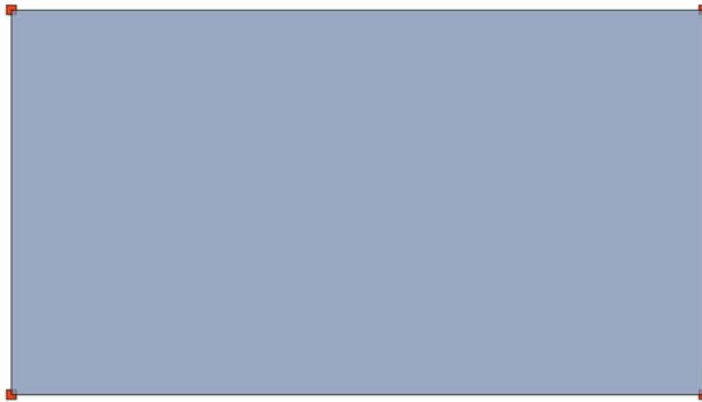


Slab - move node

Offset 2D Member edge

1. Select the 2D Member and click the edge. (Not available for straight Wall on Floor Plan)
2. From the pet palette, choose **Offset Member's edge**.





Slab - offset edge

Insert new node in 2D Member

1. Select the 2D Member and click the edge. (Not available for straight Wall on Floor Plan)
2. From the pet palette, choose Insert new Node.



Slab - insert node

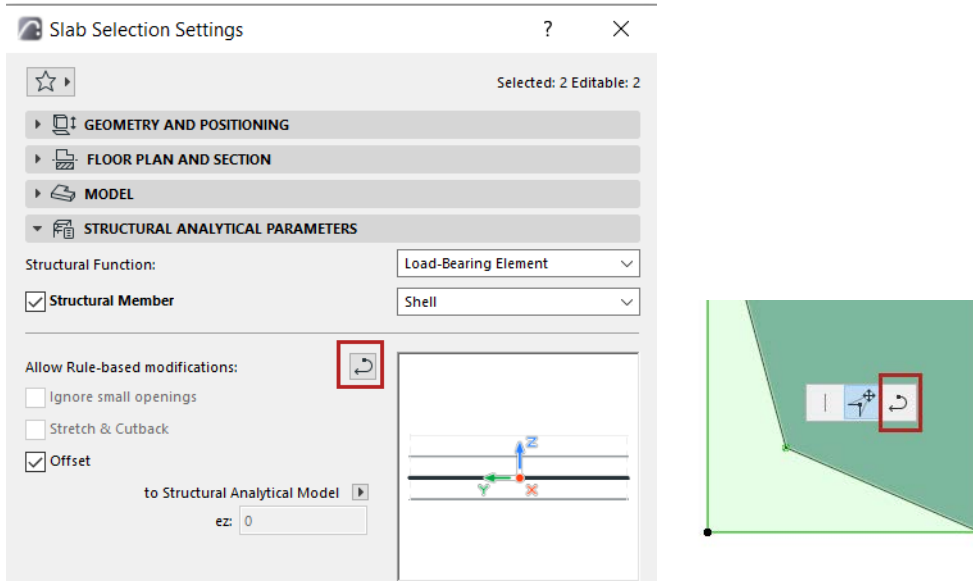
To delete a node, drag it onto another node. (There is no dedicated interaction to delete nodes.)

Revert manual editing

Revert any manual editing to the Member's original geometry.

For a selected element whose Structural Member you edited manually, find the Revert command in:

- Pet palette
- Element Settings or Info Box (Structural Analytical Parameters panel)



The Revert command does not affect the manual Eccentricity or Edge Release settings.

Properties of manually edited Members

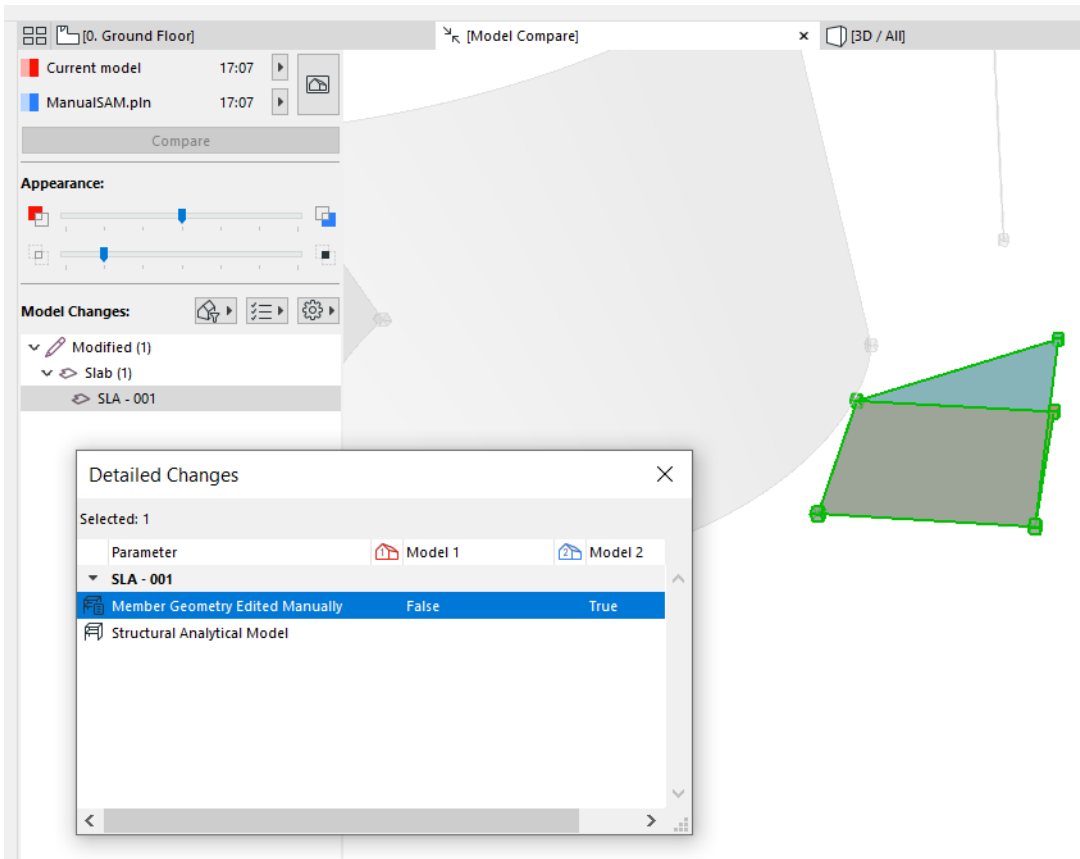
Element properties are available to help you identify 1D and 2D Members that have been manually edited:

For 1D Members: Member Stretched Manually

For 2D Members: Member Geometry Edited Manually

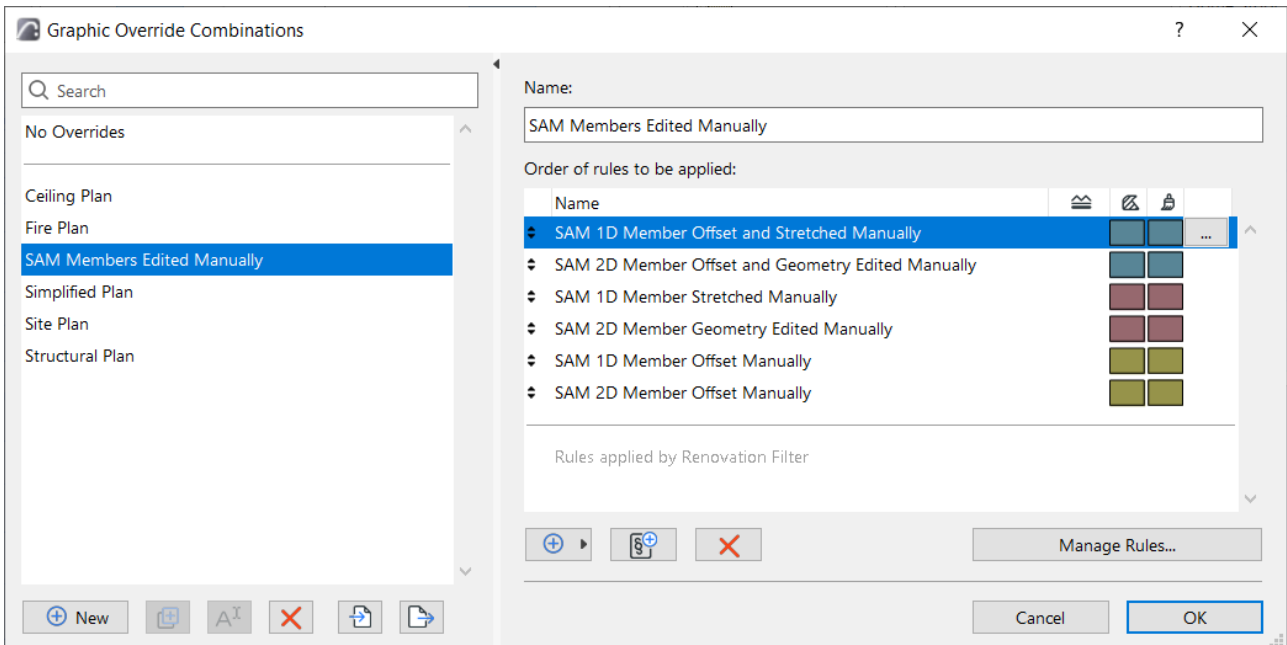
Use these properties throughout the Archicad interface, including:

- Find & Select
- Property Manager and Expression Editor
- SAM Model Compare (Detailed Changes)



View manual edits in Structural Model Compare

- Graphic Override Rules and Criteria - Use a Graphic Override Combination for instant visual feedback on manually edited Structural Members.



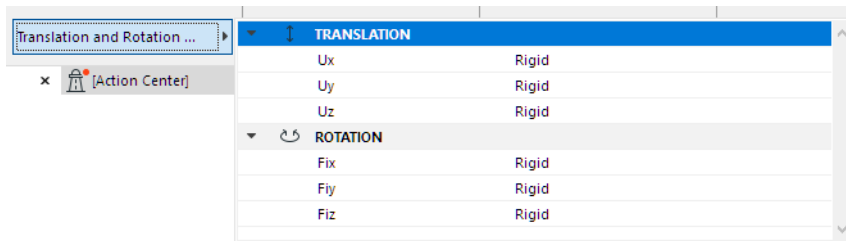
End/Edge Releases

For each 1D and 2D Member in the Structural Analytical Model, you can set End and Edge Release values, respectively.

Translation (U) and Rotation (Fi) values express the transfer of forces at the ends or edges of a Structural Member, in the three directions of its local coordinate system (x, y and z).

These values have no effect on the Physical Model, but are essential for structural analysis.

End/Edge Release controls are available in the Structural Analytical Parameters panel of applicable element Settings dialogs and the Info Box.



- For 1D Members: set Translation and Rotation values for Ends (either uniform values, or separately for each end).
- For 2D Members: set Translation and Rotation values for all Edges.

Notes:

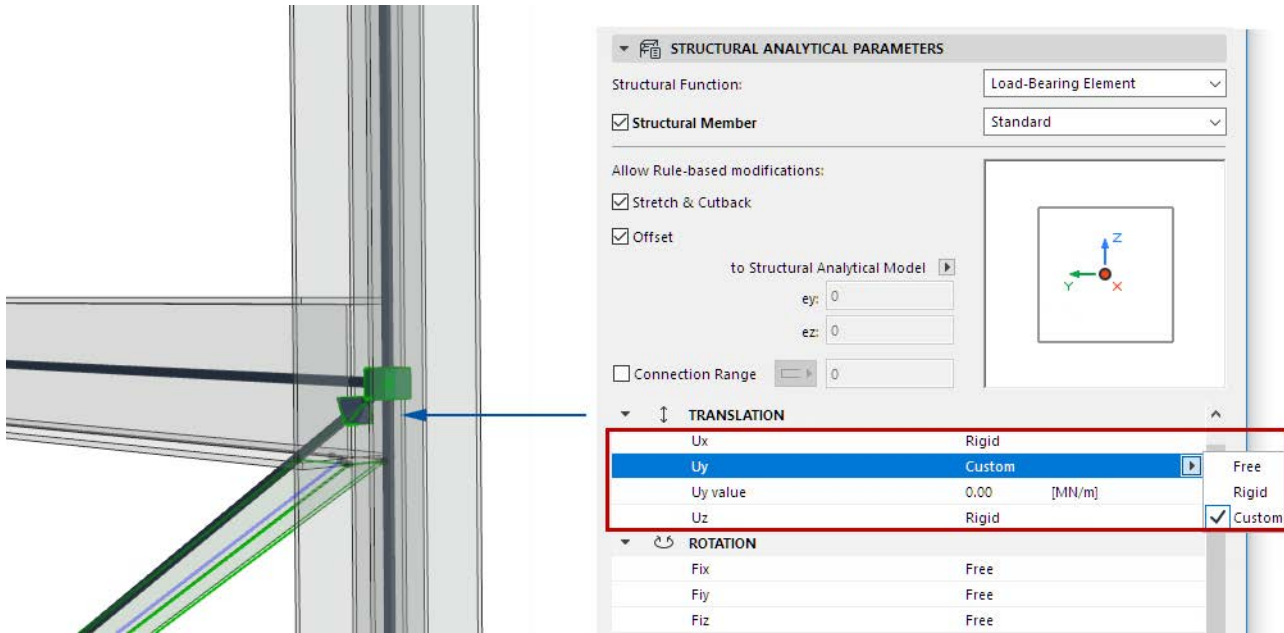
- To set custom edges for 2D Members, use the pet palette. See [Set Custom Releases for Each Edge](#).
- For the purposes of exporting the Release values to SAF format: in the Generation Rules dialog, you can apply an “All Rigid” override to all *unconnected* Ends, Edges and Edge segments of Structural Analytical Members.

[See End/Edge Release \(Generation Rules\)](#).

Release Values: Rigid, Free, Custom

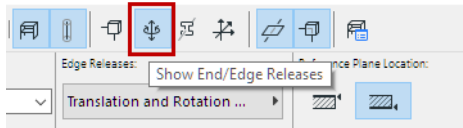
All releases are set to Rigid by default, but you can set alternative values for each direction and component to achieve fixed, hinge and roller-type releases:

- From any selected component: choose either Rigid, Free or Custom
- For Custom: manually enter the desired value.



End/Edge Release Symbols

Turn on the **End/Edge Releases** toggle to display Release symbols on Member ends or edges that you have customized. (By default, all ends and edges are set to Rigid.)



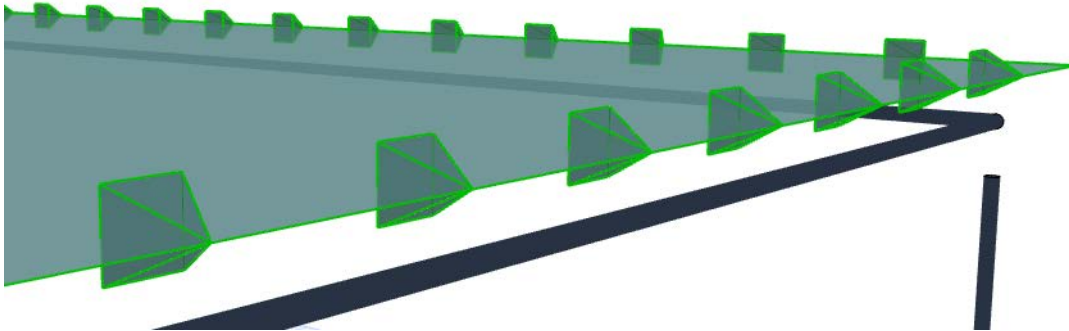
All Rigid

If the Member's release values are all Rigid - the default value - then no symbol is shown.

Hinge Release

For a Member with a “hinge” or “pinned” release (translations all rigid, rotations all free): the symbol is a pyramid

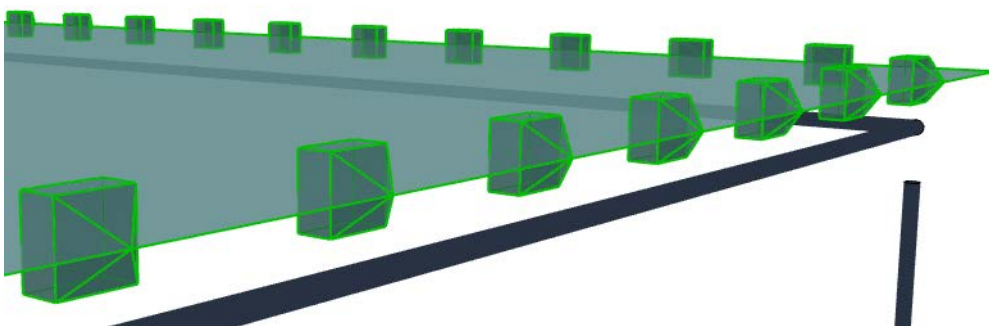
TRANSLATION	
Ux	Rigid
Uy	Rigid
Uz	Rigid
ROTATION	
Fix	Free
Fixy	Free
Fixz	Free



Custom Release

For a Member using any other Translation or Rotation value: the symbol is a “house”

TRANSLATION	
Ux	Rigid
Uy	Rigid
Uz	Rigid
ROTATION	
Fix	Custom
Fix value	999.00 [MNm/rad/m]
Fixy	Free
Fixz	Free

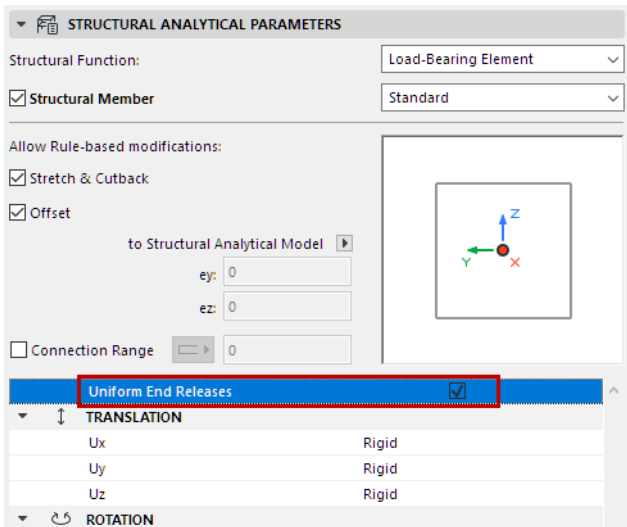


End/Edge Release Color

Customize these colors at [Structural Analytical Model Representation \(Preferences\)](#).

Uniform End Releases

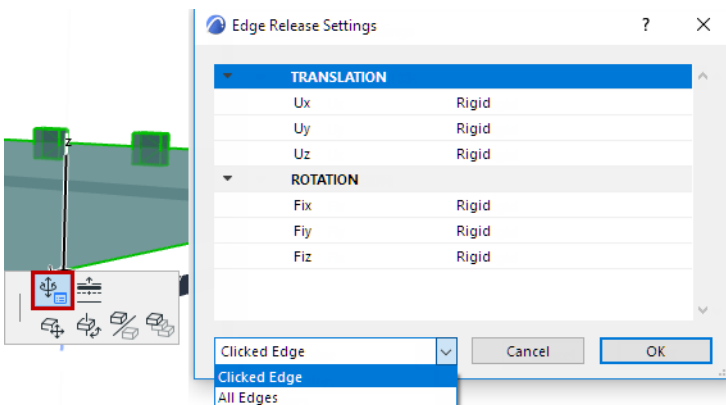
For 1D Members, check this box to set equal values at both ends. Otherwise, set values separately for each end.



Set Custom Releases for Each Edge

For 2D Members, you can customize edge release settings via the pet palette:

1. Select a 2D Member.
2. From the pet palette of an edge, choose Custom Edge Settings.
3. Set Edge Release values.
 - Activate “Clicked Edge” to apply the values only to the edge you clicked.
 - Or active “All Edges”
4. Click OK to apply.



Model Check - Overview

Use the Model Checking functions to identify collisions, potential inadequacies or contradictions in the model.

[Collision Detection](#)

[Physical Model Quality Check](#)

[Structural Analytical Model Quality Check](#)

For all three functions:

- Only visible elements are checked
- If elements are selected: only the selected elements are checked
- All results are listed in the Model Check Report

Related Topics:

[Model Check Report](#)

[Archicad Design Checker](#)

Physical Model Quality Check

Introduction

The Physical Model Quality check helps you ensure that the Physical Model is correct, by filtering for certain modeling and geometric anomalies. As with other types of Model Checking, the results (listed in the Model Check Report) draw your attention to potential issues.

Thus, you will probably run the Physical Model Quality Check *before* applying the Adjustment Rules and generating the Structural Analytical Model.

The Physical Model Quality Check dialog contains a checklist of parameters. You can check the model based on all the listed parameters, or just some of them. Each parameter is briefly described below (see [Physical Model Check Parameters](#)).

The limit/tolerance values of these parameters can be customized - e.g. to conform to modeling standards, or to the rules of your structural analysis software.

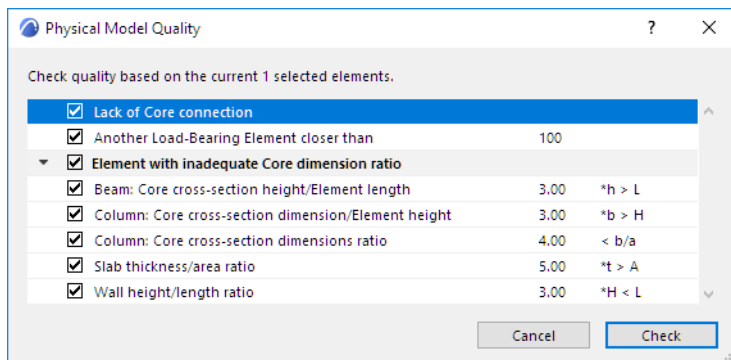
How the Physical Model Quality Check Works

- Focuses on physical construction elements (not 1D and 2D Members)
- Construction elements are considered regardless of their Structural Function (except for one parameter: see [Another Load-Bearing Element closer than](#))
- Only visible elements are checked
- If elements are selected: only the selected elements are checked

Run the Physical Model Quality Check

In the 3D/Floor Plan/Section window:

1. Open **Design > Model Check > Physical Model Quality**



2. The Physical Model Quality Check is based on the listed parameters (see [Physical Model Check Parameters](#) for details). Uncheck any parameter that should not be considered.

3. Click **Check**.

4. The **Model Check Report** opens to show results.

See [Model Check Report](#).

Physical Model Check Parameters

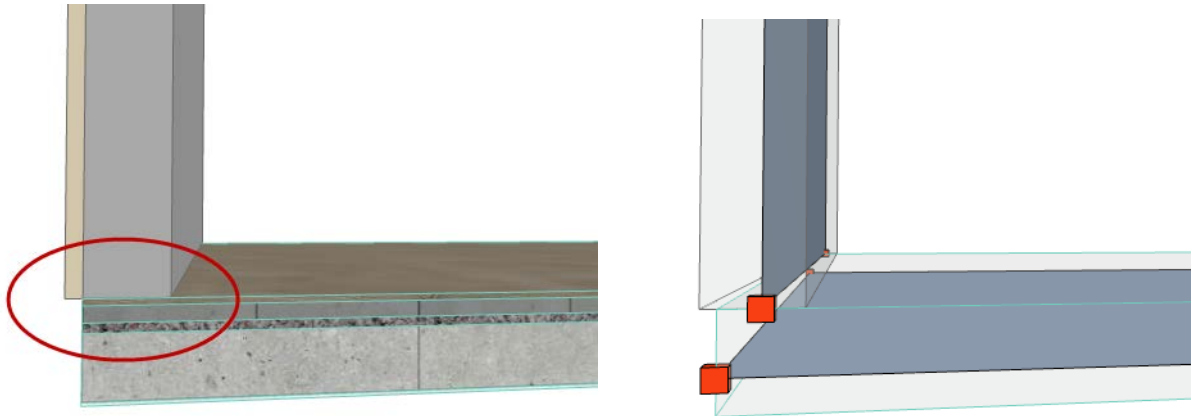
Lack of Core connection

This check identifies any two *intersecting* structural elements whose cores do not connect to each other.

In this example:

The Wall intersects the Slab only as far as the Slab finish: the cores are not connected!

This problem affects the Structural Analytical Model, because if the element cores are not connected, the Adjustment Rules cannot take effect, and the 2D Members remain disconnected.



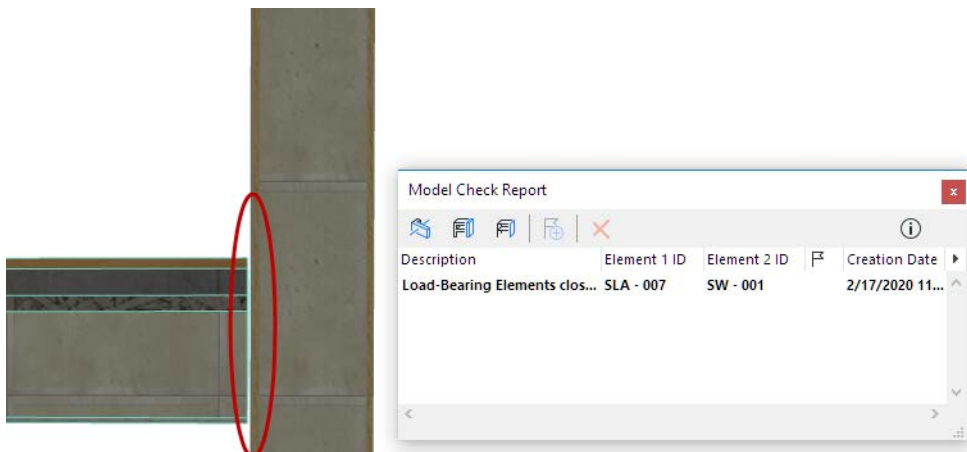
Notes:

- All elements with a Basic Structure have a core, so they are always considered in this check
- Complex Profiles and composite structures are considered only if they include at least one component that is defined as "Core"
- All elements that contain a core are considered (not just the load-bearing elements)

Another Load-Bearing Element closer than

This check identifies any pair of *load-bearing* elements which do not intersect at all, yet are closer to each other than the given tolerance value.

This situation is probably a modeling error, as shown in this image: a tiny gap exists between the Slab and the Wall.

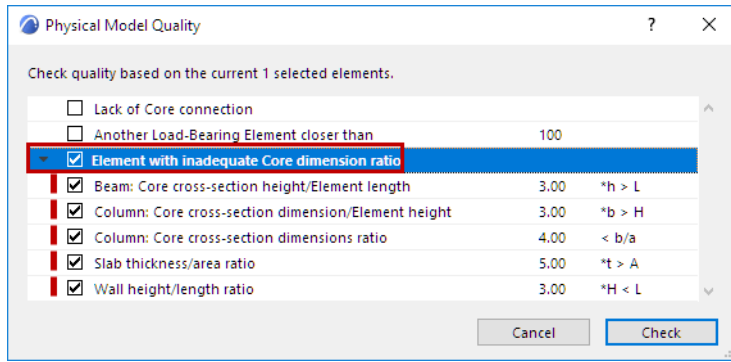


Elements with inadequate Core dimension ratio

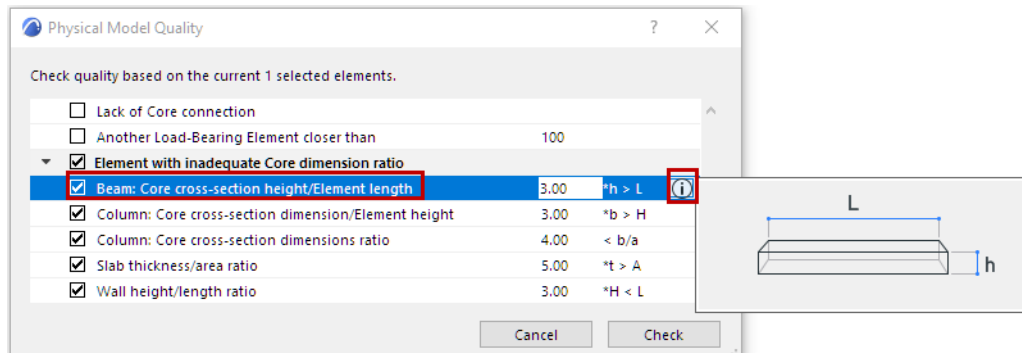
Use this checkbox to turn this group of parameters on or off (or deactivate any parameter individually).

Each parameter examines the ratios of certain core dimensions of Beams, Columns, Slabs and Walls, and checks if the result ratios are within the defined range.

If the ratio is not within the defined range, these elements are identified as “Element with inadequate Core dimension ratio” in the Model Check dialog.



- All core-containing elements are considered, regardless of their Structural Function (not just the load-bearing elements).
- The checked dimensions refer to the core part of the element.
 - For example, for a composite Slab that is 300 mm thick (200 mm core component plus 100 mm of insulation and plaster skins), **only the core thickness of 200 mm is considered** as part of the check.
- To see an illustration of the checked ratio: select an item, then click the Info button.



To See Illustration: Select Parameter, Click “Info”

Tips for Using “Inadequate Ratio” Parameters

This set of parameters in the Physical Model Quality Check dialog are based on the Eurocode standard for concrete structures. (See below for a brief description of each.)

For example, a Column whose cross-section width and height are out of proportion might be considered as a 2D Member rather than a 1D Member.

Results listed in the Model Check dialog are not necessarily errors. However, to promote consistency, you can use these “inadequate ratio” parameters to fit your specific workflow and the requirements of the receiving software. For example:

- Project team members can agree to model certain structures (such as a parapet) with a Wall rather than a Beam, or vice versa.

- Project team members can agree to model the building-length Walls using separate segments, rather than a single extra-long Wall.
- Use the Model Check to find any discrepancies

Check any of the following parameters related to inadequate core dimension ratios:

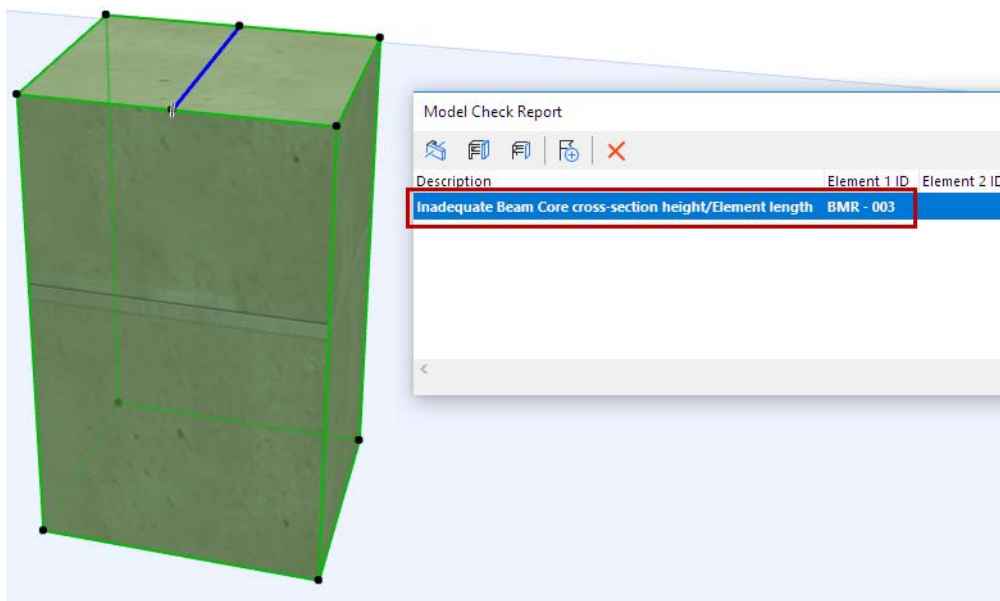
Beam: Core Cross-section height/Element length

Ratio of the Beam's total length to its (core cross-section) height.

For example: if the Beam is “too tall” compared to its length (based on the given relation values), then it is listed in the Model Check Report.

The rules of the structural analytical software might consider this element as a 2D Member (like a Wall) rather than a 1D Member, or as a Column rather than a Beam.

Note: the “Wall height/length” parameter (described below) checks for the inverse of this situation.



Beam “Too Tall” Compared to Length

Column Dimension Parameters

The next two parameters check Column dimensions in two ways:

Core cross-section dimension/Element height

Core cross-section dimensions ratio

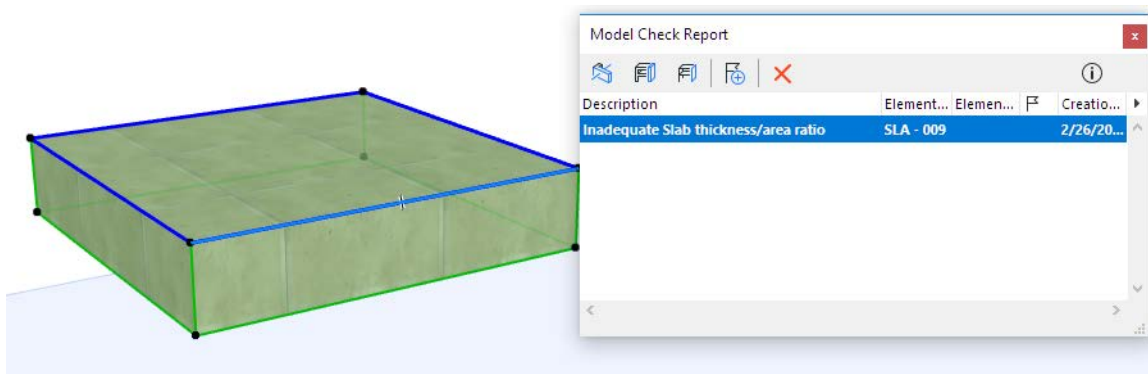
Notes:

- These parameters consider only the parts of the column defined as “Core”.
- If the Column has a Veneer: the Veneer thickness is considered if that Veneer component is defined as “Core” (in Column Settings, “Segment” page.) If the Veneer is defined as “Finish” or “Other”, it is not considered.

Slab thickness/area ratio

Compares Slab core thickness to the Slab area.

If this ratio is greater than the given value (if the Slab is too “thick” compared to its area), it is listed in the Model Check Report.



Slab “Too Thick” Compared to Area

Wall height/length ratio

Ratio of the Wall height to its total length.

For example: if the Wall is “too tall” compared to its length (based on the given relation values), then it is listed in the Model Check Report.

The rules of the structural analytical software might consider such an element as a 1D Member (like a Beam) rather than a 2D Member.

Note: The “Beam height/length” parameter (described above) checks for the inverse of this situation.

Structural Analytical Model Quality Check

Introduction

The Structural Analytical Model Quality Check helps you ensure that the generated Structural Analytical Model is correct. The results (listed in the Model Check Report) draw your attention to potential issues.

The quality check dialog contains a checklist of parameters, in four categories. You can check the model based on all the listed parameters, or just some of them. Each parameter is briefly described below (see [Structural Analytical Model Check Parameters](#)).

You will probably run the Structural Analytical Model Quality Check *after* you have applied Adjustment Rules, and *before* you export it to a structural analysis application.

Note: However, it might be worth checking for certain quality check parameters - such as Overlaps - even before you apply the Adjustment Rules. If the model contains overlapping Members, your Adjustment Rules might not have the desired effect.

All results are listed in the Model Check Report.

Even if the Model Check report lists various anomalies, this does not necessarily mean you cannot export the Structural Analytical Model. For example, the Structural Analytical Model Check can filter for sharp angles on a slab edge, but this situation may well be deliberate.

Note: You must turn on the display of the Structural Analytical Model. Otherwise, this dialog is not available.

How the Structural Analytical Model Quality Check Works

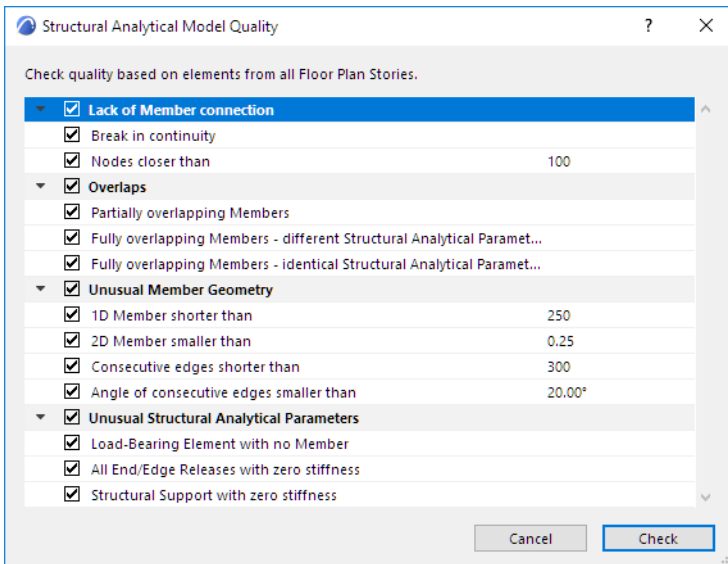
- Focuses on 1D and 2D Members (not physical model elements)
- If any elements are selected in the model, only the selected elements are checked
- Checks only elements which *exist in the Structural Analytical Model* and are *visible*

Exception: One of the parameters identifies elements that are not Members of the Structural Analytical Model: see [Load-Bearing Element with no Member](#).

Run the Structural Analytical Model Quality Check

In the 3D/Floor Plan/Section window:

1. Turn on the display of the Structural Analytical Model.
2. **Design > Model Check > Structural Analytical Model Quality**



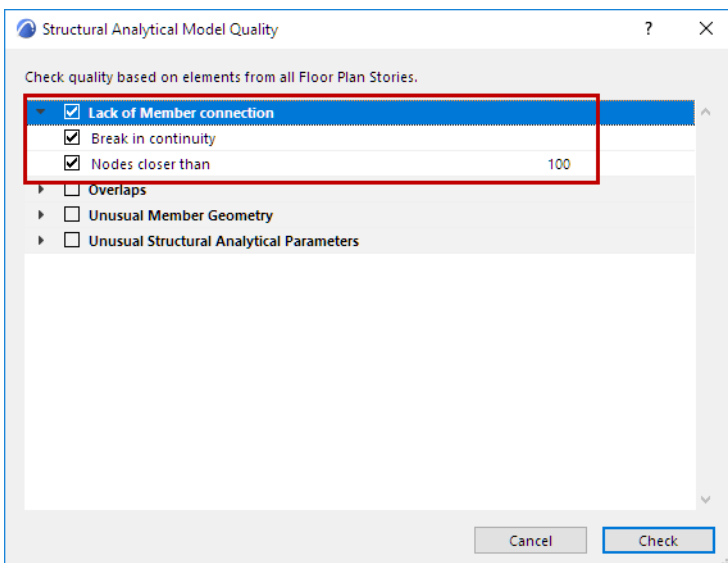
3. The dialog lists checking parameters in four categories (see below for more details).
4. Uncheck any category and/or any parameter that you want to ignore.
5. Click **Check**.
6. The **Model Check Report** opens to show results.

[See Model Check Report.](#)

Structural Analytical Model Check Parameters

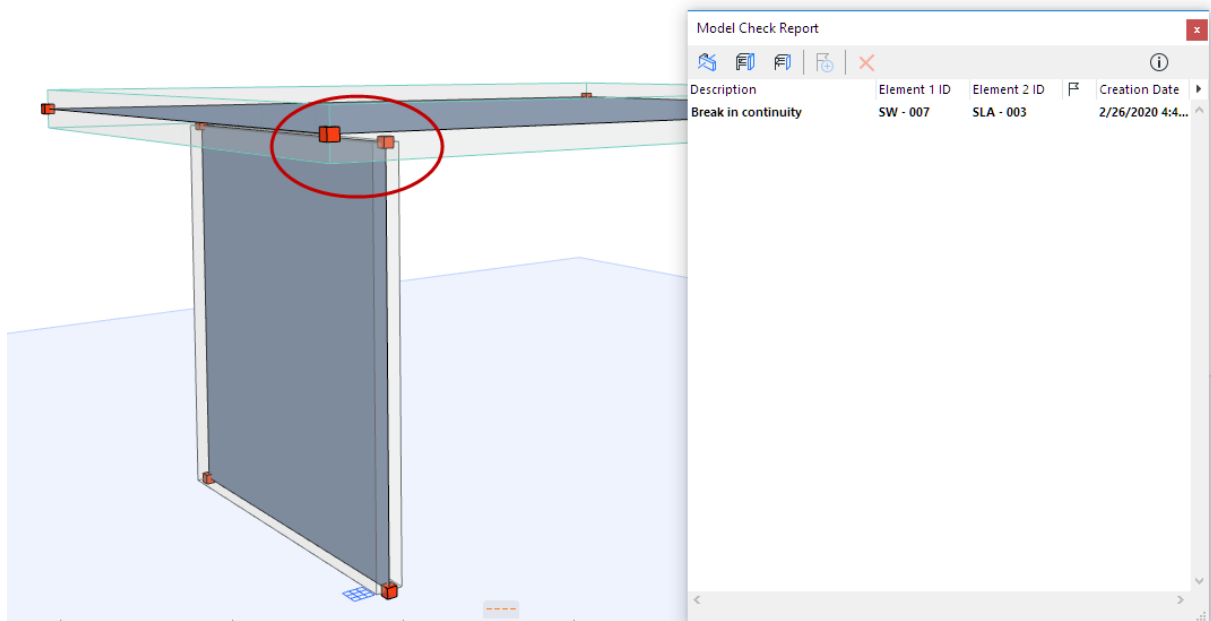
Lack of Member Connection

These two parameters find intersecting or nearby Members that are not connected (but probably should be):



Break in continuity

The cores of two load-bearing elements intersect, but their 1D or 2D Members are not connected.



Cores Intersect, but Members Do Not Connect

For 1D Members: You can “virtually” extend their connection range to avoid such breaks in continuity.

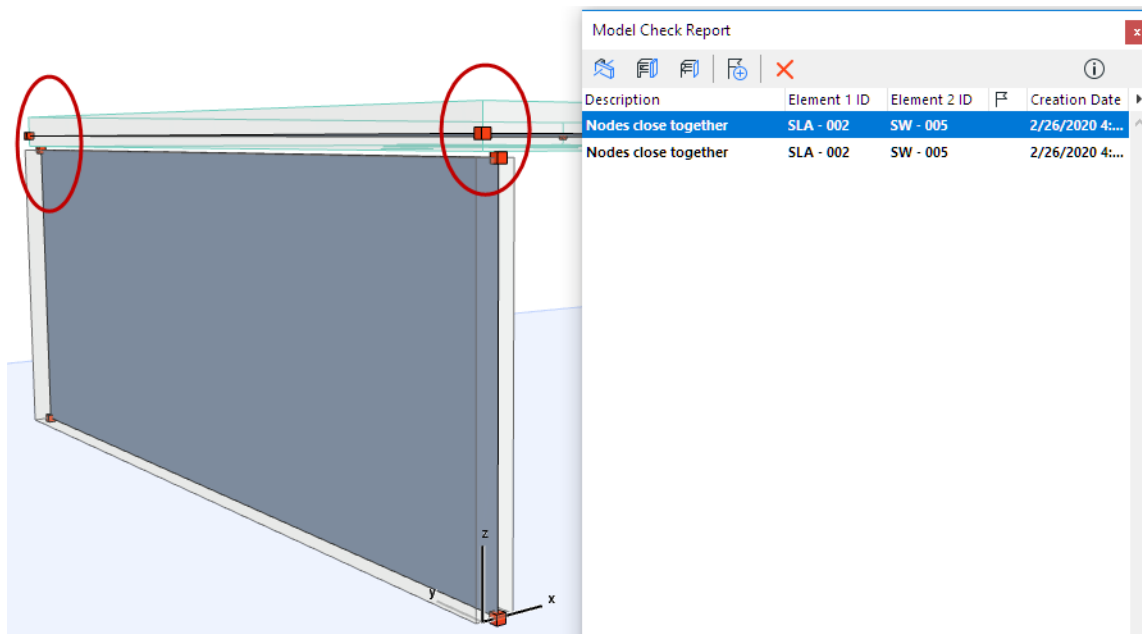
[See Connection Range \(Extend Core to Enable Connection\).](#)

Nodes closer than

Any two nodes are closer to each other than the given distance.

The nodes can be on the same Member or two different Members.

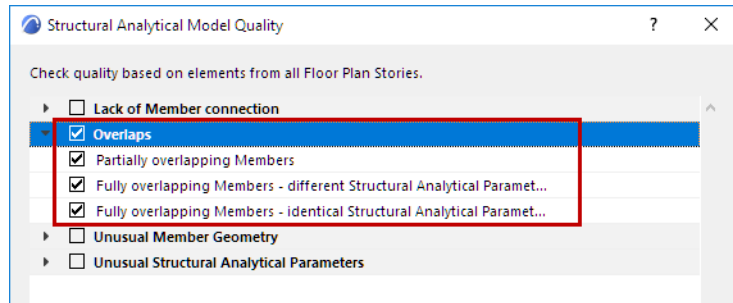
Element cores do not need to intersect.



Nodes are Close Together, but Members Do Not Connect

Overlaps

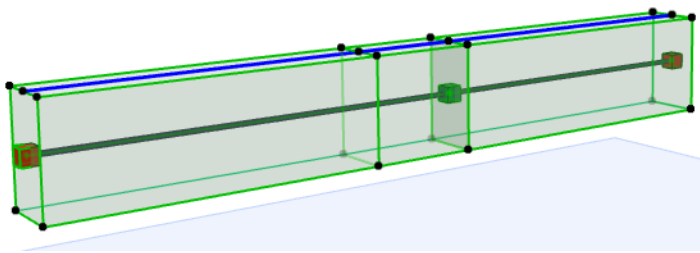
The following three parameters identify overlaps among 1D and 2D Members (*not* overlapping physical model elements!)



Partially overlapping Members

Two 1D Members or two 2D Members partially overlap. Only Member position is considered - no other parameters are considered.

For example, two Beams overlap due to a modeling error (e.g. after stretching).



Fully overlapping Members - different Structural Analytical Parameters

Two 1D Members or two 2D Members completely overlap in 3D.

However, their Structural Analytical Parameters are different. (e.g. one Member uses Offset rules, while the other Member's eccentricity was manually edited).

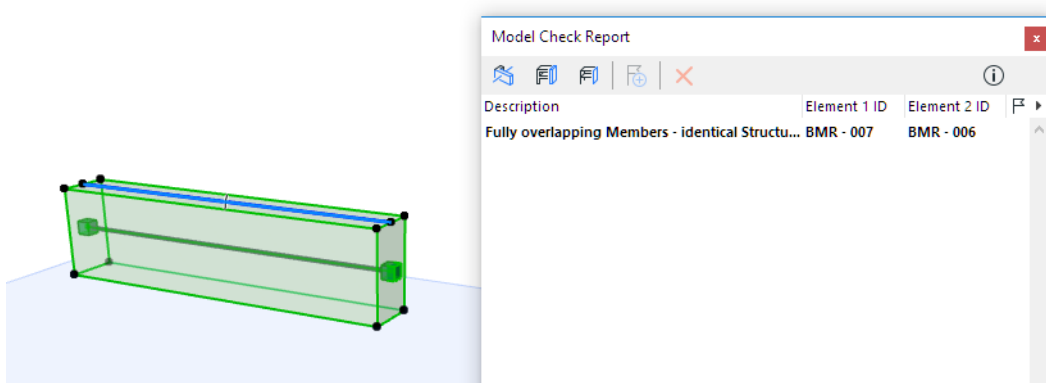
- Structural Analytical Parameters: all settings on the Structural Analytical Parameters page of the element's Tool Settings.
- Only the Member's position and Structural Analytical Parameters are considered.

This result probably indicates an inadvertent overlap. For example, one Member's eccentricity was adjusted so that it ended up outside the physical model element, right on top of the other Member.

Fully overlapping Members - identical Structural Analytical Parameters

Two 1D Members or two 2D Members overlap completely in 3D space. Their Structural Analytical Parameters are entirely identical.

- This result indicates duplicated elements, caused by an error in modeling.



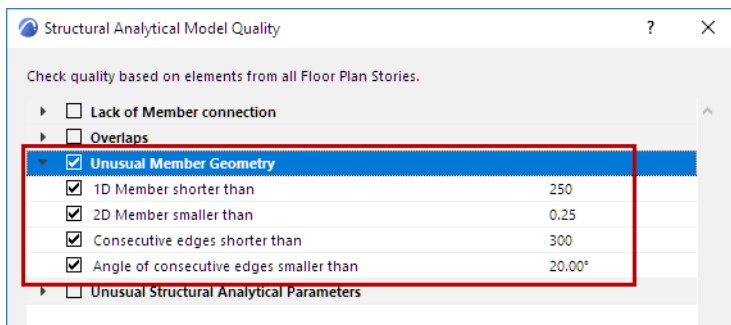
Unusual Member Geometry

These four parameters are helpful in identifying modeling errors. Such errors are worth finding, because even small geometric inaccuracies can undermine the precision of the exported Structural Analytical Model and affect the FEM analysis.

Of course, the check might also identify unusual modeling solutions that were created deliberately.

For any of these parameters: hover over it to see the current Length or Area unit (as set in Options > Project Preferences > Working Units).

You can enter precise tolerance values here - set the desired number of decimals at Working Units.



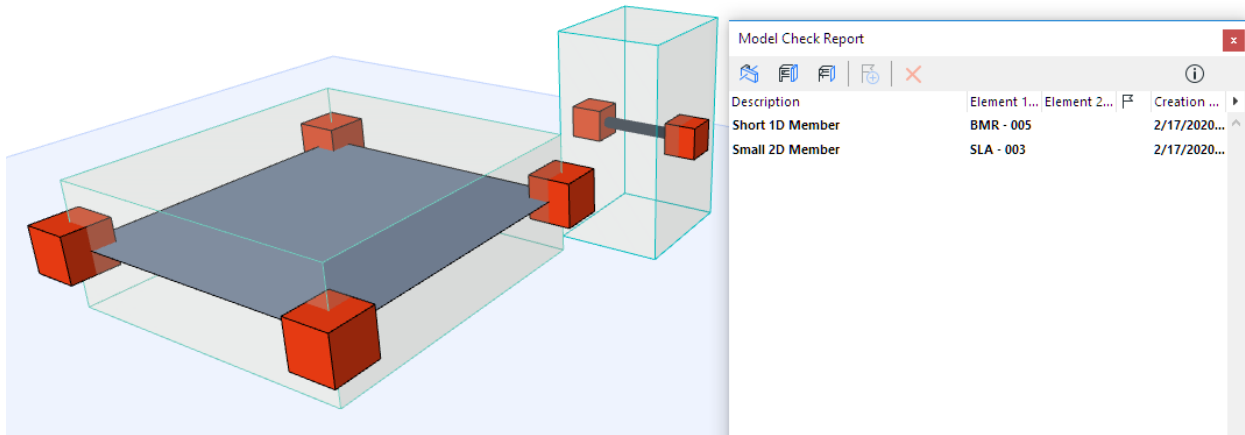
The following parameters check for “Unusual Member Geometry”:

1D Member shorter than

The length of the 1D Member is shorter than the given length.

2D Member smaller than

The area of the 2D Member is less than the given area.



2D Member Too Small, 1D Member Too Short

Consecutive Edges

The following two parameters are applicable to consecutive edges of any given 2D Member.

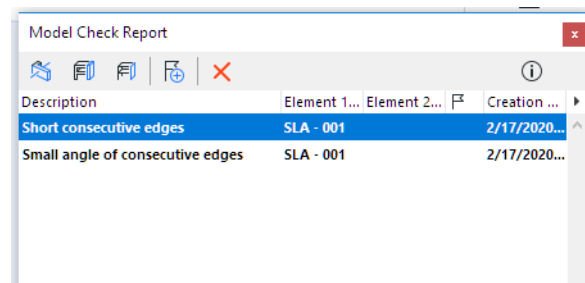
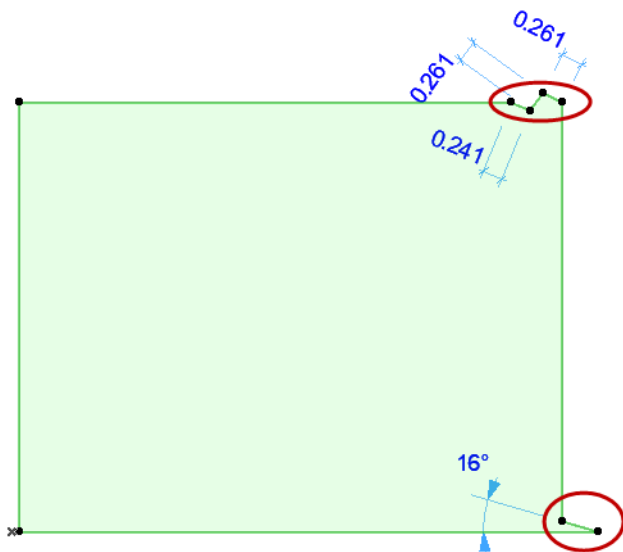
They identify uneven edges and protrusions which might be modeling errors.

- **Consecutive edges shorter than**

For any single 2D Member, finds any adjoining edges that are both shorter than the given tolerance value.

- **Angle of consecutive edges smaller than**

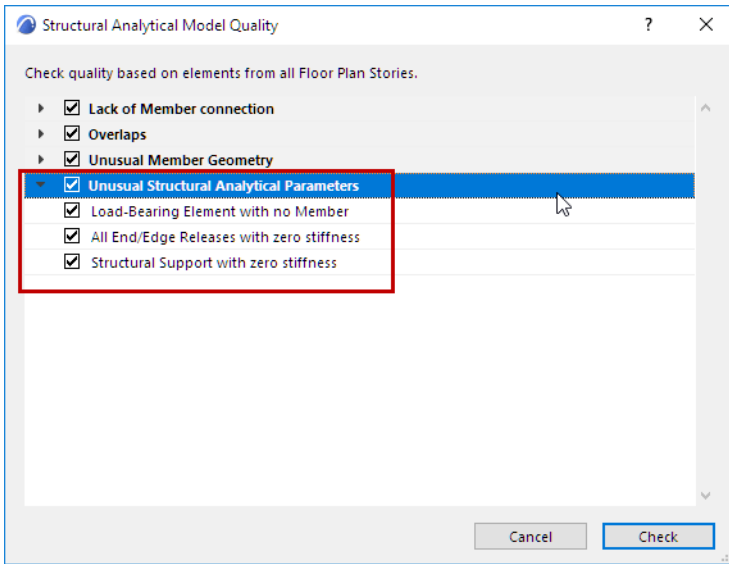
For any single 2D Member, finds any adjoining edges that form an angle less than the given tolerance value.



Consecutive Edges: Too Short, Too Sharp

Unusual Structural Analytical Parameters

These three checks identify elements that might be anomalies, based on their Structural Analytical Parameters:

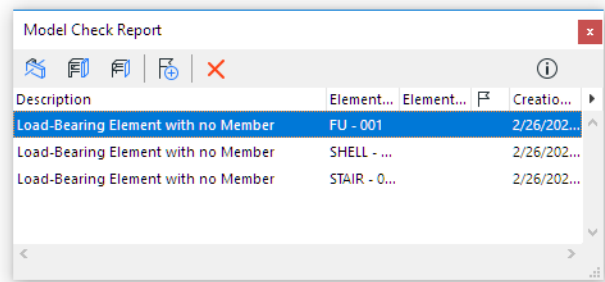
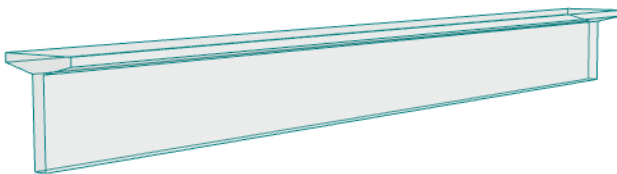


Load-Bearing Element with no Member

Finds any construction element whose Structural Function is set to “Load-Bearing”, yet does not have a Structural Analytical Member. This may or may not be an error.

Possible reasons:

- This load-bearing element could be part of the Structural Analytical Model, but its 1D Member or 2D Member checkbox is not activated in its Tool Settings dialog.
- This type of load-bearing Archicad element cannot be part of the Structural Analytical Model (e.g. Morph, Stair, Multi-plane Roof, Object)



Load-Bearing Object: No Member

All End/Edge Releases with zero stiffness

Finds any Structural Member in which *all* of the End Releases (for a 1D Member) or Edge Releases (for a 2D Member) are set to “Free”.

This is probably an error, because no forces are translated.

[See End/Edge Releases.](#)

Structural Support with zero stiffness

Finds any Structural Support element in which *all* of the Translation and Rotation Stiffness values are set to “Free”.

This is probably an error, because no forces are translated.

Model Check Report

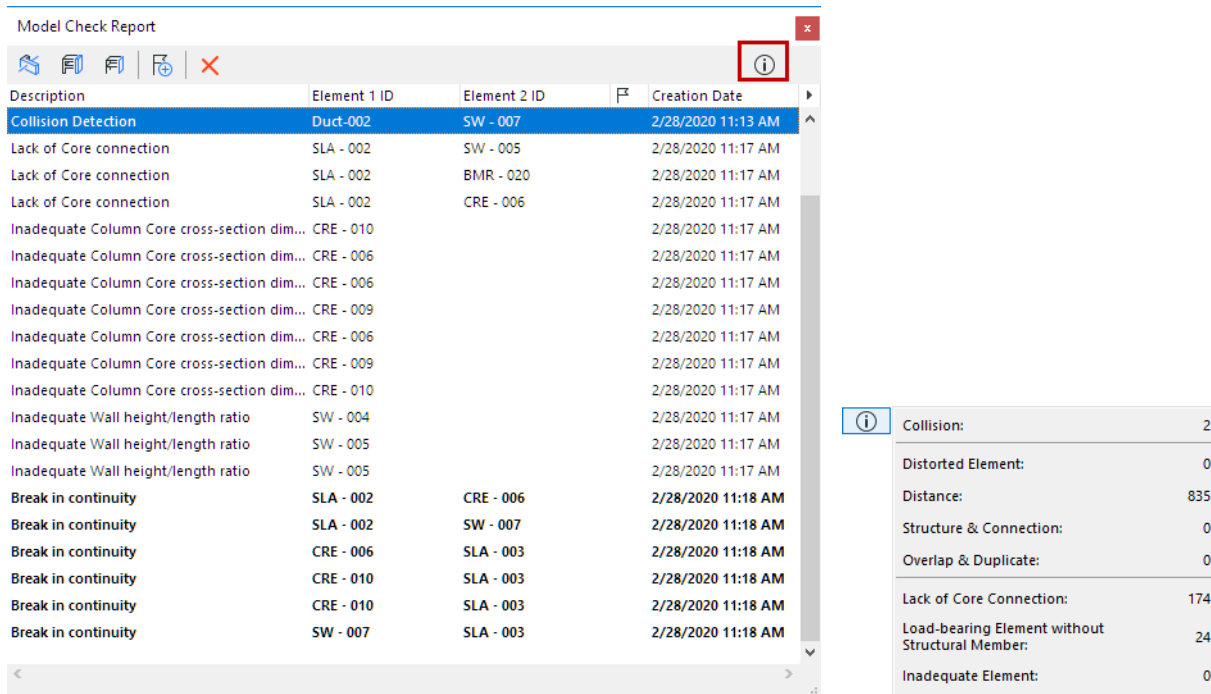
The Model Check Report lists results from all three of the Model Check functions

- [Collision Detection](#)
- [Physical Model Quality Check](#)
- [Structural Analytical Model Quality Check](#)

The list is refreshed each time you run or re-run any of the checks.

New items (compared to the previous status) are shown in bold.

Every item remains on the list until you delete it, or until you close the project.



Description	Element 1 ID	Element 2 ID	Creation Date
Collision Detection	Duct-002	SW - 007	2/28/2020 11:13 AM
Lack of Core connection	SLA - 002	SW - 005	2/28/2020 11:17 AM
Lack of Core connection	SLA - 002	BMR - 020	2/28/2020 11:17 AM
Lack of Core connection	SLA - 002	CRE - 006	2/28/2020 11:17 AM
Inadequate Column Core cross-section dim...	CRE - 010		2/28/2020 11:17 AM
Inadequate Column Core cross-section dim...	CRE - 006		2/28/2020 11:17 AM
Inadequate Column Core cross-section dim...	CRE - 006		2/28/2020 11:17 AM
Inadequate Column Core cross-section dim...	CRE - 009		2/28/2020 11:17 AM
Inadequate Column Core cross-section dim...	CRE - 006		2/28/2020 11:17 AM
Inadequate Column Core cross-section dim...	CRE - 009		2/28/2020 11:17 AM
Inadequate Column Core cross-section dim...	CRE - 010		2/28/2020 11:17 AM
Inadequate Wall height/length ratio	SW - 004		2/28/2020 11:17 AM
Inadequate Wall height/length ratio	SW - 005		2/28/2020 11:17 AM
Inadequate Wall height/length ratio	SW - 005		2/28/2020 11:17 AM
Break in continuity	SLA - 002	CRE - 006	2/28/2020 11:18 AM
Break in continuity	SLA - 002	SW - 007	2/28/2020 11:18 AM
Break in continuity	CRE - 006	SLA - 003	2/28/2020 11:18 AM
Break in continuity	CRE - 010	SLA - 003	2/28/2020 11:18 AM
Break in continuity	CRE - 010	SLA - 003	2/28/2020 11:18 AM
Break in continuity	SW - 007	SLA - 003	2/28/2020 11:18 AM

Collision:	2
Distorted Element:	0
Distance:	835
Structure & Connection:	0
Overlap & Duplicate:	0
Lack of Core Connection:	174
Load-bearing Element without Structural Member:	24
Inadequate Element:	0

Click the Info button to see a summary of the reported problems

The results listed in Model Check Report are not necessarily errors - they may simply reflect a one-off, specific solution, or a particular client request. For each listed item, you can decide to fix it, ignore it, or create an Issue, which is easily done straight from the Model Check Report dialog.

- To create an Issue: See [Create Issue from Model Check Report](#).
- Double-click any item to zoom to the element in 3D.
- Right-click any item for further context menu commands

Model Check Report				
Description	Element 1 ID	Element 2 ID		Creation Date
Collision Detection	Duct-002	SW - 007		2/28/2020 11:13 AM
Lack of Core connection	SLA - 002	SW - 005		2/28/2020 11:17 AM
Lack of Core connection	SLA - 002	BMR - 020		2/28/2020 11:17 AM
Lack of Core connection	SLA - 002	CRE - 006	✗ Delete	
Inadequate Column Core cross-section dim...	CRE - 010		Show on Floor Plan	
Inadequate Column Core cross-section dim...	CRE - 006		Show in 3D	
Inadequate Column Core cross-section dim...	CRE - 006			2/28/2020 11:17 AM
Inadequate Column Core cross-section dim...	CRE - 009			2/28/2020 11:17 AM
Inadequate Column Core cross-section dim...	CRE - 006			2/28/2020 11:17 AM
Inadequate Column Core cross-section dim...	CRE - 009			2/28/2020 11:17 AM
Inadequate Column Core cross-section dim...	CRE - 010			2/28/2020 11:17 AM
Inadequate Wall height/length ratio	SW - 004			2/28/2020 11:17 AM
Inadequate Wall height/length ratio	SW - 005			2/28/2020 11:17 AM
Inadequate Wall height/length ratio	SW - 005			2/28/2020 11:17 AM
Break in continuity	SLA - 002	CRE - 006		2/28/2020 11:18 AM
Break in continuity	SLA - 002	SW - 007		2/28/2020 11:18 AM
Break in continuity	CRE - 006	SLA - 003		2/28/2020 11:18 AM
Break in continuity	CRE - 010	SLA - 003		2/28/2020 11:18 AM
Break in continuity	CRE - 010	SLA - 003		2/28/2020 11:18 AM
Break in continuity	SW - 007	SLA - 003		2/28/2020 11:18 AM

Collaboration with Structural Analysis Software

Archicad can share the Structural Analytical Model with structural analysis applications via the Structural Analysis Format (SAF). This is a global, open-source file format, developed by Nemetschek Group and optimized for this purpose.

Import and export of the Structural Analysis Model in IFC format (IFCStructural Analysis View, developed by buildingSMART) is available in Archicad as an Experimental Feature.

Both collaboration methods are described in the following sections:

[Structural Analysis Format](#)

[IFC Structural Analysis Model Import/Export - Experimental Feature](#)

IFC Structural Analysis Model Import/Export - Experimental Feature

You can open, compare or save files in IFC Structural Analysis Model format (*.ifc). This is an IFC file that uses the “Structural Analytical View” Model View Definition.

Enable this feature

Go to Options > Work Environment > More Options > Experimental Features.

Check the box to **Enable IFC Structural Analysis Model Import-Export**.

This is an experimental feature, which may change or disappear in a future Archicad release. Please try it out!

See [Experimental Features](#).

Import

From the Floor Plan or 3D Window:

1. Use **File > Open**
2. Choose the IFC Structural Analysis Model (*.ifc) file format. IFC 2x3 and IFC4 are both supported.
3. Click **Open**.

Design > Model Compare

Browse for Structural Analysis models in *.ifc format.

Element Handling at Import

When importing a IFC Structural Analysis Model file, elements are converted automatically. There are no mapping rules.

1D and 2D Members are converted automatically based on geometry: a vertical Member is a Column, a horizontal Member is a Beam.

Building Materials

If the imported element's Building Material **already exists** (by name) in the Archicad template, it uses the corresponding Archicad Building Material.

Note: Only the names are compared. Building Materials of the same name are considered as a match.

If the Building Material **does not exist** in the Archicad template:

- the Building Material name (as in IFC) is imported and added to the list of Archicad Building Materials.
- its settings use the Building Material defined for the corresponding default tool in Archicad (e.g. the default Column's Building Material attributes)

Profiles

If the imported 1D Member's Profile **already exists** (same name and geometry) in the Archicad template, it uses the corresponding Profile.

If the incoming Profile does not exist in Archicad, then it is imported and added to the Archicad Profiles.

Other Element Settings at Import

All other settings of imported elements use the defaults of the respective Archicad tool (Layer, Classification, user-defined Properties, etc.)

Structural Analytical Model Settings

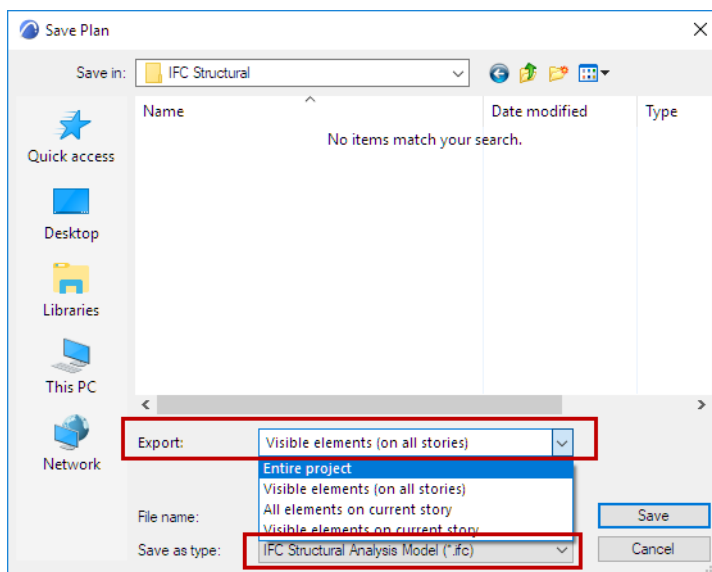
The imported model uses the following settings in Archicad to prevent automatic adjustments and to correspond as exactly as possible to the original structural model:

- Layer Intersection Group settings are reset to zero (intersecting elements will not connect)
- Adjustment Rules (in the Structural Analytical Model Generation Rules dialog) are set to “No Rules”
- Beam End Cuts are all set to vertical, Column End Cuts are all set to horizontal

Export

From the Floor Plan or 3D Window:

1. Use File > Save as
2. Choose the IFC Structural Analysis Model (*.ifc) file format. The file can be exported as IFC2x3 only. (IFC4 is not supported)



3. From the **Export** pop-up, choose to save the entire project, or only the visible elements. When saving from the Floor Plan, you can save elements from the current or all stories.
4. Click **Save**.

Element Handling at Export

All 1D and 2D Members are saved with their current settings (geometry, profiles, materials).

Structural Analysis Format

Structural Analysis Format (SAF) is an Excel-based, open-source and global schema optimized for storing and sharing structural analytical model data.

SAF supports bi-directional collaboration between Archicad and structural analysis apps. The structural analytical model of load-bearing elements in Archicad can be used for analysis and design in external applications that use the finite element method (FEM).

For further information about the SAF schema and its detailed content, visit <http://saf.guide/>.

Topics in this Section:

[Export SAF File from Archicad](#)

[Import SAF File to Archicad](#)

[SAF Translators - Overview](#)

[Detailed SAF Translator Settings](#)

[Create new SAF Translator](#)

[Import/Export SAF Translators](#)

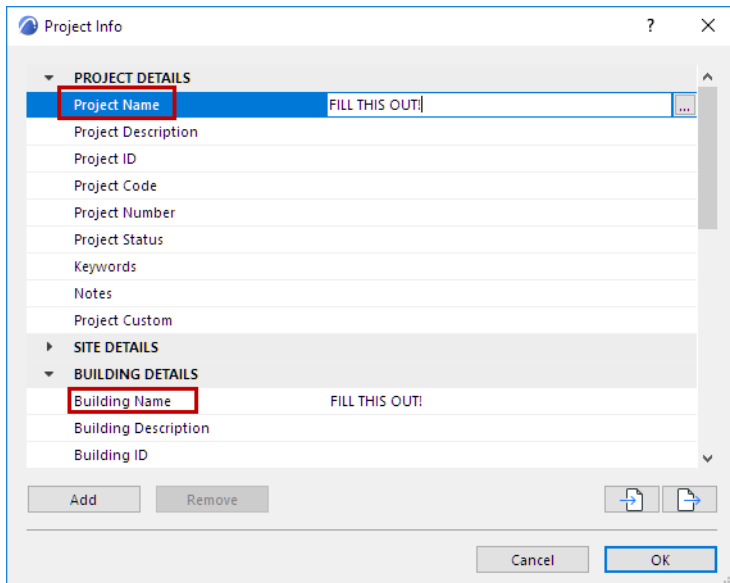
Export SAF File from Archicad

Export is based on the SAF Translator settings of the current Archicad project. (See [SAF Translators - Overview](#).)

Hint: Before export, it is recommended to fill out the following fields in the Project Info dialog box (**File > Info > Project Info**).

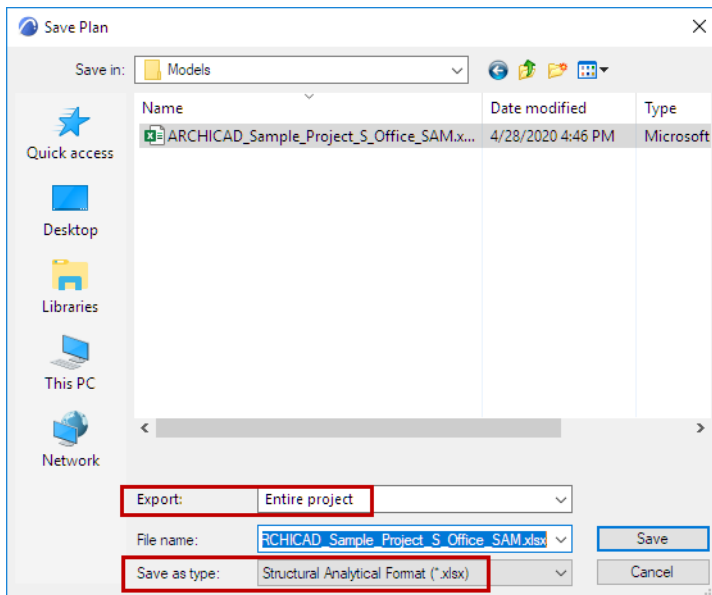
- Project Name
- Building Name

These data will be visible on the SAF file's Project and Model Data tab pages.



To export the structural analytical model, do one of the following:

- From the Archicad project (Floor Plan or 3D Window), use **File > Save as**
 - Choose Structural Analytical Format (*.xlsx)
 - From the Export pop-up, define whether to export the Entire project, the visible elements in your current view, or selected elements.



- Use **Document > Publish** to export the analytical Members visible in your published views. (See [Publisher.](#))

You can also use a connection tool to specific analysis tools, for a faster workflow:

- Frilo Connection (developed by Graphisoft): see [Export to FRILO.](#)
- SCIA Connection (developed by SCIA), available for download here: [Add-ons by Partners](#)

Import SAF File to Archicad

There are two import workflows:

- Open the SAF file in Archicad
- Use the Model Compare to visualize the differences as part of a bi-directional workflow

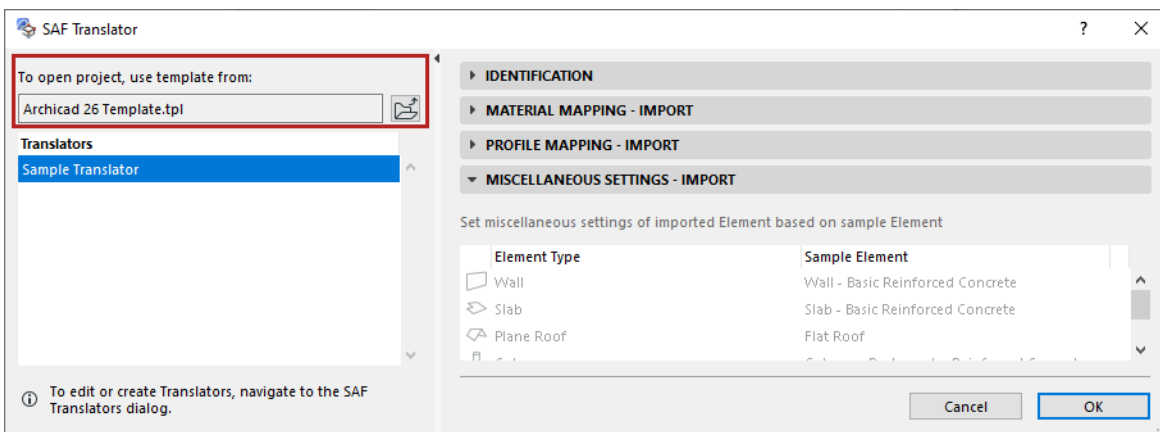
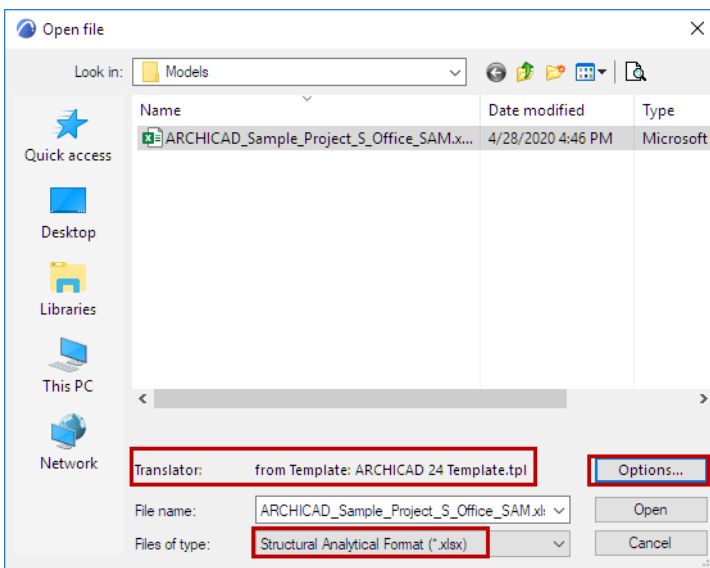
Each workflow is summarized below.

Use File > Open

1. Use **File > Open** and browse for the desired SAF file
2. The Open process uses the SAF Translator settings of the template file you choose.

Check the current template file, shown in Open dialog box at “Translator”.

If needed, click **Options** to choose a different template.



3. The SAF file opens in Archicad.

- Incoming elements are mapped to Archicad counterparts based on the settings of the SAF Translator being used. These translator settings include Layer definition for each element. (See [Miscellaneous Settings - Import.](#))

- After import, a “Custom” Layer Combination is activated automatically, in which the Layer Intersection Class is set to zero for all Layers. (This way, automatic priority-based connections do not take place, and the Structural Analytical Model connections are not disturbed.)
- The Structural Analytical Model Adjustment Rules are set to “No Rules” - again, to avoid altering the structural analytical model connections.

Use Model Compare

If you are importing back the changes or checking the modifications in the SAF file, it is recommended to use the Model Compare feature (**Design > Model Compare**).

When opening an SAF file using Model Compare, the Translator settings of the current project are applied by default. However, when selecting the SAF file in Model Compare, you can choose a different template for the SAF Translator, if needed.

See [Compare Structural Analytical Models](#).

SAF Translators - Overview

When you import and export SAF files between Archicad and external structural analysis softwares, the data exchange occurs based on SAF Translator settings. SAF Translators ensure that the Archicad structural analytical model is correctly recognized and interpreted by the external software, and vice versa.

Translator settings include

- Building Materials (in Archicad) and corresponding SAF Materials
- Archicad Profiles (in Archicad) and corresponding SAF Cross Sections
- Local Coordinate System standards (how to interpret the orientation of 1D Members)
- Archicad element default settings to use for incoming SAF elements
- How to interpret curved Walls exported from Archicad if the receiving software cannot handle them

Access SAF Translator settings at **File > Interoperability > SAF > SAF Translators**.

During structural model exchange, the applied SAF Translators come from either the current Archicad project or a template file, as summarized in this table:

	Which SAF Translators Are Applied?
Export Archicad to SAF	Translators in the Current Project
Open SAF in Archicad	Choose a Template File
Open SAF in Model Compare	Current Project OR Chosen Template

Since different softwares may use different naming conventions, you should fine-tune the translator settings to ensure correct data exchange based on the specific software you are collaborating with.

Mapping the Cross Section/Profile and Material names between SAF data and Archicad can be done by hand in the corresponding Translator settings.

[Detailed SAF Translator Settings](#)

Shortcuts to ensure standard mapping of materials and profiles

For Archicad Profiles and 1D SAF Profiles that are NOT mapped by name in the Profile Mapping panels: you can activate automatic mapping based on a default database. This can save you a lot of time and effort compared to mapping by hand.

[Use external database to translate unmapped profiles](#)

Graphisoft also provides predefined data files to make mapping easier.

[Download SAF Data Files optimized for structural applications](#)

You can also create your own SAF data file based on your analysis software

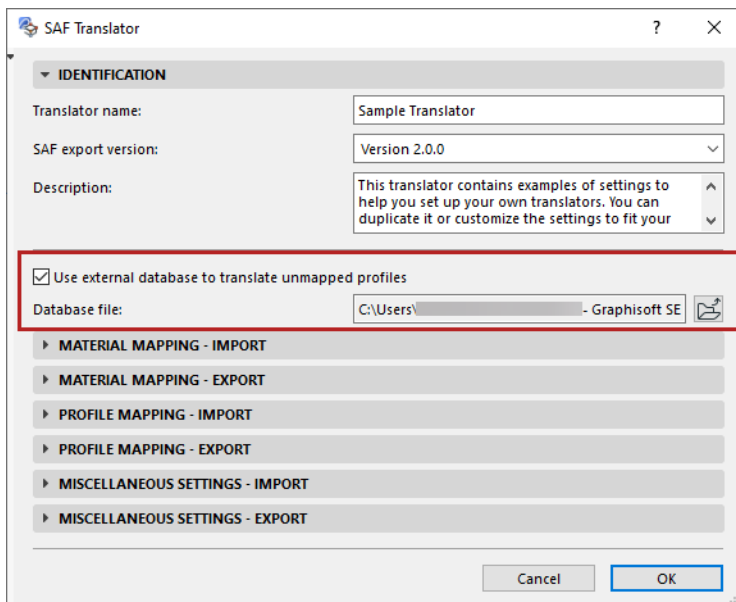
[Prepare and export a SAF Data File to ensure standard mapping](#)

Use external database to translate unmapped profiles

Outgoing Archicad Profiles and incoming 1D SAF Profiles are mapped based on the definitions in the Profile Mapping panels (for Export and Import, respectively).

For those that are NOT mapped by name in the Profile Mapping list: you can activate automatic mapping based on a default database. This can save you a lot of time and effort compared to mapping by hand.

Activate this checkbox in the Identification panel of the SAF Translator.



If you check this box, then profile mapping will refer to a predefined list (an Excel file called “ExternalDB”) to map all outgoing Archicad profiles and incoming, manufactured Cross Sections that are NOT mapped by name in the Profile Mapping panels.

Note: For each incoming SAF Cross Section which does not have a corresponding Profile in Archicad, a new Profile attribute is automatically created, based on the Standard Steel Profile database. See [Use Standard Steel Column or Beam Profile](#).

Location and editing of external database file (Manufactured SAF Cross Sections)

The “ExternalDB” Excel file is located in your Defaults folder, where you have installed Archicad.

Also check the Downloads site for predefined database files you can use for this purpose. ([See Download SAF Data Files optimized for structural applications.](#))

It is possible to create your own database file, save it, and then browse your custom file to use. You must use an Excel file consisting of a single sheet, with the same two columns and column names as in the provided default file.

Important: You can copy and edit the default ExternalDB file provided in your Archicad template, but do NOT rename or move it!

Priority of Mapping Rules

Rules for mapping Archicad Profiles and manufactured SAF cross-sections are applied in the following order:

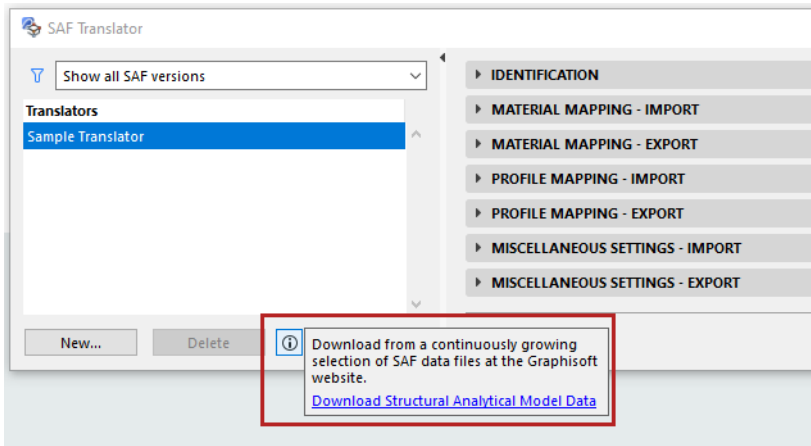
1. Mapping by name, based on the list in the Profile Mapping panel
2. External database (provided that the “Use external database” box is checked).

-
-
- 3.** “Other Profiles” setting. This is the catch-all definition for Archicad Profiles that have not been mapped in steps 1 and 2 above.

Download SAF Data Files optimized for structural applications

Do one of the following:

- go to the [Downloads](#) page at graphisoft.com
- or click the info link at the bottom of the SAF Translators dialog



Available content on this page, which is expanded periodically, includes:

- SAF Material catalog in SAF.xlsx format for various Analysis Softwares (e.g. SCIA)
- SAF Manufactured Cross Section catalog in SAF.xlsx format for various Analysis Softwares (e.g. SCIA)

Download the package you need, then import or merge the file in the relevant SAF Translator dialog.

Note: The available predefined content represents extensive and comprehensive catalogs, including many items that will not be used in your specific project.

If you only need a limited set of SAF Materials/Cross Sections, consider creating your own SAF data file based on the external analysis software, which includes only the items you need:

See [Prepare and export a SAF Data File to ensure standard mapping](#).

Prepare and export a SAF Data File to ensure standard mapping

Use this method if your project uses a limited number of specific SAF Materials/Cross Sections. This way you can easily access the exact Material/Profile names used in the external software.

Note: Graphisoft also provides several predefined SAF data files - these are much more comprehensive catalogs. [See Download SAF Data Files optimized for structural applications](#)

In the external software, create a file that contains all the SAF Materials and SAF Cross Sections used by the external software (e.g. SCIA Engineer or RISA-3D). Then export the file as SAF (*.xls). (This saved file does not need to contain any actual structural Members, just the SAF Cross Section and material lists.)

Name	Material	Cross-section Type	Shape	Parameters [mm]	Profile	Form coc	Description ID of th
6	CS5	S 235	Manufactured		CHS48.3/5.0	3	
7	CS6	S 235	Manufactured		UPE200	5	
8	CS7	S 235	Manufactured		T100	6	
9	CS8	S 235	Manufactured		FLB200/10	7	
10	CS9	S 235	Manufactured		RD20	11	
11	CS10	S 235	Manufactured		Z200	102	
12	CS64	S 235	Manufactured		HEB240	1	
13	CS65	S 355	Manufactured		HEB240	1	
14	CS11	S 235	Compound	Double i section	10	IPE300	
15	CS12	S 235	Compound	Double channel ftf	10	UAP200	
16	CS13	S 235	Compound	Double channel btb	10	UAP200	
17	CS14	S 235	Compound	Starred angle	10	RSUA100/75/12	
18	CS15	S 235	Compound	Double angle btb II	5	RSUA100/75/12	
19	CS16	S 235	Compound	Double angle ftf	5	RSUA100/75/12	
20	CS17	S 235	Compound	Double angle btb	5	RSEA60/60/10	
21	CS18	S 235	Compound	Double angle ftf II	5	RSEA60/60/10	
22	CS19	S 235	Compound	Four angle btb	5;10	RSUA100/75/12	
23	CS20	S 235	Compound	Four angle ftf	5;10	RSEA60/60/10	
24	CS21	S 235	Compound	Four angle i	220;160	RSUA100/75/12	
25	CS22	S 235	Compound	Double pipe	200	CHS168.3/12.5	
26	CS30	C25/30	Parametric	Oval	200;300		
27	CS31	C25/30	Parametric	Circle	300		
28	CS32	C25/30	Parametric	Rectangle	500;300		
29	CS33	C14 (EN 338)	Parametric	Double rectangle	100;250;50		
30	CS34	C14 (EN 338)	Parametric	Triple rectangle	100;250;50		
31	CS35	C14 (EN 338)	Parametric	Rectangle with plates	80;230;50;120		
32	CS36	C14 (EN 338)	Parametric	Double rectangle with plates	50;150;80;230		
33	CS37	C25/30	Parametric	I section	800;450;450;180;180;160		
34	CS38	S 235	Parametric	I section with haunch	300;300;25;20;12;12		
35	CS40	C25/30	Parametric	T section	600;450;160;200		
36	CS42	S 235	Parametric	C section	100;20;20;100;20;30		
37	CS43	C25/30	Parametric	L section	600;450;180;160		
38	CS44	C25/30	Parametric	L section opposite	600;450;180;160		

Sample SAF file, containing SAF Materials and Cross Sections

Now you can import or merge the file in the relevant SAF Translator dialog.

See these sections:

[Add More SAF Materials to Mapping List \(Export to SAF\)](#)

[Add More SAF Cross Sections to Mapping List \(Export to SAF\)](#)

[Add More SAF Cross Sections \(Import SAF\)](#)

[Mapping Imported "General" or "Parametric" SAF Cross Sections](#)

Detailed SAF Translator Settings

[Material Mapping - Import](#)

[Material Mapping - Export](#)

[Profile Mapping - Import](#)

[Profile Mapping - Export](#)

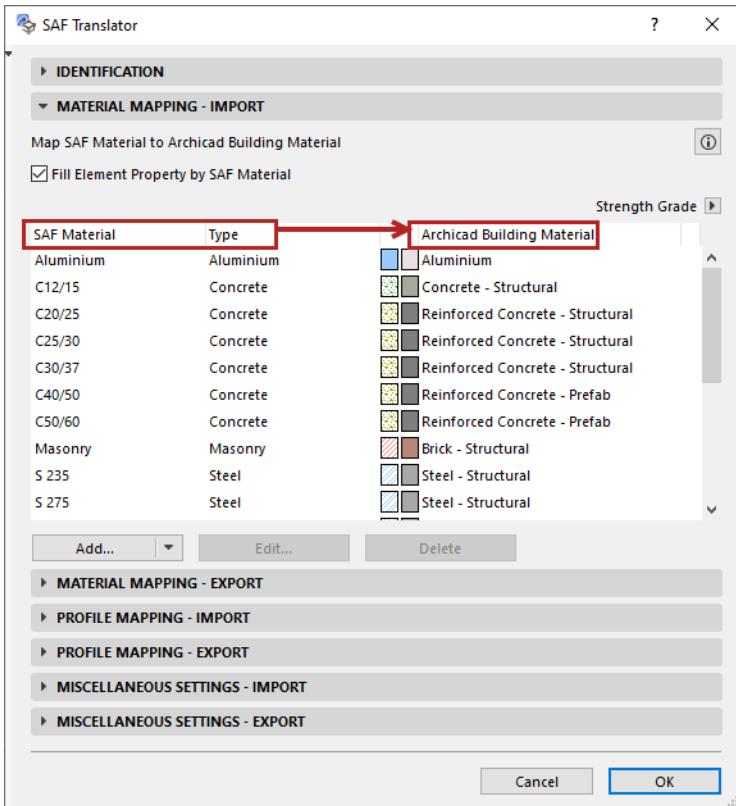
[Miscellaneous Settings - Import](#)

[Miscellaneous Settings - Export](#)

Material Mapping - Import

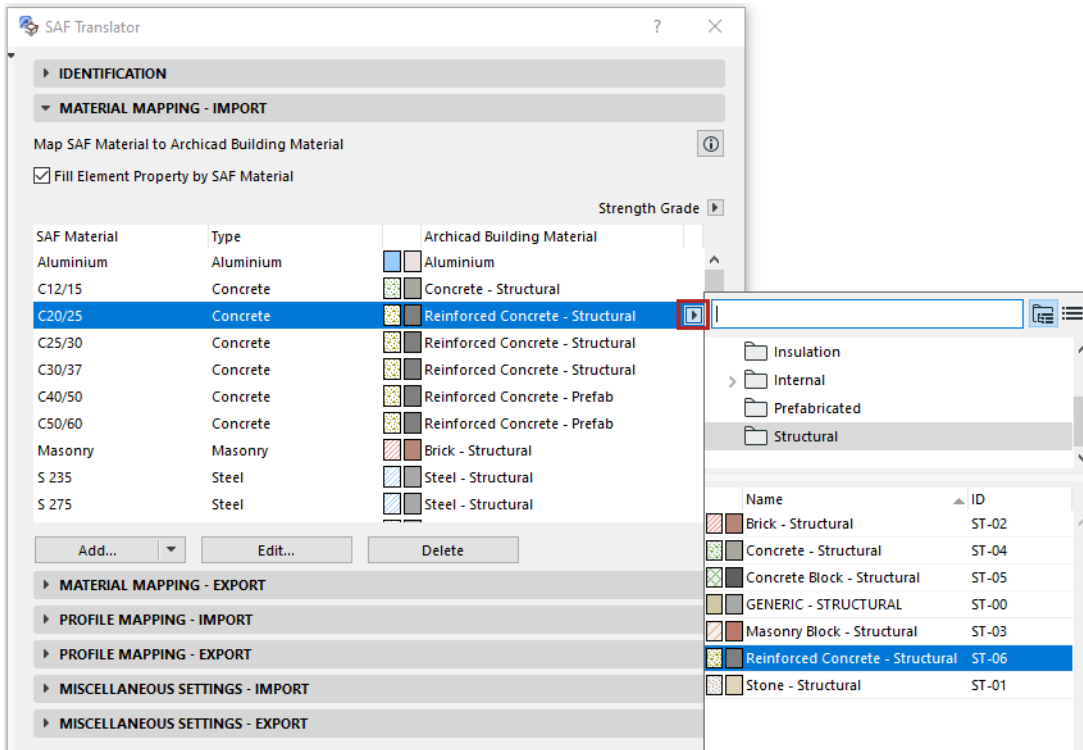
Go to **File > Interoperability > SAF > SAF Translators**.

Use the **Material Mapping - Import** panel to map incoming materials (“SAF Material”) from the external software to their counterpart Archicad Building Materials.



For each SAF Material:

Click the pop-up to choose from the current Archicad project's Building Materials.



To add more SAF materials to the mapping list, click **Add**.

[See Add More SAF Materials \(Material Mapping - Import SAF\).](#)

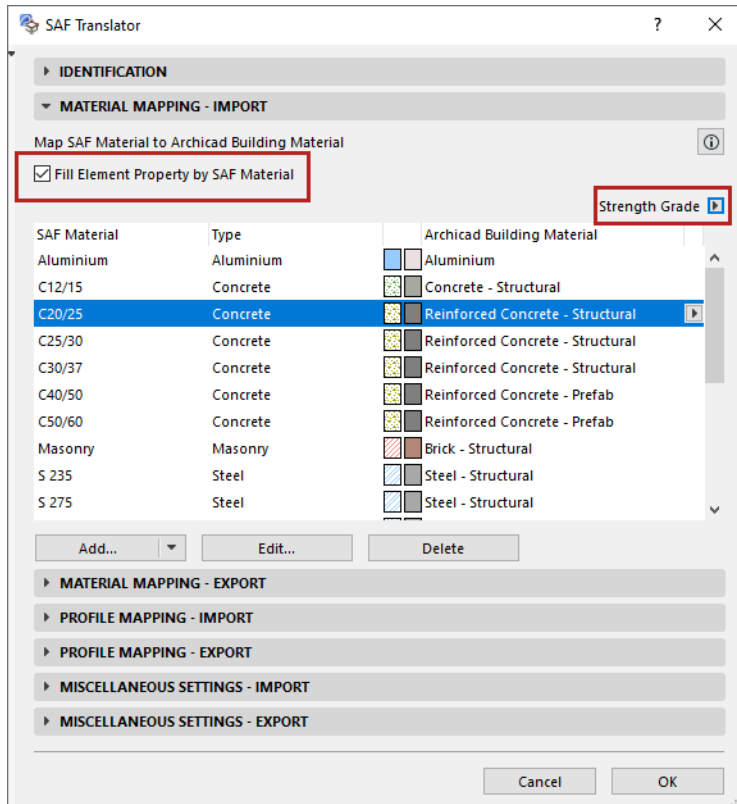
Note: If you map incoming SAF Cross Sections to Archicad Profiles by name (Profile Mapping - Import panel), then you do not need to map their materials, because Archicad Profiles include a Building Material.

Additional Material Mapping: By Element Property Value

In some cases, you may need to map multiple SAF materials to a single Archicad Building Material.

To add an additional level of mapping:

1. Check the box to include 'Fill Element Property by SAF Material'.
2. The Property pop-up is now available. Click to choose the Element Property (e.g. Strength Grade) by which to differentiate the Building Material.



By including this property value in the mapping definition, you can create multiple mappings, based on the differing property values.

Important Notes:

- In a given Archicad model, only a single Element Property can be used as a value in SAF Material Mapping.
- Only **string-type** Element Properties can be used for SAF Material Mapping.
- Property values must be assigned to Archicad elements (e.g. to Beams and Walls) - not to their Building Materials.

Add More SAF Materials (Material Mapping - Import SAF)

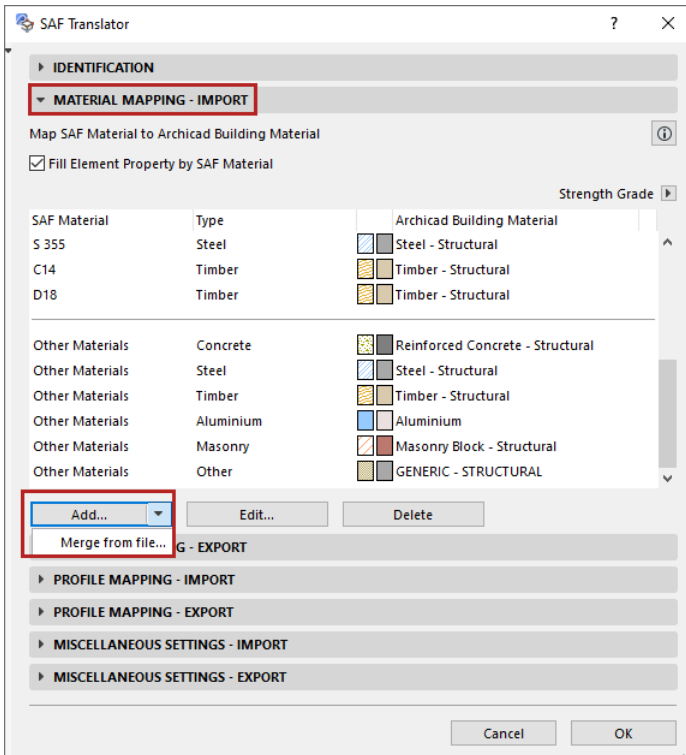
To expand the list of mappable SAF Materials, do one of the following in the Material Mapping - Import panel:

- Merge from file
- Add manually

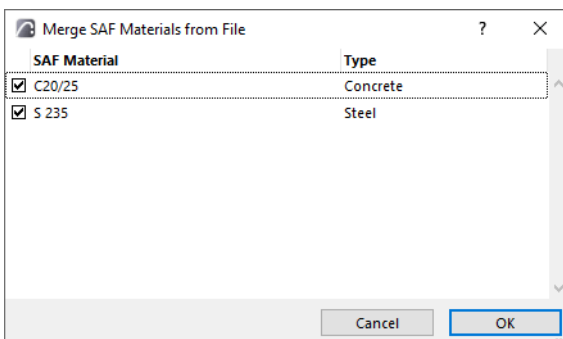
Merge from SAF File

This is the preferred method, because you can avoid typing errors.

1. From the Add button, click **Merge from file**.



2. Browse for an.xlsx (SAF) file exported for this purpose. (See [Prepare and export a SAF Data File to ensure standard mapping](#)).
3. Materials from the SAF file appear in a dialog. Review this list and check the ones to import.
4. Click **OK**.



Add or Edit Manually

1. Click **Add** or (for a selected Material in the list) **Edit**.
2. In the appearing dialog, manually enter or edit the SAF Material name/type.

Material Mapping - Export

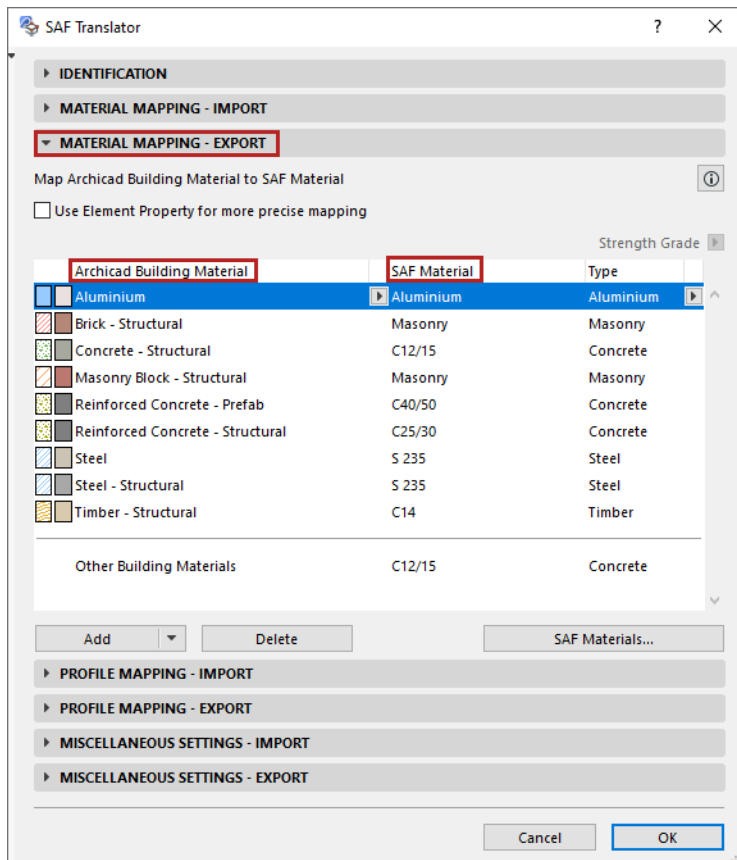
Go to **File > Interoperability > SAF > SAF Translators**.

Use the **Material Mapping - Export** panel to map Archicad Building Materials to their exact counterparts (“SAF Material”) in the receiving software.

You can map each Archicad Building Material by name to a specified SAF Material.

See [Map Archicad Building Material \(By Name\)](#).

Any Building Material that is not mapped by name uses the default specified at “Other Building Materials” - this is a catch-all definition.

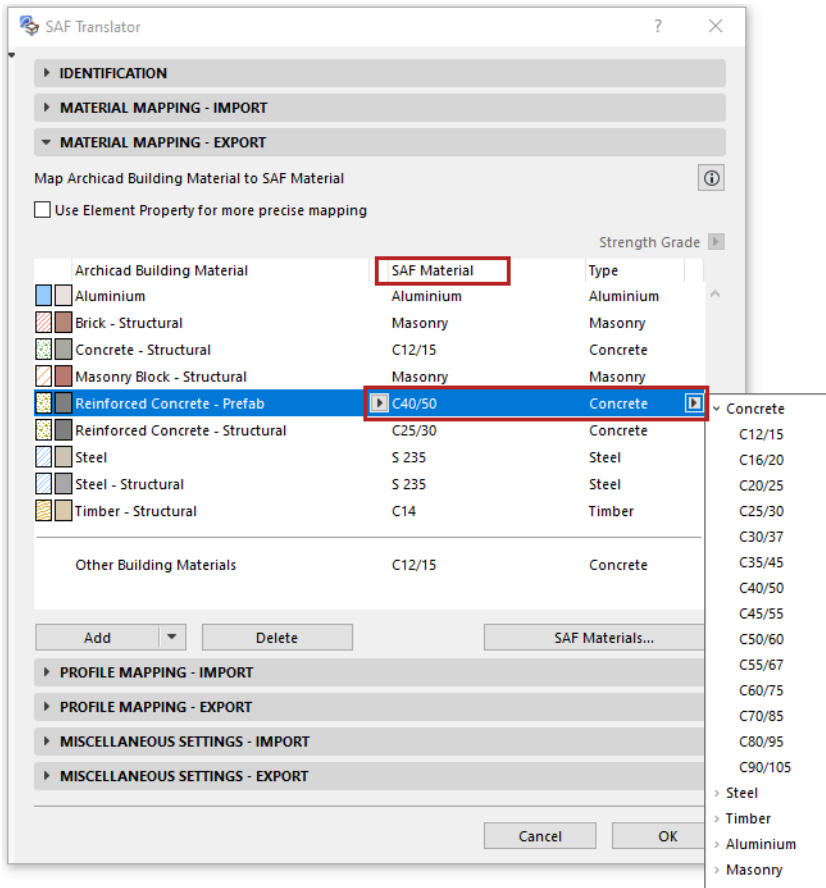


Material mapping definitions in this panel apply to the following exported Archicad structures:

- All 2D Members
- 1D Members with Rectangular and Circular Profile
- Complex Profiles that are exported as “General” Types in the Profile Mapping panel. (These have no corresponding named SAF Cross-Section which would include a material.)
 - For all other Archicad Complex Profiles: Material mapping does not apply. These Complex Profiles are mapped by name (in the Profile Mapping panel) to a corresponding SAF Cross-Section, which includes a material definition.

Map Archicad Building Material (By Name)

- For each listed Archicad Building Material, choose a corresponding SAF Material definition.
 - In the pop-up, available SAF Materials are listed according to their predefined SAF Material Types (e.g. Concrete, Steel...)



- If you don't find the material you need, add more materials by clicking the **SAF Materials** button.

See [Add More SAF Materials to Mapping List \(Export to SAF\)](#).

- To map an additional Archicad Building Material, do one of the following:
 - From the **Add** button, click **Add used Core Building Materials from Current Project**, or
 - Click **Add**, then choose from the Archicad project's list of Building Materials

SAF Translator

IDENTIFICATION

MATERIAL MAPPING - IMPORT

MATERIAL MAPPING - EXPORT

Map Archicad Building Material to SAF Material

Use Element Property for more precise mapping

Archicad Building Material	Value	SAF Material	Type	Strength Grade
Aluminium		Aluminium	Aluminium	
Brick - Structural		Masonry	Masonry	
Concrete - Structural		C12/15	Concrete	
Masonry Block - Structural		Masonry	Masonry	
Reinforced Concrete - Prefab		C40/50	Concrete	
Reinforced Concrete - Prefab	C50/60	C50/60	Concrete	
Reinforced Concrete - Structural	C20/25	C20/25	Concrete	
Reinforced Concrete - Structural		C25/30	Concrete	
Reinforced Concrete - Structural	C30/37	C30/37	Concrete	
Steel		S 235	Steel	
Steel - Structural		S 235	Steel	
Steel - Structural	S 275	S 275	Steel	

Add Delete SAF Materials...

Add used Core Building Materials from Current Project - IMPORT

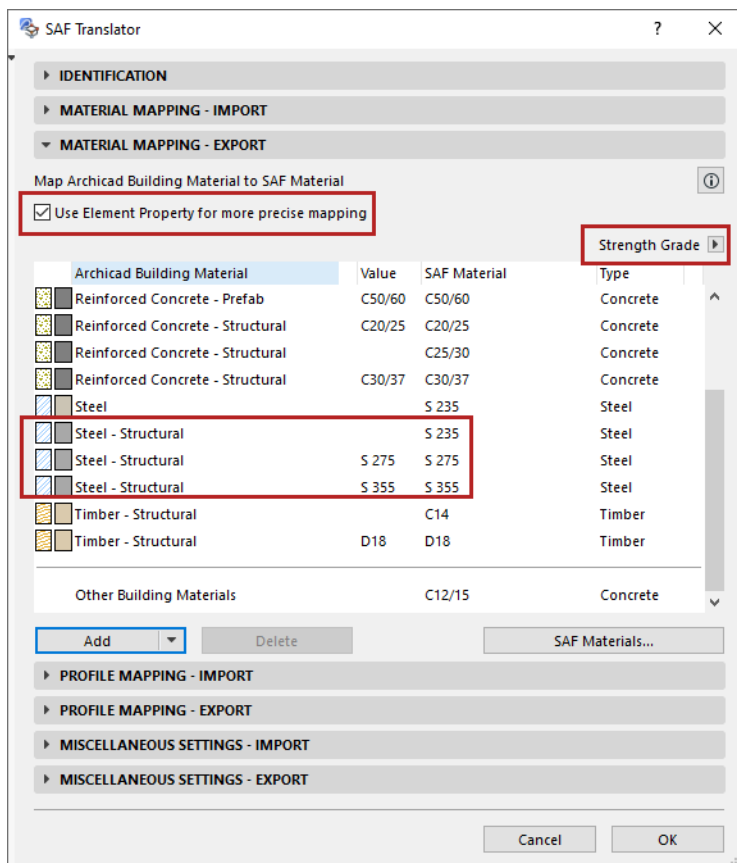
Use Element Property for More Precise Material Mapping

You may need to map a single Archicad Building Material to multiple SAF materials. In Structural Analysis applications, various strength grades might be used for the same material type (e.g. Concrete - C20/25, C30/37, etc.).

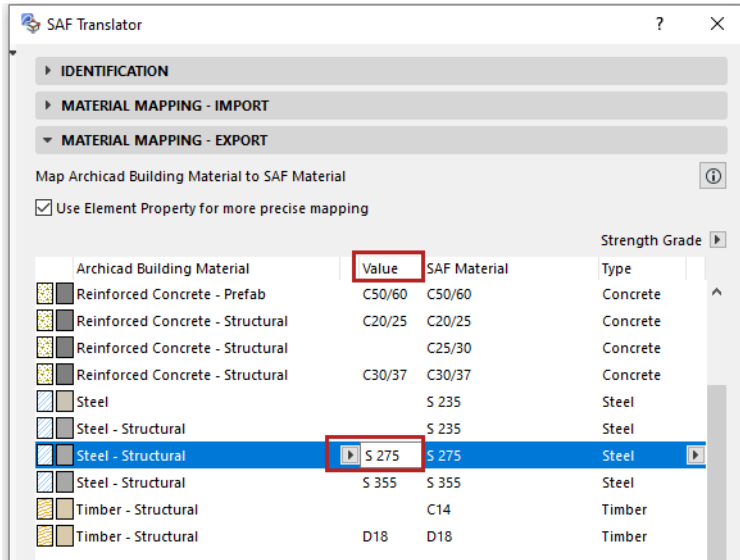
1. Check the box to include 'Use Element Property for more precise mapping'.
2. The property selection pop-up is now available. Choose a single Archicad property to differentiate the Building Material (here, Strength Grade).

Note: The pop-up contains all Properties of the Archicad project.

For example, the "Steel - Structural" Building Material in this Archicad model requires three different SAF mapping definitions, to reflect the three different Strength Grade values: S 275, S 355 and undefined.



3. The “Value” column is added to the mapping rule: click in the Value field to enter the relevant data (e.g. S 275 or S 355 for the Steel-Structural Building Material).

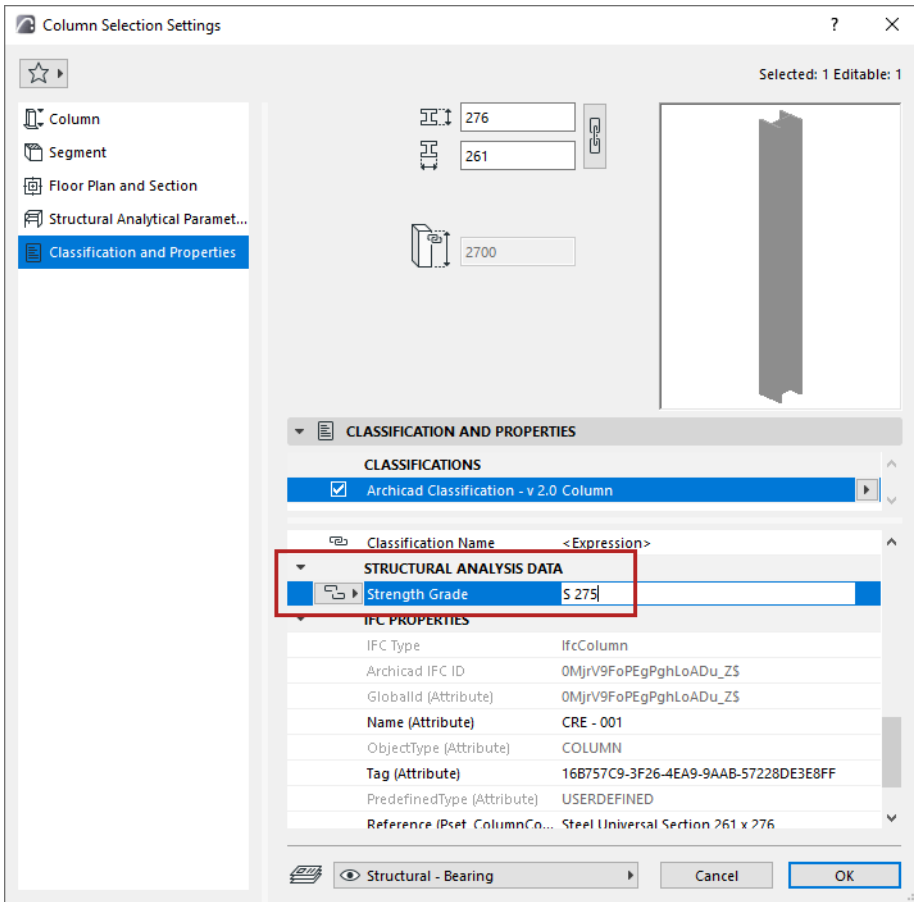


Important Notes:

- In a given Archicad model, only a single Element Property can be used as a value in SAF Material Mapping.
- Only **string-type** Element Properties can be used for SAF Material Mapping.
- To map by property value, the property value must be assigned to Archicad elements (e.g. in Beam or Column Settings) - not to Building Materials.

Example: Building Material with various Strength Grade property values

In this model, several Archicad elements use the Steel-Structural Building Material, but with varying values assigned to their Strength Grade property, in the Classification and Properties Panel of Element Settings.



Add More SAF Materials to Mapping List (Export to SAF)

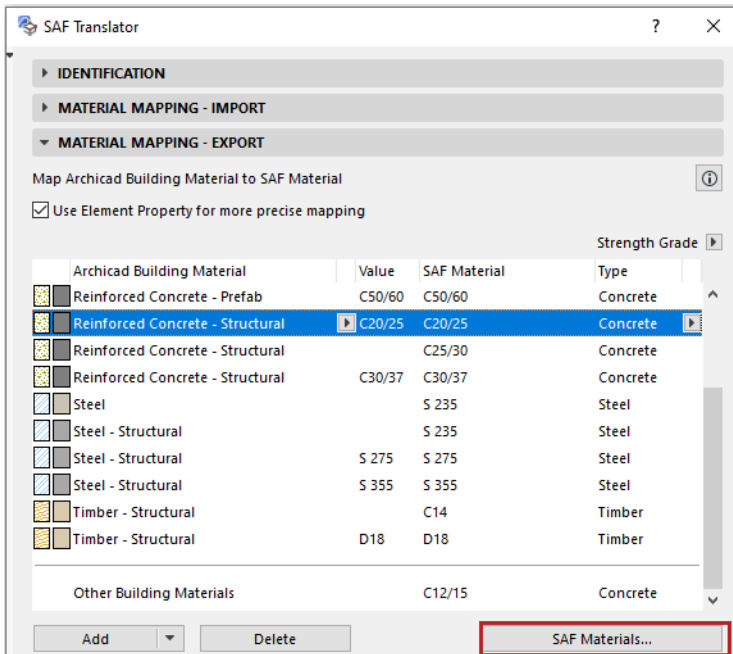
In the Material Mapping - Export panel of the SAF Translator:

There are two ways to expand the browsable list of SAF Materials: Import a file, or add by hand.

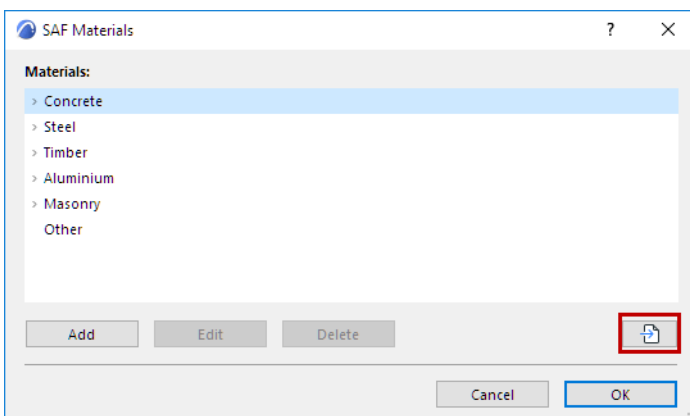
Import SAF Material List from SAF File

This is the preferred method, because you can avoid typing errors.

1. Click **SAF Materials**.



2. Click the **Import** button.



3. Browse for an.xlsx (SAF) file prepared for this purpose. (It should contain the list of Materials used by the receiving structural analytical program.)

See [Prepare and export a SAF Data File to ensure standard mapping.](#)

4. As needed, select and edit any of the materials.

5. Click OK. The chosen SAF file's Materials are automatically added to the SAF Material list in Archicad's SAF Translator dialog.

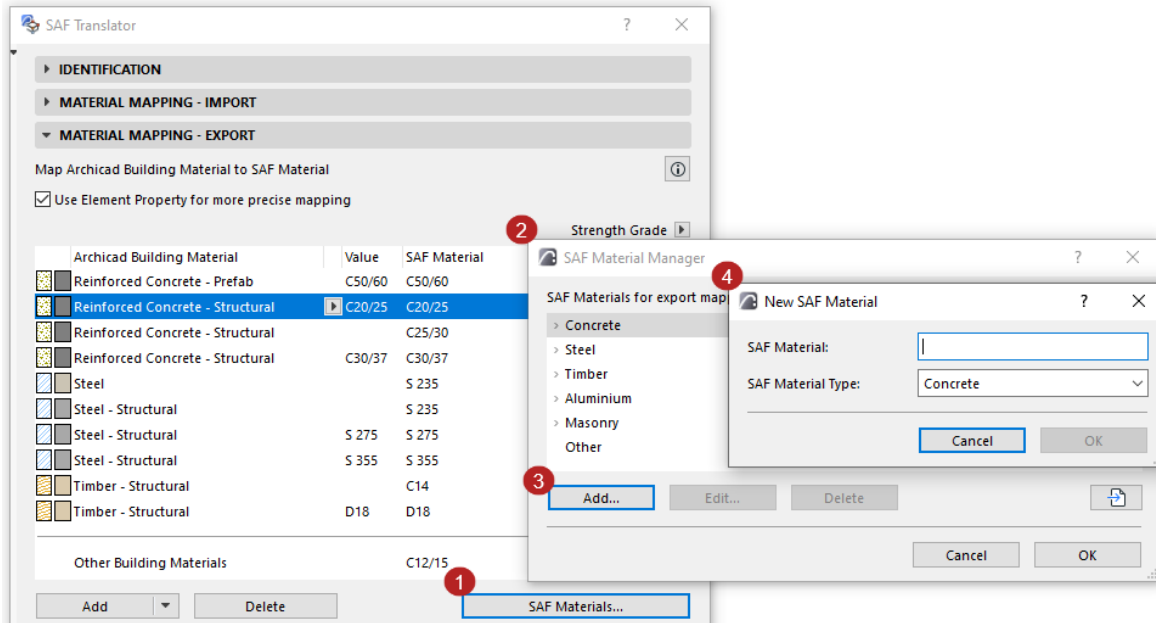
Add or Edit SAF Materials Manually

1. Click **SAF Materials**.
2. Select one or more Materials from the dropdown list (these are standard SAF Material types), then click **OK**

Note: To edit data for any listed Material, select it in the list and click **Edit**.

Or:

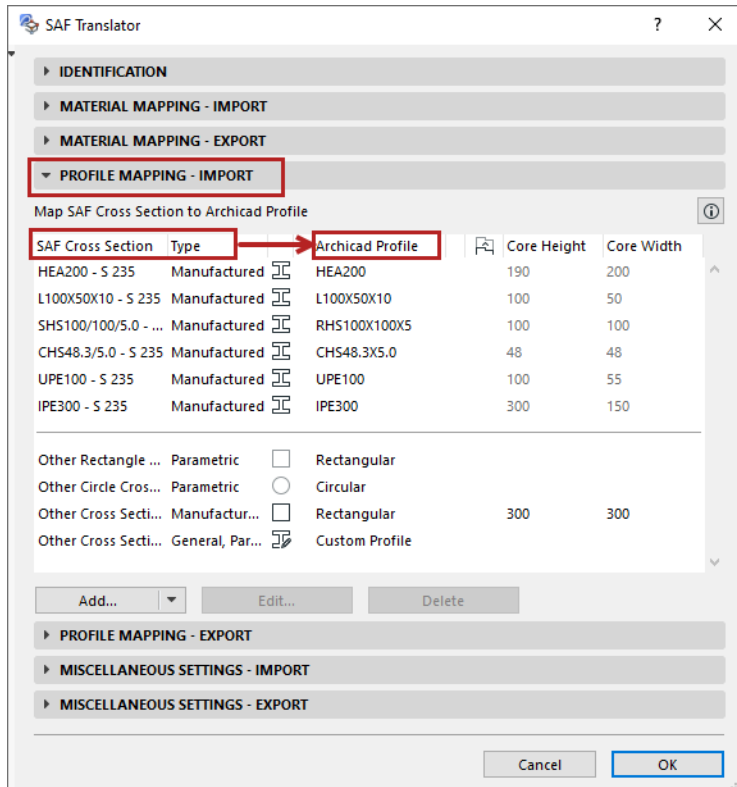
3. Click **Add**
4. Enter the SAF Material name and Material type, then click **OK**.



Profile Mapping - Import

Go to **File > Interoperability > SAF > SAF Translators**.

Use **Profile Mapping - Import** panel to map incoming SAF Cross Sections (1D Members only) to Archicad Profiles.



Topics in this Section

[Map SAF Cross Sections By Name](#)

[Add More SAF Cross Sections \(Import SAF\)](#)

[Use external database to translate unmapped profiles](#)

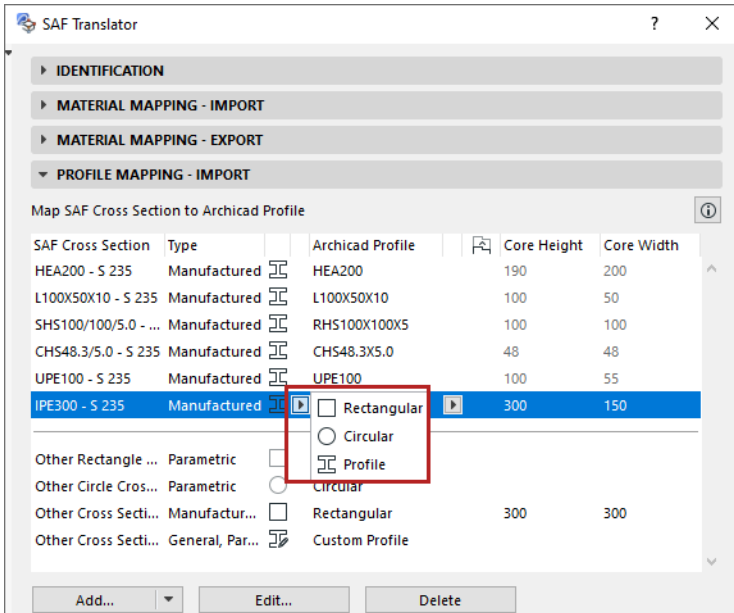
[Mapping Imported “General” or “Parametric” SAF Cross Sections](#)

[Parametric SAF Cross Sections Recognized by Archicad](#)

Map SAF Cross Sections By Name

In the Profile Mapping - Import panel of the SAF Translator:

1. From the Type pop-up, choose **Profile**.

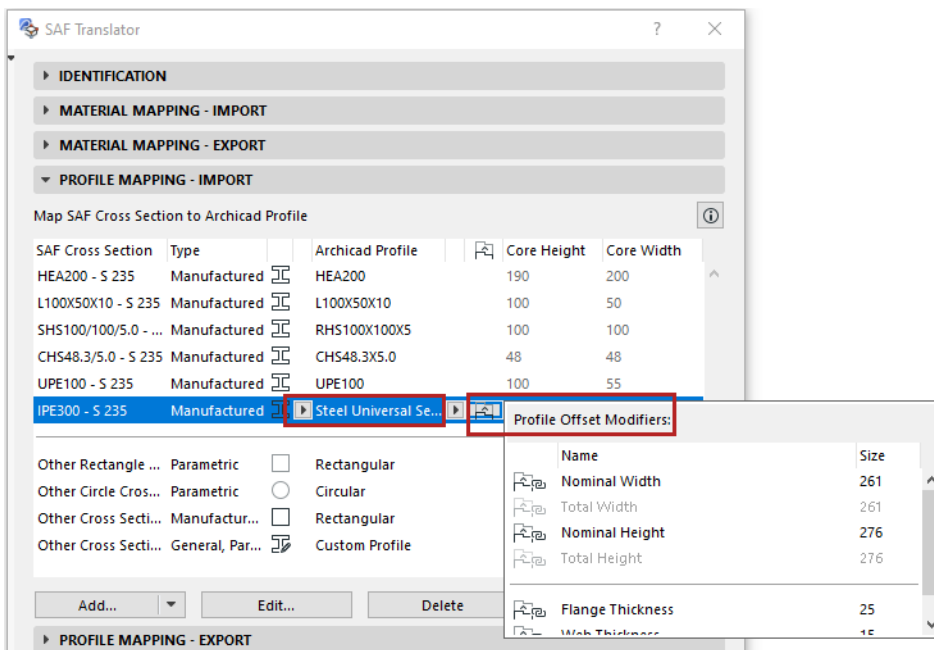


2. From the Archicad Profile pop-up, choose a Profile from the current project. (Available Profile attributes: those with a core and used with Beams or Columns.)

Note: You can use **Options > Complex Profiles > Import Standard Steel Profile** to add new Profile attributes to the project.

See also [Use Standard Steel Column or Beam Profile](#).

3. View and edit the Complex Profile's Offset Modifiers as needed, to best match the incoming Cross Section.



To expand the list of mappable SAF Cross Sections: see [Add More SAF Cross Sections \(Import SAF\)](#).

Automatic Mapping of Manufactured Cross Sections

For incoming, manufactured SAF profiles that are NOT mapped by name in the Profile Mapping list: you can activate automatic mapping based on a default database.

See [Use external database to translate unmapped profiles](#).

Add More SAF Cross Sections (Import SAF)

In the Profile Mapping - Import panel of the SAF Translator:

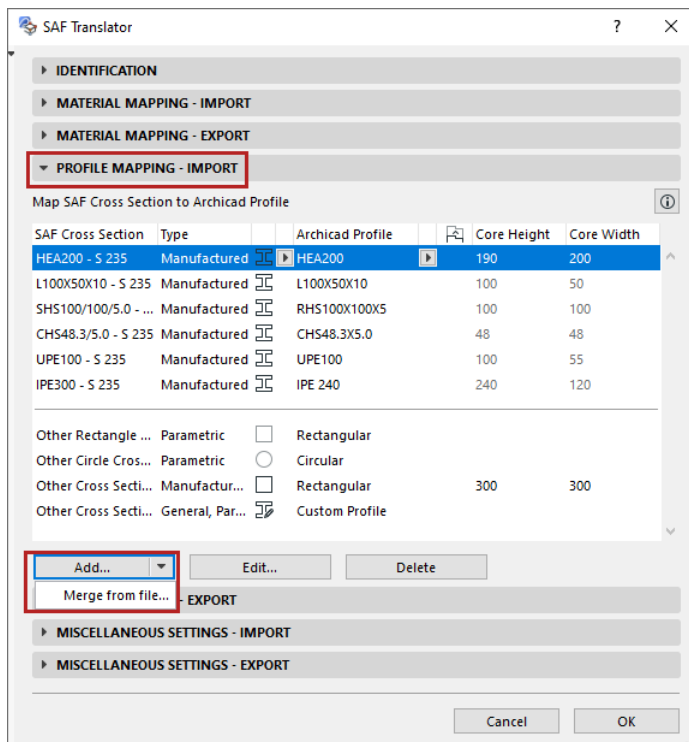
There are two ways to expand the list of mappable SAF Cross Sections:

- Merge from file
- Add manually

Merge Cross Sections from SAF File

This is the preferred method, because you can avoid typing errors.

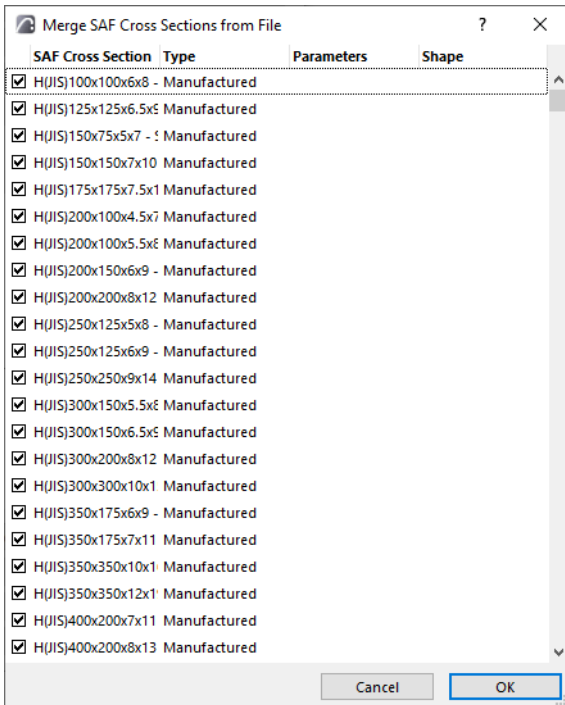
1. From the Add button, click **Merge from file**.



2. Browse for an .xlsx (SAF) file prepared for this purpose.

See [Prepare and export a SAF Data File to ensure standard mapping](#).

3. Cross Sections from the SAF file appear in a dialog. Review this list and check the ones to import.
4. Click OK.



Add or Edit SAF Cross Sections Manually

1. Click **Add** or (for a selected Cross Section in the list) **Edit**.
2. In the appearing dialog, manually enter or edit the new SAF Cross Section name/material.
Note: Only “Manufactured” type SAF Cross Sections can be added here, since they use standard names.
3. Choose a SAF Cross Section Formcode from the pop-up.
4. Choose a SAF Material type (steel or aluminum).

Priority of Mapping Rules for Manufactured Cross Sections

Mapping rules are applied in the following order:

1. Mapping by name, based on the list in the Profile Mapping - Import panel
2. External database (provided that the “Create Profiles for Unmapped Manufactured Cross Sections” box is checked).
3. “Other Cross Sections - Manufactured” setting. This is the catch-all definition for all manufactured Cross Sections that have not been mapped in steps 1 and 2 above.

Mapping Imported “General” or “Parametric” SAF Cross Sections

You can map any “General” or “Parametric” SAF Cross Section to a particular named Archicad Profile (as with Manufactured types). ([See Map SAF Cross Sections By Name.](#))

However, unlike Manufactured types, the SAF definition of the “General” and “Parametric” Cross Sections includes geometric and parametric information that can be recreated or recognized in Archicad.

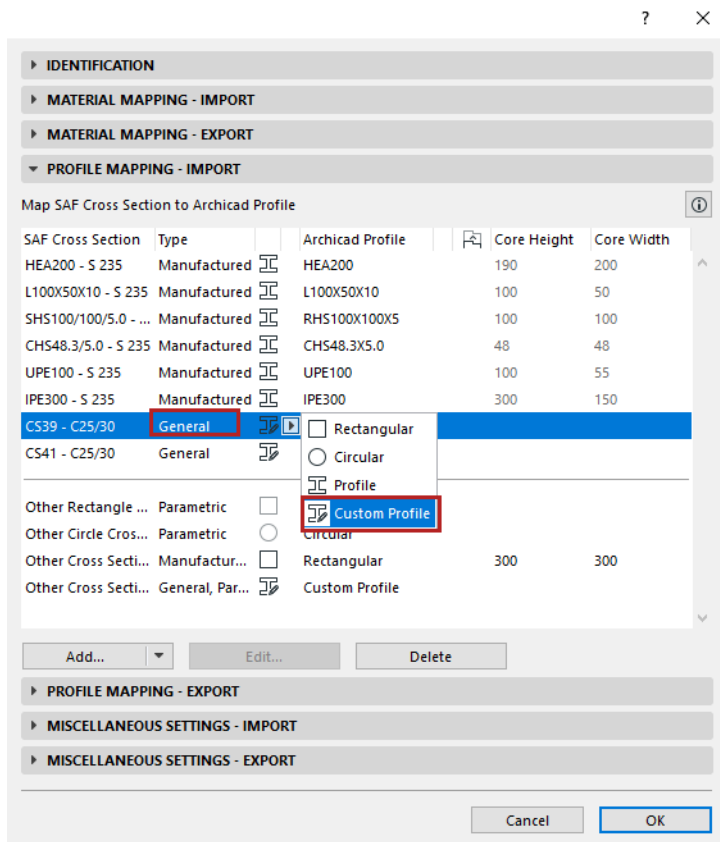
To map these incoming Cross Sections to specific Archicad Profile attributes, use one of the following two workflows.

Option 1: Map as Custom Profile

Any SAF Cross Section of “General” type can be imported as a Custom Profile.

In addition, certain SAF Cross Sections of “Parametric” types can also be imported as Custom Profiles.

[See Parametric SAF Cross Sections Recognized by Archicad.](#)

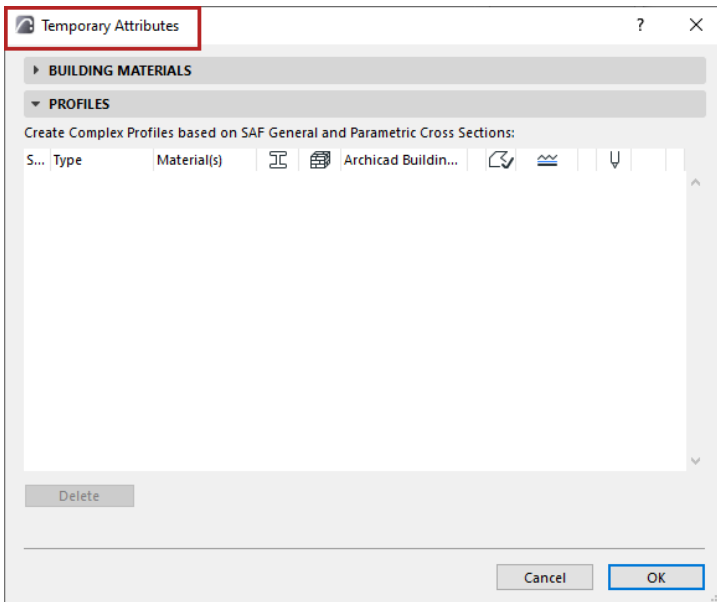


- The polygon of the SAF Cross Section is correctly interpreted and embedded within the Custom Profile, but without any material information. (Their materials are assigned based on the Material Mapping panel.)
- This Custom Profile is not listed among Archicad’s attributes, and you cannot list it by name or otherwise refer to it in the Archicad project.

Note: Once imported to Archicad, you can place this Custom Profile in the project and then save it as a named Profile attribute.

Option 2: Create New Archicad Profile Attribute Through Attributes Palette

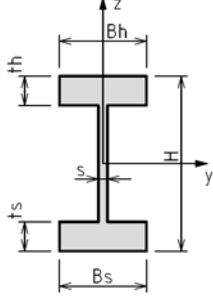
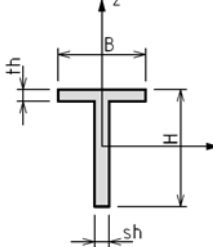
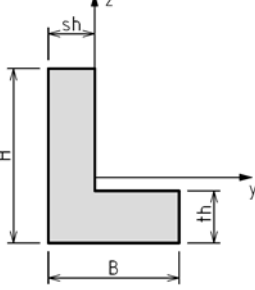
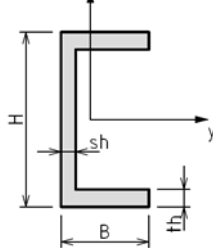
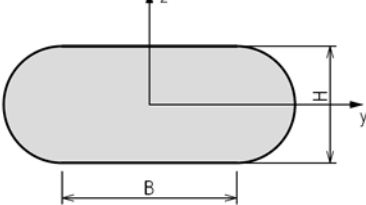
1. Open Options > Element Attributes > Attributes.
2. Click the **Import** button.
3. From the Import Attributes dialog: browse for an .xlsx (SAF) file prepared for this purpose.
[See *Prepare and export a SAF Data File to ensure standard mapping.*](#)
4. A Temporary Attributes dialog appears. The Profiles panel lists the potential new Complex Profile attributes, based on the SAF Cross Sections (General types and supported Parametric types) listed in the incoming file.

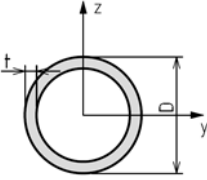
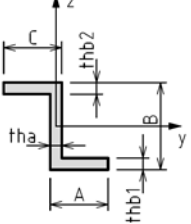
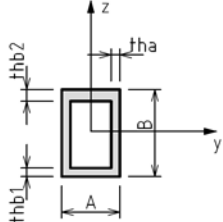


5. Assign Archicad Building Material, Fill Type, Line, and Pen to each SAF Cross Section you want to append.
Note: If a Cross Section contains multiple SAF Materials, only one Archicad Building Material can be assigned to it. However, after you add it to the Archicad Project as a new Profile, you can use Profile Editor to set different Building Materials for each Core component.
6. Click OK to return to Attributes.
7. Append the new Profiles to the Archicad project.

Now you can map the incoming SAF Cross Section to the corresponding, named Archicad Profile attributes (instead of "Custom Profile").

Parametric SAF Cross Sections Recognized by Archicad

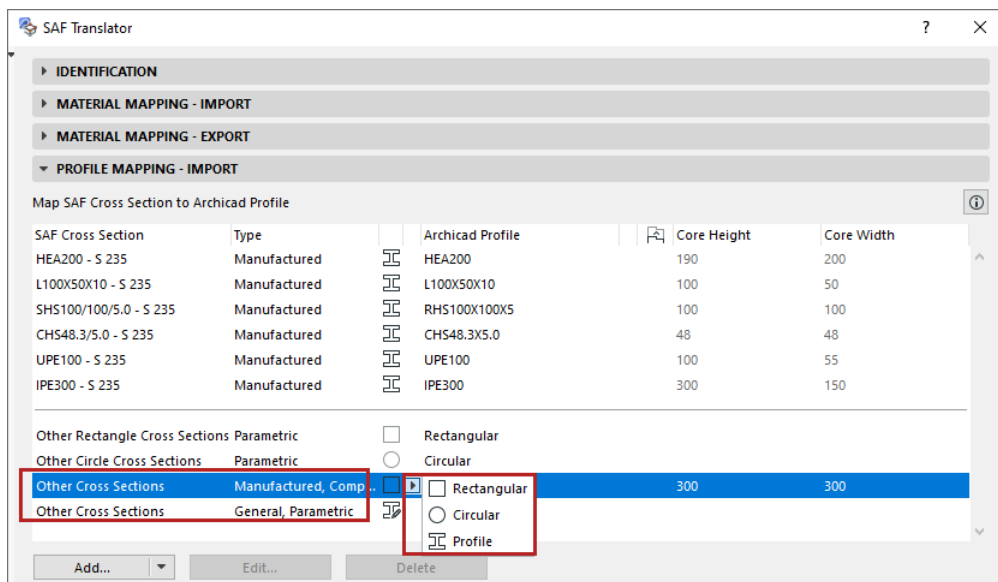
Name	Parameters	CS Shape	CS Name in SCIA Engineer
I Section	H; Bh; Bs; ts; th; s		I ng (H; Bh; Bs; ts; th; s)
T Section	H; B; th; sh		T g (H; B; th; sh)
L Section	H; B; th; sh		L g (H; B; th; sh)
U Section	H; B; th; sh		U g (H; B; th; sh)
Oval	H; B		Oval (H; B)

Name	Parameters	CS Shape	CS Name in SCIA Engineer
Pipe	D; t		Tube (D; t)
Z Section	A; tha; B; thb1; thb2; C		Z (A; tha; B; thb1; thb2; C)
Box	A; tha; B; thb1; thb2		O (A; tha; B; thb)

Default Mapping for “Other” Cross Sections (Import SAF)

- The “**Other Rectangular/Circular**” default rules generically map these incoming Cross Sections as “Rectangular” or “Circular”, with their actual dimensions (height/width or core diameter).
- The “**Other Cross Sections**” mapping rules apply to all other incoming SAF Cross Sections that are not mapped by name.
 - **Manufactured or Compound** Types: Choose a default Archicad structure (Rectangular, Circular, Profile)
 - **General or supported Parametric** Types: You have the additional option to import these as an Archicad “Custom Profile”.

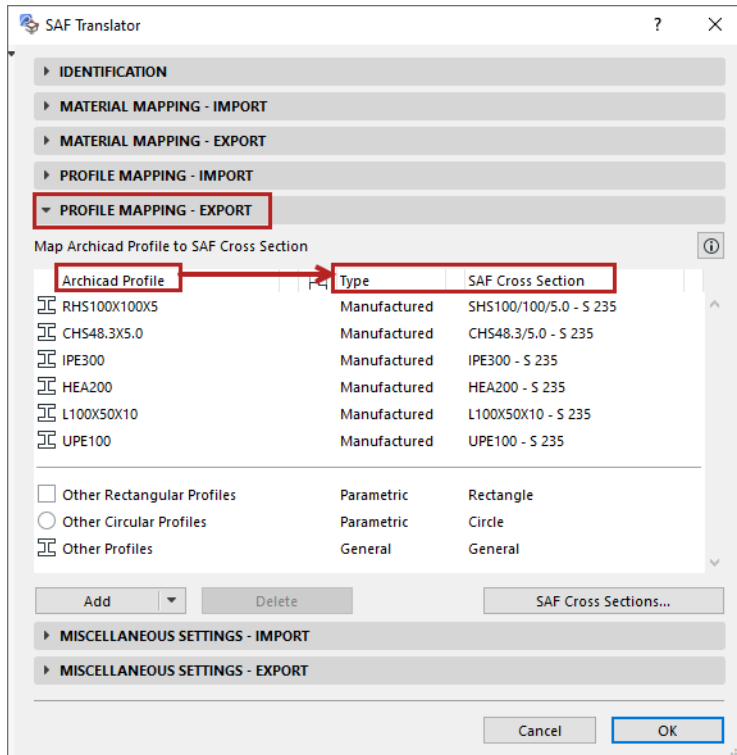
See also [Mapping Imported “General” or “Parametric” SAF Cross Sections](#).



Profile Mapping - Export

Go to **File > Interoperability > SAF > SAF Translators**.

Use the **Profile Mapping - Export** panel to map **1D Members** that are Complex Profiles to their exact counterparts (“SAF Cross Section”) in the receiving software.



Cross Section Types in SAF

The SAF standard defines four Types of Cross Sections:

- **Manufactured:** Industrially manufactured cross-sections, with standard naming.
- **Compound:** Compounded section based on multiple manufactured profiles (e.g. two I-sections). Example of compound cross-section input: profile = IPE200 and parameters=10mm (distance between profiles).
- **Parametric:** Cross Sections defined by shape and dimensions (parameters).
- **General:** Serves to define any general shape of the cross-section. Consists of one or more closed polygons including openings. These do not use standard naming.

Topics in this Section

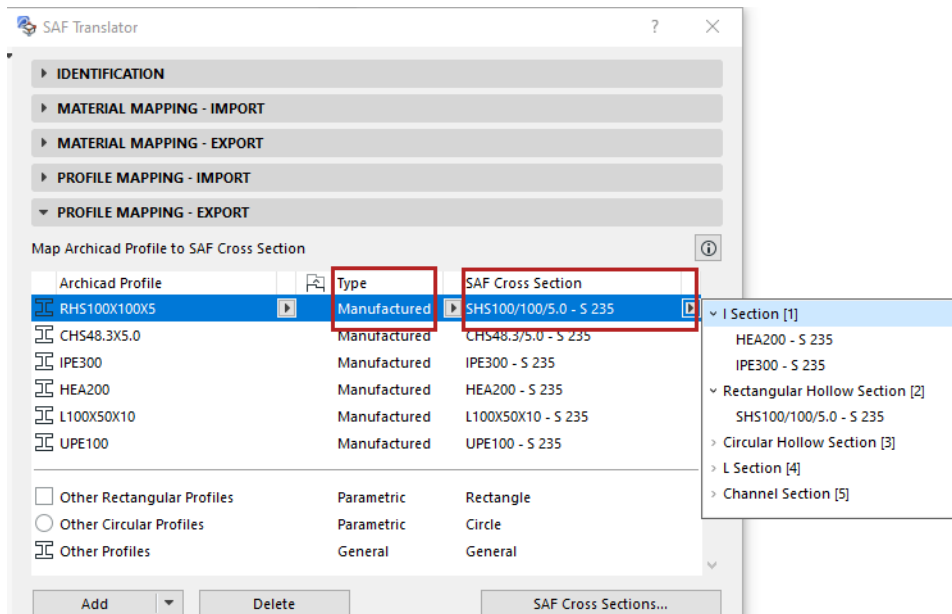
[Map Complex Profiles \(By Name\)](#)

[Use external database to translate unmapped profiles](#)

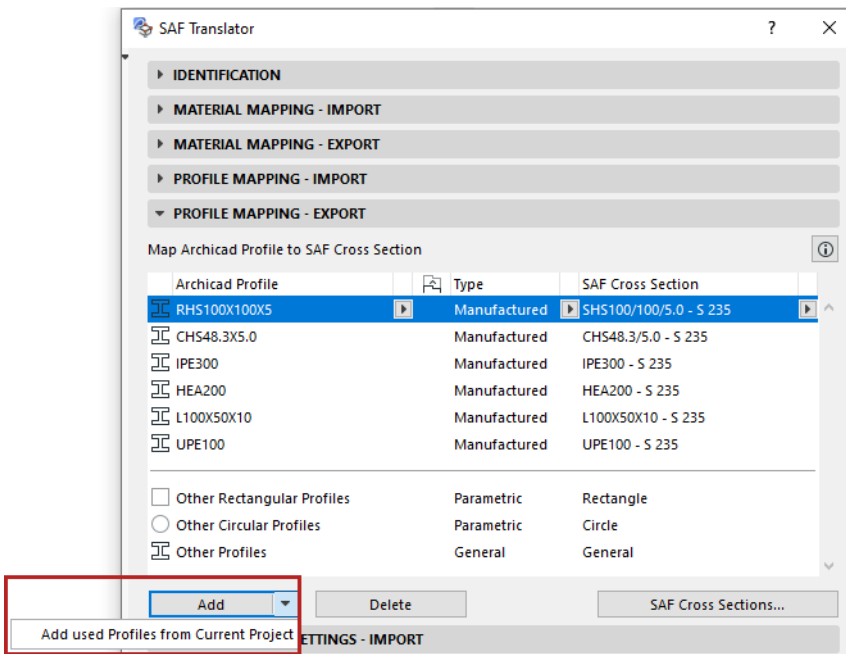
[Default Mapping for Exported Cross Sections](#)

Map Complex Profiles (By Name)

1. For each listed Archicad Profile, choose a corresponding Type and Cross Section.



- The SAF Cross-section pop-up is filtered by the chosen type (Manufactured, Parametric or Compound) and listed according to SAF Formcode.
 - If you don't find the SAF Cross Section you need in the pop-up, you can add more to the list. See [Add More SAF Cross Sections to Mapping List \(Export to SAF\)](#).
 - For all Profiles mapped by name, the SAF Cross Section *includes the material definition*.
 - Profiles whose type is "General" are not mapped by name. They will be recreated as SAF Cross Sections based on the Profile polygon or geometry. Their material is defined in the Material Mapping panel.
2. To add an additional Archicad Profile to the list, do one of the following:
 - From the Add button, click **Add used Profiles from Current Project**



- Click **Add**, then choose from the Archicad project's list of Complex Profiles. (Available Profile attributes: those with a core and used with Beams or Columns.)

Add More SAF Cross Sections to Mapping List (Export to SAF)

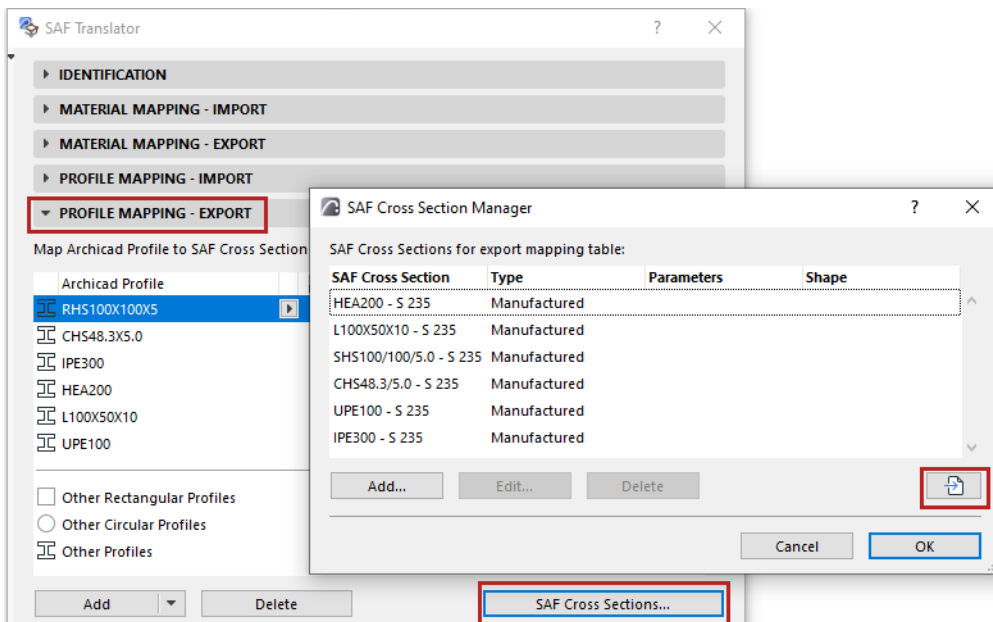
In the Profile Mapping - Export panel of SAF Export Translator: there are two ways to expand the list of available SAF Cross Sections:

- Import a file
- Add manually

Import SAF Cross Sections from SAF File

This is the preferred method, because you can avoid typing errors.

1. Click **SAF Cross Sections**.
2. Click the **Import** button.



3. Browse for an .xlsx (SAF) file prepared for this purpose.

See [Prepare and export a SAF Data File to ensure standard mapping](#).

4. The SAF Cross Sections from the .xlsx (SAF) file (Parametric, Manufactured and Compound types) are now listed in the dialog.

Note: General types and Rectangular/Circular parametric types are not listed!

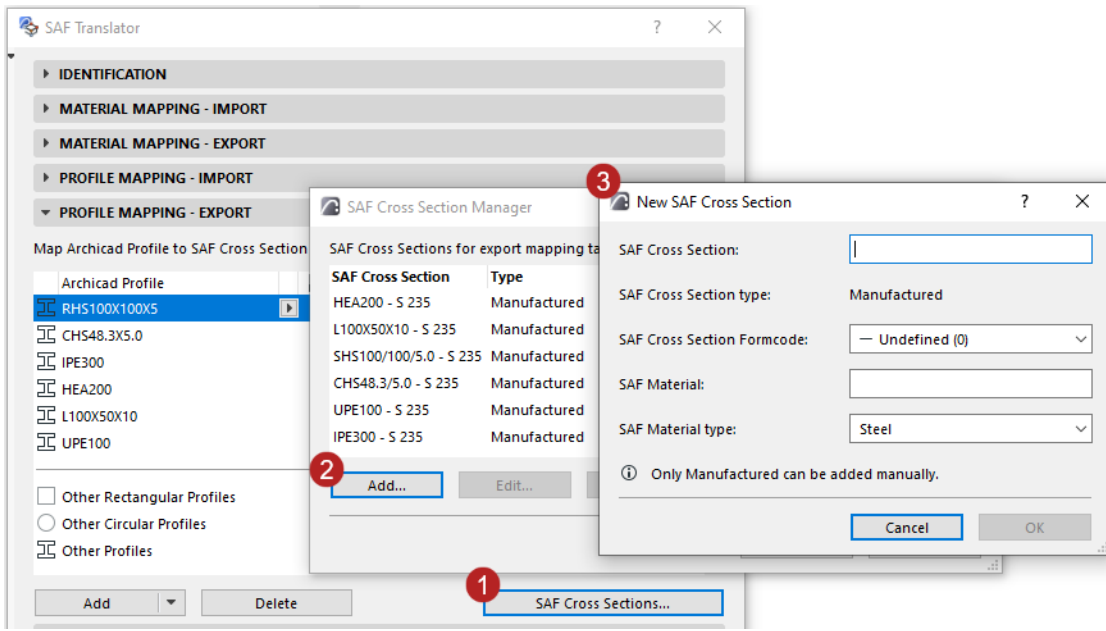
5. Sort and select the listed Cross Sections as needed.
6. Click **OK** to add them to the Translator.

Add or Edit SAF Cross Sections Manually

1. Click **SAF Cross Sections**.
2. Click **Add**
3. In the New SAF Cross Section dialog.

– Enter the SAF Cross Section name and Material type.

Note: Only “Manufactured” type SAF Cross Sections can be added here, since they use standard names.

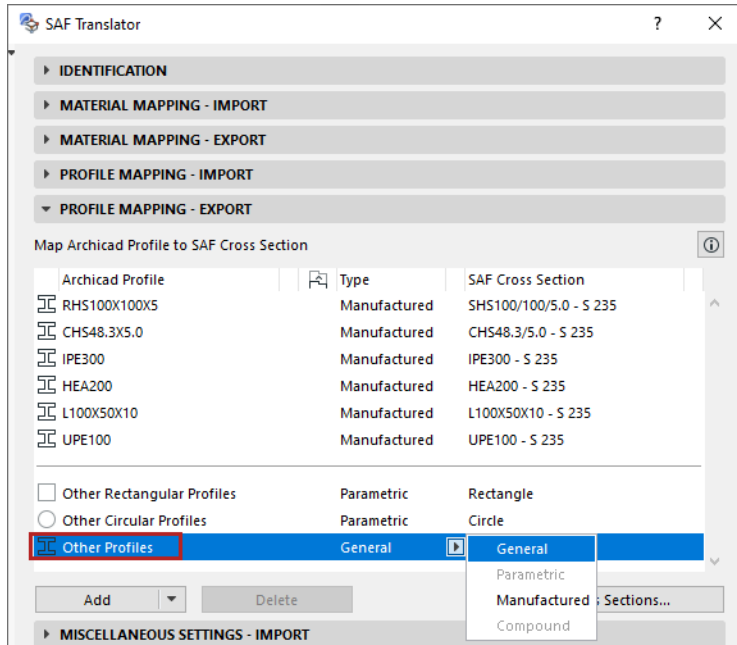


- Choose a SAF Cross Section Formcode from the pop-up.
- Choose a SAF Material type (steel or aluminum).

To edit data for any listed Cross Section, select it in the list and click **Edit**.

Default Mapping for Exported Cross Sections

- Rectangular and circular (solid) Beams and Columns are handled automatically by the “Other Rectangular Profiles” and “Other Circular Profiles” default rules: they are converted to “Parametric - Rectangle” or “Parametric - Circle” SAF Cross Sections, with their actual dimensions (height/width or core diameter). (Their materials are defined based on the Material Mapping panel.)
- Rectangular or circular profiles which are *hollow* (tubular) are handled as “Other Profiles”.
- The “Other Profiles” default mapping rule applies to all other exported Complex Profiles that have no specific mapping. Its default Type can be either “General” or “Manufactured”.

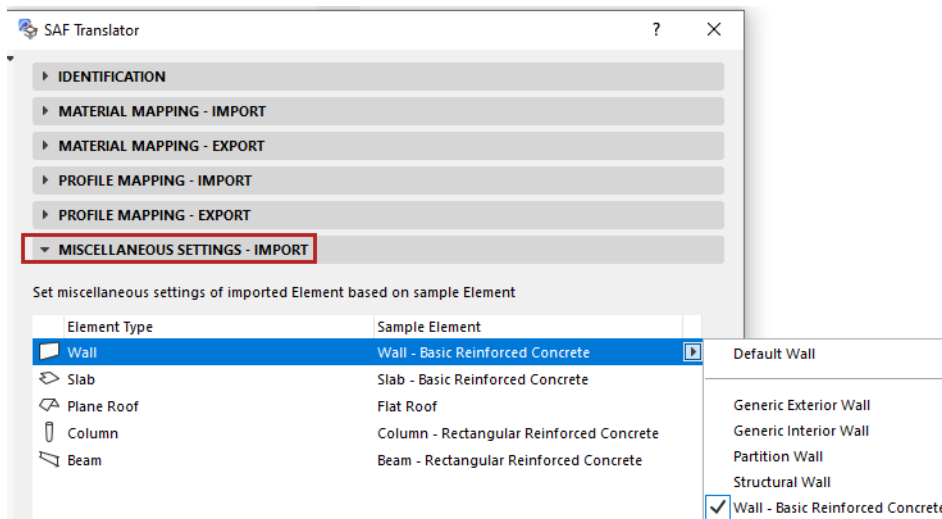


Miscellaneous Settings - Import

Go to **File > Interoperability > SAF > SAF Translators**.

Use the **Miscellaneous Settings - Import** panel to choose default Archicad tool settings for each imported analytical Member type (Wall, Slab, Roof, Column and Beam).

For each element type, use the pop-up to choose either the Archicad Tool default settings, or one of its Favorites.



What are these settings used for?

Many structural analytical parameters of imported SAF elements are correctly recreated (or mapped by hand) in Archicad: geometry, location, material and cross section parameters, etc. When importing a SAF file, these SAF data will overwrite the corresponding values in Archicad.

However, analytical elements imported to Archicad are transformed into physical elements that have many other parameters (e.g. layer, pen set, classification) that are *not* defined in SAF.

Therefore, to define its physical appearance in Archicad, each imported analytical element will use the tool settings definitions you choose in the Miscellaneous Panel of the SAF Import Translator (for all settings that are undefined in SAF).

The following table summarizes the Archicad elements created out of 1D and 2D Members imported from SAF:

Imported 1D Member		Archicad Element
	Slant angle < 45 deg	Beam
	Slant angle > 45 deg	Column
Imported 2D Member		Archicad Element
	Plane is horizontal	Slab
	Plane is 0 > 45 deg	Single-plane Roof
	Plane is 45-90 deg	Wall

Miscellaneous Settings - Export

Go to **File > Interoperability > SAF > SAF Translators**.

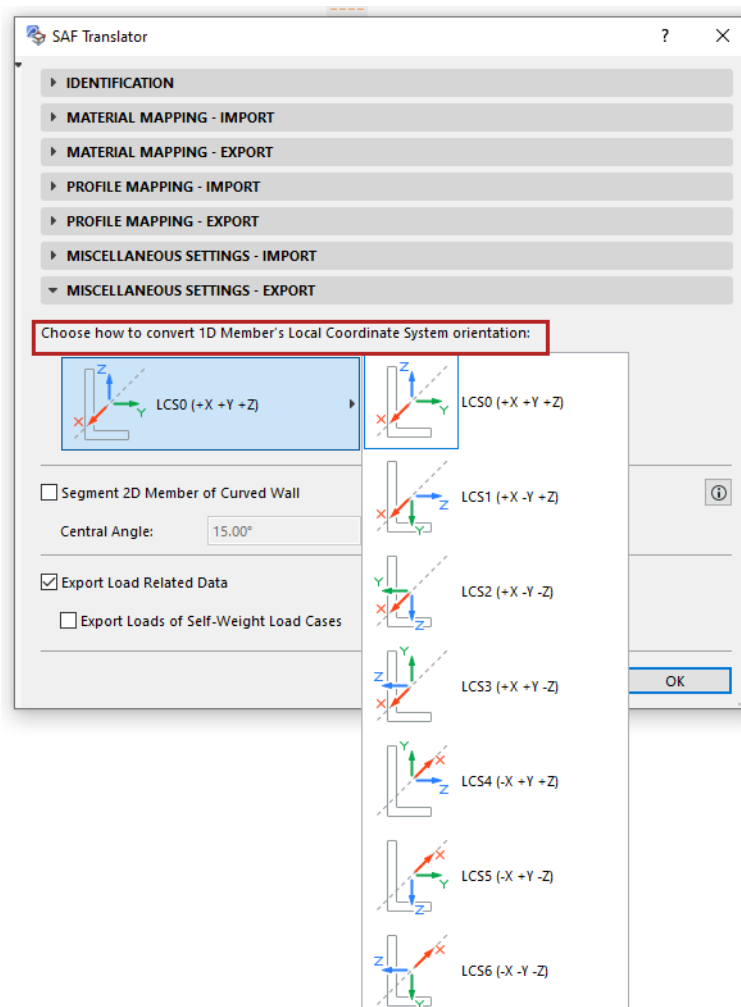
Use the **Miscellaneous Settings - Export** panel for the following options:

Local Coordinate System (Export to SAF)

In each receiving software, a cross section from the catalog is placed into the workspace based on its default XYZ orientation, as defined in the Local Coordinate System (LCS) of the host 1D Member.

Since the LCS standard varies by software, you must choose the standard which the receiving software uses: click one of the LCS standards shown in this panel of the SAF Export Translator.

At SAF file export, Archicad will convert the outgoing elements accordingly, so that 1D Member Cross Sections will be oriented correctly in the receiving software.



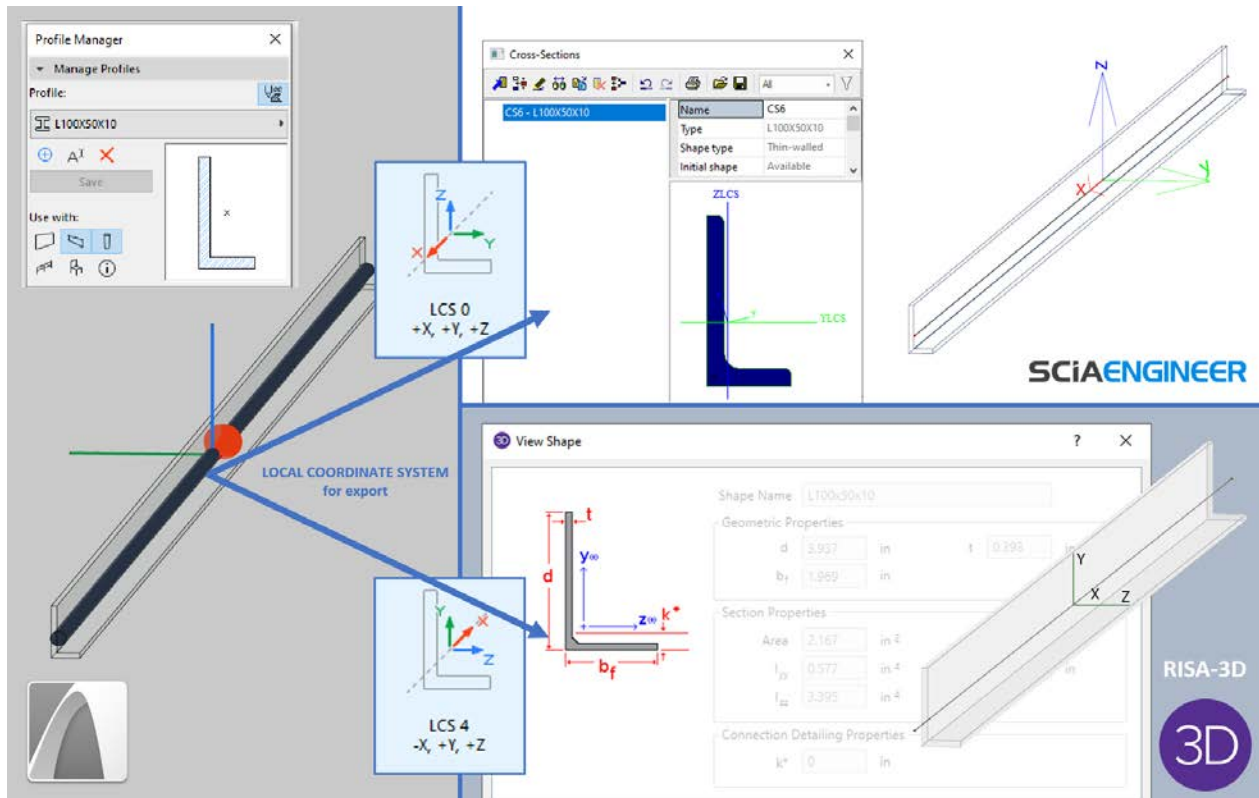
How Do I Know Which Local Coordinate System to Choose?

If you have imported predefined SAF Translator Settings optimized for a particular software, the Local Coordinate System is correctly defined.

[See Download SAF Data Files optimized for structural applications.](#)

To identify the LCS used by the receiving software, do the following (and view the image below):

1. Map an L-shaped Archicad Profile Attribute to its counterpart in the receiving program. (Use a non-symmetrical profile - such as L - whose orientation is obvious.)
2. In the receiving software, place the corresponding L Cross Section using its default placement (without extra rotation or mirroring), so that it “faces” the same way as the Archicad profile.
3. View the placed Cross Section position in the LCS of the 1D member. Typically, European software (e.g. SCIA Engineer) use LCS 0, while U.S. software use LCS 4 (e.g. RISA-3D) or LCS 3.
4. In the SAF Translator for Export, choose the LCS option that corresponds to your receiving software.

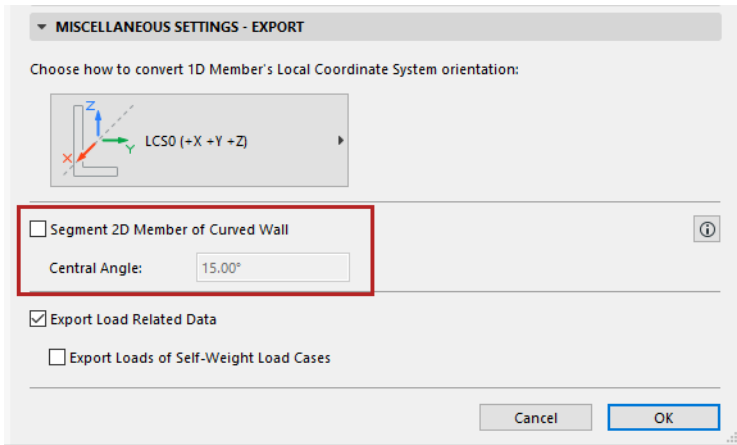


Note: You do not have to set an LCS in the SAF Import translator. When importing a SAF file, Archicad automatically converts the incoming element positions, based on the “LCS of cross-section” Model data.

Segment 2D Member of Curved Wall

This control affects the SAF export of 2D Members that are generated from curved Walls.

Segmentation is off by default. Leave it off if your receiving structural analysis software can handle curved 2D Members generated from curved Walls.



When Should You Enable Segmentation?

Check the box only if your receiving structural analysis software *cannot* handle curved Walls. The curved 2D Member will be exported as multiple segments.

In the **Central Angle** field: Define a maximum central angle for the resulting segments.

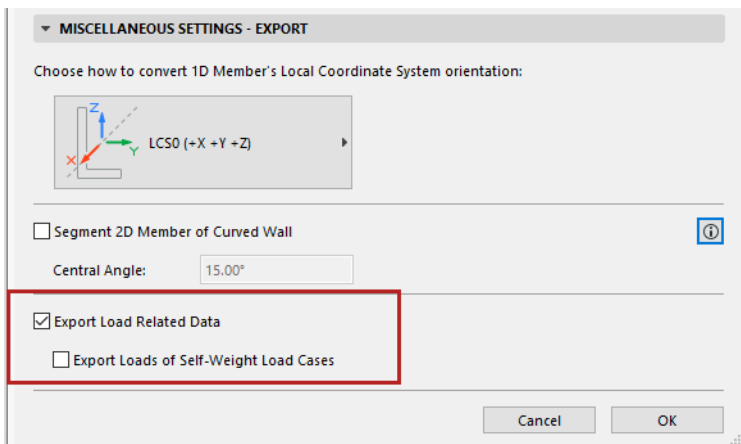
- The resulting segments will approximate the central angle you have defined (without exceeding it), but are subject to the following adjustments:
 - If the curved Wall contains additional nodes besides its endpoints (e.g. structural nodes where a connecting element meets the Wall), then segments will be created so that their endpoints coincide with these existing nodes.
 - Based on these node locations, actual Central angle values (and segment lengths) will vary.

Important Notes

- If you enable **Segment 2D Member of Curved Wall**: the receiving software will generate additional segment endpoints to coincide with the nodes of any Structural Support and Structural Analytical Links that are attached to the curved Wall.
- The newly generated, inner edges of the exported Wall segments will have Edge releases set to Rigid (regardless of the Release values of the original Wall edges).

Export Load-Related Data

On by default.



Disable this control if you want to ignore all Load-related data when exporting the Structural Analytical Model to SAF.

For example, in the following scenarios involving data roundtrips:

- the structural software uses Load data not recognized by Archicad
- the architect chooses not to incorporate the SAF file's Load data into the Archicad model

In such cases, roundtrip data exchange can result in the loss of Load-related data. To avoid this loss, you may wish to disable the checkbox. Load data will be ignored.

Export Loads of Self-Weight Load Cases

Off by default.

Most analytical programs do not handle self-weight Loads from external programs.

Create new SAF Translator

Access SAF Translator settings at **File > Interoperability > SAF > SAF Translators**.

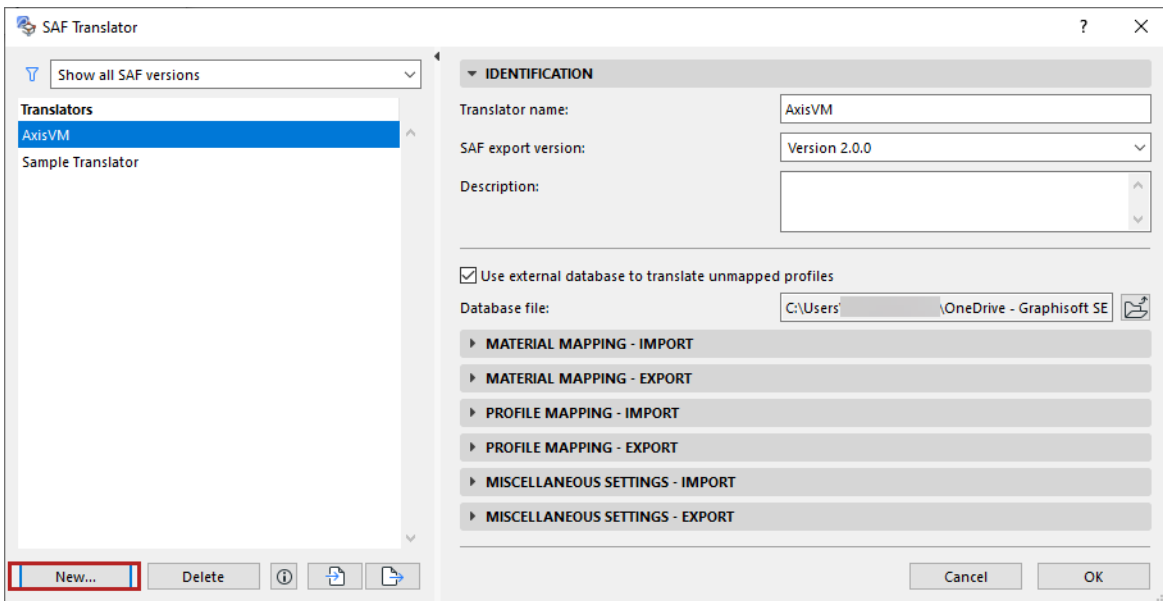
You can set up and store multiple translators for SAF-based file exchange. Customize each translator to meet the particulars of any structural application.

From the SAF Translator dialog:

1. Click the New button.
2. Choose to create either a brand-new translator, or duplicate an existing one, then click OK
3. On the Identification panel: choose a SAF export version to optimize the file to match your analysis program's capabilities.

Note: This is relevant for export only. Archicad can import all SAF versions.

4. Enter an optional description for the translator.



See also:

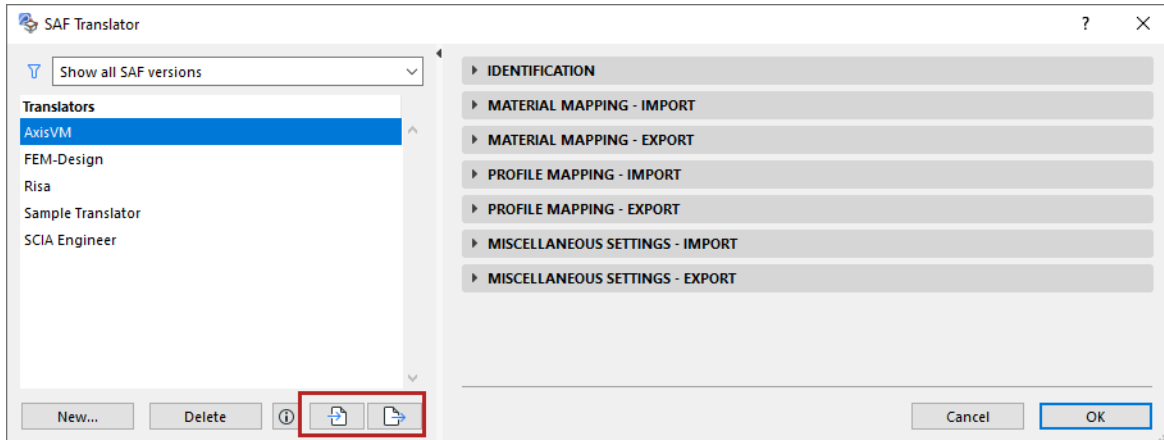
[Import/Export SAF Translators](#)

Import/Export SAF Translators

Access SAF Translator settings at:

File > Interoperability > SAF > SAF Translators.

Use the “Export” and “Import” buttons to save SAF Translator Settings and/or exchange them in XML format.



See also [Download SAF Data Files optimized for structural applications.](#)

Visualization

Visualization in Archicad encompasses all the tools and techniques that allow you to display and present your architectural design to your customer.

You can visualize your entire project or parts of it by adding dedicated **cameras** to the floor plan and then viewing the data from that vantage point.

For **photorenderings**, the built-in Cineware Engine comes with pre-set scenes and user-friendly Basic Settings. Cineware is optimized to work with corresponding Surfaces (with a catalog of hundreds of predefined surfaces) and Light objects. Even if you only use the Basic-level settings, you can quickly create good renderings with a minimum of effort.

However, the full menu of Detailed Settings is also available for Cineware. These correspond to Maxon Cinema 4D version R21, and incorporate most of its parameters.

Archicad offers various options for exporting the entire model into a number of common file formats used by professional rendering software.

- Artlantis is a rendering solution developed by Graphisoft's French distribution partner, Abvent SA. It can handle still images and animations exported directly from Archicad (File > Save as). Moreover, an intelligent link is established between the original model and the Artlantis file.

[See *Export to Artlantis*.](#)

- You can also export the floor plan or 3D view into a variety of applications such as Piranesi, 3D Studio or Photoshop for fine-tuning it or adding special effects.

Graphisoft BIMx lets you create an interactive Hyper-model from your Archicad project. The Hyper-model can be explored in the context of the hyperlinked 2D project documentation and embedded BIM data, using a mobile or desktop app, or in a web browser.

The most commonly used commands related to Visualization techniques can be found on the predefined **3D Visualization** toolbar. Choose **Window > Toolbars > 3D Visualization** to display this toolbar as needed. (This toolbar also contains 3D Window display and **3D Navigation** commands.)

Multi-core processors are recommended to support 3D navigation and 3D model generation.

Topics in this section:

[Cameras](#)

[PhotoRendering](#)

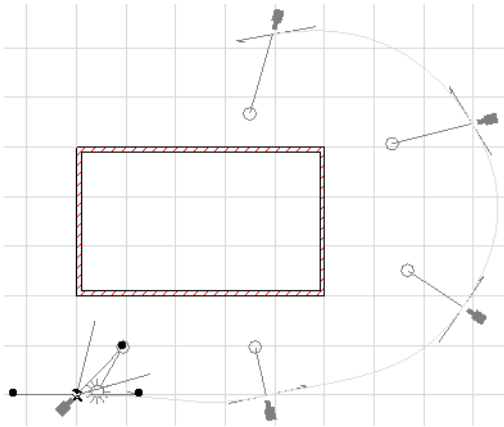
[Visualization Extras](#)

[Graphisoft BIMx](#)

Cameras

You can place any number of Cameras in the Floor Plan for generating perspective views in still image or animation (fly-through) format.

Perspective Cameras allow you to view and edit the entire project or a selected part of it in the 3D Window and to create renderings that represent a snapshot of the current state of the project. They are defined by a viewpoint, a target point and an opening angle.



On the Floor Plan, all Cameras are shown with their target points and paths, and the handles that can be used to manipulate them. Only one camera is active at a time, as indicated by the sun and view angle markers.

Double-click the Camera tool to open Camera Tool Settings.

For details, see [Camera Tool Settings](#).

Camera Path Options

The **Display Options** of perspective Cameras can be set with a pop-up menu in the Path Options dialog box accessed from **Camera Settings**. You can hide the Cameras altogether, show the Cameras with or without the Path line and with or without the in-between frames.

See [Path Options](#).

When the 3D Window is active, you can modify Cameras with the **Navigator** and **Navigator Preview** palettes (click the desired Camera from the list in the Navigator, then modify it in the Navigator Preview). You can also modify the settings of placed cameras by changing the 3D view with the navigation commands, then using the **3D Navigation Extras** commands to adjust the placed cameras to the new viewpoints.

For more information, see [3D Navigation Extras](#).

When you have made your choices in the **Camera Settings** dialog box, you are ready to place Cameras on the Floor Plan.

Your first click on the Floor Plan defines the location of the Camera itself. Draw a rubber band line to set the view direction, and click again to define the target's horizontal position. When the operation is completed, the sun icon appears with default settings.

To apply the camera-defined view defined to the 3D model, select the camera and activate the 3D Window. When you do this, the parameters of the perspective defined by the camera are copied into the **3D Projection Settings dialog box** and the **Navigator** and **Navigator Preview** palettes.

PhotoRendering

The Archicad PhotoRendering feature allows you to create photorealistic images of the 3D model.

Creating quality renderings is a complex function that depends on the interaction among:

- Surfaces
- Light effects
- Rendering settings

Archicad's built-in Cineware Engine comes with pre-set scenes and user-friendly Basic Settings to make this process easier. Cineware is optimized to work with corresponding Surfaces (with a catalog of hundreds of predefined surfaces) and Light objects. Even if you only use the Basic-level settings, you can quickly create good renderings with a minimum of effort.

However, the full menu of Detailed Settings is also available for Cineware. These correspond to Maxon Cinema4D version R21, and incorporate most of its parameters.

The Redshift renderer (GPU based) from Maxon is included in Archicad as a Technology Preview, with limitations. See [Using Redshift Renderer](#).

Photorendered images are generated in a separate picture window and cannot be edited.

Important: Photorendered images are not saved with your Project. If you wish to keep them, you have to save them manually.

[PhotoRendering Settings Palette](#)

[Cineware Basic Settings](#)

[Create Rendered Image](#)

[Detailed Cineware Settings](#)

[Using Redshift Renderer](#)

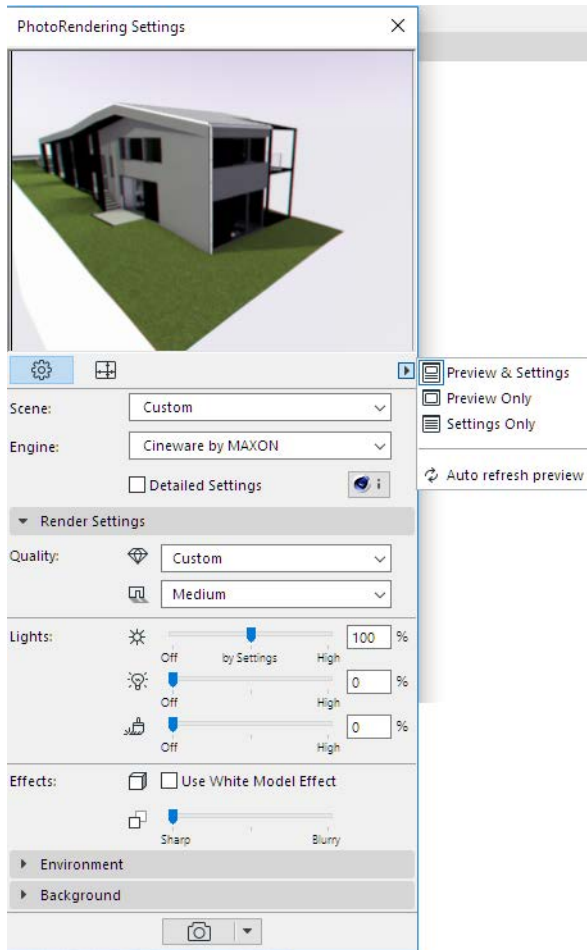
PhotoRendering Settings Palette

The **PhotoRendering Settings** palette is accessible from **Window > Palettes > PhotoRendering Settings**.

By default, the Preview window is at the top, and the settings of the selected Rendering engine are below.

Click the pop-up below the Preview window to show:

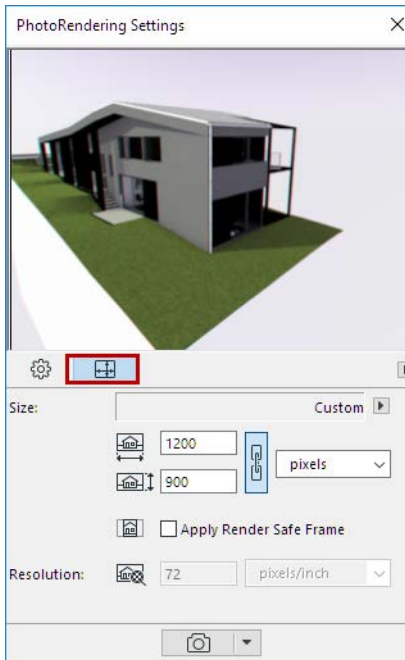
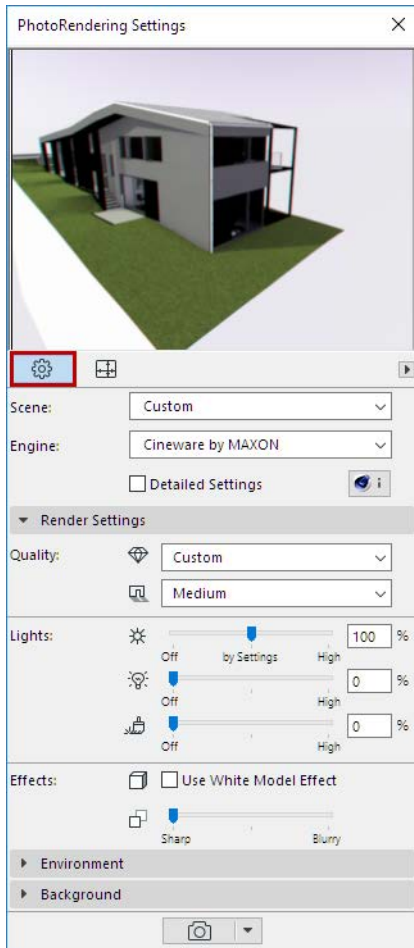
- both Preview and Settings
- Preview only; or Settings only.



In the top part of the Palette, there are two main views: click the left button for **Settings**; click the right button for **Size**.

Note: If you save a Rendering view, you can edit the size it should render, in View Settings.

[See View Settings 3D Only Panel.](#)



- Use the **Settings** view to adjust the parameters of your PhotoRendering Scene; parameters vary depending on the Engine you choose. You can save and manage your scenes in folders, so you can re-use them easily in subsequent renderings and export/import them among different projects.

For Cineware's Basic settings: see [Cineware Basic Settings](#), below.

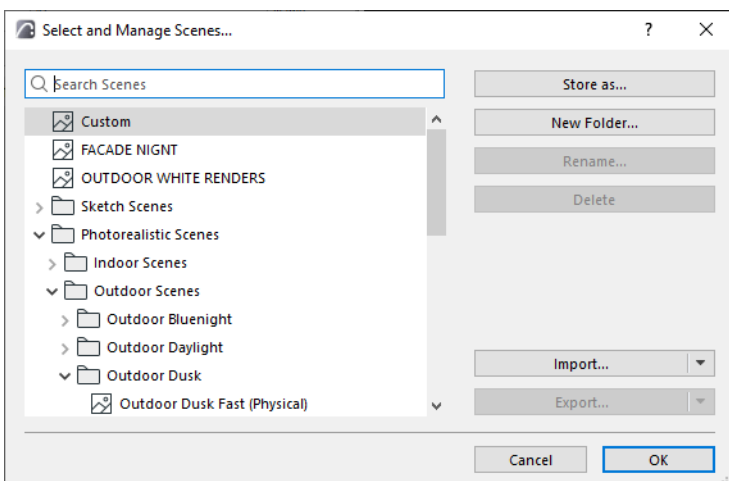
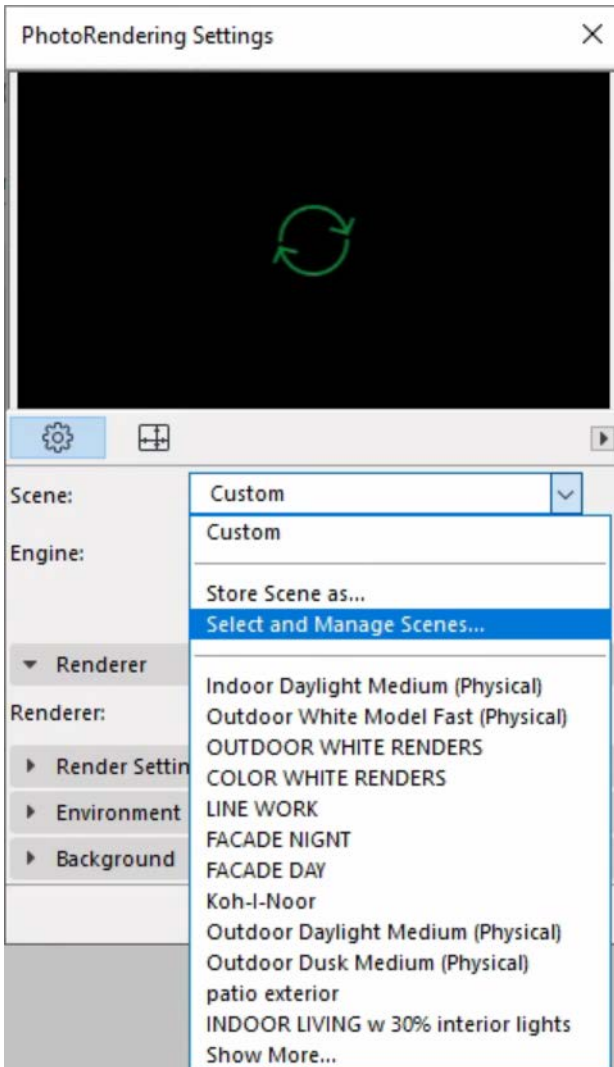
For details on other engine-specific settings: see [PhotoRendering Settings](#).

- The **Size** parameters will not be saved as part of the Rendering Scene. Thus, you will be able to use the scene in a variety of contexts, regardless of size.

Note: for a rendered view, you can also edit Size parameters in View Settings.

Select and Manage Rendering Scenes

A Rendering Scene is a set of predefined Rendering Settings that can be stored and reapplied using the **Select and Manage Scenes** dialog box.



For each Rendering engine, Archicad’s factory template provides one or more predefined Scenes. These are listed in a tree structure in the Select and Manage Scenes dialog box.

Use the Select and Manage dialog box to organize into folders, and to export/import Scene files among different Archicad's. (Scene files have the extension "renderingscene".)

If you have loaded a Scene into the PhotoRendering Settings Palette, and then you change any parameter, the name of the Scene changes to "Custom" until you store it under a new name.

To store the settings of a Rendering Scene, open the Select and Manage Scenes dialog box and click **Store as**.

A stored Scene includes all the parameters on the *Settings view* of the PhotoRendering Settings Palette. Note that this includes the Rendering engine (e.g. Cineware or Sketch). However, a Rendering's output Size is *not* stored as part of the Scene.

Before closing a project, make sure you store any customized Scene settings under a new name if you wish to reapply its settings later. Also, when saving a 3D view, you can save any named Scene as part of its View Settings (in the 3D Only panel of View Settings).

Note: You cannot use a PhotoRendering Scene in any version of Archicad earlier than Archicad 18.

Choose an Engine

Choose an Engine from the Engine pop-up. Archicad comes with the following built-in rendering engines.

- **Cineware Engine:** This renderer is based on Maxon Cinema4D version R21 and incorporates most of its parameters.

Note: Copyright MAXON Computer GmbH. All rights reserved. <http://www.maxon.net>. (Click the Info button to access additional information on this engine.)

- This engine enables you to create very high-quality, highly realistic images, with more exact texture and light reproduction than the other engines.

See [Cineware Basic Settings](#).

See also [Detailed Cineware Settings](#).

- **Basic Rendering Engine:** The Basic Engine (whose settings are analogous to the Hardware acceleration engine) is suitable for producing draft-quality or simple renderings offering textured image surfaces, simple shadows and transparencies.

See [Basic Rendering Engine](#).

- **Sketch:** This produces non-photorealistic renderings that replicate sketched lines - best suited for draft views in the early project stages.

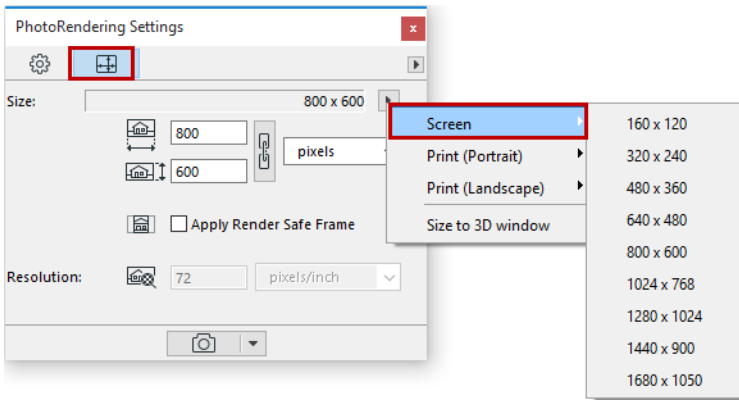
For more information, see [Sketch Rendering Engine](#).

- **Redshift:** The Redshift renderer (GPU based) from Maxon is included in Archicad as a Technology Preview, with limitations. See [Using Redshift Renderer](#).

Set Size

The Size parameters you set up here will not be saved as part of the Rendering Scene. Thus, you will be able to use the scene in a variety of contexts, regardless of size.

Click the pop-up menu to access convenient predefined Size presets (corresponding to screen size in pixels, paper size, or the current 3D window size).



Size Presets available (e.g. Screen, in pixels, or Print, to match paper size).

Click the chain icon to keep these values proportionally constant: if you change one of the values manually, the other value will be updated accordingly.

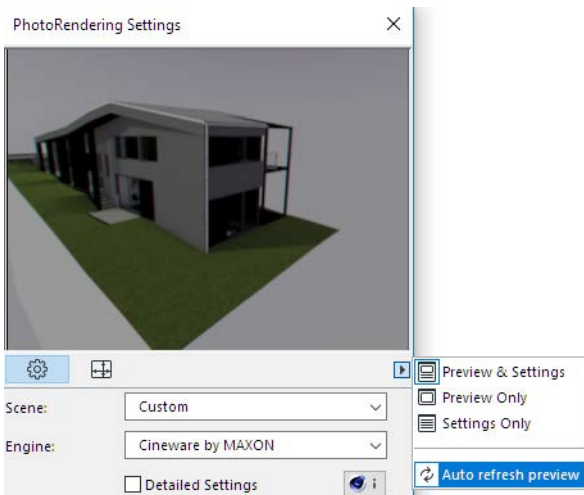
If you choose a Paper size preset, you must also enter a **Resolution** value in dpi. The Size tab also displays the final image resolution.

Preview Picture

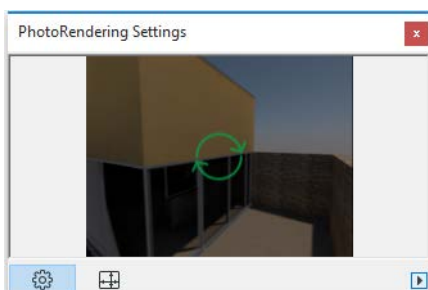
Generation of the Preview picture is optimized for speed. Consequently, it has certain limitations: for example, the 3D grass feature is not shown in the preview (the Grass channel in Surfaces has no effect on the preview.)

The Preview can be refreshed manually or automatically.

- **Automatic refresh:** To turn on automatic Preview refresh, choose this option from the pop-up



- **Manual refresh:** just click the picture. A Preview that requires refreshing is shown with an update icon, and the Preview is grayed out.



Notes on the PhotoRendering Preview:

- The 3D window must be active in order for automatic refresh to occur.
- While the PhotoRendering Palette can be displayed over any Archicad window, the 3D window is always regenerated and brought to the fore every time its Preview is refreshed.
- While rendering with the Sketch Render engine, the preview will not use the actual 3D model. Instead, to speed up the preview feedback, a “dummy” basic model is shown, in which your current rendering settings are reflected.
- To refresh the Preview, Archicad will regenerate only the changes resulting from modifications to the model, surface settings and rendering settings. However, if you close the PhotoRendering Settings Palette, Archicad will have to re-render the entire model the next time you open it. Thus, to save time, you should keep the PhotoRendering Palette open while you work.

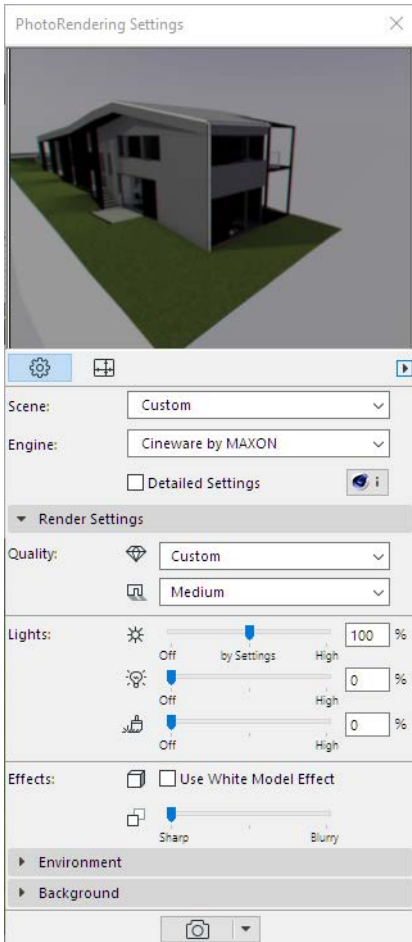
Cineware Basic Settings

The Cineware engine's Basic Settings are enough to create a quality rendering for most purposes. By using the Basic Settings interface, you need only set a few types of parameters.

The following section describes these Basic Settings.

For an overview of the Rendering process, see [Create Rendered Image](#).

By default, the PhotoRendering Settings palette shows the Cineware engine's Basic Settings. (Make sure the "Detailed Settings" box remains unchecked.)



Note: All parameters in Basic view are also available in Detailed view. The settings of these parameters are always identical; it doesn't matter where you set them.

Related Topic

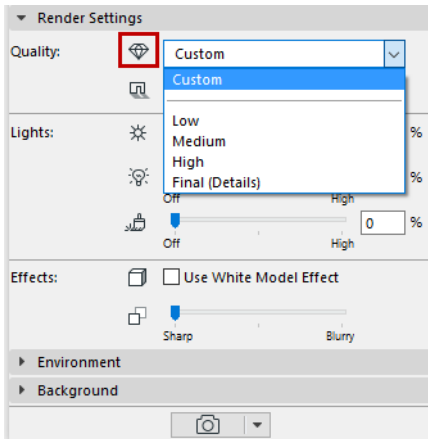
[Detailed Cineware Settings](#)

Render Settings (Basic) Panel

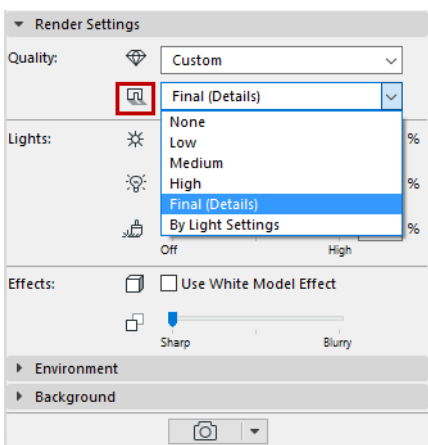
Quality

The two quality pop-ups refer to the Rendering Quality and Shadow Quality of your rendering.

- **Rendering Quality:** The first pop-up sets Rendering quality. Choose Low, Medium, High or Final (Details). (The quality you set here will have an effect on the antialiasing and Global Illumination processes, which produce better quality but require more render time.)



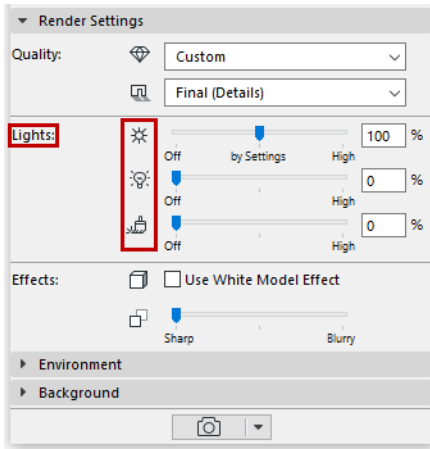
- **Shadows:** The second pop-up sets Shadow quality. (Varies depending on whether you are using a Standard or Physical Renderer.)
 - Choose **None** for no Shadows.
 - **By Light Settings** means that the **Shadow Casting** setting for each individual placed light object (as defined on the Lamp Settings Light Parameters page) will be in effect in this Scene. ([See also Shadow Casting.](#))



Light Sources

Use three sliders to set the intensity of three different types of light used in the rendering: Sun intensity; Lamp intensity; and Illuminating surfaces.

Each slider can override a particular light source intensity that is set elsewhere in the project.



- **Sun Intensity**

- If you are using a **Physical Sky** as your Environment, then this slider affects the sun that is part of that Physical Sky setting.

See [Cineware Basic Settings: Environment Panel](#).

- If you are using an HDRI image as your Environment, then this slider affects the Archicad Sunlight settings.

See [Sunlight Settings](#).

- The midpoint of the slider, at “By Settings”, corresponds to 100 percent: this means that the rendering will use the sun intensity defined in current settings.

- **Lamp Intensity:** Can override the Main Intensity settings (set in Lamp Settings) for all individual lamps placed in the project. The midpoint of the slider, at “By Settings”, corresponds to 100 percent: this means that the rendering will use the lamp intensity exactly corresponding to the intensity set for lamp objects placed in the model.

See [Light Color and Intensity for Lamp Tool Settings](#).

[Watch the Video](#)

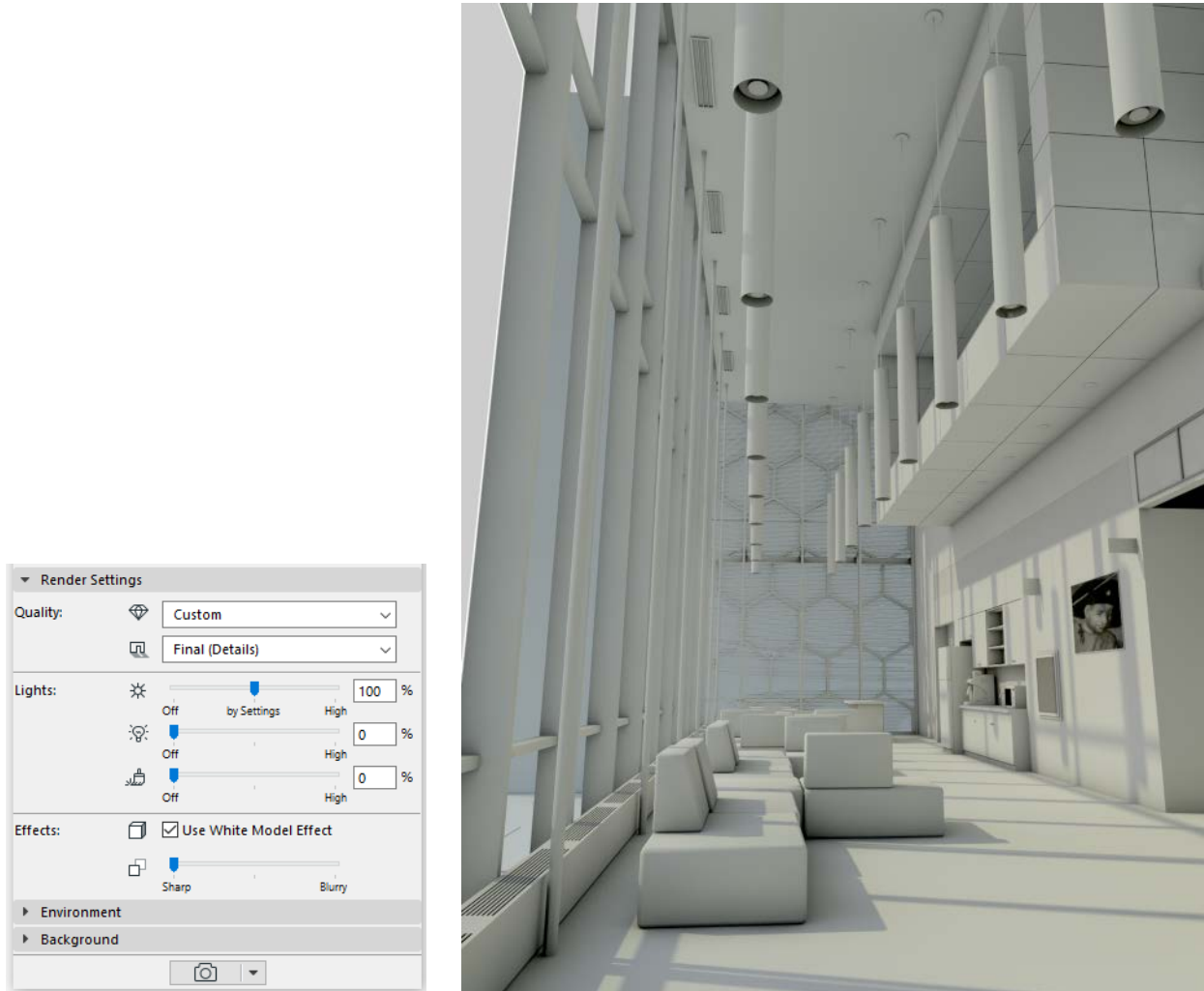
Note: These Rendering settings consider the settings that affect the 3D window, but the reverse is not true: the 3D Window does not consider what you adjust in Rendering settings. So a lamp that is turned off in the 3D window cannot be turned on in Rendering Settings.

- **Illuminating Surfaces:** This setting overrides the surface-level **Luminance** channel.

Note: All surfaces also have an Illumination channel, which affects Global Illumination. But the Illuminating Surfaces control in Photorendering Settings affects the Luminance only.

Only two effects are part of the Basic Settings parameters: the White Model Effect and Depth of Field.

White Model Effect

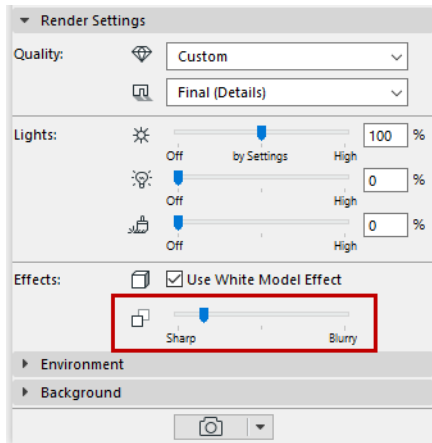


All surfaces are white, except those which use transparency or alpha channels (those remain transparent). This control is useful, for example, when showing an early design phase to a client, to convey the essential forms and space of the project plus light effects, without getting bogged down in colors and textures.

Note that applying this effect involves changing *all* surfaces in the model. Consequently, the entire model must be converted during the preview generation process, which takes time.

Depth of Field

This parameter controls the sharpness of the depth view. (That is, to what extent should it blur over distance.) By default, this is set to sharp.



Limitations:

Keep in mind that if you are using Standard rendering mode, Depth of Field functions as a post-effect: that is, it is only generated after the photorendered image is completed. Consequently, if:

- you are using Standard rendering mode, and
- your project contains alpha-effect objects (e.g. a bitmap tree object that uses an alpha effect), and
- you have set a value for Depth of Field:

then the Depth of Field may not have the desired effect.

To solve this problem, switch to Physical Renderer mode. (In Physical Renderer, Depth of Field works like a camera setting (F-stop). [\(See also *Physical vs. Standard Renderer.*\)](#))

Cineware Basic Settings: Environment Panel

Use this panel to set your rendering into a particular environment: using an option sky backdrop, plus weather, and time settings. Location data is taken from Project Location settings: see [Location Settings](#).

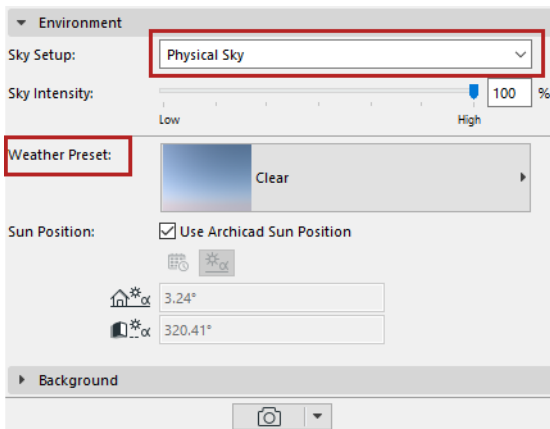
Sky Setup

From this pop-up, choose None, HDRI Sky, or Physical Sky. The most common setting is Physical Sky.

Physical Sky

The Physical Sky is an entire constellation that includes multiple objects having properties (sun, weather, clouds, fog, rainbow, atmosphere). A Physical Sky provides both a background image and lighting effects. It is “smart” and the parameters are interconnected.

With Physical Sky selected, click the Weather Preset pop-up to choose a named set of weather parameters that combine the sky, atmosphere, clouds, fog, rainbow and sunbeams.



Use Archicad Sun Position

Check this box to reset the location according to the Archicad Sun Position. Use this to fine-tune the weather preset.

Set Sun Position manually

Uncheck the Archicad Sun position box.

Choose an option to set sun position:

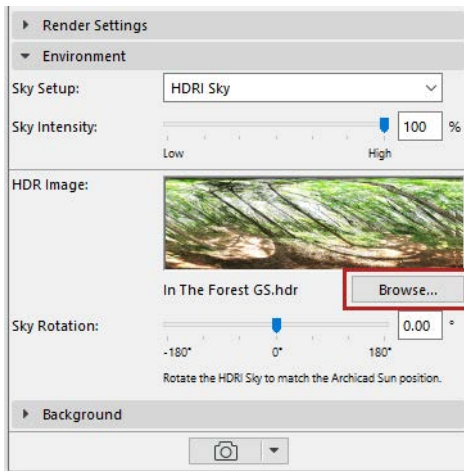
- by date/time
- by entering azimuth/altitude data

[See also Note on Sun Settings in Archicad.](#)

HDRI Sky

Click **Browse** to select an image which provide a background picture and ambient light source for the rendering.

- Use a predefined image from the library ([Himg] HDF Images folder), or create your own. (If you create your own, you must use a spherical view, otherwise it will be distorted.)



- An HDRI image provides diffuse light and blurred shadows (no sharp shadows)
- The HDRI Sky Setup does not include a separate Sun. To use the effects of the Archicad sun, make sure that the Sun slider on the Basic Settings panel is turned on. If you increase the Sun slider value, this will also increase shadow sharpness.
- Use the Sky Rotation slider to rotate the HDRI Sky image so that it matches with the Archicad sun position of the 3D model. Check the rotation in the preview.

Note: The intensity of sunlight is determined in the Sunlight Settings dialog box, but that setting can be overridden (or turned off) in the Light Sources section of the Render settings (Basic) panel. [See Light Sources.](#)

Note on Sun Settings in Archicad

- Archicad **Sun Position** is set in the 3D Projection Settings dialog boxes.

[See Set Sun and Camera Positions Graphically.](#)

- The Archicad Sun's intensity is set in **Sunlight Settings** (available from the 3D Projection Settings dialogs as well as Camera Settings).

[See Sunlight Settings.](#)

Using a More Sophisticated Sun with HDRI

1. Turn off the Archicad Sun (In Light Sources, move the slider to Off).
2. Using the Lamp tool, place a Sun Object into the model. The Sun Object can be positioned manually, has more parameters than the Archicad Sun and takes better advantage of Cineware capabilities.

Note that the PhotoRendering Palette's Environment settings are not reflected in the 3D model, so in order to gauge the effect of the Sun Object, you must watch its effect in the Rendering Palette Preview, while moving it in the 3D window.

Cineware Basic Settings: Background Panel

This panel of PhotoRendering Settings is identical for all PhotoRendering engines.

A Background is a plain 2D picture that is placed “behind” the rendering. (You can use it to place the 3D model into a context.)

Note: In contrast, the Environment (available with Cineware) is a genuine 3D environment which covers up the background. Thus, if you are using an Environment with Cineware, the Background has no purpose.

Use the Background controls to adjust parameters of your rendering’s background image.

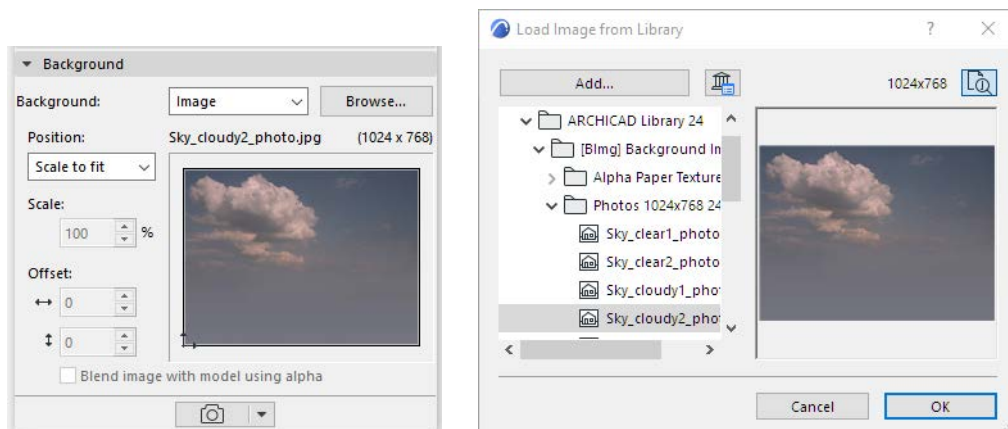
- **None:** This option will give you a non-transparent black-and-white background if you are using the Basic or Sketch Engine.

With the Cineware Engine, this setting gives you a transparent background. The same effect can be achieved with Cineware Detailed Settings: In the General Options section, enable Generate Alpha Channel.

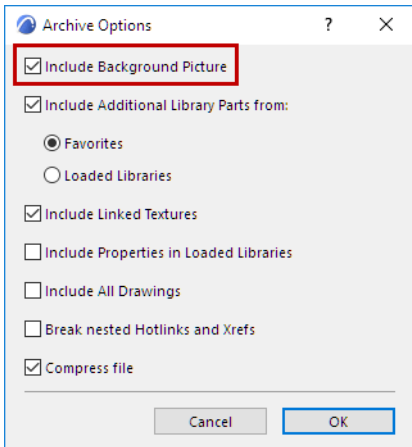
- **Colors:** Set the color of the sky and the ground by clicking the respective color-picker fields. During the photorendering process, Archicad will automatically locate the horizon line of the background to correspond with the horizon of the picture.

In parallel projections, if the bottom of the elements is visible, sky color is used; if not, ground color is used. The 3D Window can also use the colors specified here. If you just need a single-color background, click the **Chain** icon and set either the Sky Color or the Ground Color.

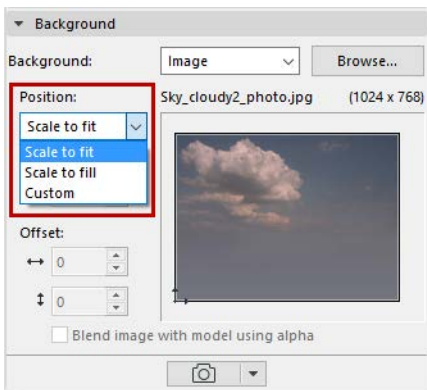
- **Image:** Use this option to browse a background image from the library.



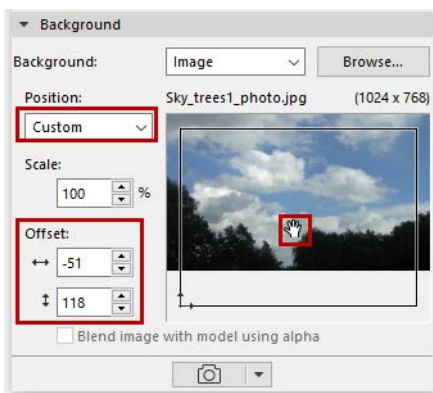
Note: You can save the background picture together with your Project. Use the **Save As** command to save the project in archive format, then check the **Include Background Picture** checkbox in the dialog box.



Position: Use this pop-up to fit the image to the 3D window in several ways:



- **Scale to Fit:** Rescales the image so it fits within the 3D window.
- **Scale to Fill:** Rescales the image so that it fills out the full 3D window.
- **Custom:** The control switches to Custom whenever you manually reposition the image to the frame: either by clicking and dragging the preview, or by filling out new values in the Offset fields.

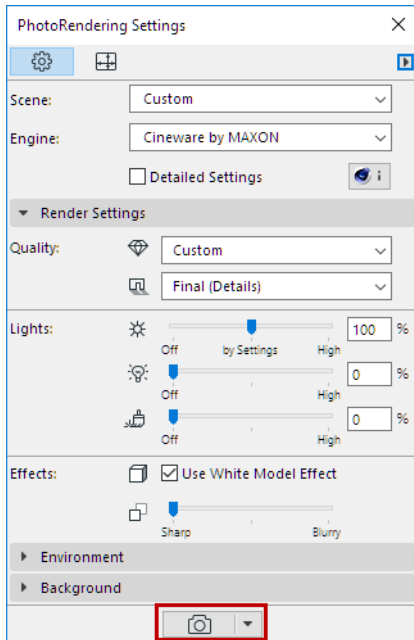


Blend Image with model using alpha: This checkbox affects renderings done with the Sketch rendering engine; it produces a blending effect. (An alpha image used as a Background with Cineware will also have this effect.)

Create Rendered Image

To create a rendered image of the 3D window contents, subject to render size settings:

- Click the render icon at the bottom of the PhotoRendering Settings Palette, or
- Use **Document > Creative Imaging > PhotoRender Projection**



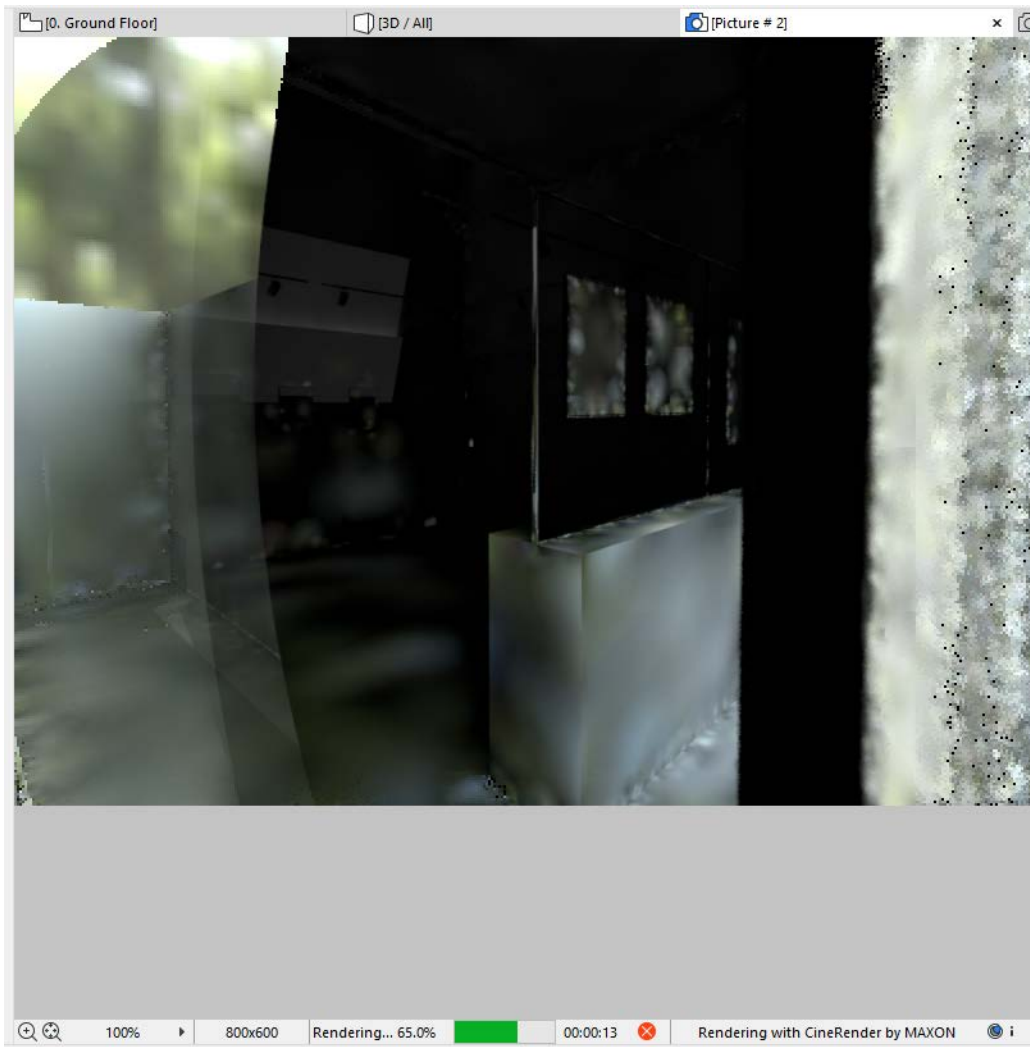
- The picture window appears, with a progress bar and Stop button, feedback on elapsed rendering time, plus the familiar Zoom and Fit in Window window controls.
- In Tab Bar, Picture window tab shows feedback on background rendering progress (Cineware only). A checkmark indicates that the render is complete (but not yet opened).



Rendering in progress



Rendering complete, not yet viewed



- If needed, click Stop, or just close the window to stop the render.

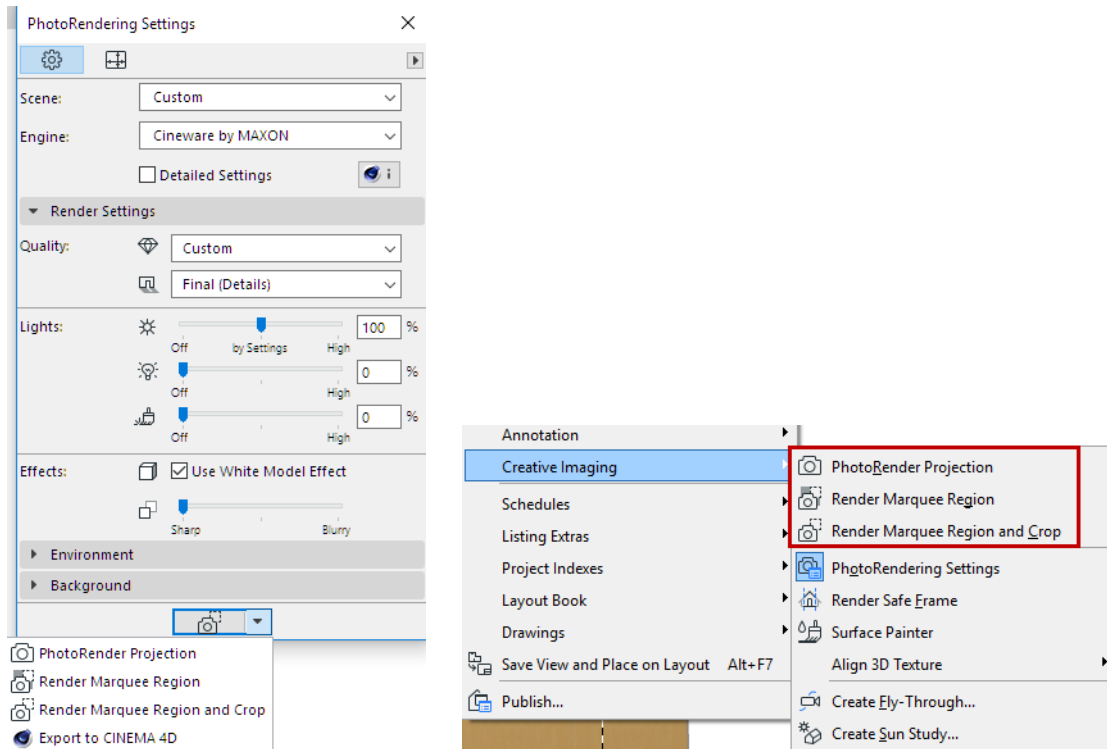
Note: Rendering time varies depending on many factors, including size and complexity of the rendering parameters, and hardware capabilities.

Render Marquee Options

By placing a Marquee in the 3D window, you have additional ways to limit the rendered model.

Note: You must use the 2D Marquee option. It will always assume a rectangular shape (even if you draw a custom-shaped Marquee).

Once you have placed the Marquee, two Render Region commands are available (in the **Document > Creative Imaging** menu and as pop-up options of the Render button of the Render palette):



- **Render Marquee Region and Crop:** With a Marquee placed, this is the default rendering command. This renders and outputs only the marquee region.
- **Render Marquee Region:** Use this function if you only want to render a small part of the 3D window. It will render the scene as a whole, but only the marquee region is rendered in full; the rest of the scene is shown as transparent.

This is useful, for example, if you have already fully rendered the scene, but then change or correct one small part of it, and only need to re-render that part.

For either option, the Preview shows only the marquee region.

Note: If the Render Safe Frame mode is on (see below), then your Marquee will only have an effect to the extent that it falls within the Render Safe Frame. If Render Safe Frame is off, then you will render the Marquee area regardless.

Render Safe Frame

Turn on this 3D view feature at **Document > Creative Imaging > Render Safe Frame**.

This is an Archicad mode that affects the 3D window; it displays a frame that indicates which part of the model will be rendered.

This might not be obvious, because the output is affected by the Size setting in Render Settings. Depending on the Size settings, your model could be cut off in the render.

[See Set Size.](#)

The Render Safe Frame gives you feedback on this and a chance to adjust the Size parameters.

Note: The Render Safe Frame setting to OFF corresponds to Rendering Settings from Archicad versions 17 and older. In this situation, the full content of the 3D window - even content that is not visible on screen - will be rendered, with its size determined by Render Size.

This setting (whether Render Safe Frame is applied or not) is noted in the 3D Only panel of View Settings and on the 3D Visualization toolbar.



Phases of Rendering with Cineware

Although the Cineware engine is built in to Archicad, it is an add-on that is a separate program component. The rendering process involves data exchange between the Archicad model and the engine, in the following steps:

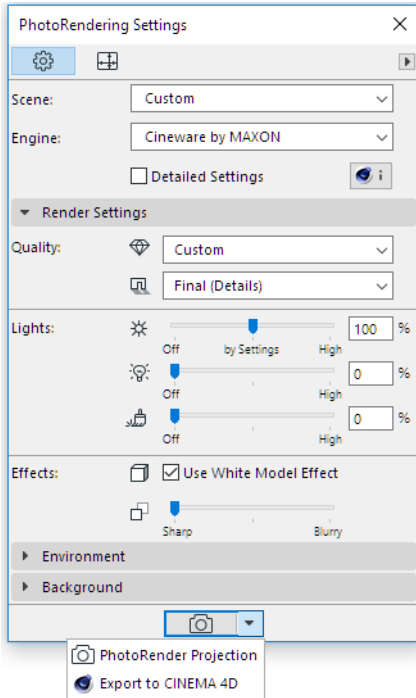
1. Converts the 3D model. This occurs in Archicad; this goes quickly, but during this time you cannot continue working in Archicad. During this interval, you will see the “Preparing” tag in the top right corner of your screen in a separate Progress Window, and also and in the rendering window.
2. Model is sent to the Cineware engine. This takes only a few seconds. To optimize the process, only those changes are sent which occurred since the last render.
3. Rendering calculations are underway. During this time, you can keep working in Archicad.

Meanwhile:

- Global Illumination (if applied) is calculated. You can recognize this phase by the sampling on the picture, while the lighting of the scene is calculated.
 - The rendering itself takes place, recognizable by the squares revealing the image from the inside out, in a circular manner. Note that the number of squares corresponds to the number of cores in your computer (more cores result in faster renderings)
 - Finally, post effects (if applied) are calculated.
4. The Render picture is sent back - this is an ongoing process whose progress you can see in the picture window.

Export to Cinema 4D

The Cineware engine in Archicad does not support multi-pass or net render functions. If you want to continue working on your model with additional Cinema4D functions, use the **Export to Cinema 4D** command from the Render pop-up button of the PhotoRendering palette. This will export your model, surfaces and rendering settings.



However, to improve the post-processing of images, it is possible to save model pictures with a color depth of 32-bit per channel in certain image file formats (e.g. TIF).

Note: This color-depth feature is not available if the model picture is created using the Render Region & Crop command, which creates the image using only 8 bits per channel.

Limitation on Axonometric 3D Views Handled by the Cineware Engine

The following axonometric views in Archicad are not supported by the Cineware engine:

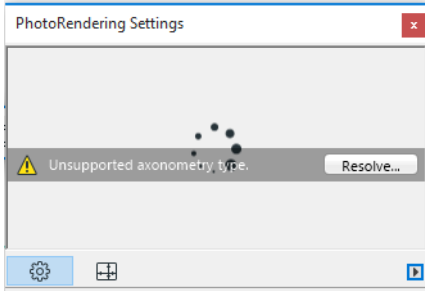
- monometric axonometry
- custom axonometry (if the Z-axis is not vertical, or if the x-y axes have been distorted)

[See *Parallel Projection Settings*.](#)

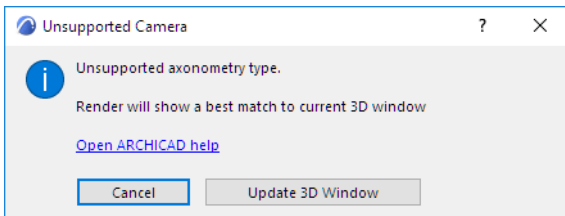
If you are working with such a 3D view in the Photorendering environment, you will be alerted to the problem and options for resolving it.

In the Preview window:

The preview window will display a warning notice:



Click **Resolve** to bring up further information:



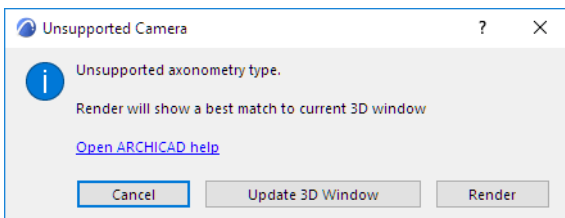
The preview will be generated using a “best match,” that is, the view axes will be modified as needed to achieve an axonometric view that the Cineware engine can handle.

You have two options here:

- Click **Cancel** to proceed - the Preview and the 3D view will not be totally identical, because the Preview will show a “best match”.
- Click **Update 3D Window** to modify the 3D view to match the view in the Photorendering Preview.

During the final PhotoRender

The following dialog box appears:



Choose one of the following options:

- Click **Render** to continue the rendering process. Cineware will create a rendering using a “best match,” that is, the view axes will be modified as needed to achieve an axonometric view that the Cineware engine can handle. Thus, the final rendering will correspond to the Preview, but the 3D window remains unchanged.
- Click **Update 3D Window** to modify the 3D view to match the view in the Photorendering Preview. (If you are not satisfied with this modified view, you can adjust it as needed.) The rendering will not take place until you issue the PhotoRender Projection command again.
- Click **Cancel** to end the process without rendering.

Visualization Extras

You can enhance your visualization output with dynamic content such as fly-throughs or virtual reality. This section is devoted to these “extras.”

[Fly-Through](#)

[Sun Study](#)

[Align View](#)

Fly-Through

Use the Fly-Through feature to create a movie of the project.

Access this function from **Document > Creative Imaging > Create Fly-Through**.

The **Create Fly-Through** command will only be active if you have at least one Parallel Projection pre-set, or if a Camera path with at least two cameras is active on the Floor Plan.

The Camera path is created by the Camera Tool. Placed cameras along this path are defined as **key frames**, and the number of images between each camera is set by the **in-between frames** (defined at Camera Path Options). Only one camera path will be active at a time (cameras and their markers visible on the Floor Plan). Use Camera Tool to select the active Camera path.

The Parallel Projection Pre-sets are created from the **3D Projection Settings > Parallel Projection Settings > Pre-Sets**: these are defined as **key frames**, and the number of images between each Pre-set is set by the **in-between frames**.

The frame can be saved from either the 3D window or PhotoRendered, as an animated movie file or as a sequence of still images. If you save the Fly-Through as an external movie file, you can view the animation with most common movie viewers on any platform. Alternatively, immediately view the animation on screen.

Note: A large number of frames means a smoother movie but also a much larger file.

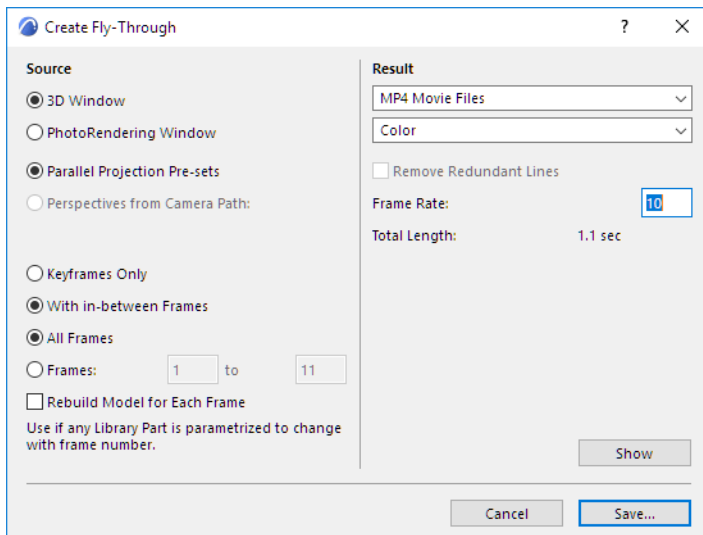
Related Topics

[Camera Tool Settings](#)

[Sun Study](#)

Create Fly-Through

The Fly-Through dialog box allows you to set the source and results of the Fly-Through.



Source

- Click **3D Window** to use the 3D Window as the image source and preview window.

Note: Choose a **3D Style** and set the current 3D window options before creating the Fly-Through.

- Click **PhotoRendering Window** to use the PhotoRendering Window as the image source and preview window.

Note: Choose a PhotoRendering Scene to generate the photorealistic images before creating the Fly-Through.

Define the route of the Fly-Through:

- Click **Parallel Projections Pre-sets** to create animated transitions between parallel projections.

Note: Set up at least two pre-set projections. Otherwise, this option is disabled.

- Click **Perspectives from Camera Path** to create a fly-through animation along the active Camera Path. The name of the current Path is shown below.

Note: This option is disabled if fewer than two cameras are defined.

Define the content of the Fly-Through:

- Click **Keyframes Only** to image key frames without any in-between frames.
- Click **With in-between Frames** to image both key frames and in-between frames.
- Click **All Frames** to image all animation frames in the current Path.
- Click **Frames** to specify a range of animation frames (first and last) for imaging from the current Path.

Note: The numbers shown by default are the first and last frames in the entire animation.

Rebuild Model for Each Frame: If you check this option, Archicad rebuilds the project after each frame and creates a new view from the rebuilt model.

Note: This option lets you create additional animation effects, provided that your project contains any Library Parts whose appearance changes with every frame. This can be achieved if the Library Part parameter is set to change along with the frame number.

Result

Specify the format of the Fly-Through.

- Select a **file format**, such as a single movie file or set of images.
- Specify the **color resolution**

Note: Higher color depths result in increased file size.

Remove Redundant Lines: Check this box to remove overlapping lines from the saved image(s).

This option is available if:

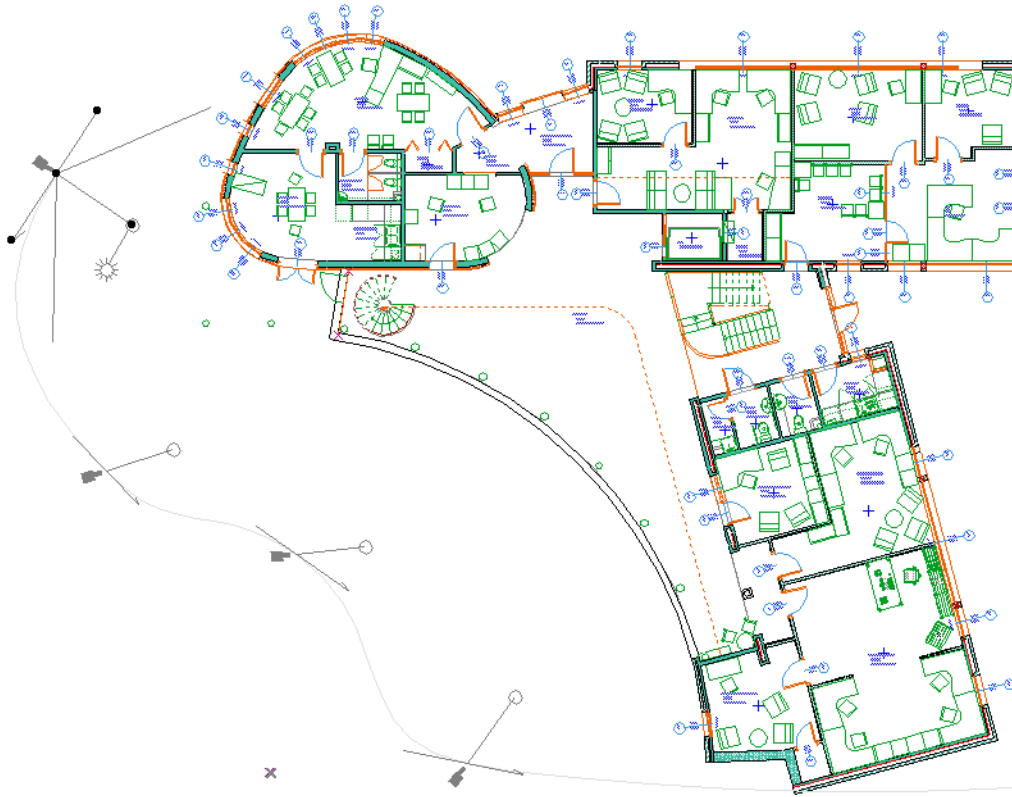
- the 3D Engine is the Vectorial Engine (not Hardware acceleration)
- you are exporting particular Frames (not All Frames)
- the chosen file format is either PMK or Archicad 2D Lines.

Frame Rate: Enter the frame rate per second value here.

Total Length: Length of the animation is displayed in seconds, which divides the total number of frames with the frame rate, as an indication of how long the saved animation will be.

Show: Click to process the Fly-Through and preview the images without saving them. You can save the resulting animation afterwards by reopening the dialog and clicking Save.

Hint: To preview only a short sequence of frames, adjust the **Frames** range.



Note: Collections of 3D parallel projections can also be linked to create a Fly-Through, but these are handled differently from perspectives. Only a single set of parallel projections can exist, since the parallel projection viewpoint distance is infinite - they cannot be displayed on the Floor Plan.

Sun Study

To use this feature, use **Document > Creative Imaging > Create Sun Study**.

Sun Studies allow you to view an entire project or a selected part of it in a real-life situation by choosing a geographic location and defining the time period over which you wish to examine lighting and shadow casting. What you actually see is a series of simple or photorendered 3D images defined by the current **3D Projection Settings**, the current **3D Style** or **PhotoRendering Settings**, and the **Sun Position** settings.

The geographic location of your project, including Project North, is defined at **Options > Project Preferences > Location Settings**.

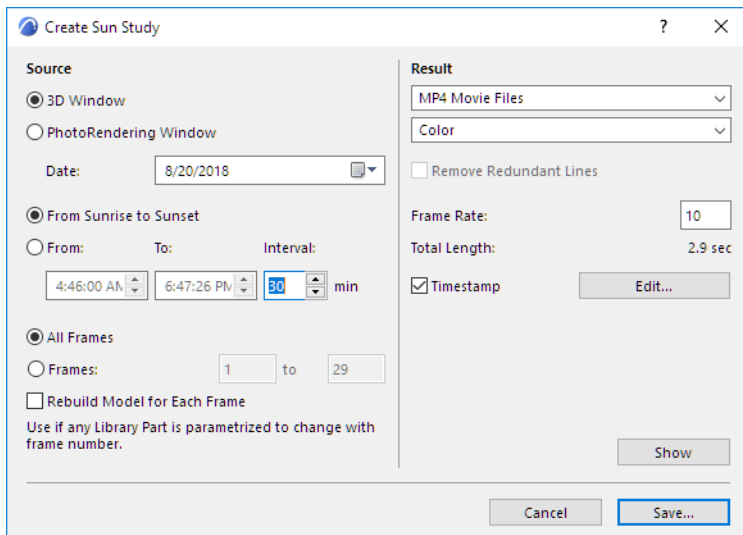
See [Location Settings](#).

Note: Based on these data, Archicad uses the Solar Position Algorithm from the National Renewable Energy Laboratory (USA) to calculate the solar zenith and azimuth angles.

For information on this algorithm, see: [Solar Position Algorithm](#).

You can either have the resulting animation shown immediately on the screen, or save these images in the desired file format and show them in the future using a movie viewer on any platform.

Create Sun Study



All the projections in a Sun Study will be generated in accordance with the current parallel projection or perspective view, including Sun Position, defined in the **3D Projection Settings** dialog box.

See [3D Projection Settings](#).

Source

- Click **3D Window** to use the 3D Window as the image source and preview window.
Note: Choose a 3D Style and set the current 3D window options before creating a Sun Study.
- Click **PhotoRendering Window** to use the PhotoRendering Window as the image source and preview window.
Note: Choose a PhotoRendering Scene to generate the photorealistic images, before creating the Sun Study.

Date

- Choose the month and day to begin the Sun Study.
- Choose the length of your sun study: either **From Sunrise to Sunset**, or enter a specific time span, in hours and minutes.
- **Interval:** Set the interval, in minutes, for taking the Sun Study images during the time range set above.
- Click **All Frames** to image all animation frames in the current Path.
- Click **Frames** to specify a range of animation frames (first and last) for imaging from the current Path.

Note: The numbers shown by default are the first and last frames in the entire animation.

Rebuild Model for Each Frame: If you check this option, Archicad rebuilds the project after each frame and creates a new view from the rebuilt model.

Note: This option lets you create additional animation effects, provided that your project contains any Library Parts whose appearance changes with every frame. This can be achieved if the Library Part parameter is set to change along with the frame number.

Result

Specify the format of the Sun Study.

- Select a **file format**, such as a single movie file or set of images.
- Specify the **color resolution**.

Note: Higher color depths result in increased file size.

Remove Redundant Lines

Check this box to remove overlapping lines from the saved image(s).

This option is available if:

- the 3D Engine is the Vectorial Engine (not Hardware acceleration)
- you are exporting particular Frames (not All Frames)
- the chosen file format is either PMK or Archicad 2D Lines.

Frame rate: Enter a frame rate per second value here.

Total Length: Length of the animation is displayed in seconds, which divides the total number of frames with the frame rate, as an indication of how long the saved animation will be.

Timestamp

- Check to add this information to the Sun Study (available if saving to a movie format)
- Click **Edit** to specify the content, appearance and location of the Timestamp.

[Watch the Video](#)

Show

Click **Show** to process the Sun Study and preview the images without saving them. You can save the resulting animation afterwards by reopening the dialog and clicking Save.

Hint: To preview only a short sequence of frames, adjust the **Frames** range.

The pictures below show the same part of a building at different times of day, including sunrise, one hour later, two hours later and in the afternoon.



Align View

When using a scanned photograph as the background of your rendering, you may want to fit your Archicad model into the photograph's environment. In the photorendering made from the Camera placed by the **Document > Creative Imaging > Align View** command, the perspective of your building in Archicad will match the perspective of the background.

For more information, see [PhotoRendering](#).

For this feature, you need to select two clearly visible vertical segments on the photograph (e.g., the sides of a door or a window, two vertical edges of a wall, or two flagpoles), and connect them with their exact position in the virtual model or its environment. This is done by connecting each of four points on the placed photograph (the tops and bottoms of the two vertical segments) with their position on the Floor Plan, and then entering their Z coordinates in the **Align View dialog box**.

Note: This function is mainly useful in refurbishing, when a photograph of the existing environment is available.

Align View: Interior Example

Using the **Figure** tool, place the scanned image on the Floor Plan at any size or resolution.

For more information, see [Figures](#).

You then have two possibilities for aligning the camera view to the photograph: you can either use existing lines or can manually draw lines between the corresponding points of the Floor Plan and the photograph.

Draw Connection Lines Automatically

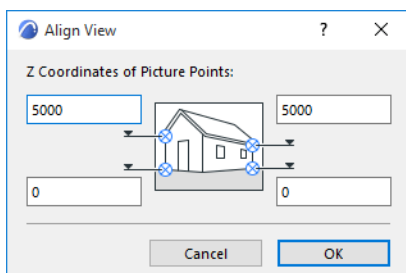
1. Select the placed photo.
2. Choose the **Document > Creative Imaging > Align View** command. Archicad will prompt you to make six clicks on the Floor Plan with the Pencil cursor, identifying the corresponding points between the picture and the plan. The connection lines are drawn automatically.

The Align View dialog box appears, prompting you to enter the height values for the four points.

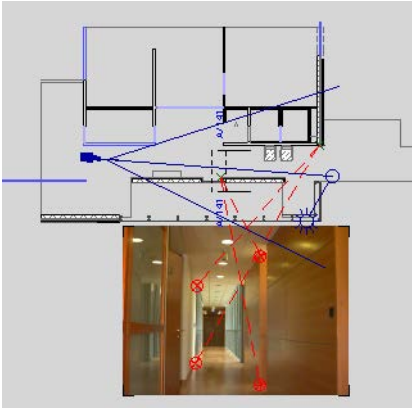
3. Click **OK**. The camera will be placed on the Floor Plan on a new path.

Draw Connection Lines Manually

1. Using the **Line** tool, connect the tops and bottoms of two vertical lines visible on the photograph with their location on the Floor Plan (e.g., the sides of a door or a window, two vertical edges of a wall, or two flagpoles). This will give you four lines, each extending from a floor plan location to a point on the photograph, with the two line pairs connecting where they meet on the Floor Plan.
2. Select the photo and the four lines.
3. Choose the **Align View** command. The dialog box appears, prompting you to enter the height values for the four points.



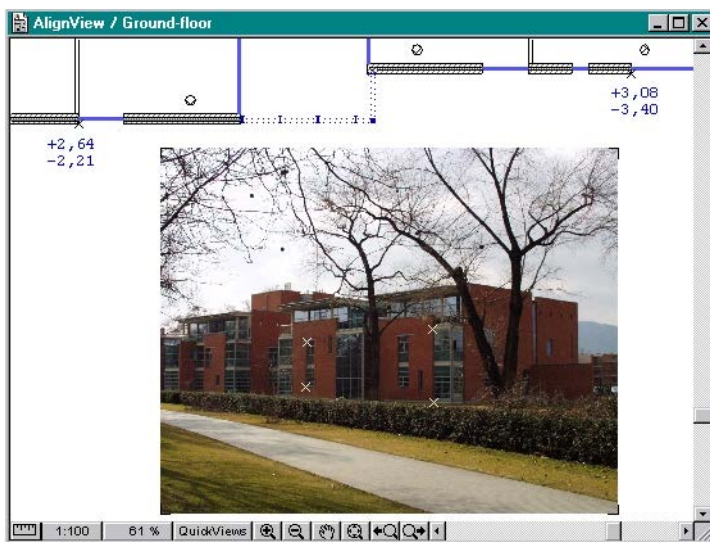
4. Click OK to close the Align View dialog box. The camera will be placed on the Floor Plan, on a new Path.



Note: For greater precision, you can enter the horizon of the photograph by simply rotating the picture on the Floor Plan so that the horizon is exactly parallel to the unskewed x-axis (otherwise the command will calculate the horizon for you). The precision of the result depends largely on that of the entered points, but even very careful preparation may result in a Camera that is slightly off position. In this case, set the photograph as the background of your 3D window, and fine-tune the Camera's position in Wireframe or Hidden Line mode using Archicad's 3D navigation tools.

Align View: Exterior Example

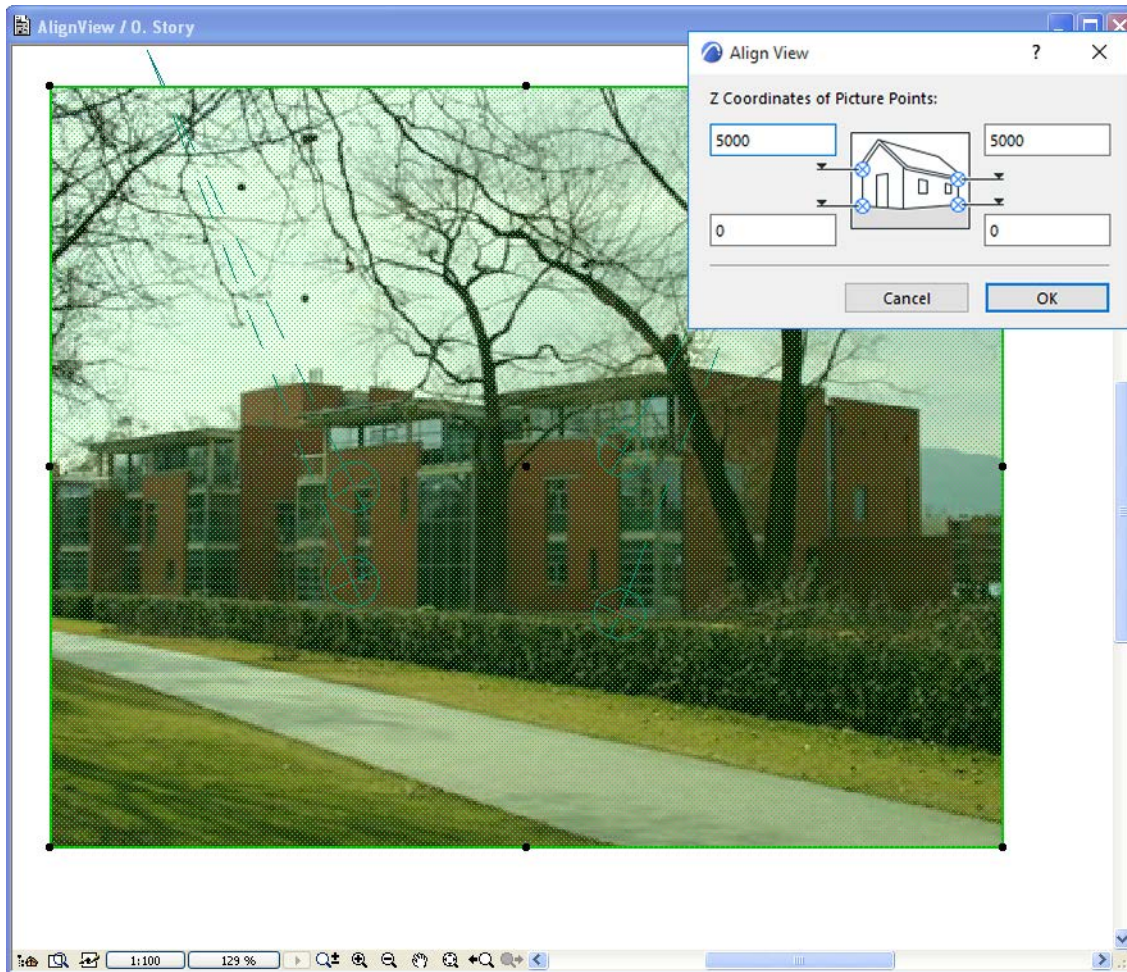
Let's say you have a photograph of a building and its plan as an Archicad file. Place the original photo on the Floor Plan of the building. If the picture is placed without any rotation then the horizon is calculated automatically as horizontal. You can, of course, override this horizon calculation manually, and enter the horizon angle yourself; this is done by rotating the picture on the floor plan. When the picture is rotated with an angle other than 90° (or 180° or 270°) on the floor plan, then Align View calculates the horizon by itself, otherwise the horizon is assumed to be horizontal. You need to know the height values of two vertical lines (four points in all) on the building and their locations on the photo.



Two black hotspots on the Floor Plan and the heights of two distinct points of the building below each are shown on the illustration. Zoom in on the photo and place four hotspots at the locations of the four points with known height values.

Select the photo and choose **Document > Creative Imaging > Align View**.

To link the hotspots, click first on the point on the Floor Plan that corresponds to hotspots on the photo. (In the above example, the side of the window on the left where you see the black hotspot with two heights, +2.64 and -2.21.) Now click the pair of hotspots on the photo, one after the other. Repeat this operation with the other point on the plan and the corresponding pair of hotspots. It does not matter which point on the Floor Plan you start with. In addition, after marking a point on the Floor Plan, you can select the two corresponding points on the photo in either order.



When correctly done, the result will look as shown above, and the Align View dialog will appear.

Type the height of the hotspots in the appropriate fields in the dialog box. The upper points are located along the same vertical line as the lower points. In this example, these will be 2.64 and -2.21 meters (left upper and lower) and 3.08 and -3.40 meters (right upper and lower). After you click OK, a new camera will be placed on the Floor Plan.

Select the original scanned photo to be the background picture in the **PhotoRendering Settings** dialog box. Make sure the width-height ratio of the resulting PhotoRendering is the same as that of the original photo. You can also modify the size of the photo to suit the size of your PhotoRendering window.

Note: You cannot crop the photo. If you must, crop it in a way that the original proportions of the photo stay the same (i.e. cover a part of the photo with a solid white fill), because Align View needs the original proportions of the picture to locate the original target point of the photo. The view will not be aligned precisely if the proportions of the photo have been changed.

For example, if you have a scanned picture of 1200 x 900 and you want a 3D window or PhotoRendering window image of 600 x 450, you would reduce the size of the background photo to 50 percent in the PhotoRendering Settings dialog box. (You can choose this same background in the **View > 3D View Options > 3D Styles** dialog box: in the Background panel, check “As in PhotoRendering”.) By selecting the created camera and generating a 3D View or a photorendering, you will see the result - the building is accurately placed in the existing environment.

Graphisoft BIMx

Graphisoft BIMx lets you create an interactive walk-through model out of your Archicad project. The functions needed to create and share a BIMx model are included in Archicad.

The BIMx model is known as a “Hyper-model” (file format .bimx). The BIMx model can contain the full documentation of your Archicad project: the 3D model, plus views, layouts, project and element-level BIM data, and camera paths.

BIMx apps

Explore the BIMx Hyper-model on any of the following:

- **BIMx Mobile** (iOS, iPadOS, or Android). Free from [Apple App Store](#) and [Google Play](#).
- **BIMx Desktop** (Windows and Mac). Free from [Apple App Store](#) and [Google Play](#).
- **BIMx Web**. Open Hyper-models shared from BIMx Model Transfer or uploaded to BIMcloud.

Extra collaboration features are available with **BIMx Pro**, included in certain subscriptions - see details at: [BIMx Licensing](#).

Share BIMx model

- to the BIMx Model Transfer site at bimx.graphisoft.com

Related Topics:

[Publish BIMx Hyper-Model from Archicad](#)

[Republish Hyper-model](#)

[Architect Credits in BIMx](#)

[BIMx: Notes on Model Data, 3D Model Settings, and Textures](#)

Read More:

[Graphisoft BIMx page](#)

Publish BIMx Hyper-Model from Archicad

The BIMx model is known as a “Hyper-model” (file format .bimx). The BIMx model can contain the full documentation of your Archicad project: the 3D model, plus any views, layouts and camera path.

To create a Hyper-model in Archicad, you must either use the Wizard or set up a Publisher Set manually in Archicad.

There are two ways to publish a Hyper-model:

[Publish BIMx Hyper-Model Using a Wizard](#)

[Publish BIMx Hyper-Model: Publisher method \(detailed method\)](#)

Publish BIMx Hyper-Model: Publisher method (detailed method)

Create and publish a Hyper-model in Archicad, using the Publisher. This method is recommended (as opposed to the quick Wizard method) if you are familiar with Publisher Sets, because it provides more precise, extensive export options:

- Choose specific items from the Publisher Set to be published in the Hyper-model
- Save multiple 3D views, each with different settings (model background, texture limitation, Sun settings)
- Control which element information should be part of the Hyper-model

You may need to optimize the Archicad model before saving it in BIMx format to ensure the best possible performance.

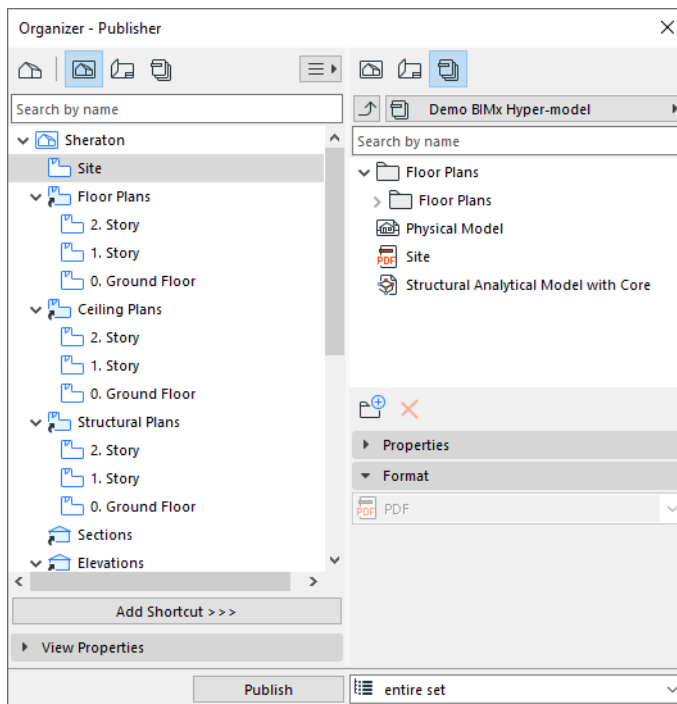
[See *BIMx: Notes on Model Data, 3D Model Settings, and Textures*.](#)

Step 1: Create Publisher Set

Use the Organizer mode of the Publisher to create the desired Publisher Set.

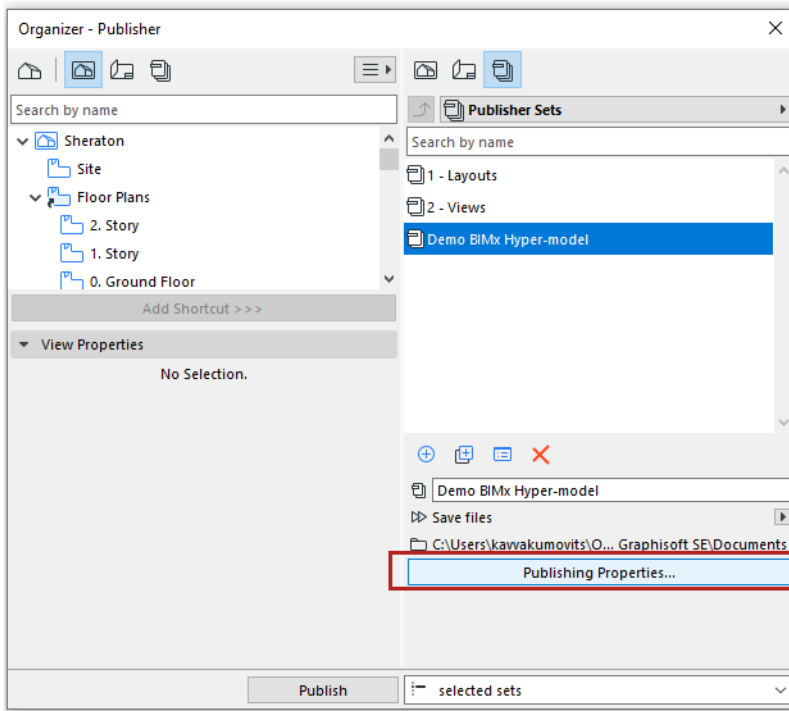
[See also *Publisher*.](#)

- Assemble the combination of views (including a camera path if needed), Layouts or subsets, and any number of 3D views to include in the Hyper-model
- Optimize the content of each 3D view - for example, using Partial Structure Display, Layer Combinations or Marquee to limit the 3D views



Step 2: Set Publishing Properties

Open the Publishing Properties dialog box.



Choose how you would like to publish the Hyper-model: either **Upload** or **Save**.

Upload BIMx Hyper-model

Choose this publishing method to upload the file to the BIMx Model Transfer site at bimx.graphisoft.com.

The screenshot shows the 'Publisher Set Properties' dialog box with the following settings:

- Publishing method:** Upload BIMx Hyper-model
- User profile:** Name: Upload BIMx Hyper-model, Company: GRAPHISOFT-DEV
- Upload to:** GRAPHISOFT-DEV
- To this folder:** Home
- Private Quota:** 49 GB used of 505 GB total
- Options:** Private (selected), Password Protected (unchecked)
- Project details:** Name: Sheraton - Demo BIMx Hyper-model
- Save local copy:** Unchecked
- Info Set:** Single Selection
- Architect Credits:** Use Architect Credits in BIMx Hyper-models (unchecked)

- **Sign in** to your Graphisoft account, if you are not signed in already.
If you don't have an account, please **Sign up** from the appearing dialog.
- **Upload to:** Choose company's or your own storage location.
- **To this folder:** Choose whether to upload to a Public or a Private Folder. If you choose Private Folder, you can optionally password-protect the file. Available storage space is shown at left.
Note: The Private Folder option is available only in your company storage location, to users who have a Software Service Agreement (SSA), or a subscription, or a Forward agreement for Archicad.
- **Name:** Optionally rename the Hyper-model file.
- **Save local copy:** Check this box and enter a destination file path if you wish to save a local copy of the Hyper-model in addition to uploading it to the BIMx Model Transfer site.

See also [Step 3: Info Set](#) and [Step 4: Use Architect Credits](#) (below) for information on these options.

Save BIMx Hyper-model

Publisher Set Properties

Publishing method: Save files

Create single file
 Create a real folder structure
 Create flat file structure

File Details

Name: Sheraton - Demo BIMx Hyper-model

Path: C:\Users' Browse...

Info Set: Single Selection

Format: BIMx Hyper-model

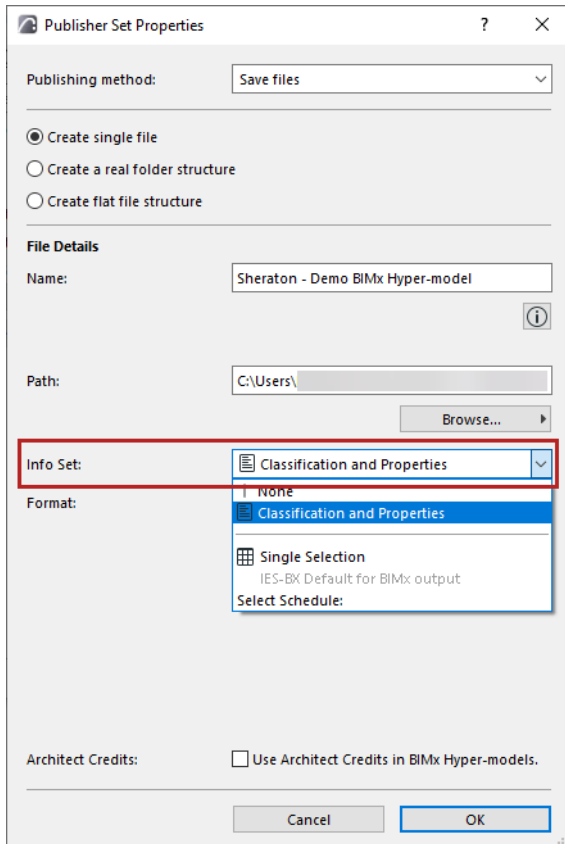
Architect Credits: Use Architect Credits in BIMx Hyper-models.

Cancel OK

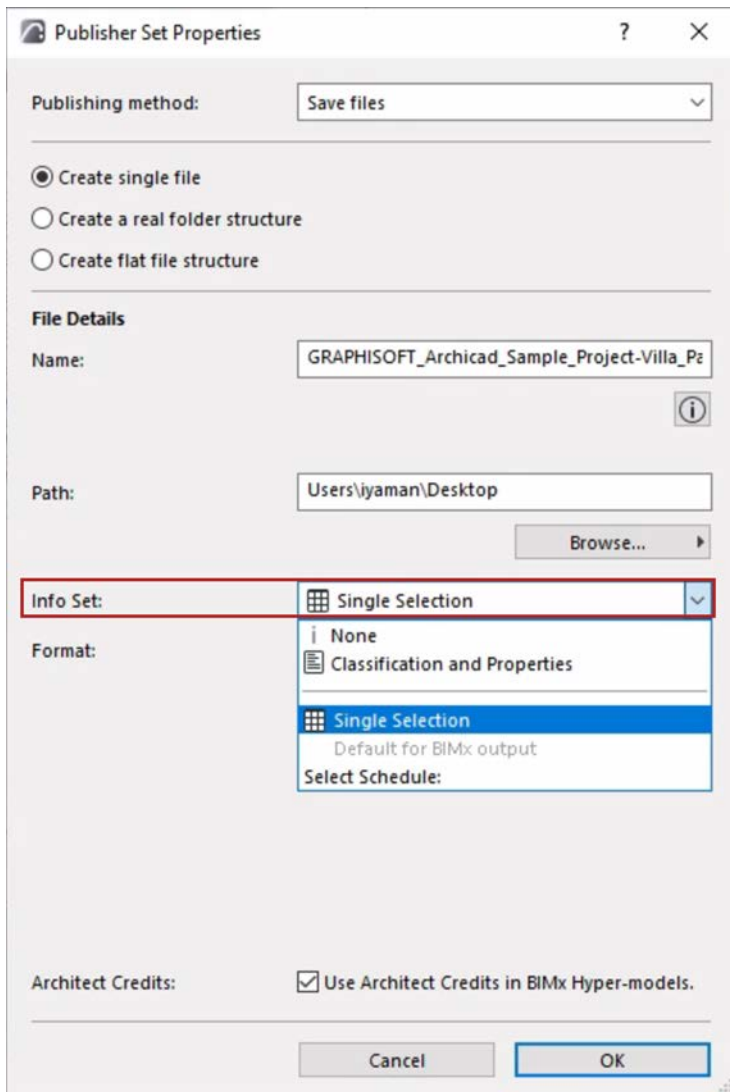
- Choose **Save files** publishing method
- Choose **Create single file**
- **Path:** Enter destination
- Choose **BIMx Hyper-model** format

Step 3: Info Set

Use the **Info Set** pop-up to control which model element information should be available to viewers of the BIMx model:



- The **Classification and Properties** option uses the data defined for each element in the Classification and Properties panel of its Element Settings dialog box.
- Choose **None** if you don't want the recipient to view any element information (e.g., if the recipient will just be viewing the model geometry as a whole, and doesn't need any details on the various component elements).
- **Select Schedule:** Click this to open a pop-up where you can choose one or more of your Element Schedule Schemes. The BIMx model will use the data defined in the selected schedule(s). Since you can choose one or more Schedules here, you may find it useful to divide the exported custom data into several Schedules (for example, by element type). This speeds up the update and publishing process, and makes it easier to manage data.
- **IES Default for BIMx output** option uses the data defined in an Interactive Element Schedule of this name. This schedule is provided in the factory-default Archicad template (for the International market), but you are free to modify the scheme as needed, or to choose any other schedule(s).



Step 4: Use Architect Credits

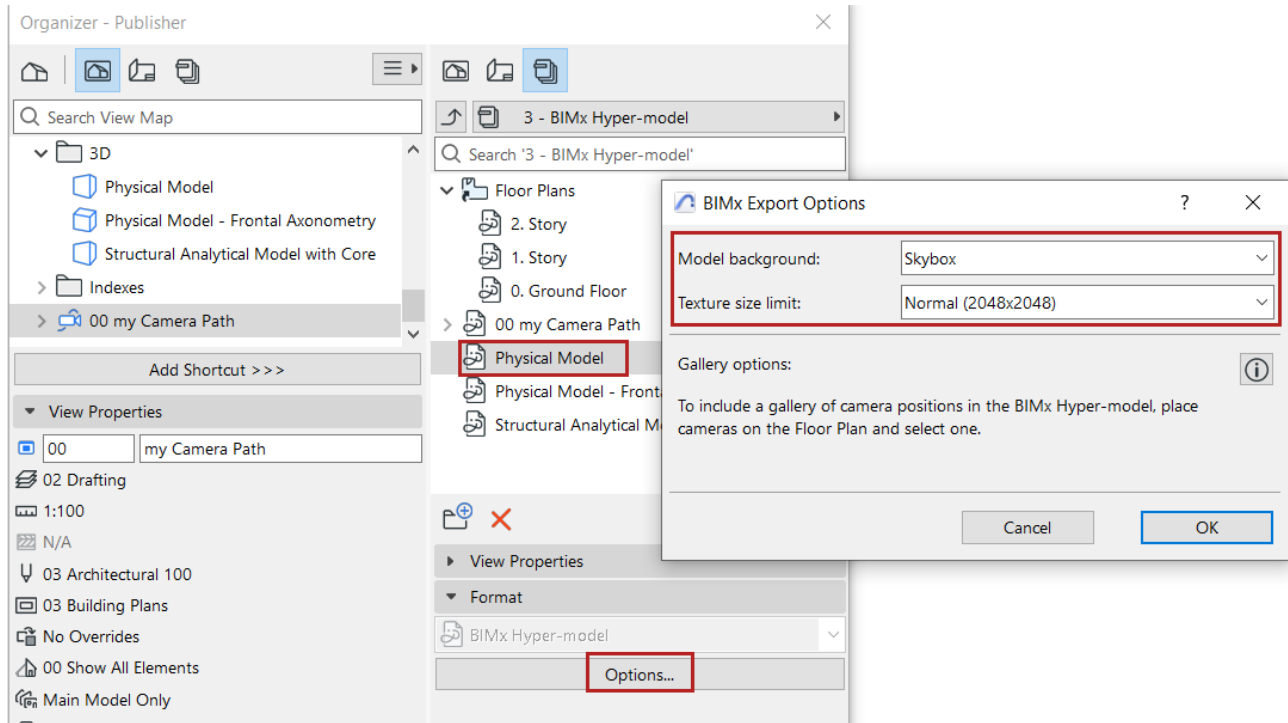
Check this box to display information about the project's designer when the Hyper-model is opened in a BIMx application.

You must first define this information in Archicad at **File > Info > Project Preview > Architect Credits in BIMx**.

[See *Architect Credits in BIMx*.](#)

Step 5: Set BIMx Export Options - for each 3D view

From the Publisher set: select a 3D view, then click **Options**.



Model background

Choose a background to use while exploring the 3D model in BIMx.

“Skybox” is the default BIMx background.

Texture size limit

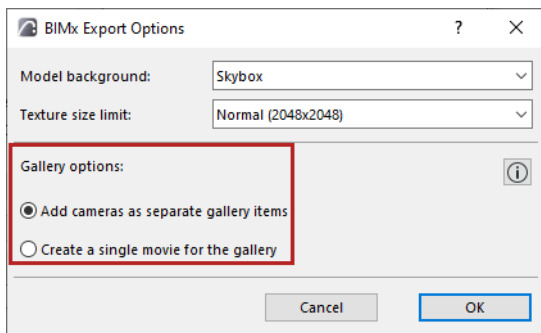
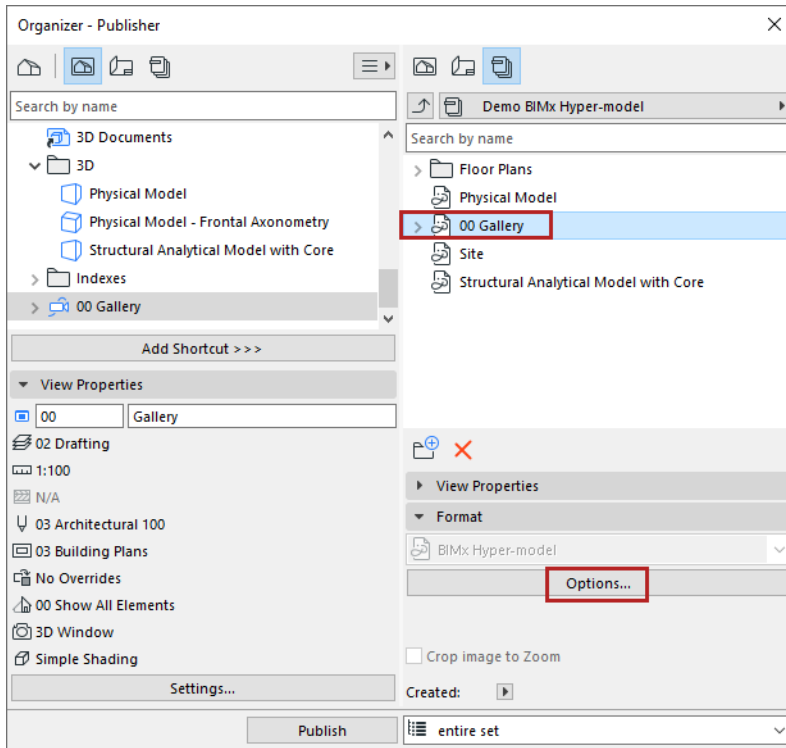
3D model textures can provide attractive visual effects, but also affect model size and export speed.

Reset the “Normal” default limit depending on your needs:

- lower limit can speed up export time and limit model size
- higher limit can result in enhanced visual details (but larger file size).

Step 6: Gallery options for camera path

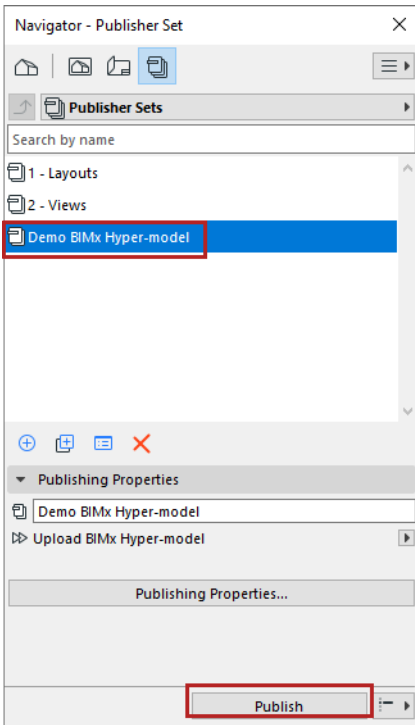
If your Publisher Set includes a camera path: select the camera path from the Organizer to set its Gallery options.



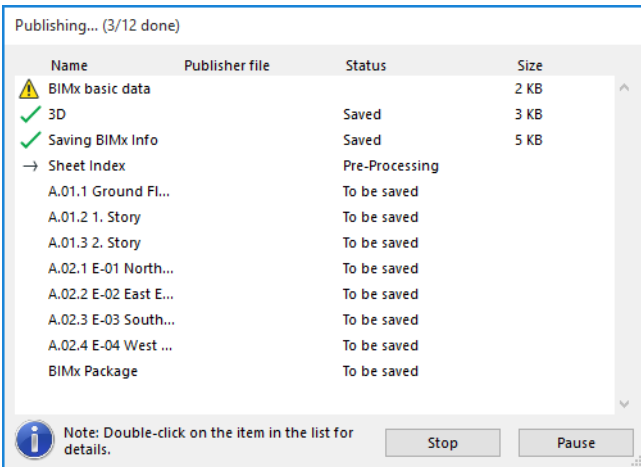
- **Add cameras as separate gallery items:** A separate gallery item (an image viewpoint) will be created out of each camera in the current Archicad camera path. (In BIMx, you can then combine these gallery items into a presentation.)
- **Create a single movie for the gallery:** A single movie-type gallery item will be created out of the current Archicad camera path.

Step 6: The Publish Process

In Publisher, select your Publisher Set and click Publish.

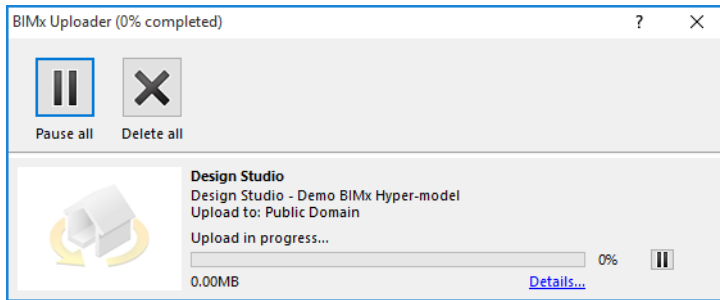


The Publisher process dialog box appears:



If you are uploading the BIMx Hyper-model to the BIMx Model Transfer site, the BIMx Uploader will track the status of the upload. Use this dialog box to stop the upload or re-start the process as needed.

Click Details for more information.



Publish BIMx Hyper-Model Using a Wizard

Use this method (**File > Publish BIMx Hyper-model**) to create a new Hyper-model Publisher Set from your Archicad model a few easy steps.

Compared to the Publisher method, the wizard is easier, but customization options are more limited:

- You can save only a single 3D view to the Hyper-model.
- You cannot pick and choose among individual layouts: you can include all Layouts, no Layouts, or a single Layout Subset.
- You have fewer options to limit model size or to fine-tune the exported element information.

To take full advantage of Archicad model data in BIMx, try saving the Publisher set using the Publisher-Organizer instead. See [Publish BIMx Hyper-Model: Publisher method \(detailed method\)](#).

Steps to Publish BIMx Hyper-model Wizard

1. In the 3D window, set up the desired view. This will be the basis of your Hyper-model.

If you want to include a gallery of camera positions in the BIMx model, place cameras on the Floor Plan and select one.

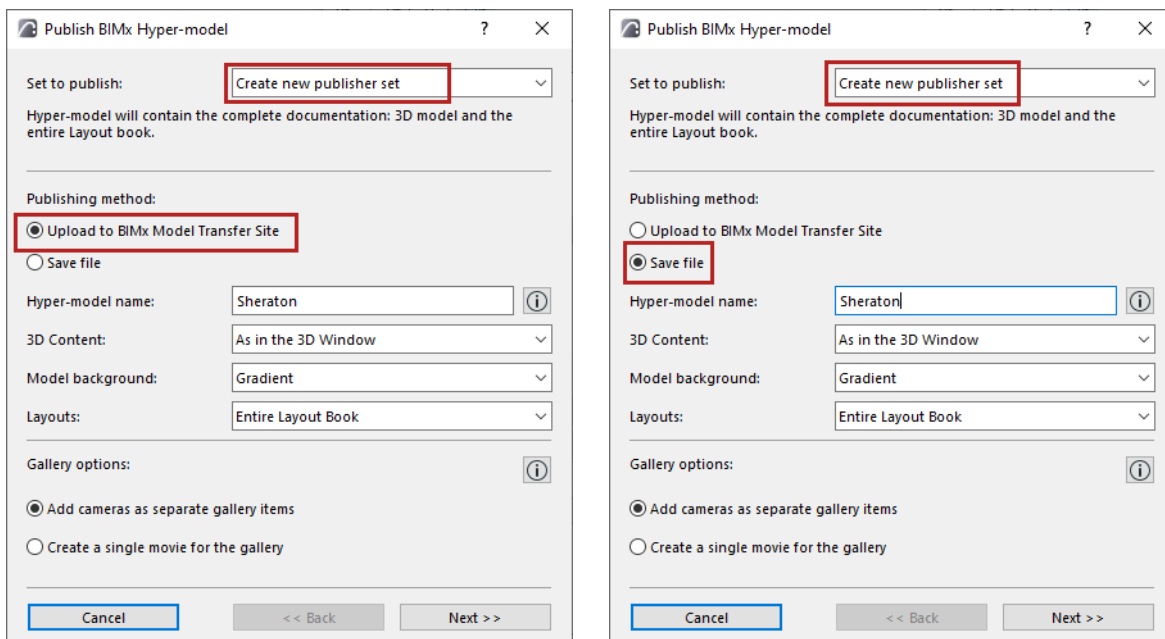
Note: You may need to optimize the Archicad model before saving it in BIMx format to ensure optimal performance. See [BIMx: Notes on Model Data, 3D Model Settings, and Textures](#).

2. Go to **File > Publish BIMx Hyper-model**.

3. In the appearing dialog box, choose **Create new publishing set**.

4. Choose a publishing method: **Upload to BIMx Model Transfer site** or **Save file**.

- **Upload to BIMx Model Transfer site** will upload the file to bimx.graphisoft.com. (You must register before using this site.)

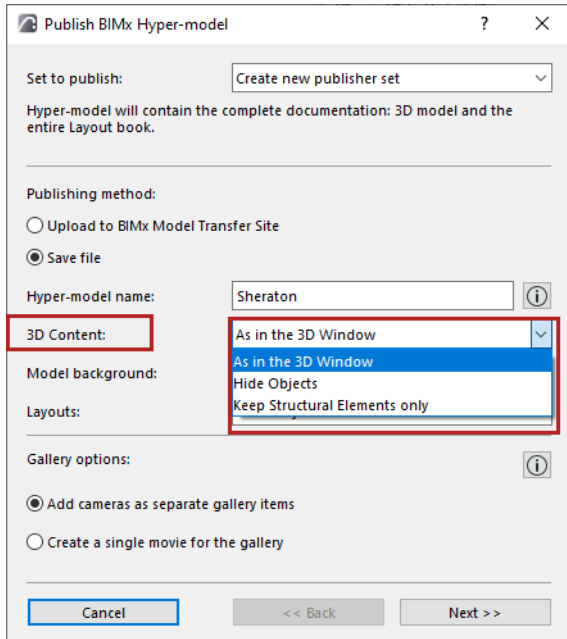


5. **Hyper-model name:** Optionally, use this field to rename the Hyper-model.

Note: The Hyper-model will include the Archicad Project Info and Project Preview data.

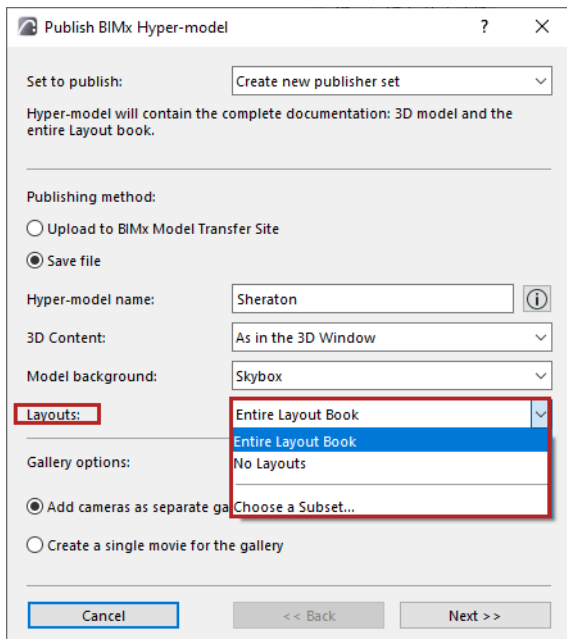
6. 3D Content: Choose an option to filter the model's 3D content:

- **As in the 3D Window:** Use the current view
- The **Hide Objects** and **Keep Structural Elements only** options limit the elements included in the 3D view, thus reducing file size.



7. Model background: Choose a background to use while exploring the 3D model in BIMx.

8. Layouts: Choose the Layout content to include in the BIMx Hyper-model: Entire Layout Book; No Layouts; or a selected subset.



- Click **Choose a Subset** to select one Layout Book Subset.

Note: When using this wizard, you cannot add or remove individual Layouts to the new Publisher Set. To do that, you must use the Publisher-Organizer.

9. Set Gallery options:

Gallery options for camera path

If the Publisher Set contains a camera path: set Gallery options.

- **Add cameras as separate gallery items:** A separate gallery item (an image viewpoint) will be created out of each camera in the current Archicad camera path. (In BIMx, you can then combine these gallery items into a presentation.)
- **Create a single movie for the gallery:** A single movie-type gallery item will be created out of the current Archicad camera path

10. Click **Next**.

11. Set the options depending on the publishing method you choose (either Upload or Save).

If you are **uploading** the BIMx Hyper-model:

Publish BIMx Hyper-model

Please provide your Graphisoft ID and the destination of the published Hyper-model.

User profile

Name: Katica Avakumovits
Company: GRAPHISOFT-DEV

[My profile...](#) Sign Out

Upload to: GRAPHISOFT-DEV

To this folder: Home

Private Quota:
49 GB used of
505 GB total

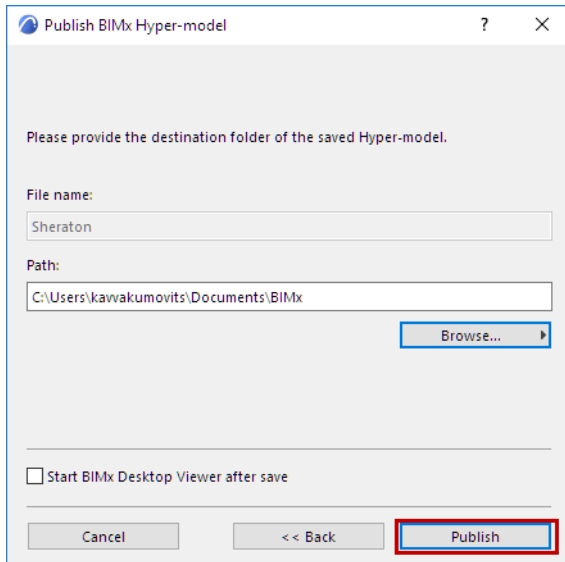
Public
 Private
 Password Protected

Cancel << Back Publish

- Choose whether to upload to the Public Domain, or to a Private Folder. If you choose Private Folder, you can optionally password-protect the file.

Note: The Private Folder option is available only in your company storage location, to users who have a Software Service Agreement (SSA), or a subscription, or a Forward agreement for Archicad.

If you are **saving** the BIMx Hyper-model:



- Enter a destination file path.
- Optionally, check **Start BIMx after save** to open the BIMx model in BIMx Desktop.

Note: The latest version of the BIMx Desktop application is available only in Microsoft Store and Apple App Store. Please use the store version to stay up-to-date.

[Read more here.](#)

12. Click **Publish**.

For details, see [Step 6: The Publish Process](#).

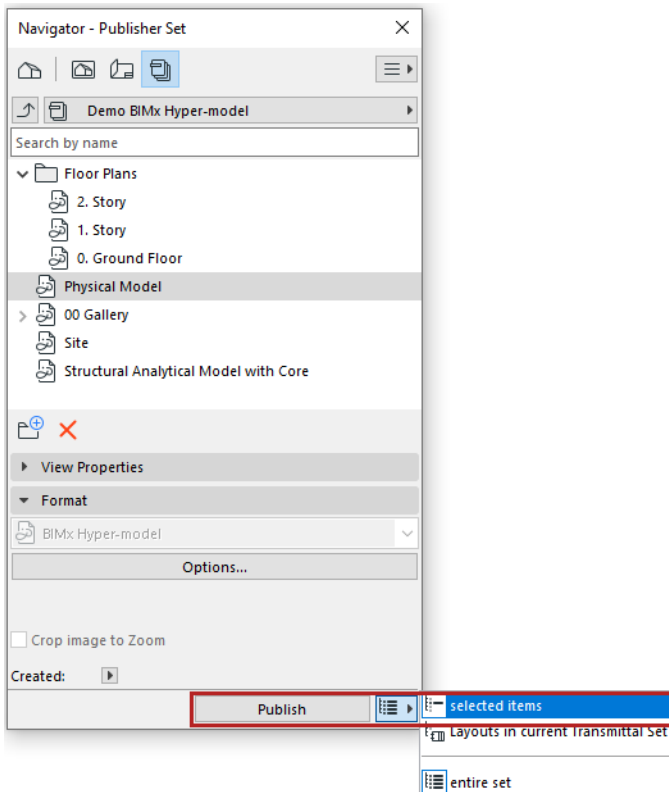
Republish Hyper-model

You can republish an existing Hyper-model either from Publisher or the wizard.

Note: The Publisher method allows you to republish selected parts of the Hyper-model as needed. The wizard always republishes the whole thing.

From Publisher:

- Select the Hyper-model and click Publish



Republish selected items to update existing Hyper-model

Choose **Selected Items** to publish only the selected items (layouts or 3D models) and merge them into the previously published, full Hyper-model

You can then open the revised Hyper-model:

- BIMx Mobile: If the Hyper-model is revised on BIMx Model Transfer, the Mobile app displays green update icons next to the publisher items' names in the Hyper-model index. The transfer service also sends model update notifications to your mobile device (if notifications are allowed for BIMx).
- BIMx Web automatically opens the latest version of the Hyper-model, which is stored on BIMx Model Transfer or BIMcloud.
- BIMx Desktop also always opens the latest version if you republish selected Hyper-model items to the same location (path: local folder, mapped folder, synced cloud storage).

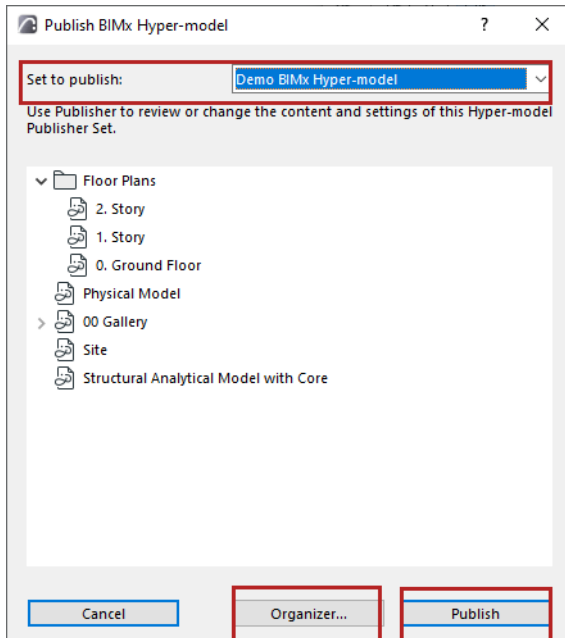
Republish selected items as new Hyper-model

Before republishing, remove the previously published Hyper-model from its saved location.

Archicad will publish only the selected items as a new, smaller Hyper-model, in place of the one you deleted (or moved).

From wizard:

1. Use **File > Publish BIMx Hyper-model**.
2. In the appearing dialog box, choose the desired Hyper-model.



3. Do one of the following:
 - Click **Publish** to republish it unchanged; or
 - Click **Organizer** to access the Organizer and modify the contents of the Publisher Set. In this case, you will follow the steps detailed at: [Publish BIMx Hyper-Model: Publisher method \(detailed method\)](#).

BIMx: Notes on Model Data, 3D Model Settings, and Textures

These tips and technical details are intended for optimizing the Archicad model before saving it in BIMx. When you open a BIMx model in any BIMx app, the model will use the Model Unit defined in Archicad by default.

For a more detailed description plus examples, consult the article on “Archicad export optimization for BIMx” at this [link](#).

Sun Settings for Enhanced BIMx Presentations

As of Archicad 27: If you use Date and Time sun positioning in [3D Projection Settings](#) or Camera Settings, Archicad will save the realistic sun position to the Hyper-model, and BIMx's Sun settings palette will open Date and time by default instead of custom.

See

Using Date and time as default helps you create dynamic presentations in BIMx that display sun and shadow effects reflecting the model's real appearance at different times of the day.

You can save different camera positions with different time settings in Archicad, and include them in the Hyper-model as BIMx Gallery items. You can also save favorites in BIMx for the same purpose on your device.

Read more: [Accurate Sun Positioning in BIMx](#).

Use Element ID Fields

Each Archicad element has an Element ID, defined by default in its Tool Settings dialog box, under the Classification and Properties Panel. Make sure that your Archicad model elements include this Element ID data for best results in BIMx.

[Read more here](#).

Background: When you are working In Teamwork, creating issues, or generating hyperlinks, BIMx can provide smart feedback on selected elements by identifying each element by its ID. If the Archicad model does not include this element ID data, BIMx cannot provide this ID-based feedback.

Change in BIMx 3D format (Archicad 26 and later)

As of Archicad 26, BIMx models use a single, modern 3D model format.

In earlier versions, exporting to BIMx format required generating the 3D model twice - to accommodate two different formats (older and newer). The renewed mobile, web, and desktop apps already use only the new 3D format.

As of Archicad 26: The previous, outdated 3D model format is no longer exported, and all models are exported only with the newer format. Export from Archicad is considerably faster in many cases, and the file size is smaller.

Advantages of the new format are visible in both BIMx Desktop and BIMx Web:

- BIMx model reflects Archicad Location Settings:
 - Direct light makes the model more realistic (north-facing sides are darkened). Set Project Location and/or North Angle in Project References > Location Settings to open the 3D model with the desired and realistic light direction in BIMx.
 - Automatic rotation of “Skybox” background

- Archicad-defined View Cone is also reflected in BIMx. Define the desired View Cone in 3D Projection Settings. You can set different values for e.g. interior and exterior 3D models included in the same Hyper-model.
- On-the-fly rendering (Ambient Occlusion) is available in all BIMx apps.

Textures

BIMx supports the following surface texture formats: jpg, bmp, tif, tga, png, gif.

If a texture would fail to load for any reason, the object that uses it will be drawn with the surface color instead. As of Archicad 26, when exporting an Archicad model to BIMx, you can set your own texture size limit for a 3D view - for better control over texture quality, model size, and the time needed for export. The previous limit was hard-coded at 1024 x 1024 pixels in Archicad 25 and earlier versions.

The new default is 2048 x 2048 pixels, and you can max this out to 4096 x 4096 pixels for best detail level, or choose a smaller texture size to limit file size and save time.

Alpha Channel Texture Effects

Use texture mapping and alpha-channel transparent pictures instead of physical models. This solution can be extremely useful if you do need a large number of trees, vehicles, fences and other complex objects in the model.

BIMx supports alpha channel texture effects with 32-bit tif, tga and bmp images. For the alpha channel effects to be detected inside BIMx, you must enable transparency in the Archicad surface/texture properties.

However, transparent objects are not necessarily visualized using the correct display order - to do so would put a big dent in your performance.

Energy Evaluation

Integrated in the Archicad environment, Energy Evaluation offers an easy-to-use workflow for performing dynamic building energy calculations on projects of any size.

Energy Evaluation is an energy evaluation tool that enables architects to monitor and control all architectural design parameters that influence building energy performance. The Energy Evaluation tool performs reliable dynamic energy evaluation at all stages of the design process, so that architects can make informed decisions regarding their buildings' energy efficiency. Including Energy Evaluation in the architectural design workflow makes it easy to create projects that conform to or even exceed energy efficiency regulations and comply with building energy standards.

- Create and visualize multiple thermal block Building Energy Model (BEM) geometry directly from the Archicad BIM using the Energy Model Review Palette.
- Use the Model-Based Solar Study to determine the intensity of solar irradiation on each external glazed opening individually, on every hour of the reference year - including the shading effect of the surroundings (buildings, plants etc.) and shading devices.
- Export the Archicad model geometry and material property data via IFC or as an XLS spreadsheet, to be processed by external energy analysis applications
- Run the dynamic energy simulation, using the VIP Core engine that is integrated in Archicad, to produce an Energy Performance Report. This report provides information on the project's yearly energy consumption, carbon footprint and monthly energy balance.

Note: EcoDesigner STAR, available for SSA/Forward customers and Education license holders, is an Archicad extension that enables designers to further extend and fully utilize the building energy modeling capabilities of Energy Evaluation.

The details of the Energy Evaluation feature are described in these sections:

[Energy Evaluation Workflow: Overview](#)

[General Settings for Energy Evaluation](#)

[Energy Evaluation: Direct BIM to BEM](#)

[Energy Model Review Palette](#)

[Additional Data Assignment and Input](#)

[Energy Performance Evaluation](#)

Energy Evaluation Workflow: Overview

This section summarizes the steps of the Energy Environmental Workflow.

Prepare the Architectural BIM for Energy Evaluation

For successful evaluation, the **building model** must contain at least the enveloping structures and fenestration, as well as all major internal structures that represent significant heat storage mass.

Furthermore, Archicad **zones** must be placed in every conditioned space of the building, since the model's geometry analysis is based on these Archicad zones.

The Energy Model Review works only on visible elements, so you must define an Archicad **view** dedicated to the energy model. Use layer visibility to achieve the view you need. Make sure that zones are visible.

Note: Marquee selection does not filter building elements for the purpose of Energy Model Review.

Define Thermal Blocks

Spaces are represented by 3D Zones in the Archicad Building Energy Model. For Energy Evaluation purposes, you will group these Zones together in "thermal blocks", using the Thermal Blocks page of the Energy Model Review palette (**Design > Energy Evaluation > Energy Model Review**).

Thermal blocks are a collection of one or more rooms or spaces in a building that have similar orientation, operation profile and internal temperature requirements (also called thermostat control requirements). Zones need not be contiguous to be combined within a single thermal block.

Automatic Model Geometry and Material Property Analysis

After the thermal blocks have been defined, the architectural model (BIM) is transformed into a Building Energy Model (BEM) by the automatic model geometry and material property analysis functionality of Archicad. This analysis does the following:

- analyzes the visible structures and openings according to their orientations and positions relative to zones and generates the space boundaries on them. (Space boundaries describe the building's geometry in a format that works for energy simulation input.)
- populates the space boundary lists. Structures and Openings are automatically listed with their properties that are relevant for the energy evaluation.

Assign and Input Additional Data to Complete the BEM

The Energy **Model Review palette** is the main user interface of Archicad's Energy Evaluation function. Use the palette to edit the input data for the building energy simulation, as well as to add information:

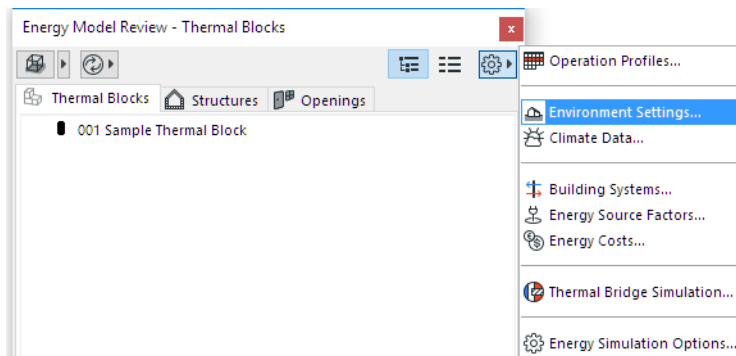
- Assign Building Systems and Operation Profiles to Thermal Blocks
- Space boundary property settings

Note: Most space boundary properties are extracted from the architectural model by the automatic model geometry and material property analysis functionality of Archicad. Space boundary property settings allow users to fine-tune this data and add information that does not come directly from the architectural building model.

- Structures property settings: Use the U-value calculator or U-value override functionality and the Infiltration and Surface material settings to define the physical properties of the opaque space boundaries listed on the Structures list.

- Openings property settings: Assign Frame and Glazing performance data to fenestration from the Openings catalog. Select Shading devices from predefined databases to complete the Openings list.

Besides displaying the Structures and Openings list, the user-friendly Energy Model Review palette also contains links to dialogs.



These Additional Data Input dialogs enable quick definition of the following parameters needed to run the Energy Evaluation:

- **Environment Settings:** Provides links to the Location Settings, Climate Data and Wind Protection dialogs. Grade level, Soil and Surrounding surface types are also set here.
- **Operation Profile:** Select the building's function(s) to assign related Internal temperature and heat gain profiles in the Operation Profile dialog. If needed, customize an Operation Profile to fine-tune occupancy according to local regulations or to match actual building usage.
- **Building Systems:** Provide information essential for the energy calculation, regarding the building's MEP Systems (for heating, cooling, ventilation, hot water generation)
- Finally, in order to enable primary energy, CO2 emission and energy cost calculations, define the **Energy Source Factors and Energy Costs**, using the dedicated dialogs.

Evaluate Building Energy Performance

Click the Start Energy Simulation button of the Model Review palette to start the **Energy Performance Evaluation** of your project. The built-in, certified VIP-Core engine executes the dynamic energy simulation that calculates the building's hourly energy balance and outputs a Building Energy Evaluation Report. The report contains information such as the project's energy-related structural performance, yearly energy consumption, energy balance and carbon footprint.

[Read More](#)

General Settings for Energy Evaluation

This section contains information on setting your general preferences for the Energy Evaluation function.

Units

Energy Evaluation uses the calculation units defined by you in Archicad, at **Options > Project Preferences > Calculation Units**. Thus, Energy Evaluation uses the same units as the Archicad Project to which it is applied.

Additional unit setting options:

- When defining components of Heating and Ventilation systems in the Building Systems dialog box, choose either Celsius or Fahrenheit for the water temperature.
This way, for example, you can set the temperature scale of Hot water generation independently of other units. (This can be useful if your country uses metric dimensions and Fahrenheit temperatures concurrently.)
- On the U-value (R-value) Calculator and the U-value (R-value) Override panels, click the pop-up to alternate between U-value and R-value as needed.

General Modeling Conventions

For best results with Energy Evaluation, follow these conventions of building energy modeling in Archicad:

- Use the Mesh tool to model the building site's grade level, especially if it is uneven.
- Do not use multiple separate parallel walls (or slabs or roofs) to model composite structures. If such constructions exist in the building model, make sure that only one of the parallel structures is visible in the energy model, and that the adjacent Internal Space Zones touch that structure.
- Use fills consistently within your project: a particular fill should always indicate the same single building material or composite.

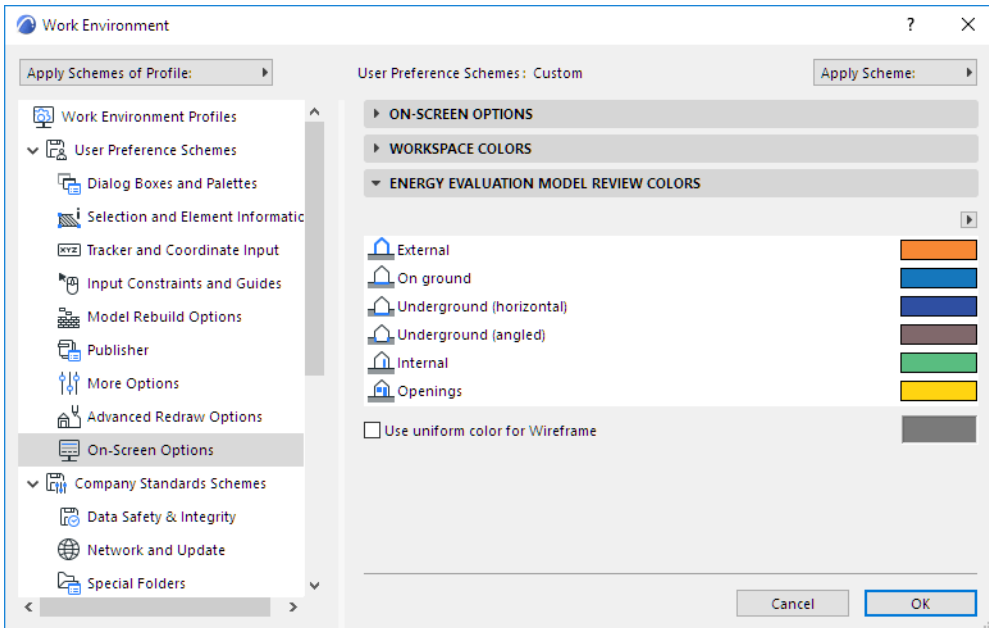
See [Energy Evaluation: Direct BIM to BEM for more details on modeling guidelines](#).

Teamwork Settings

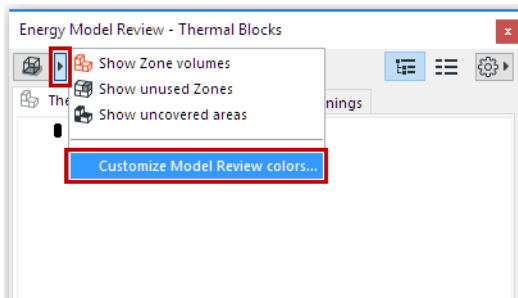
If you are working on a Teamwork project, you must first reserve the Energy Model Review palette using the control light interface on the palette itself, or the dialog boxes that open from it.

Customize Energy Model Review Colors

To define or change the 3D default colors for Structures and Openings listed in the Model Review palette: go to **Options > Work Environment > On-Screen Options**, and use the Energy Evaluation Model Review Colors panel:



A quick way to access these controls is by selecting **Customize Model Review Colors** from the pop-up to the right of the Show in 3D button, at the top of the Model Review Palette:



Energy Evaluation: Direct BIM to BEM

To be able to perform a successful evaluation with Energy Evaluation, you must create the virtual building model in Archicad. The more detailed the model, the more accurate the calculation results will be. For successful evaluation, you should model, at minimum, the enveloping building structures and openings as well as the major internal structures that represent significant heat storage mass.

Topics in this section:

[Internal Space Zones for Energy Evaluation](#)

[Building Energy Model View](#)

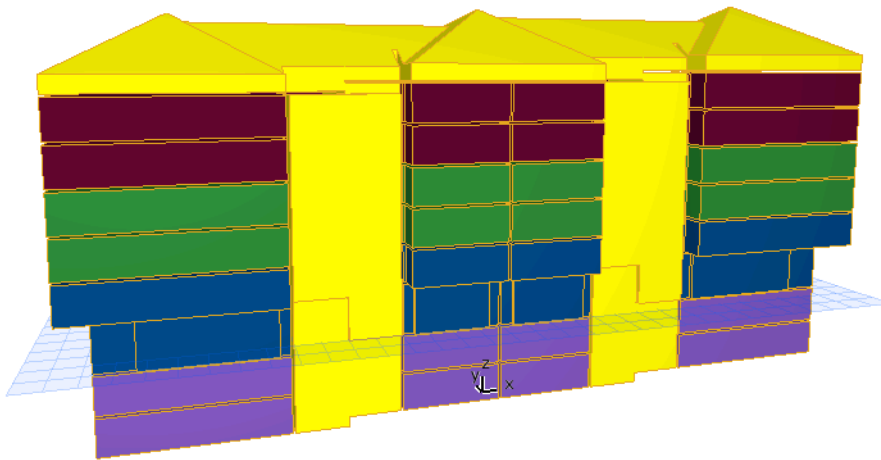
[Automatic Model Geometry and Material Property Analysis in Energy Evaluation](#)

Internal Space Zones for Energy Evaluation

Create a zone in every conditioned space of the building, using exclusively the **Inner Edge** zone construction method.



Energy Evaluation is **not compatible** with zones created with the Manual construction method. The zones must be directly adjacent to the surrounding elements' surfaces, therefore the Reference Line zone construction method is not applicable for energy modeling either.



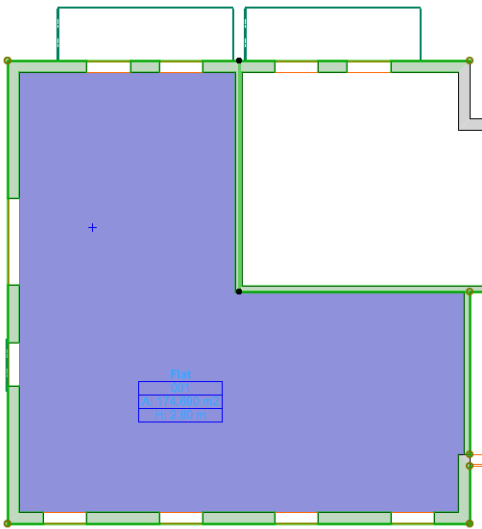
Archicad zones inside the project's internal spaces

3D Zone Boundaries

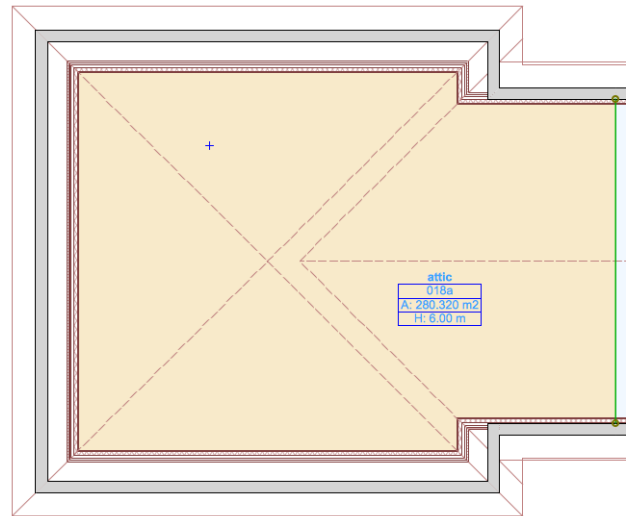
When placing zones on the floor plan, make sure that they are completely enclosed by zone boundaries.

- All Archicad walls (including profiled curtain walls) automatically behave as zone boundaries.
- When using Slabs as bottom or top zone boundaries, make sure that the zone level and/or height are set so that the horizontal zone surface(s) touch the inner edge of the Slab(s).
- If Roofs, Meshes, Shells, Morphs or Archicad library objects are used as zone boundaries, an additional interaction is required besides just creating the zone. In such situations:
 - Draw the zone manually, making sure that it extends beyond the Roof, Mesh, Shell zone boundary elements completely.

- Use a command from the **Design > Connect** menu, such as Trim Elements or Solid Element Operations, to shape the zone with the selected zone boundaries and establish the connection between them.



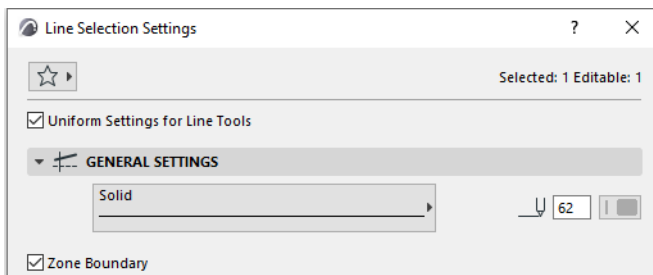
Walls as zone boundaries



Walls and line as zone boundaries

2D Zone Boundaries

It is also possible to define lines on the floor plan as 2D zone boundaries. Select the line(s) you want to behave as zone boundaries and tick the Zone Boundary checkbox on the Line Selection Settings dialog.



There are a number of modeling scenarios that make it necessary to define lines as Zone Boundaries. For example:

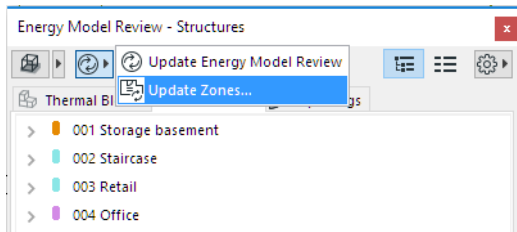
- Modeling perimeter zones in Core and Shell projects
- Atria in buildings
- Spaces that contain several, dramatically different floor and/or roof levels



Attic space with different floor and roof levels

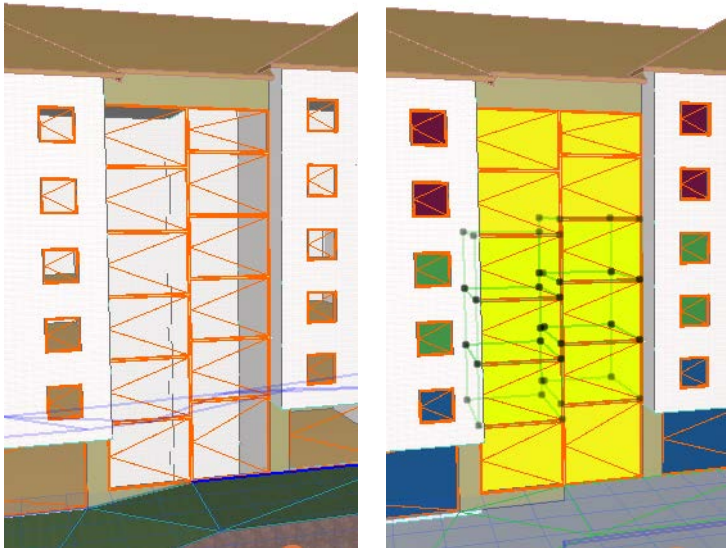
Update Zones

Use the Update Zones command on the left upper corner of the Energy Model Review palette after altering the geometry of zones or zone boundaries. This will ensure that the energy model reflects the up-to-date state of the Archicad model.



Zone Modeling Conventions

- Model multistory internal spaces (e.g. staircase blocks, atria) with separate zones on every Archicad Story.



Staircase - modeling of multistory internal space

- Adiabatic walls are walls of the building shell that separate heated spaces. They are called adiabatic due to the absence of heat transfer through them. A typical example is a fire wall separating row-houses or other adjacent buildings.

In such situations, model the surfaces of the neighboring building that are adjacent to your building, with thin Archicad zones. Use a separate zone on every story of your project.

Building Energy Model View

The Energy Model Review of the Energy Evaluation feature only works on visible elements; therefore it is necessary to define an Archicad view dedicated to the energy model. Use layer visibility to achieve the view you need. Make sure that zones are visible.

Note: Marquee selection does not filter building elements for the purpose of Energy Model Review.



Building Energy Model View

The Building energy Model View must display all the parts of the building that you want to include in the energy evaluation (internal space zones, building shell elements, internal structures with significant thermal mass, site mesh) and none that you do not (furniture, zones other than those representing the conditioned spaces of the building, separate parallel Archicad elements representing composites).

Automatic Model Geometry and Material Property Analysis in Energy Evaluation

Space Boundary Generation

Just select **Design > Energy Evaluation**, and the Automatic model geometry analysis function automatically turns the Archicad building information model (BIM) into input for building energy modeling.

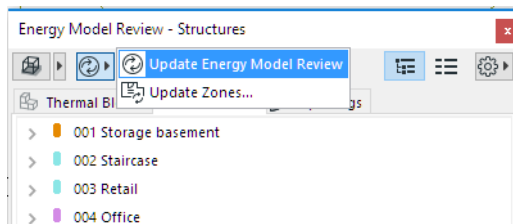
Before the Energy Model Review palette appears, Archicad analyzes the visible structures and openings according to their orientations and positions relative to the zones that represent the internal spaces, then automatically generates the space boundaries. These space boundaries make up the contents of the Structures and Openings lists of the appearing Energy Model Review palette.

Note: When Energy Evaluation is selected for the first time on a project, Archicad creates a default Sample Thermal Block. This represents a single thermal block (simplified) building energy model. You must add zones to this Thermal Block manually.

[See *Create Thermal Blocks* for details on how to define a detailed energy model that contains multiple thermal blocks.](#)

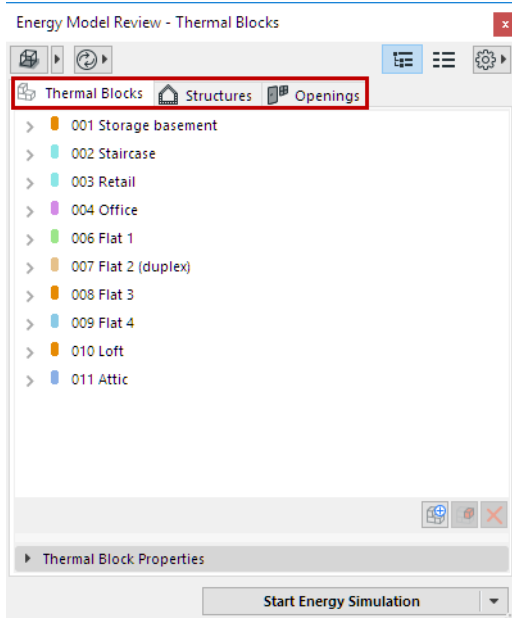
Update Energy Model Review

Use the Update Energy Model Review command on the upper left corner of the Energy Model Review palette after redefining Thermal Block data, or after altering the geometry of Archicad Structures or Openings boundaries. This function regenerates the space boundaries, ensuring that the energy mode reflects the up-to-date state of the Archicad model.



Energy Model Review Palette

The Energy Model review palette is the main user interface of Archicad's Energy Evaluation. It contains three tab pages that display input data for the building energy simulation. Use the tab pages to edit as well as to add information to this calculation input data.



Besides the three main tab pages, the Energy Model Review palette also contains:

- General function buttons:
 - BEM visualization options
 - BEM update options
 - Tab page view switch
 - Links to the Additional Energy Simulation Input dialogs
- Tab-page specific functions: see the following sections for details.

List and Tree Tab Page View Options

Thermal Blocks Tab Page

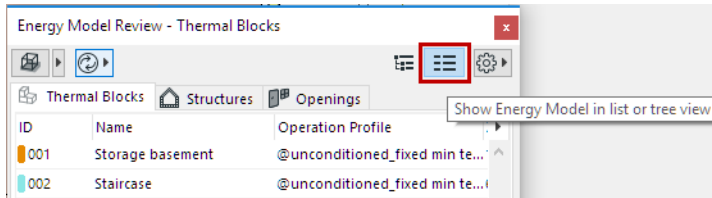
Structures Tab Page

Openings Tab Page

Building Energy Model Visualization

List and Tree Tab Page View Options

Use the “Show Energy Model in list or tree view” function buttons of the Energy Model Review palette to toggle between the two available tab page view options.

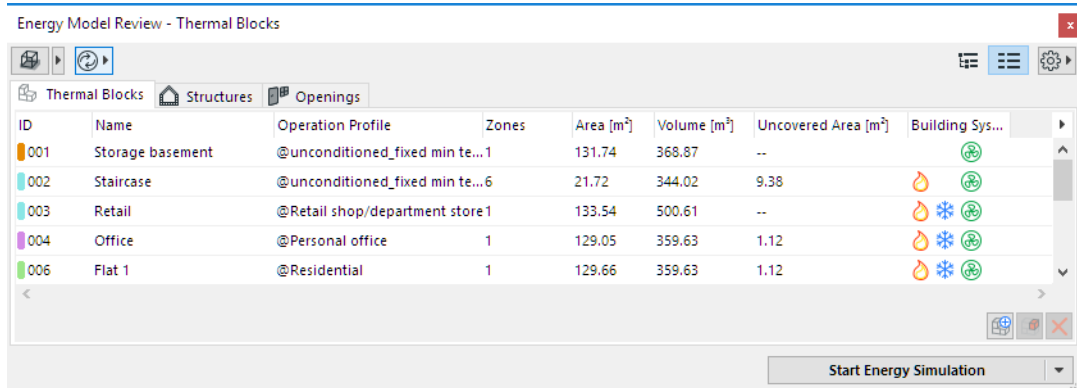


These view options display the energy simulation input data in two different ways, allowing for a wide variety of user interactions. Most interactions (data assignments, overrides, adjustments etc.) can be performed in both views, while others are view-specific (e.g.: rearranging data according properties is only possible in List View, drag-and-dropping zones into thermal blocks is only possible in Tree View).

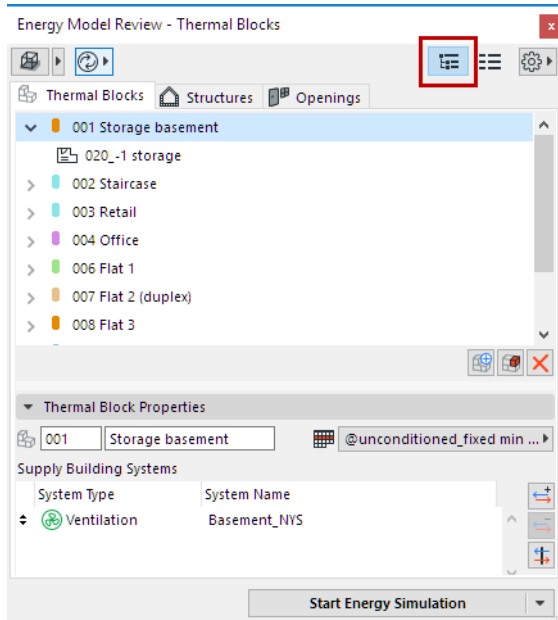
Thermal Blocks Tab Page

For Energy Evaluation purposes, you will group Archicad Zones together in “thermal blocks”. Thermal blocks are a collection of one or more rooms or spaces in a building that have similar heating or cooling requirements - also called thermostat control requirements. Thermal blocks are represented by groups of 3D Zones in the Archicad Building Energy Model.

The Thermal Blocks page of the model Review Palette lists thermal blocks with all their relevant properties.

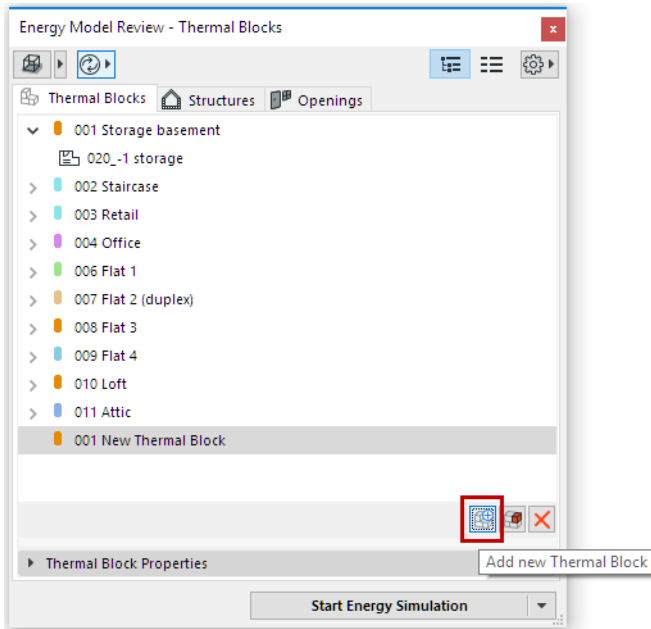


In Tree View, users can see the Zones that make up the thermal blocks as well, if they select the dropdown arrow by the thermal block's name. If a thermal block is selected, its properties are displayed on the bottom of the page, in the Thermal Block Properties section.



Create Thermal Blocks

Use the **Add new Thermal Block** button to create a new thermal block. Its name and ID can then be entered in the list.



To assign Zones to the thermal block:

- Drag and Drop 3D Zones into Thermal Block (in Tree View)
 - Use Find and Select to Select Zones and Add Zones to Thermal Block (in List View)
- Note:** Zones need not be contiguous to be combined within a single thermal block.

Thermal Blocks Data

The Thermal Blocks tab page can be viewed in either table or tree form. There are two kinds of data types on this page:

- Archicad model data types (coming directly from the BIM):
In case of Thermal Blocks: Number of Zones, Floor Area, Volume, Uncovered Area
- Additional data types (assigned to the Thermal Blocks list entries by the user):
Thermal Block ID and Name, Operation Profile, Building Systems.

[See Thermal Block Property Settings \(Energy Evaluation\).](#)

Structures Tab Page

The Structures tab page of the Energy Model Review palette lists the space boundaries on the building's external and internal structures, with all their physical properties that are relevant for the energy simulation. The opaque panels of curtain walls also appear on the Structures list.

Energy Model Review - Structures

Thermal Blocks Structures Openings

Thermal Block	Orientation	Category	Name	Type	Area [m ²]	Thickness...	U-value [W/m ² K]	Infiltration [l/sm ²]	Solar Ab: ▶
004 Office	Bottom	External	@Shell Slab_good	Slab	41.44	430	0.18	1.10	60.00
007 Flat 2 (duplex)	Bottom	External	@Shell Slab_good	Slab	5.32	430	0.18	1.10	60.00
010 Loft	Inner (011 Attic)	Internal	@Living attic ceiling_poor	Slab	9.17	200	0.24	0 %
009 Flat 4	Inner (010 Loft)	Internal	Reinforced Concrete - Structural	Slab	122.04	200	4.05	0 %
008 Flat 3	Inner (010 Loft)	Internal	Reinforced Concrete - Structural	Slab	29.26	200	4.05	0 %
010 Loft	Inner (010 Loft)	Internal	@External Wall_good	Wall	27.62	600	0.12
002 Staircase	Inner (010 Loft)	Internal	Masonry Block - Filler	Wall	12.60	200	0.55	5 %
011 Attic	Inner (010 Loft)	Internal	@Living attic ceiling_poor	Slab	9.17	200	0.24	0 %
006 Flat 1	Inner (009 Fla...)	Internal	Reinforced Concrete - Structural	Slab	128.44	200	4.05	0 %
010 Loft	Inner (009 Fla...)	Internal	Reinforced Concrete - Structural	Slab	122.04	200	4.05	0 %
008 Flat 3	Inner (009 Fla...)	Internal	Masonry Block - Filler	Wall	34.44	200	0.55	5 %
002 Staircase	Inner (009 Fla...)	Internal	Masonry Block - Filler	Wall	12.60	200	0.55	5 %
007 Flat 2 (duplex)	Inner (008 Fla...)	Internal	Reinforced Concrete - Structural	Slab	58.84	200	4.05	0 %
009 Flat 4	Inner (008 Fla...)	Internal	Masonry Block - Filler	Wall	34.44	200	0.55	5 %

Area threshold: 0.60 m²

Start Energy Simulation

The Tree View of the Structures Page shows exactly which External and Internal Structures belong to which thermal block in an easy to understand, graphic way. If a structure is selected, its properties are displayed on the bottom of the page, in the Structure Properties section.

Energy Model Review - Structures

Thermal Blocks Structures Openings

- > 001 Storage basement
- > 002 Staircase
- > 003 Retail
 - External Structures
 - Straight Wall - Existing
 - Straight Wall - Existing
 - Straight Wall - Existing

Area threshold: 0.60 m²

Structure Properties

Thermal Block: 003 Retail

Orientation: West

Category: External

Name: @Basement Wall_good

Type: Wall

Area: 37.77 m²

Thickness: 400 mm

U-value: 0.17 W/m²K

Infiltration: 1.10 l/sm²

Solar Absorptance: 60.00 %

Start Energy Simulation

Structures Data

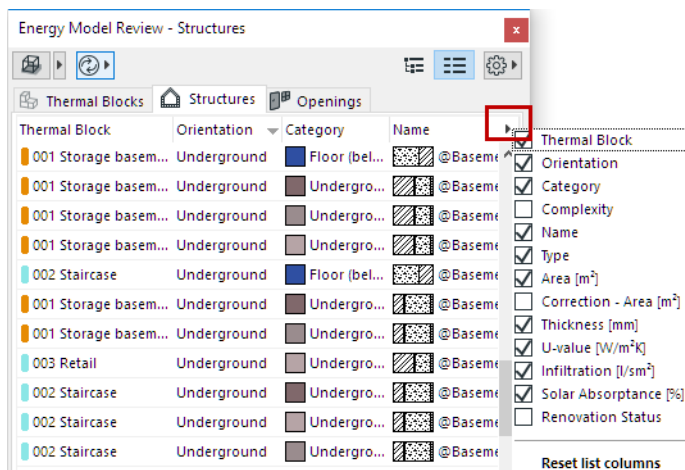
There are two kinds of data types:

- Archicad model data types (coming directly from the BIM):
In case of Structures: Thermal Block assignment, Orientation, Space Boundary Category, Archicad Element Type, Complexity, Name, Area, Thickness, Renovation Status
- Additional data types (assigned to the Structures list entries by the user):
U-value/R-value, Infiltration, Solar Absorptance, Correction-area

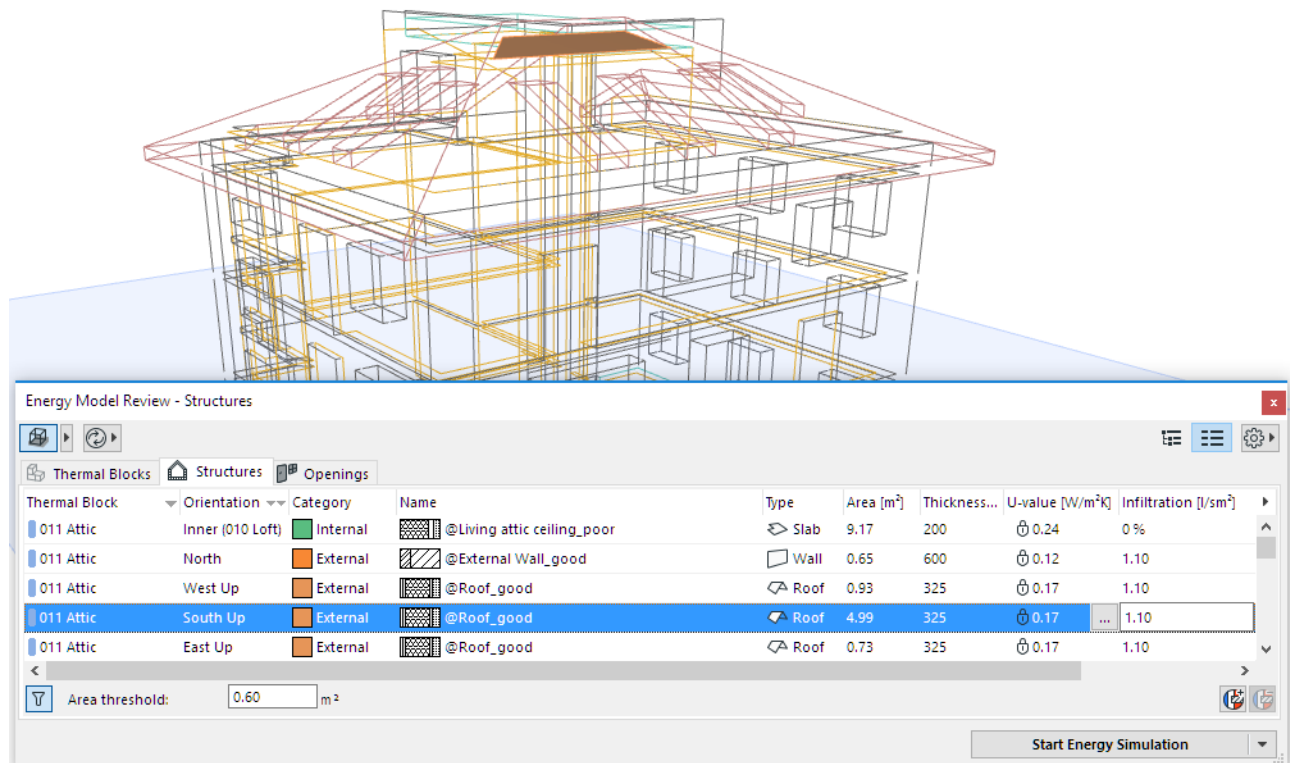
[See Structure Property Settings \(Energy Evaluation\).](#)

Show or hide any of these data types as needed:

- In tree view: right-click any Structure Properties list item to show or hide the data types
- In list view: Click the arrow pop-up at right to show or hide the table columns



One Structures List entry usually consists of multiple Space boundaries. All Space boundaries that are identical in all their Archicad model data types are aggregated in a single Structures List entry and appear with the sum of their areas on the list.



In list view, you can rearrange the list according to any property by clicking on the property's headline button.

Area Threshold

To filter out structure or opening list entries of negligible area, enter a value in the “Area Threshold” field at the bottom of the Structures list. Entries whose area is smaller than this value will not be listed, and will not be considered by the calculation engine. (If you later lower this area threshold, the filtered-out structures will reappear as applicable.)

Setting an Area threshold will help you filter out small areas that are insignificant for energy evaluation, resulting in a more manageable Structures list.

Structure Area Correction

The Structures tab page of the Model Review palette has a data type for “Correction-area”. For any listed Structure, you can adjust the area by entering a positive or negative value in this column. The value of this item in the “Area” column will be adjusted accordingly; this corrected Area value (if positive) will be used by the calculation engine.

Openings Tab Page

The Openings tab page of the Energy Model Review palette lists the space boundaries on the building's external openings, with all their physical properties that are relevant for the energy simulation.

Energy Model Review - Openings

Thermal Blocks Structures **Openings**

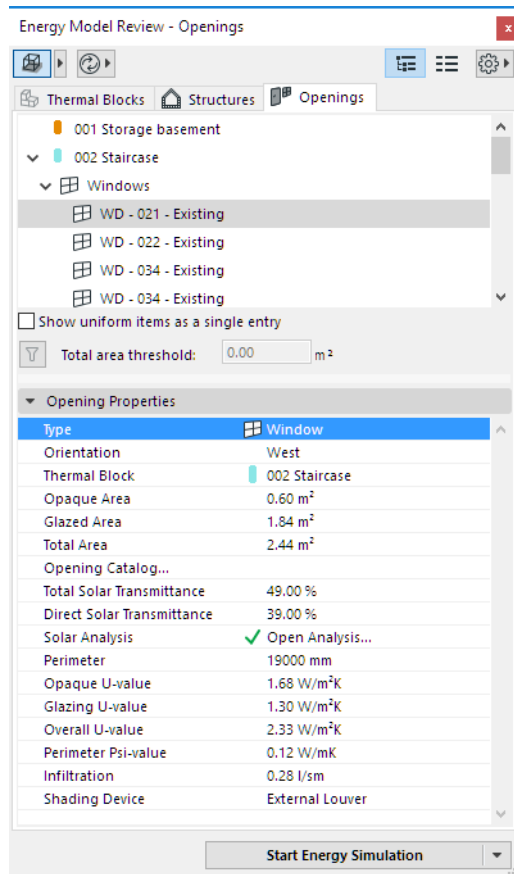
Type	Orientation	Thermal Block	Opaque Area [m ²]	Glazed Area [m ²]	Total Area [m ²]	TST%	DST%	Solar Analysis	Perimeter [mm]	Opaque U
Window	East	003 Retail	1.09	12.11	13.20	49.00	39.00	✓ Done	14200	1.68
Window	East	003 Retail	1.09	12.11	13.20	49.00	39.00	✓ Done	14200	1.68
Window	South	004 Office	0.65	3.85	4.50	49.00	39.00	✓ Done	8400	1.68
Window	South	009 Flat 4	0.65	3.85	4.50	49.00	39.00	✓ Done	8400	1.68
Window	South	006 Flat 1	0.65	3.85	4.50	49.00	39.00	✓ Done	8400	1.68
Window	West	008 Flat 3	0.56	3.04	3.60	49.00	39.00	✓ Done	7200	1.68
Window	West	009 Flat 4	0.56	3.04	3.60	49.00	39.00	✓ Done	7200	1.68
Window	South	004 Office	0.56	3.04	3.60	49.00	39.00	✓ Done	7200	1.68
Window	South	009 Flat 4	0.56	3.04	3.60	49.00	39.00	✓ Done	7200	1.68
Window	West	006 Flat 1	0.56	3.04	3.60	49.00	39.00	✓ Done	7200	1.68
Window	South	006 Flat 1	0.56	3.04	3.60	49.00	39.00	✓ Done	7200	1.68
Window	West	007 Flat 2 (duplex)	0.56	3.04	3.60	49.00	39.00	✓ Done	7200	1.68
Window	East	002 Staircase	0.70	2.50	3.20	49.00	39.00	✓ Done	9000	1.68

Show uniform items as a single entry

Total area threshold: m²

Start Energy Simulation

The Tree View of the Openings page shows exactly which Doors, Windows and transparent Curtain Wall panels belong to which thermal block in an easy to understand, graphic way. If an Opening is selected, its properties are displayed on the bottom of the page, in the Opening Properties section.



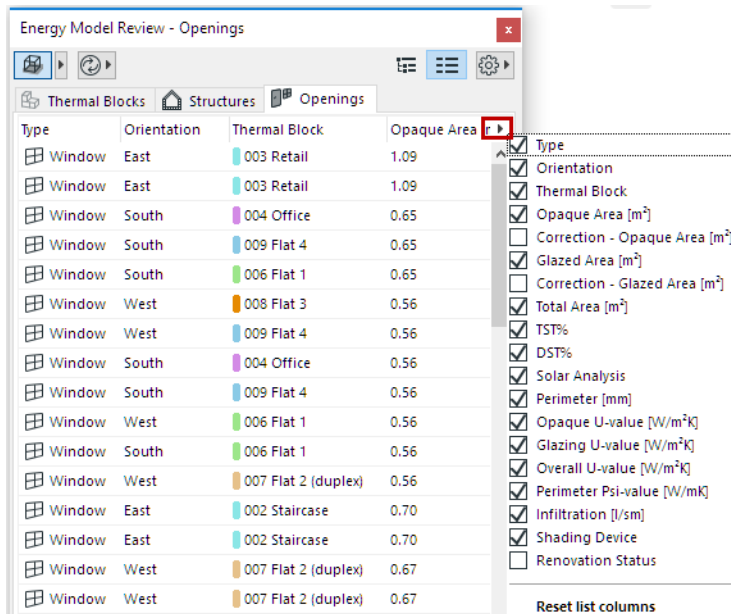
Openings Data

The Openings List can be viewed in either table or tree form; each column represents a data type. There are two kinds of data types:

- Archicad model data types (coming directly from the BIM):
In case of Openings: Thermal Block assignment, Solar Analysis results, Orientation, Archicad opening type, Area (Glazed, Opaque, total), Frame perimeter, Renovation Status.
- Additional data types (assigned to the Openings list entries by the user):
In case of Openings: Total and Direct Solar Transmittance, U-value (for glazing, opaque areas, perimeter and overall), Perimeter Psi value, Shading device, Infiltration

Show or hide any of these data types as needed:

- In tree view: right-click any Structures Properties list item to show or hide the data types
- In list view: Click the arrow pop-up at right to show or hide the table columns



In list view, you can rearrange the list according to any property by clicking on the property's headline button.

For each Openings list item, the following data are shown in columns:

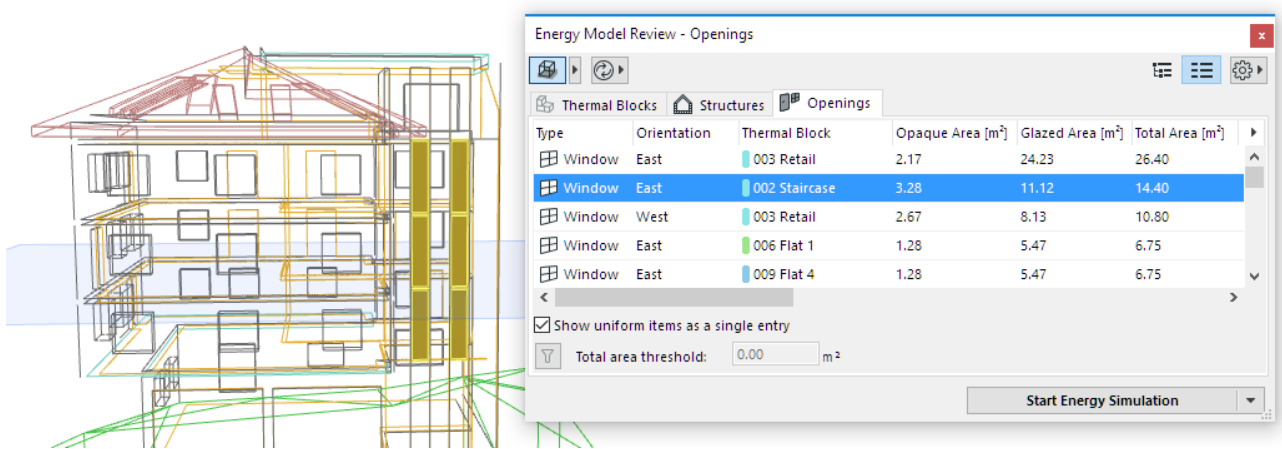
- Thermal Block Assignment
- Orientation and Archicad opening type
- Solar Analysis result

See [Solar Analysis](#).

- Data describing the transparent panels of Openings:
 - Glazed area, frame perimeter
 - Glazing U-value
 - Total solar transmittance (TST): The percentage of incident solar radiation transmitted by an object which includes the Direct Solar Transmission plus the part of the Solar Absorption reradiated inward. TST divided by 100 equals Solar Heat Gain Coefficient (SHGC) or g-value.
 - Direct solar transmittance (DST)
- Data describing the opaque panels and frames of Openings:
 - Opaque area, perimeter, U and Psi-values
 - Infiltration: air permeability of the selected Opening (typically occurs around the perimeter, at the frame to wall connection)
- Shading Device assignments
- Renovation status

Show Uniform Items as a Single Entry

Use this checkbox in the bottom left corner of the Openings page in list view to make every Space boundary which is identical in terms of all Archicad model data types appear in a single Openings List entry, with the sum of their areas on the list.



Total Area Threshold

To filter out opening list entries of negligible area, enter a value in the “Total Area Threshold” field at the bottom of the Openings list. Openings whose total (opaque + glazed) area is smaller than this value will not be listed, and will not be considered by the calculation engine. (If you later lower this area threshold, the filtered-out openings will reappear as applicable.)

Setting a Total Area threshold will help you filter out small areas that are insignificant for energy evaluation, resulting in a more manageable Openings list.

Openings Area Correction

Two columns are available for manually correcting the Opening area: the “Correction - Glazed area” and “Correction - Opaque area.” Enter a positive or negative value in this column. The value of this item in the “Area” column will be adjusted accordingly; this corrected Area value (if positive) will be used by the calculation engine.

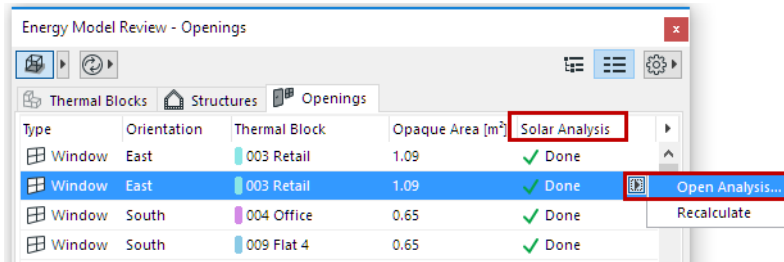
Solar Analysis

The Openings page of the Model Review Palette allows for the individual analysis (listing, editing and visualization) of every transparent space boundary of the Archicad building model. This is necessary to take advantage of the Model-Based Solar Irradiation Study, with which solar gain through each individual transparent element of the building envelope may be precisely determined. This advanced tool enables architects to strategically determine the place of each individual piece of fenestration with respect to the surroundings and the building’s geometry, in order to utilize the benefits yet avoid the unwanted effects of solar gain throughout the year.

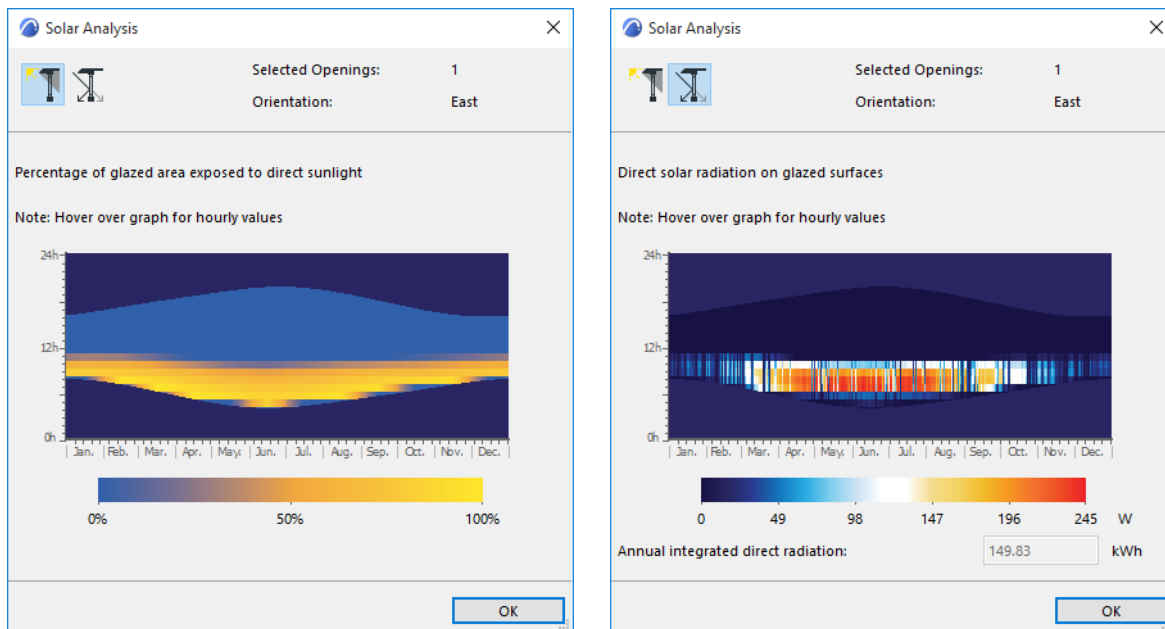
The function considers all Archicad objects and elements that are visible in Building Energy Model view to calculate direct solar irradiation on each glazed opening of the building model. Furthermore, Archicad plant objects act as intelligent shading devices, as their shading capacity is scheduled for the reference year, making it possible to factor in whether the plants used for shading are coniferous or deciduous.

[Watch the Video](#)

Select any Openings List entry, then click **Open Analysis** at Solar Analysis to activate the dialog dedicated to model-based solar irradiation study.



The Solar Analysis dialog has two tab pages. Both display diagrams that show each hour of the calculation reference year, with the days of the year along the x axis and the hours of the day along the y axis.



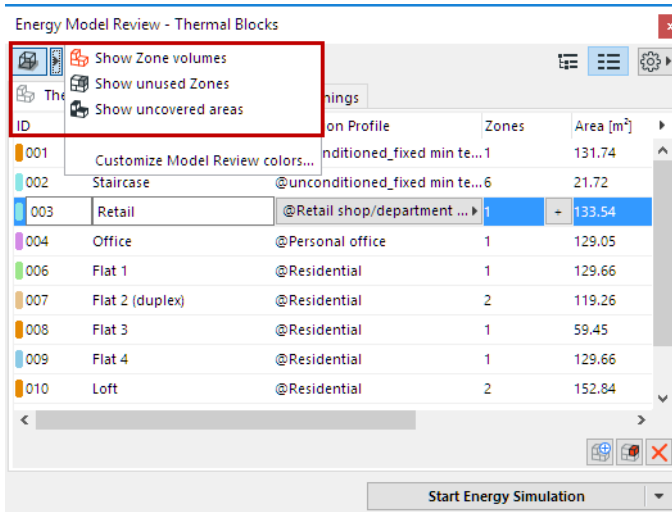
Hover over the “Percentage of glazed area exposed to direct sunlight” diagram to get a precise reading of hourly values, or use the color coding of the diagram to view 100% solar exposure with bright yellow and 0% solar exposure (totally shaded) with blue, while shades of yellow represent times of the year when the Opening in question is only partially irradiated by the sun. This view disregards cloudy days specified in the weather file.

Hover over the “Energy transmission through glazed area” diagram to get a precise reading of hourly energy transmitted through the Opening, or use the color coding of the diagram to view energy transmission intensity: most intense is bright red and 0 is blue, while colors of the gradient between these two extremes represent intermediate energy flow values. This view takes cloudy days specified in the weather file into consideration and displays the resulting “Annual integrated direct radiation value” in kWh.

Note: If multiple entries are selected on the Openings List, or a single entry is selected but the **Show uniform items as a single entry** option is active, then the Solar Analysis dialog will display data that reflect the solar irradiation on all of the selected Openings items, combined.

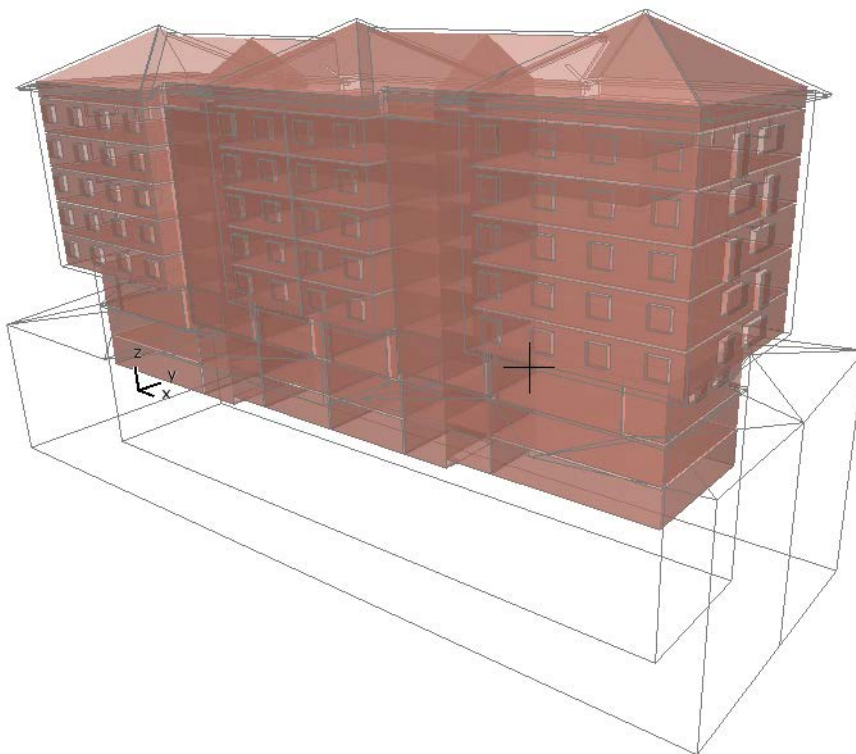
Building Energy Model Visualization

The **Show selected items in 3D** button on the upper left corner of the Energy Model Review palette enables you to visualize the energy model of your project in different ways.



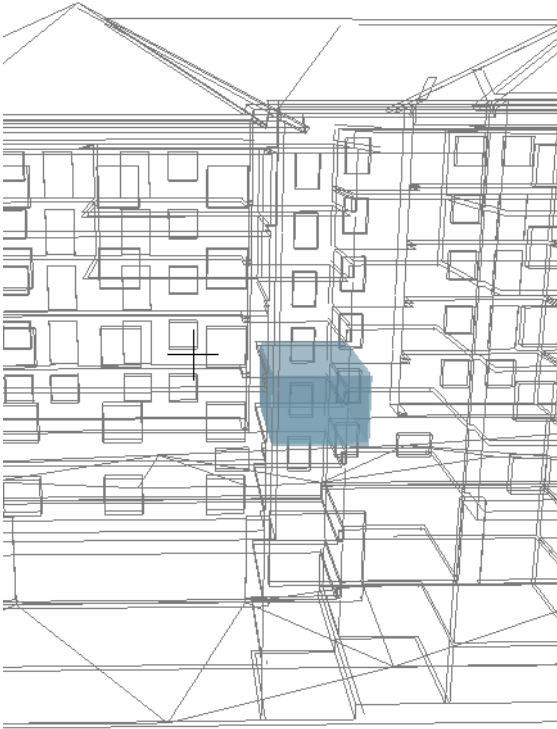
Show Zone Volumes

The Show Zone Volumes function can be used to visualize the internal spaces in 3D.



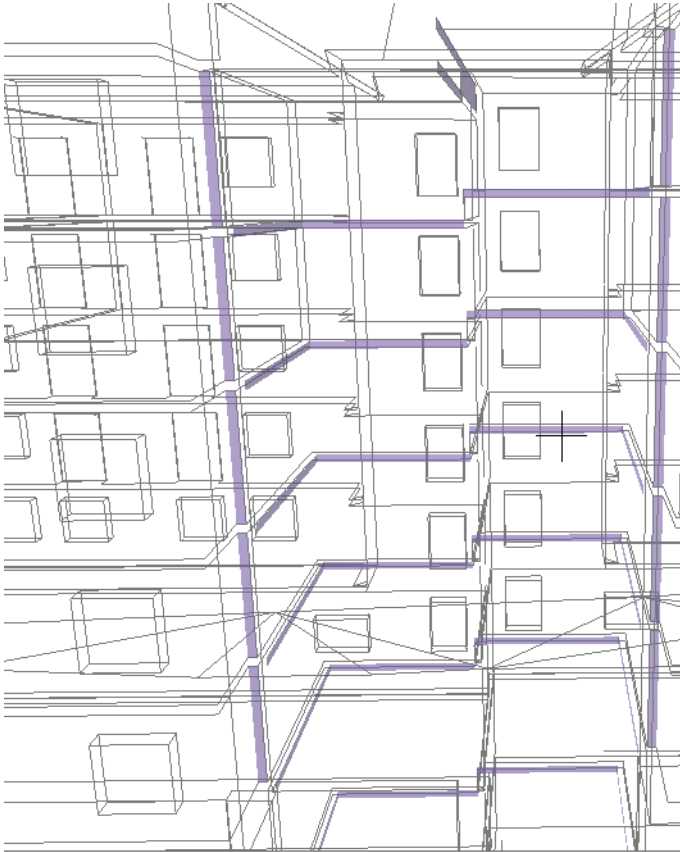
Show Unused Zones

The Show unused Zones function highlights the visible Archicad Zones that do not belong to any Thermal Blocks in the Building Energy Model.



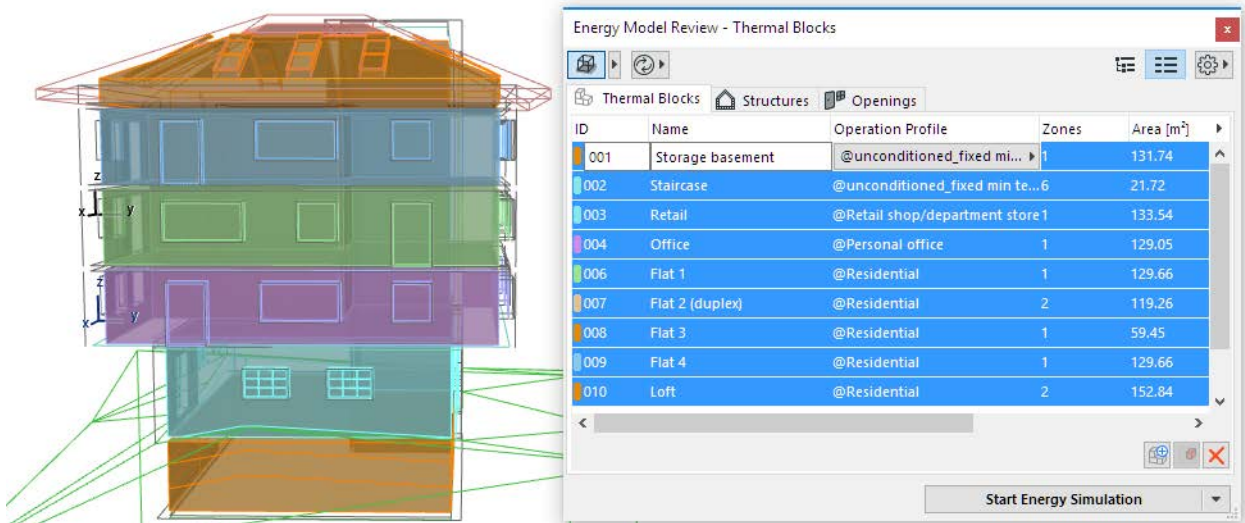
Show Uncovered Areas

The Show Uncovered Areas function highlights the surfaces that do not have space boundaries associated to them. Ideally, all building structure and opening surfaces are covered by space boundaries; so these uncovered areas indicate modeling inaccuracies.

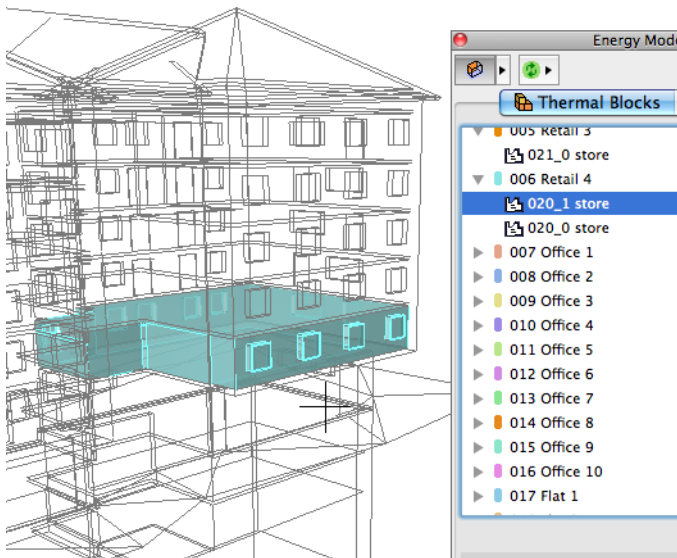
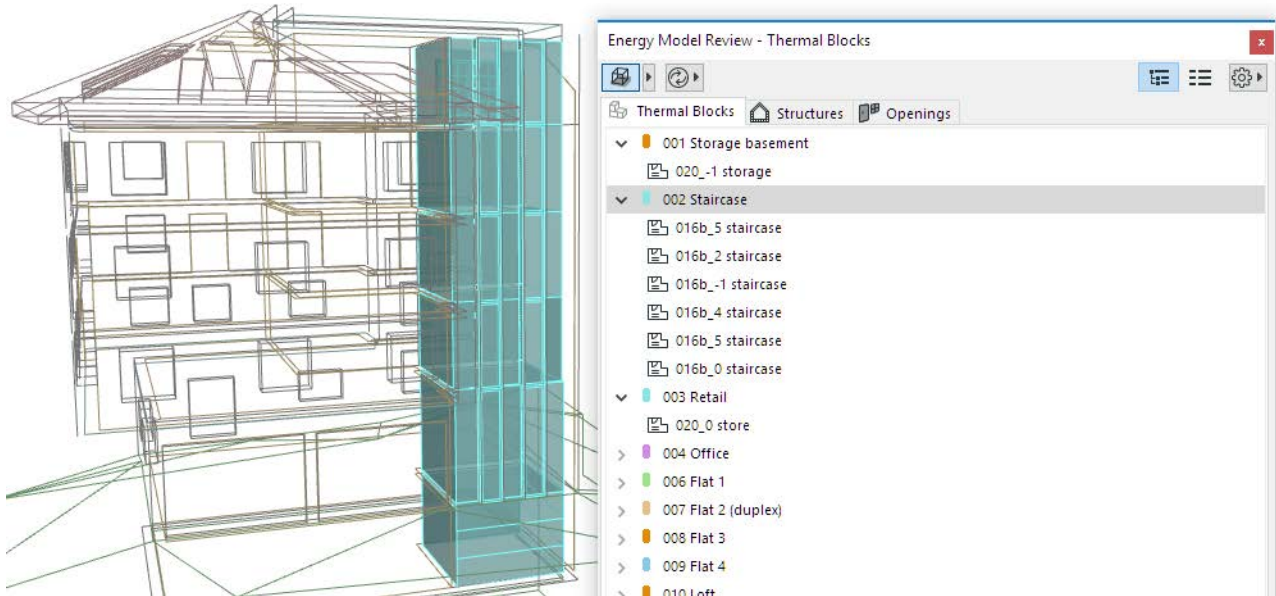


Visualization of Thermal Blocks

In the Energy Model View, the entries that are selected on the Thermal Blocks list are simultaneously highlighted with solid colors on the 3D model.

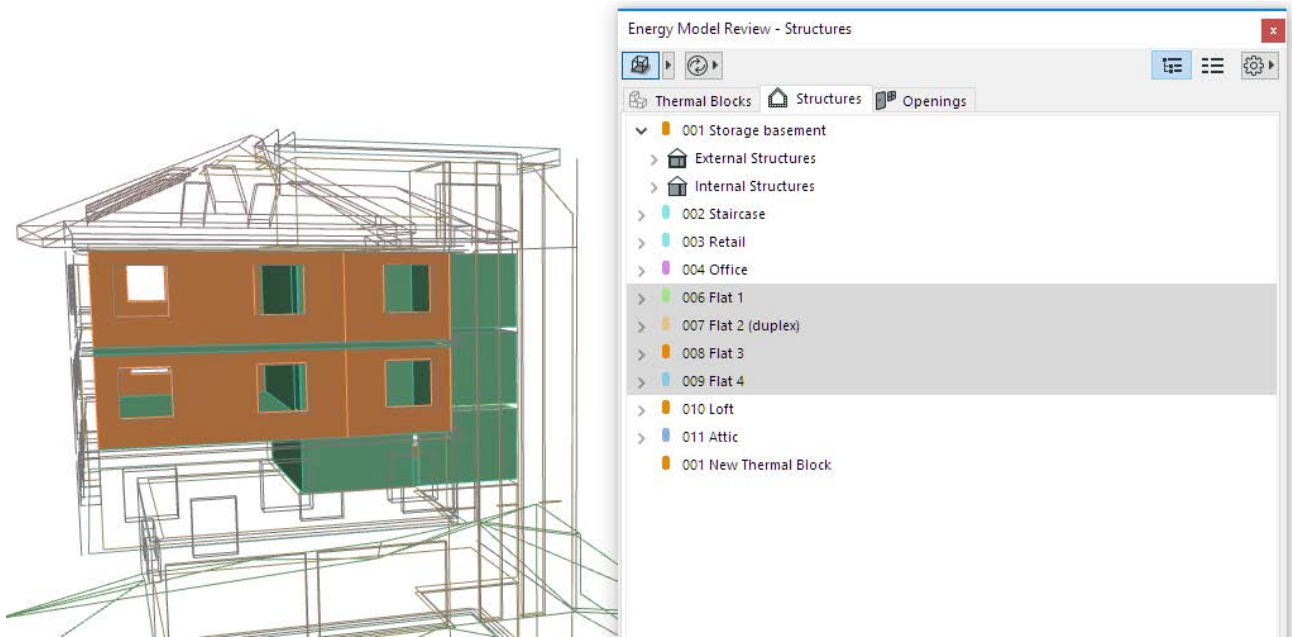


The model elements that are not selected are shown in wireframe while this function is active. In Tree View, it is also possible to select and visualize the zones that make up a thermal block, one by one.



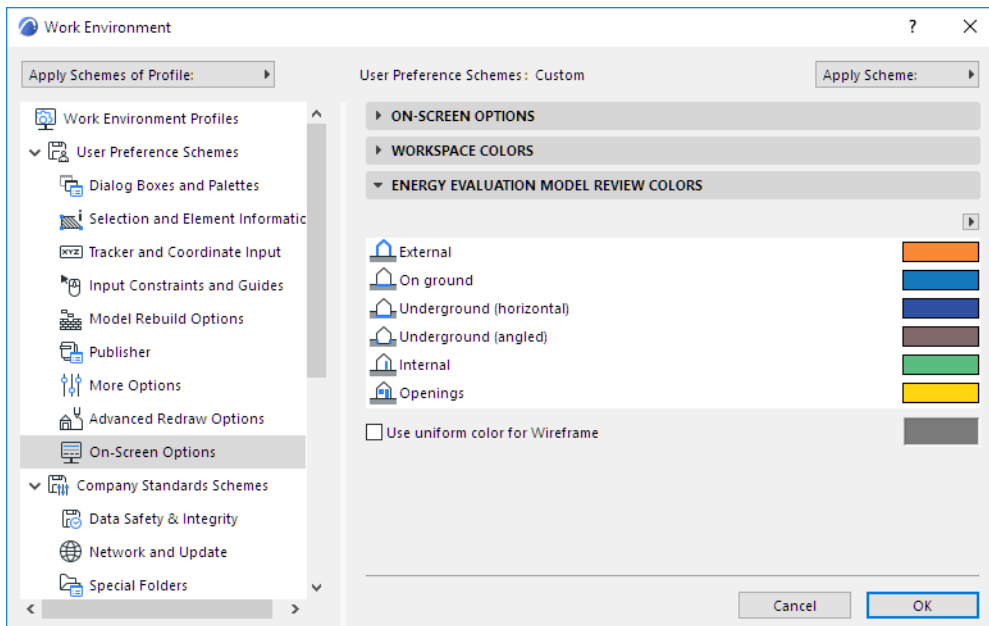
Visualization of Structures

In the Energy Model View, the space boundaries that are selected on the Structures list are displayed on the Archicad building model. Select one or more items from the Structures list, then click “Show selected item in 3D” (the model elements that are not selected are shown in wireframe while this function is active).

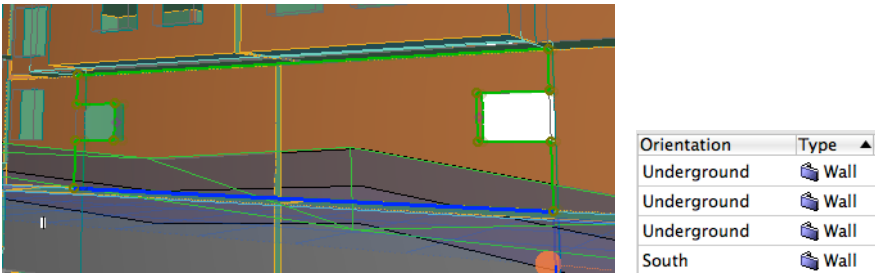


A separate color is assigned to each space boundary category in Energy Model View.

[See *Customize Energy Model Review Colors*.](#)



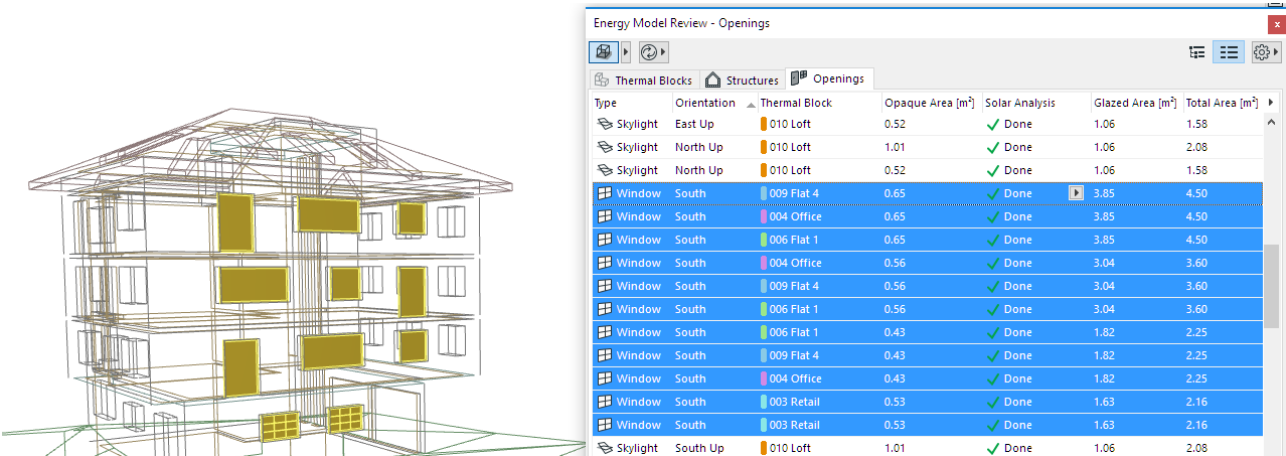
A single Archicad element (e.g. a wall that extends both above- and below-ground) might consist of multiple space boundaries, and therefore might appear as part of several different entries (with different orientations) on the Structures list.



A single Archicad wall element that consists of multiple space boundaries

Visualization of Openings

The Show in 3D button can also be applied to visualize the selected entries of the Openings list.



Additional Data Assignment and Input

In Energy Evaluation, there are two necessary energy calculation input data types in addition to the information that comes directly from the BIM:

- Space boundary (Structures and Openings) property settings
- Auxiliary data assignment and input

These input data are described in the following sections:

[Thermal Block Property Settings \(Energy Evaluation\)](#)

[Structure Property Settings \(Energy Evaluation\)](#)

[Opening Property Settings \(Energy Evaluation\)](#)

[Climate Data \(Energy Evaluation\)](#)

[Environment Settings \(Energy Evaluation\)](#)

[Operation Profiles \(Energy Evaluation\)](#)

[Building Systems \(Energy Evaluation\)](#)

[Energy Source Factors \(Energy Evaluation\)](#)

[Energy Costs \(Energy Evaluation\)](#)

Bring up the Model Review Palette by selecting **Design > Energy Evaluation**. The three tab pages of this palette (Thermal Blocks, Structures and Openings) list data populated by the Automatic model geometry analysis. Additional data beyond the information that comes from the building model must be assigned to the Archicad model data on the Thermal Blocks, Structures, and Openings lists.

Prior to manual editing, the fields of Model Review Palette tab pages that contain additional data are filled with default values. Fine-tune the building energy model by editing this data manually, using the function buttons on the lists.

To modify the editable values for any property, select the property from the list.

In List view, you can edit multiple entries. Select the first entry that needs to be modified. Then add further entries to the selection by moving the pointer with the mouse without releasing the left mouse button, or by clicking on the entries while pressing the CTRL or the SHIFT key. Use the function buttons that appear by the first selected line to modify properties of all selected list entries.

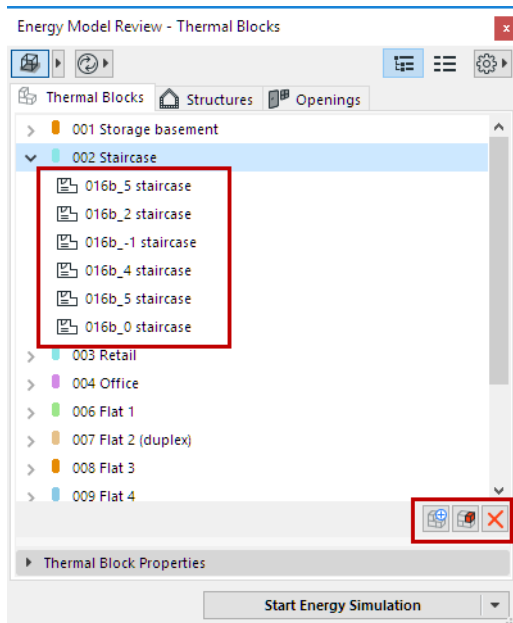
If any of the data on the lists are insufficient for simulation input, the list entry that contains it is automatically highlighted with red, and a warning symbol (exclamation mark in a yellow triangle) appears at the data field in question.

Thermal Block Property Settings (Energy Evaluation)

In order to enable the energy simulation for Energy Evaluation, it is necessary to define three property sets for each thermal block individually. These key thermal block property sets are: Space Zones, Operation Profile and a set of Building Systems.

- **Space Zones** that make up the thermal block.

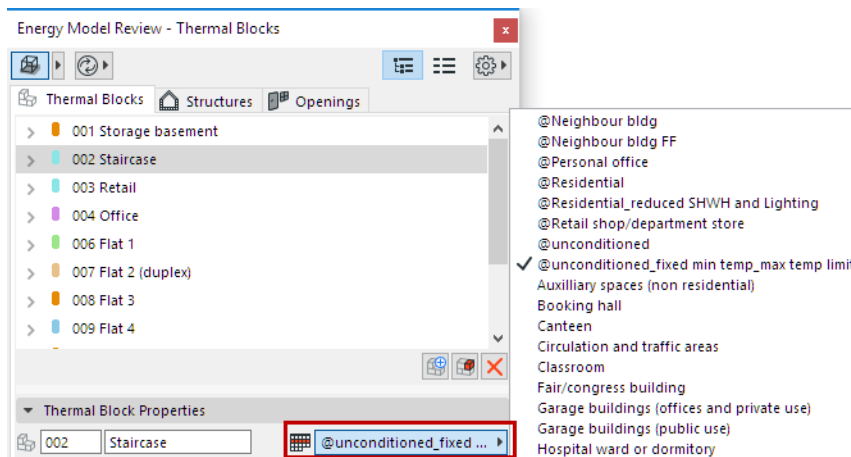
See [Create Thermal Blocks](#).



Use the control buttons below the thermal blocks tree to add or remove Archicad Zones from the selected thermal block, as well as to create a new thermal block.

- An **Operation Profile** that describes how the thermal block will be used.

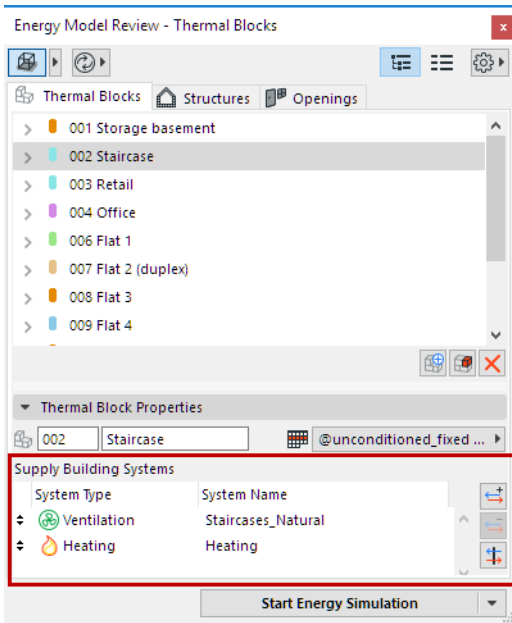
See [Operation Profiles \(Energy Evaluation\)](#).



Click the Assign Operation Profile button to select an operation profile that fits the selected thermal block's function.

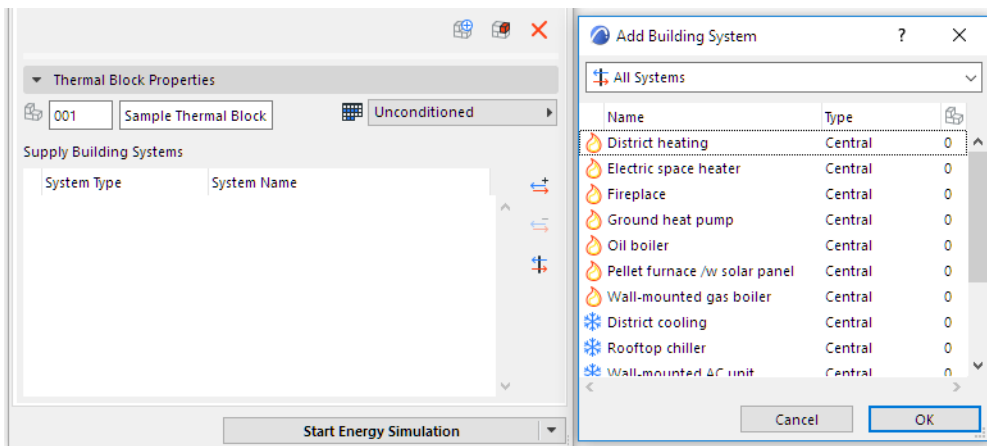
- A set of **Building Systems** that maintain the comfort requirements defined by the Operation Profile.

See [Building Systems \(Energy Evaluation\)](#).



Use the control buttons next to the list of Supply Building Systems to add or remove systems from the list.

Note: It is also possible to assign systems to thermal blocks via the Building Systems dialog.



Note: One thermal block can only be supplied by one heating system configuration, one cooling system configuration and one ventilation system configuration.

Structure Property Settings (Energy Evaluation)

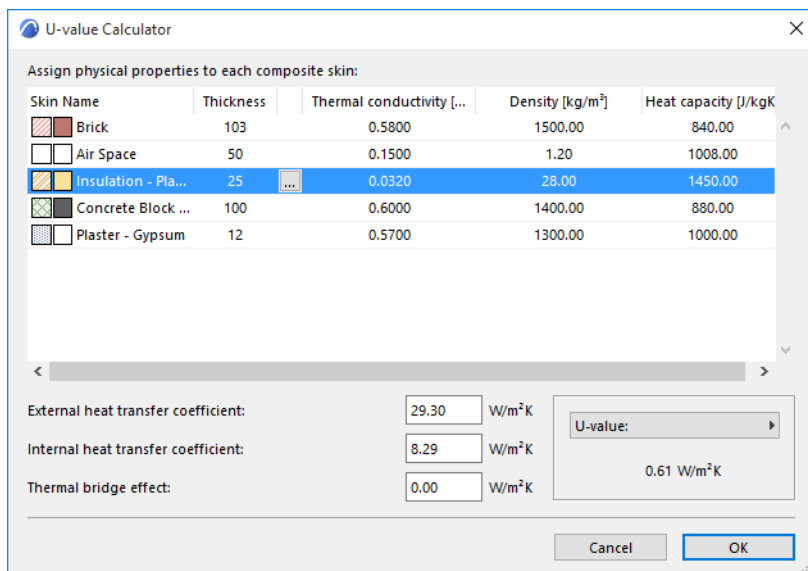
U-Value (R-Value) Calculator

U-value refers to the heat transmission coefficient of the selected structure. Use the U-value Calculator to estimate the thermal physics performance of the structures in your projects, based on the physical properties of the structure's Building Material(s).

Note: For a simpler and faster, but somewhat less accurate method, edit the U-value manually.

See [U-value \(R-value\) Override](#).

Select any structure in the list. In the U-value column, click the three-dot button - with the lock icon set to closed (red) - to bring up the U-value Calculator dialog.



In the U-value Calculator, the skin(s) that make up the selected structure are listed with their relevant properties (Thickness, Thermal conductivity, Density and Heat capacity), each showing a default value.

Note: These values derive from the Physics Properties panel of Archicad's Building Materials dialog box. You can edit any of these values manually.

Depending on local convention, you may prefer to use the R value (Thermal Resistance Coefficient), which is the inverse of the U value. To list and display R values instead of U values (both here and in the final Evaluation Report), click the U-value pop-up at the bottom right of this dialog box, and choose R-value.

External and Internal heat transfer coefficients, and the Thermal bridge effect related to structural surface area are shown at the bottom left of the U-Value (R-Value) Calculator dialog box. These data, in addition to the material properties, are necessary for the calculation.

The Calculated U-value is displayed in the bottom right corner of the panel.

Note: The Calculated U-value shown in this dialog box is based on stationary building component performance data. This kind of data is listed under Key Values/Heat Transfer Coefficients in the Energy Performance Evaluation report. However, Energy Evaluation uses a more accurate, dynamic calculation algorithm to calculate hourly heat transmission through the building envelope structures, when simulating the building energy balance for the reference year. The result of this dynamic analysis is the basis of the Energy Consumption, Carbon Footprint and Monthly Energy Balance data displayed on the Energy Evaluation Report.

U-value (R-value) Calculation Algorithm

The U-value Calculator calculates the average heat transmission coefficient of Building Materials and composite structures, based on a stationary algorithm that is used by most national standards.

U : Heat transfer coefficient (static value)

$$U = \frac{1}{\frac{1}{h_i} + \sum \frac{d}{\lambda} + \frac{1}{h_e}}$$

h_i : internal heat transfer coefficient

h_e : External heat transfer coefficient

d : skin thickness

λ : thermal conductivity

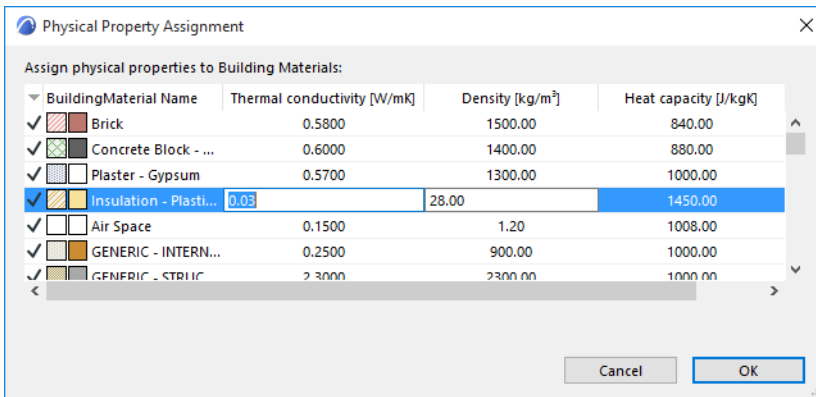
To include the effects of thermal bridges, **delta U-values** are added to the average U-values of structure group entries. The magnitudes of **external and internal heat transfer coefficients** and the delta U-value depend on the position of the evaluated structure relative to the thermal current. Default settings are offered within Energy Evaluation. However, it is advisable to review and manually override these predefined values if the structural situation demands, or if the standards for the project location are different.

Physical Property Assignment

View the skins listed in the U-value calculator. While the fill type, name and thickness of each skin is derived from the element's settings in Archicad, the three properties to the right of the vertical line - Thermal conductivity, Density and Heat capacity - can be modified by you, if the default values here do not correspond to your actual project structures.

To change the Thermal conductivity, Density and/or Heat capacity value of any skin, select it from the list in the U-value Calculator.

Click the button with the three dots to access the **Physical Property Assignment** dialog box.



The **Physical Property Assignment** dialog box lists all Building Materials defined for this project. Those Building Materials actually used in this project are shown here with a checkmark in the far left column. Each listed fill is shown with the relevant physics properties that are assigned to it.

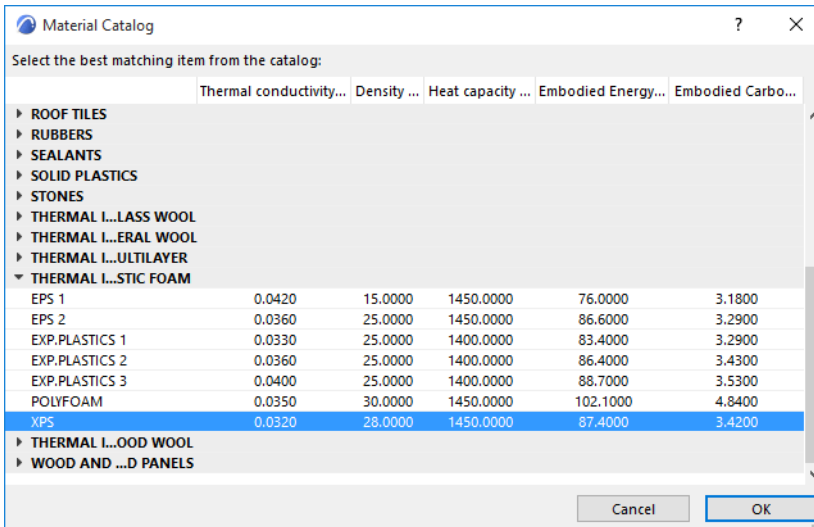
Note: These values derive from the Physical Properties panel of the Building Materials dialog box (**Options > Element Attributes > Building Materials**).

The Building Material you selected in the U-value calculator is highlighted in the list. Here you can edit any value, either directly or by choosing defined values from the **Material Catalog**.

Any change made here will automatically be reflected in all project elements that include the edited fill.

Material Catalog

Materials in Energy Evaluation represent physical properties (thermal conductivity, density, heat capacity, embodied energy, and embodied carbon). In the Physical Property Assignment dialog box, highlight a row and click the right-side Catalog button: this brings up the Energy Evaluation Material Catalog.



The Material Catalog is an extensive database integrated within Energy Evaluation that contains building material information relevant for the energy calculations (thermal conductivity, density, heat capacity, embodied energy, and embodied carbon). The building materials are grouped in main categories with drop-down detailed lists to enable easy access and quick selection.

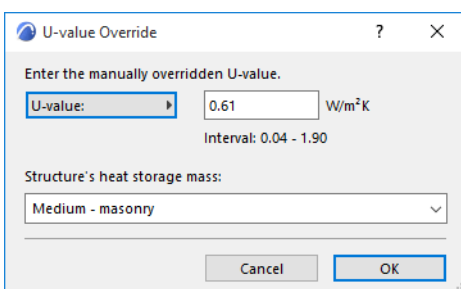
Select a material here whose physical properties correspond to your needs. When you click OK in the Material Catalog, the thermal properties of the selected material are assigned to the Building Material you selected in Physical Property Assignment.

This way, the calculation engine obtains the physical data required for the thermal evaluation without any numeric input from you.

U-value (R-value) Override

It is possible to enter a U-value (or R-value) manually, instead of using the U-value calculator and Physical Property Assignment functions. Omitting the Physical Property Assignment greatly shortens the time needed to evaluate a project, but leads to somewhat less accurate results.

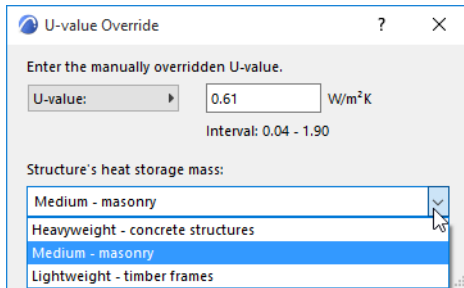
Select any structure in the list. In the U-value column, click the three-dot button - with the lock icon set to open (white) - to bring up the U-value Override dialog.



Click the pop-up to alternate between U-value and R-value as needed. Simply enter the desired value in the editable field.

Structure's Heat Storage Mass

Use this pop-up (also in the U-Value Override dialog box) to select the thermal mass (Lightweight, Medium or Heavyweight) of the selected structure, instead of using the materials coming from the building model directly.



The threshold values for these categories are:

- heavy-weight: > 400 kg/floor area
- medium: 250 – 400 kg/floor area
- light-weight: < 250 kg/floor area

Within Energy Evaluation, these predefined settings are linked to mean density values. These density values are multiplied by the total volume of the interior structures, providing the numeric Internal heat storage mass value to the calculation engine.

Infiltration Settings

Modify the default Infiltration (unit: l/s,m2) of the selected Structures List entry to precisely define its air permeability (0.6 l/s,m2 is considered as low, while 1.6 l/s,m2 is considered as high infiltration).

The Energy Evaluation function of Archicad not only simulates the effect of infiltration on the hourly energy balance, but also displays the total Air Leakage in ACH (air change per hour), in the Key Values section of the PDF Energy Performance Report.

Solar Absorptance

The Surface material property determines the Solar Absorptance properties of the selected Structure. Modify the default Solar Absorptance (unit:%) of the selected Structures List entry to precisely define its capacity to absorb solar energy.

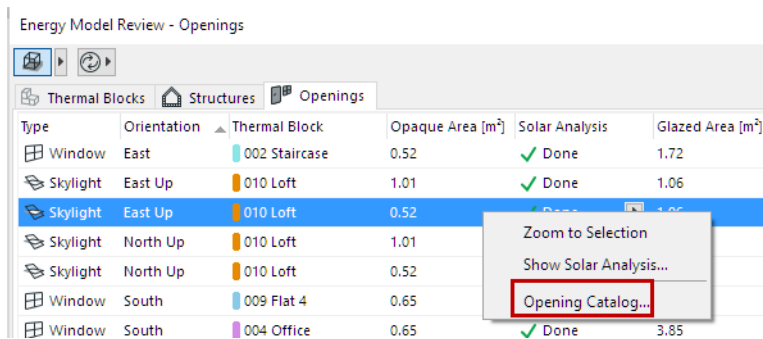
Opening Property Settings (Energy Evaluation)

The Openings tab page of the Model Review dialog box lists data for all of the Openings on the building shell and on internal thermal block boundaries.

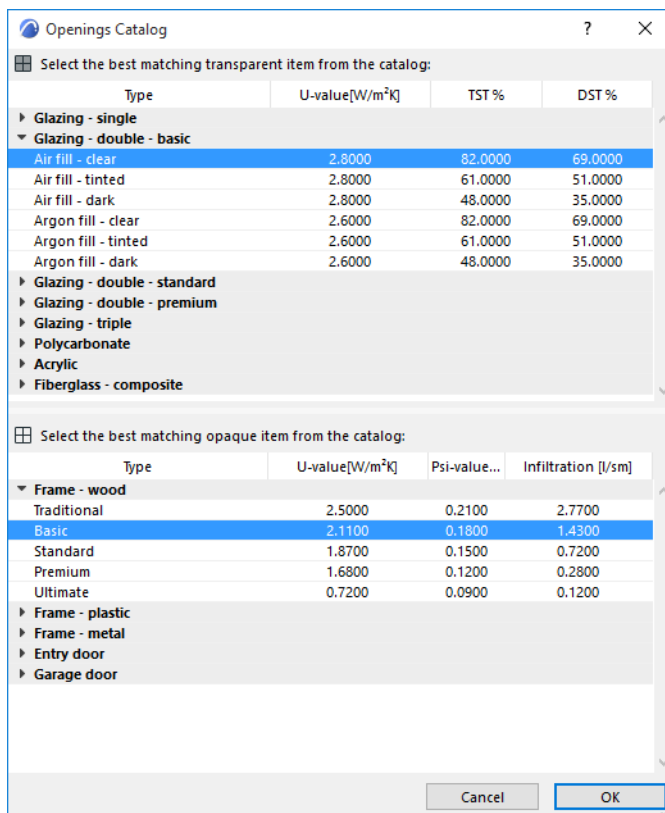
To change the physical properties of the selected Openings List item, either edit the values manually, or use the predefined value sets from the Openings Catalog.

Openings Catalog

To access the Openings catalog, select an item on the Openings tab page and right-click to bring up the context menu:



The Openings Catalog is an extensive database of building physics information relevant for the energy calculations. Transparent and opaque components of Openings are listed separately:



- **Transparent** items are listed with their U-values, Total Solar Transmissions and Direct Solar Transmissions;

- **Opaque** items are listed with their U-values, Psi-values (the linear thermal transmittance coefficient in W/m²K that are used to factor in the effect of the thermal bridges that occur at the frame-to-wall joint around Openings) and linear Infiltration properties.

Shading Devices

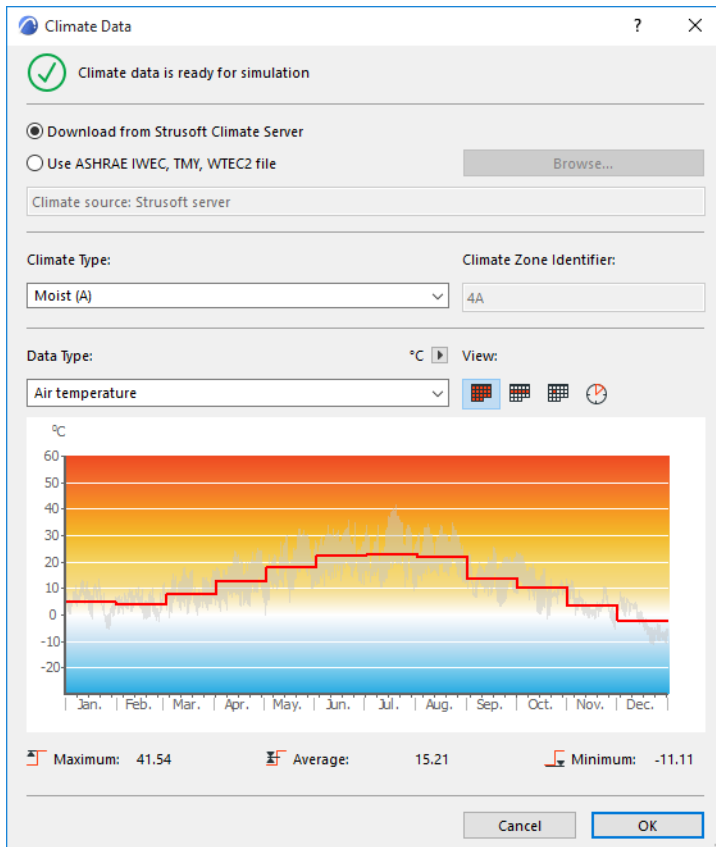
Click the arrow on the Shading Device item to apply a shading device, with its predefined physical properties, to the selected entry.

There are two types of Shading Devices on the list:

- Devices whose shading effect is additional to the effect of the model-based Solar Analysis (None, Sunscreen 10%, Sunscreen 40%, Sunscreen 60%, Sunscreen 80%, External Blind, External Louver)
- Devices that disable the effect of model-based Solar Analysis (Canopy, Canopy And Side-Fins, Solar Analysis Off). These settings are mostly used for analytic (scientific) software functionality tests. Users are advised to use Shading Devices from the previous group on architectural projects.

Climate Data (Energy Evaluation)

Open this dialog box from the Model Review Palette's pop-up at the top right; or from **Design > Energy Evaluation > Climate Data**; or from the **Environment Settings** dialog box.



Climate Data Source

Choose an option for the source of the climate data to be used for Energy Evaluation.

- Download from StruSoft Climate Server. All StruSoft climate data are created from NCEP Reanalysis data provided by the NOAA-CIRES Climate Diagnostics Center, Boulder, Colorado, USA, from their website at <http://www.cdc.noaa.gov/>.
- Use ASHRAE IWECC, TMY, WTEC2 file: Click Browse to select and import data from analytic weather files obtained from an external source (from open source e.g. <https://energyplus.net/weather>, or purchased)

Note: Use analytic (“ideal year”) weather data whenever possible, as it usually represents a given location's climate better than climate data recorded in any particular year.

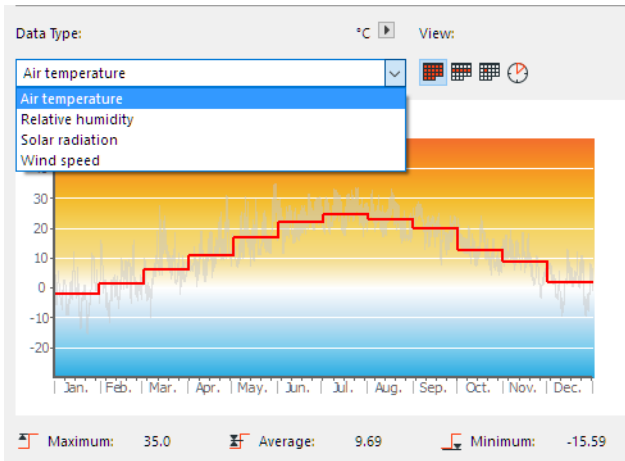
Once the weather data are assigned to a certain project, they are stored in the Archicad Cache Library, so the data remain available when opening another Archicad project later, even if the computer is not online.

Climate Type

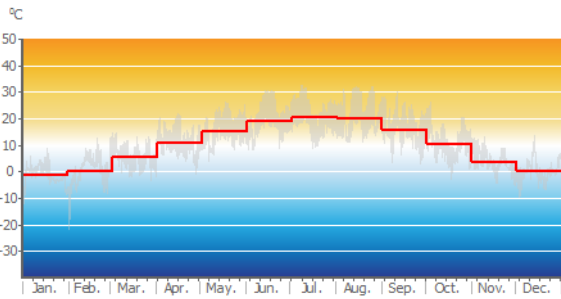
It is possible to assign a Climate Type that best describes that of the building location: Moist, Dry or Marine. This classification is necessary for compliance with certain North American Building Energy Efficiency Calculation Standards, but it has no effect on the calculation result.

View Climate Data

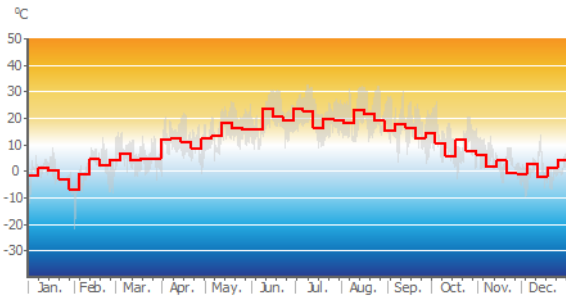
The chart below graphically illustrates climate fluctuation over the course of one year. Choose the climate data type that you want to see in the chart: either Air Temperature, Relative humidity, Solar radiation or Wind speed.



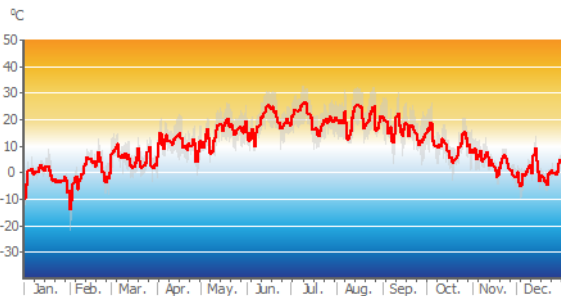
Use the four icons at right to choose the level of detail you want to view: by month, day, week or hour.



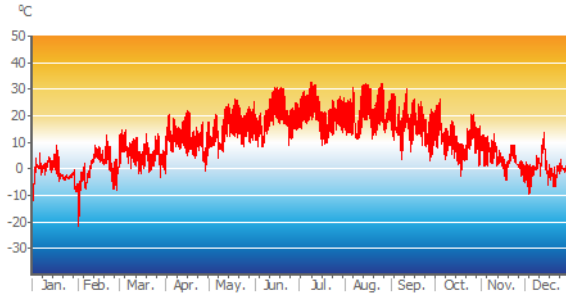
Air Temp monthly



Air Temp weekly

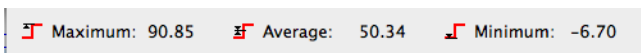


Air Temp daily

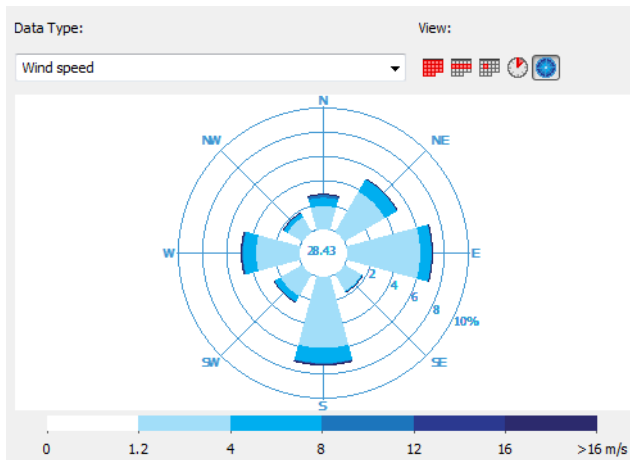


Air Temp hourly

Below the graph, the maximum, average and minimum values for the chosen climate data type are listed.



If Wind speed is selected as visualized climate data type, an additional icon is also available, which displays the wind data in wind-rose form.



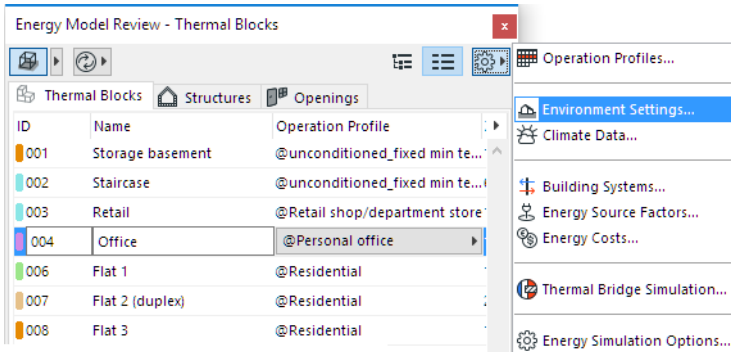
The polar diagram represents the 8 main directions. The length of each segment represents how often (as a percentage, over the course of a year) the wind blows from the given direction.

Each color within a segment represents a wind speed range (as a percentage of the total wind from that direction), in accordance with the color scheme below.

The value at the center of the wind-rose (the white area) represents the percentage of time that the wind is nonexistent or negligible.

Environment Settings (Energy Evaluation)

Open this dialog box from the Model Review Palette’s pop-up at the top right; or from **Design > Energy Evaluation > Environment Settings**.



The Environment Settings dialog contains:

- Settings: Location and Climate, Grade Level, Soil Type, Surroundings
- Links to other dialogs: Location Settings, Climate Data, Surface Heat Transfer, Wind Protection, Horizontal Shading

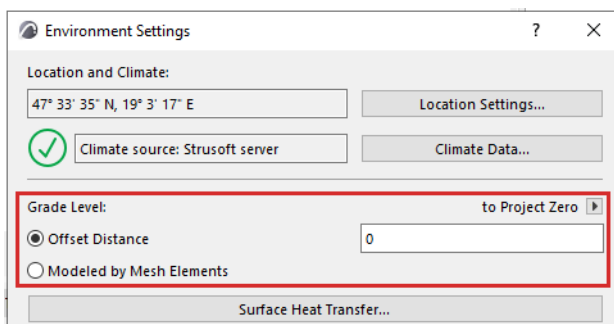
Location Settings

Energy Evaluation considers the geographic location of your building when obtaining Climate Data from the StruSoft Climate Server. Use the Location Settings button on the Environment Settings panel to access this Archicad dialog.

[See Location Settings.](#)

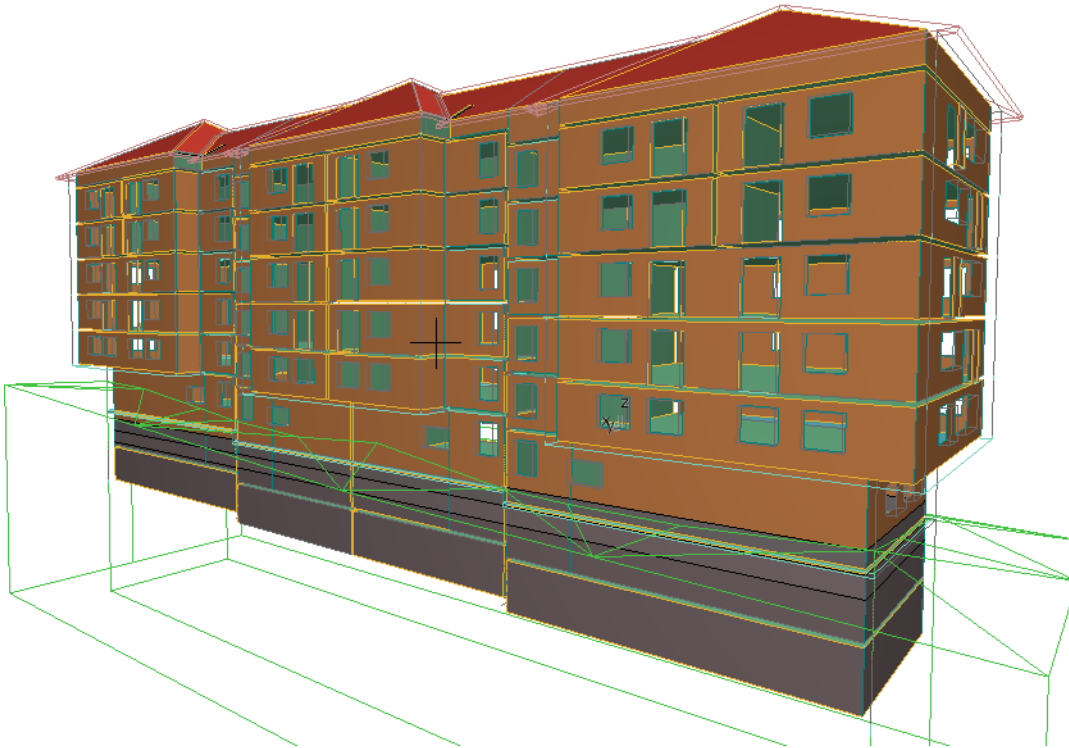
Grade Level

This control is in the Environment Settings dialog box.

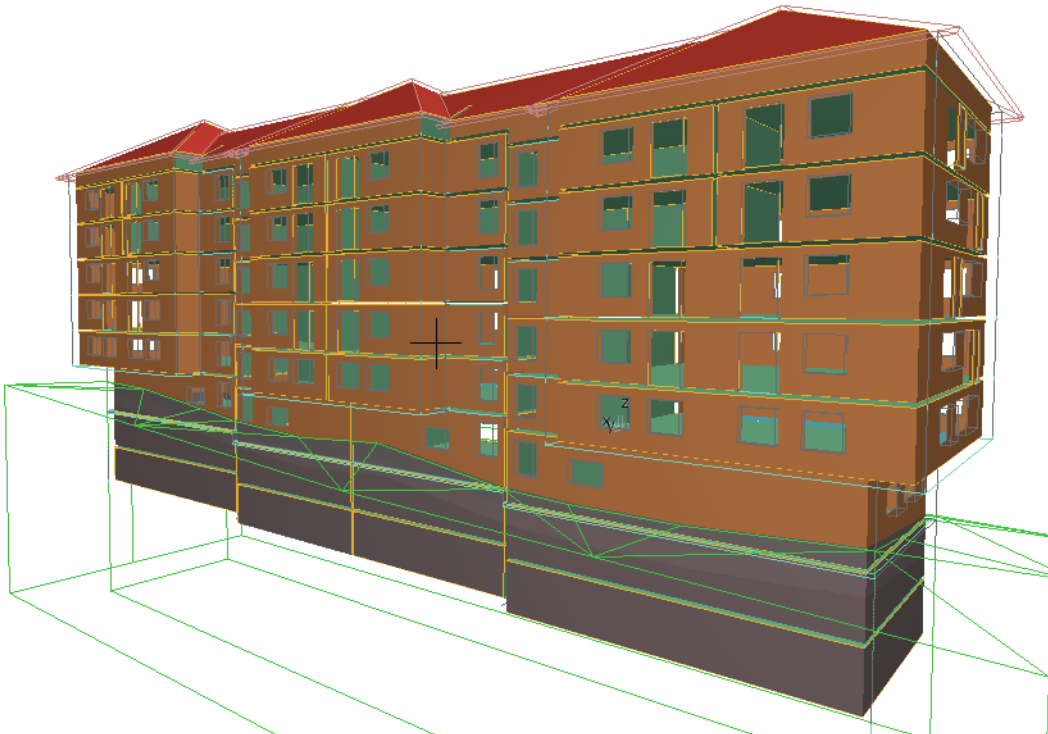


The connection of the ground with the building can be modeled in two ways in Archicad for Energy Evaluation:

- **Offset distance:** the distance by which the pavement is offset from either Project Zero or a predefined Reference level.

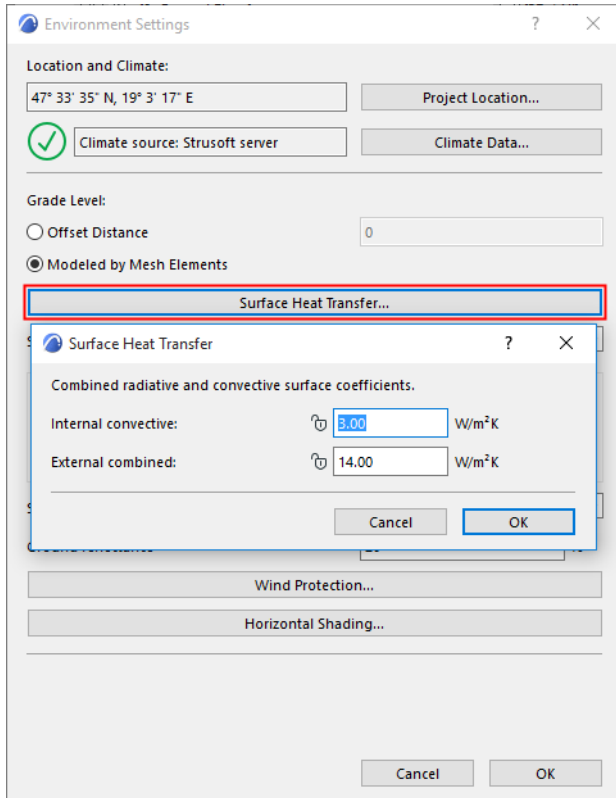


- **Modeled by Mesh elements:** use the Mesh tool to model the site around the building, if it is not completely level, for better energy modeling accuracy of structures in contact with the ground.



Surface Heat Transfer

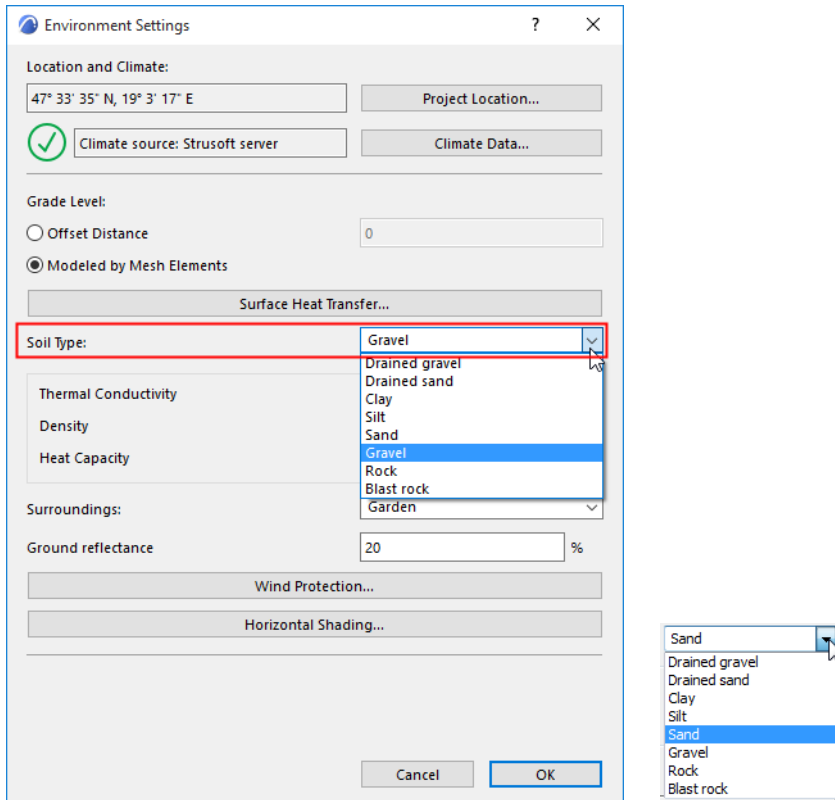
This dialog is used to define Internal and External combined heat transfer coefficients for all opaque structures in the project. While the surface coefficients entered on the U-Value Calculator influence only the stationary thermal performance calculations on the level of individual structures, the global settings on the Surface Heat Transfer panel are used for the dynamic energy balance simulation.



Users are advised to change the default global surface coefficient values only if local standards specifically require the use of other values.

Soil Type

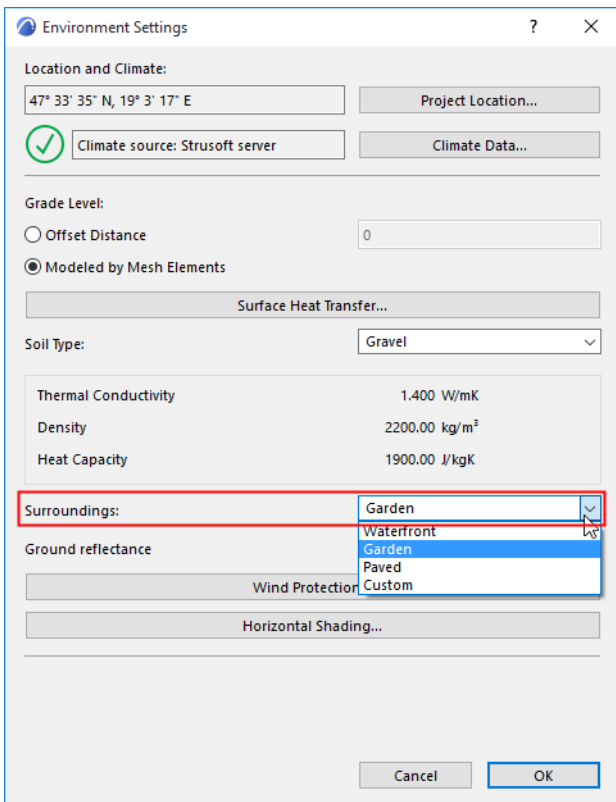
From the pop-up, choose the option that most closely describes the soil type of the building location. This setting serves as the basis for calculating heat flow through structures in contact with the ground.



The applicable Thermal Conductivity, Density and Heat Capacity values are then displayed below your chosen soil type.

Surroundings

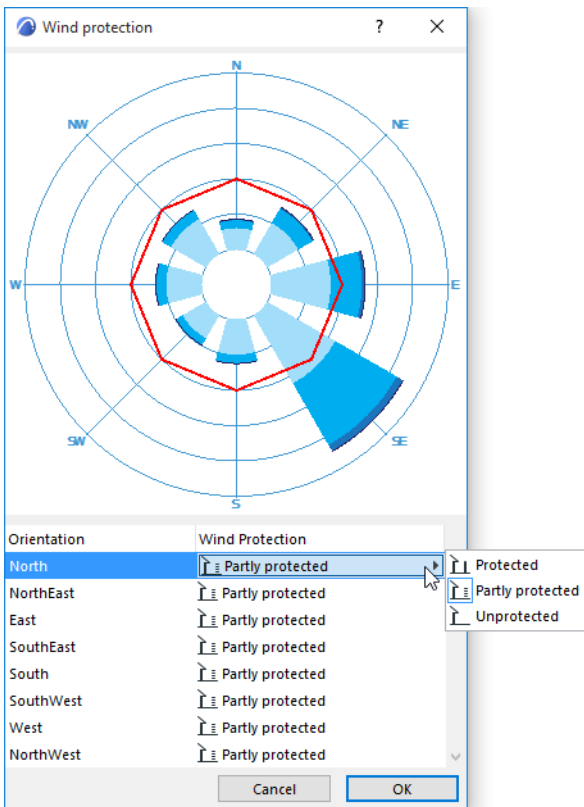
Choose the Surroundings option that best describes the environment of your building: Waterfront, Garden, Paved or Custom.



This setting is used when calculating the effect of indirect solar irradiation.

Wind Protection

From the Environment Settings dialog box, click **Wind Protection** to open the dialog box of the same name.



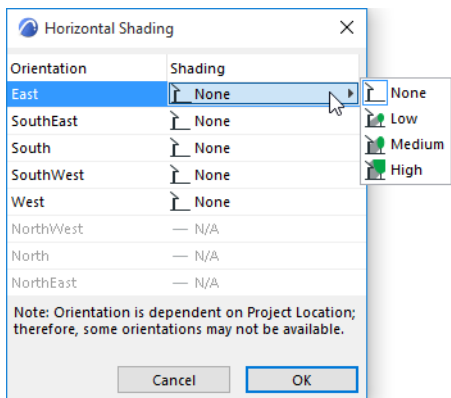
The polar diagram represents the 8 main directions. The length of each segment represents how often (as a percentage, over the course of a year) the wind blows from the given direction, and how fast.

Here, for each orientation of the building, choose the appropriate Wind Protection level: Protected, Partly Protected or Unprotected. For each orientation, a point is placed on the graph representing wind protection (the further out the point, the higher the protection factor), and the points are connected by a red line.

Horizontal Shading

The model-based Solar Analysis function of Energy Evaluation in Archicad only works on the transparent elements of the building shell. However, it does not automatically determine the extent to which shadows are cast by external objects on the opaque elements of the building envelope.

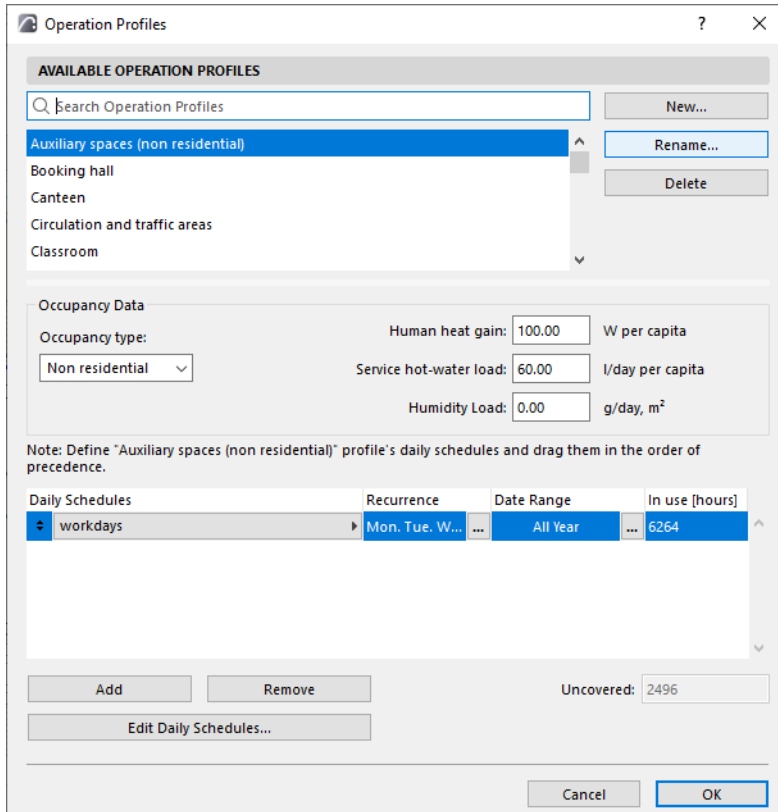
Use the Horizontal Shading button to activate a separate dialog with a list of those sides of the building that receive the sunlight (the list of Orientations can vary depending on project location).



For each orientation, choose the options that best describe the amount of shading resulting from external objects (select an option - None, Low, Medium or High).

Operation Profiles (Energy Evaluation)

Open this dialog box from the Model Review Palette's pop-up at the top right, from **Design > Energy Evaluation > Operation Profile**, or with the **Options > Element Attributes > Operation Profiles** command.



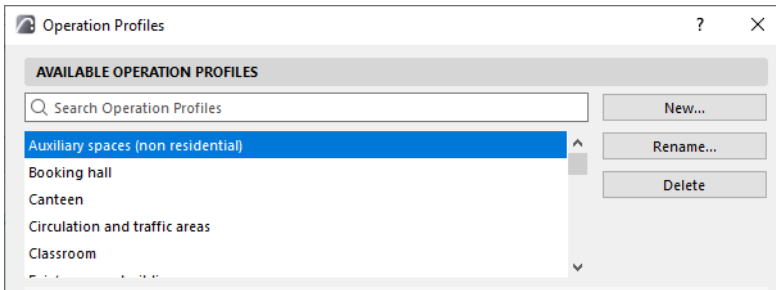
The Operation Profile is an Archicad attribute. A separate Operation Profile can be assigned to each Thermal Block.

Each Operation Profile is associated with a daily schedule comprising the following data, by hour, for a full year (8760 hours total):

- Required Internal temperature range
- Human heat gain
- Hot water needs
- Humidity loads

Available Operation Profiles

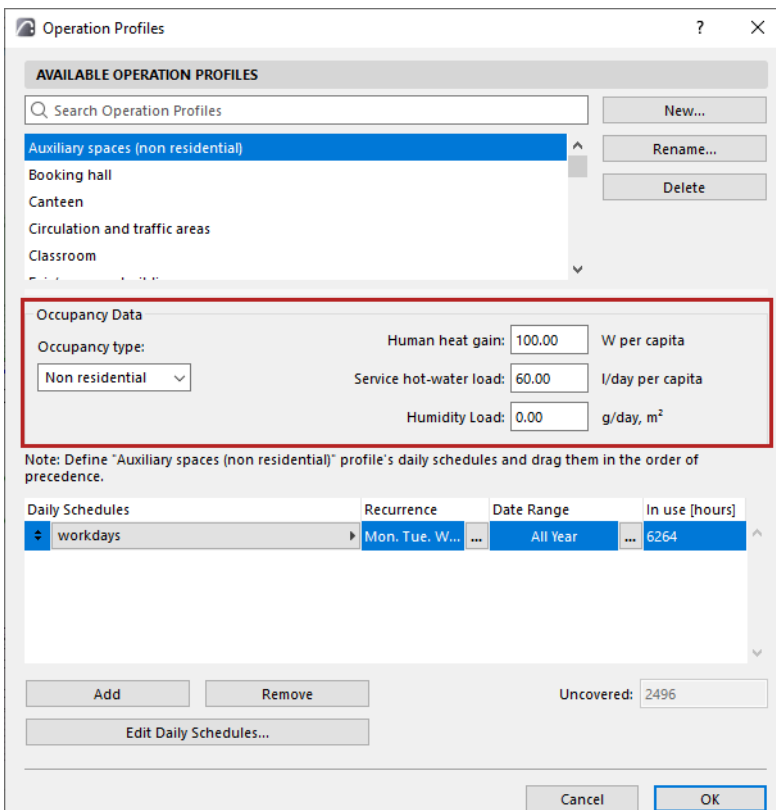
Pick the operation profile that best fits the selected thermal block’s function from the list of available defaults. These default values reflect the Operation Profile specifications of the DIN 18599 Standard - Energy Efficiency of Buildings.



It is also possible to rename or to create custom Operation Profiles with the Energy Evaluation function within Archicad, and to add them to the list of Available Operation Profiles. You can also edit them as needed, by changing the values in the daily schedule list item they reference.

Occupancy Data

Below, view the Occupancy Data of the selected Operation Profile.



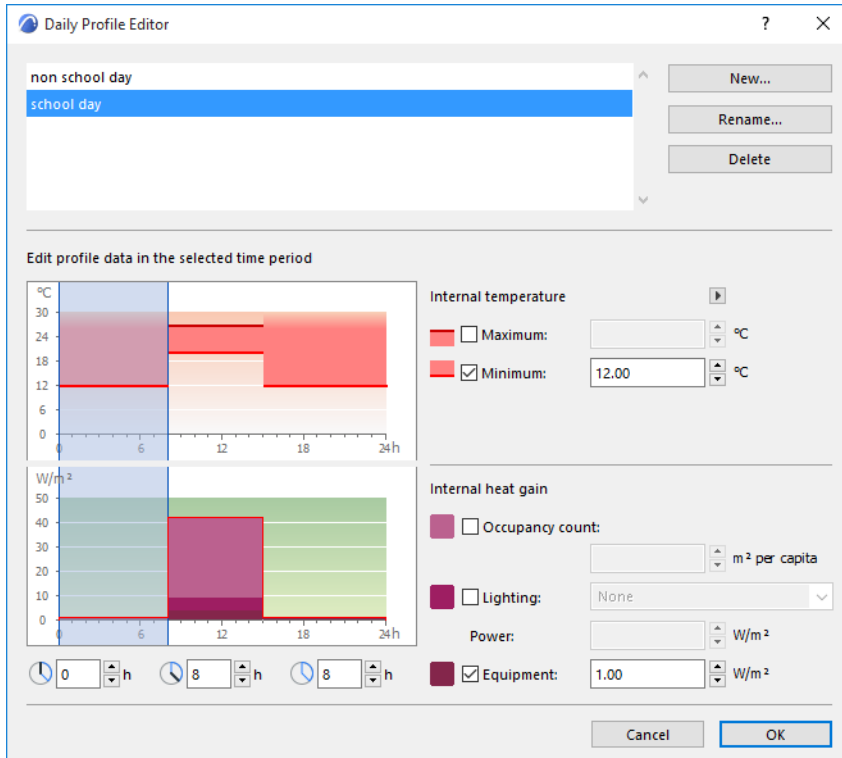
Change any of these values if needed to better reflect the thermal block’s function.

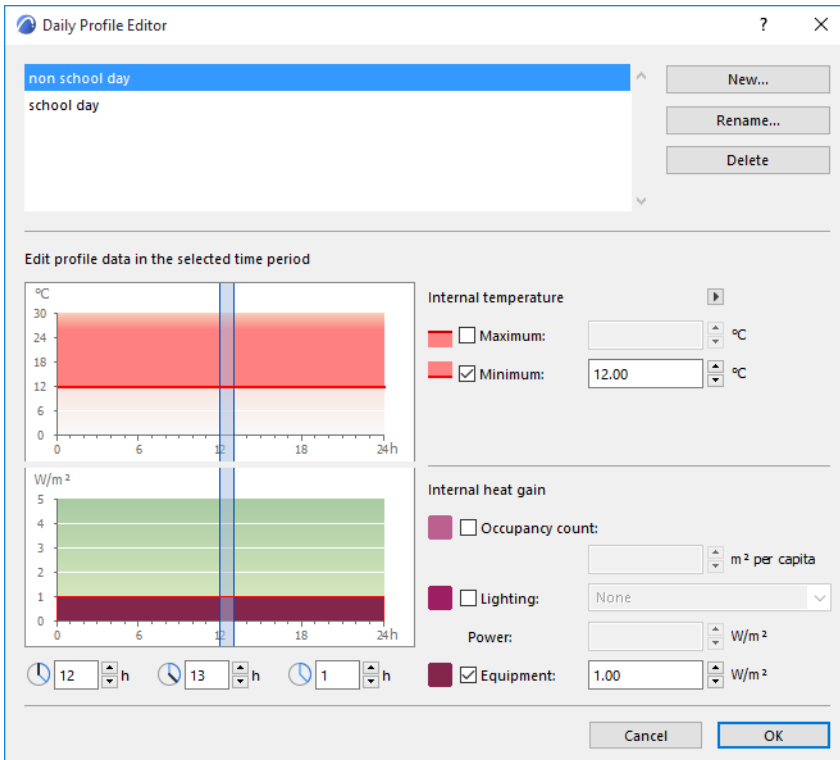
- **Occupancy type:** either Residential or Non-residential
- **Human heat gain:** the amount of heat produced by the human bodies in the building (W/capita)
- **Service hot-water load:** amount of hot water necessary per person, as associated to the selected building function (l/day, per capita)

- **Humidity Load:** the amount of water vapor that gets into the internal air as a result of the building's operation (l/day)

Daily Profile Editor

Select "Edit Daily Schedules" to open the Daily Profile Editor. This dialog displays two daily occupation profile graphs with corresponding key values to the right of each graph. These controls are used to specify the required Internal temperature range as well as Occupancy count, Lighting power density (LPD, in W/m²) and Equipment power density (LPD, in W/m²), which will drive the Internal heat gains.





Theoretically, it is possible to define different operation conditions for each hour of the reference year with the Daily Profile Editor.

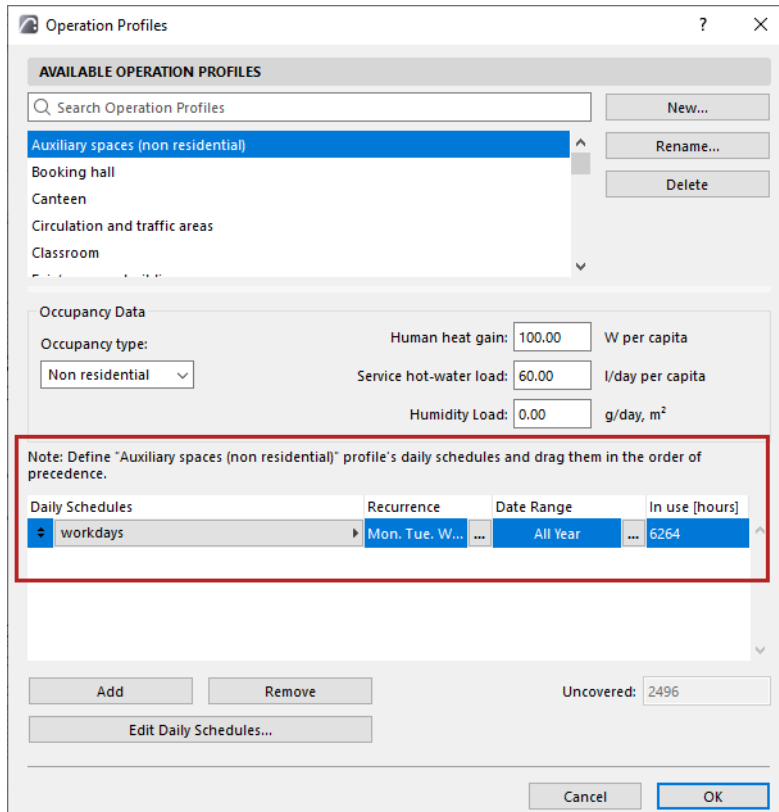
Internal temperature: Set the hourly allowed internal air temperature range (maximum and/or minimum) during the day

Internal heat gain: Define the factors that result in energy emission (internal heat gain) during the day, per m² of internal floor area (W/m²)

- Occupancy count: Refer to local regulations to enter a value that describes the density of human occupation for the selected operation.
- Lighting: Check the Lighting box, then select a lighting type from the dropdown list. The default Power value for the selected Interior Lighting type appears in the Power field. You can fine-tune this value manually if the lighting design of the project is available at the time of the evaluation.
- Equipment: Refer to local regulations to enter a value that describes the density of appliances (E.g.: TVs, Computers, servers) for the selected operation.

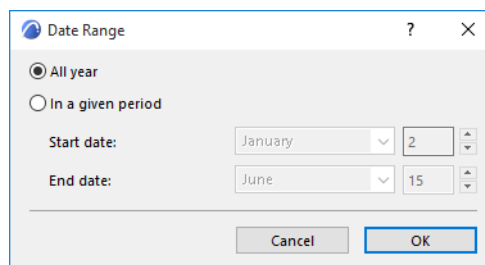
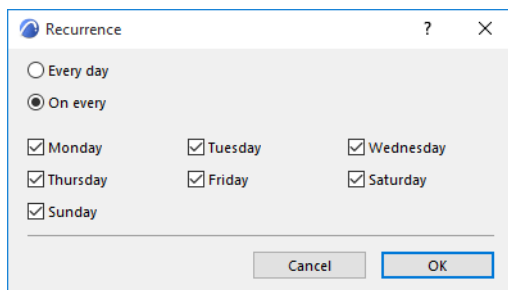
Yearly Operation Scheduler

Since these data will vary by day of the week and over the course of the year, the Operation Profile consists of any number of daily schedules, which correspond to specific date ranges and days of the week.



For each daily schedule, you can define the following characteristics:

- Recurrence: days of the week when the particular schedule is in effect
- Date Range: periods of the simulation year during which the schedule occurs



Recurrence and Date Range define the **In use hours** (the total number of hours during which the selected Daily Schedule is in effect).

If the **Uncovered hours** counter on the bottom right corner of the Yearly Operation Scheduler shows any number but 0, it means that further modifications in Recurrence and Date Range are necessary because there are some hours of the reference year without operation schedule assigned to them.

If the **Overlapped hours** counter on the bottom right corner of the Yearly Operation Scheduler shows any number except 0, it means that further modifications in Recurrence and Date Range are necessary because there are some hours of the reference year with more than one operation schedules assigned to them. Click the yellow warning tag for details on the overlapping profiles.

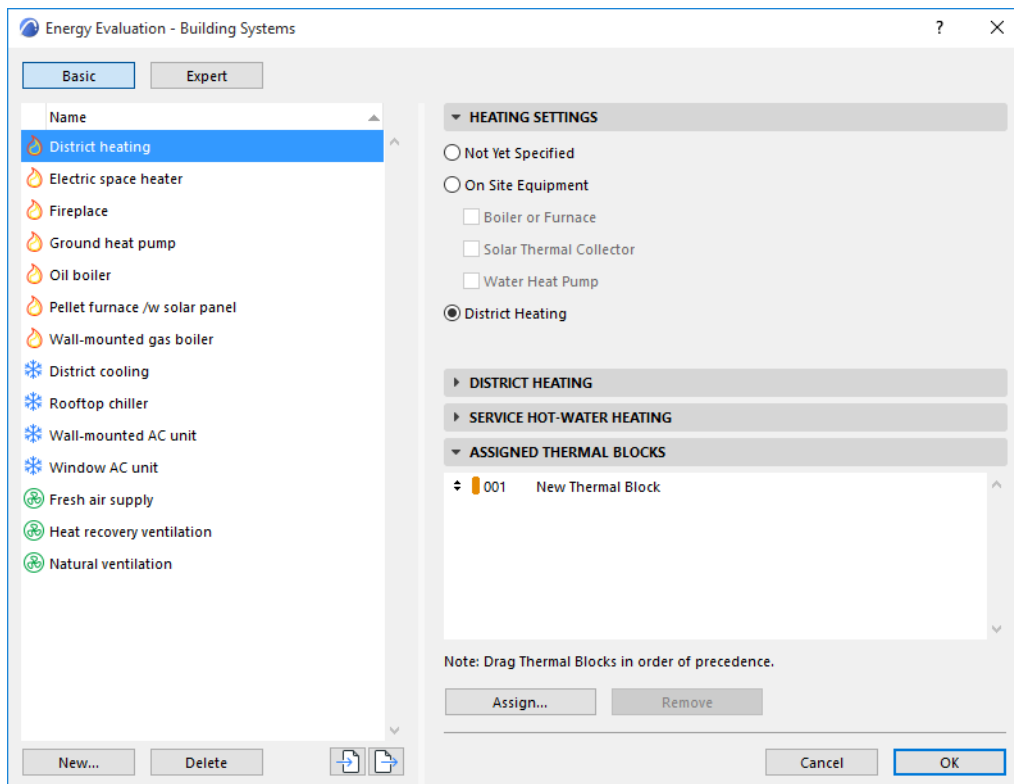
Building Systems (Energy Evaluation)

Open the Building Systems dialog box from:

- the Model Review Palette's pop-up at the top right
- **Design > Energy Evaluation > Building Systems**, or
- the Thermal Blocks page of the Energy Model Review palette

Use this dialog box to define Building Systems settings that will maintain the internal comfort conditions (as specified by the Operation Profiles dialog) for the thermal blocks of your building model.

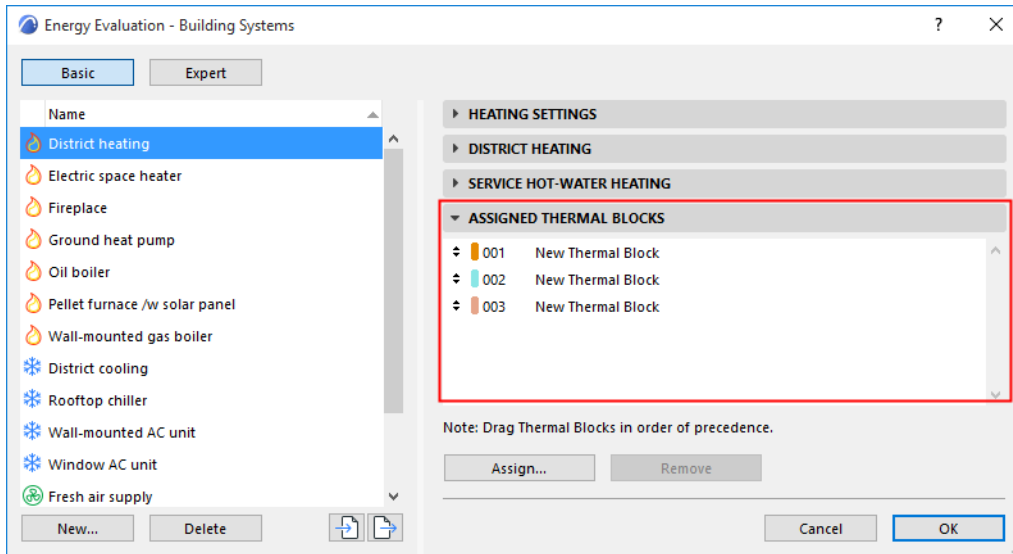
You can either select from the defaults or define custom Building Systems, which you will assign to each Thermal Block.



The Building Systems dialog box lists each Building System on the left, and the options for each on the right. The content of the building system options on the right changes depending on which system configuration is selected. Use the Building Systems dialog to edit the properties of existing building systems, to create new ones and to assign them to thermal blocks.

Assigned Thermal Blocks

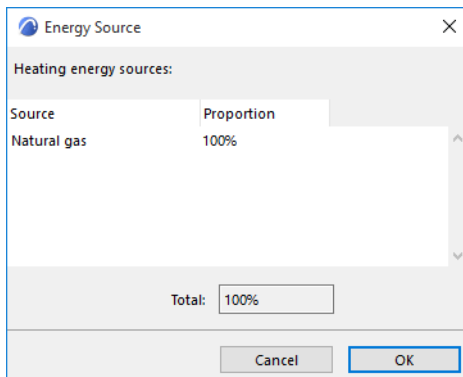
View or edit the list of thermal blocks that use the selected system.



Note: It is also possible to assign systems to thermal blocks via the Thermal Blocks page of the Energy Model Review palette.

Energy Source

Press the **Energy Source** button from either the Boiler or Furnace tab page (Heating System) or the District Cooling Settings tab page (Cooling System).



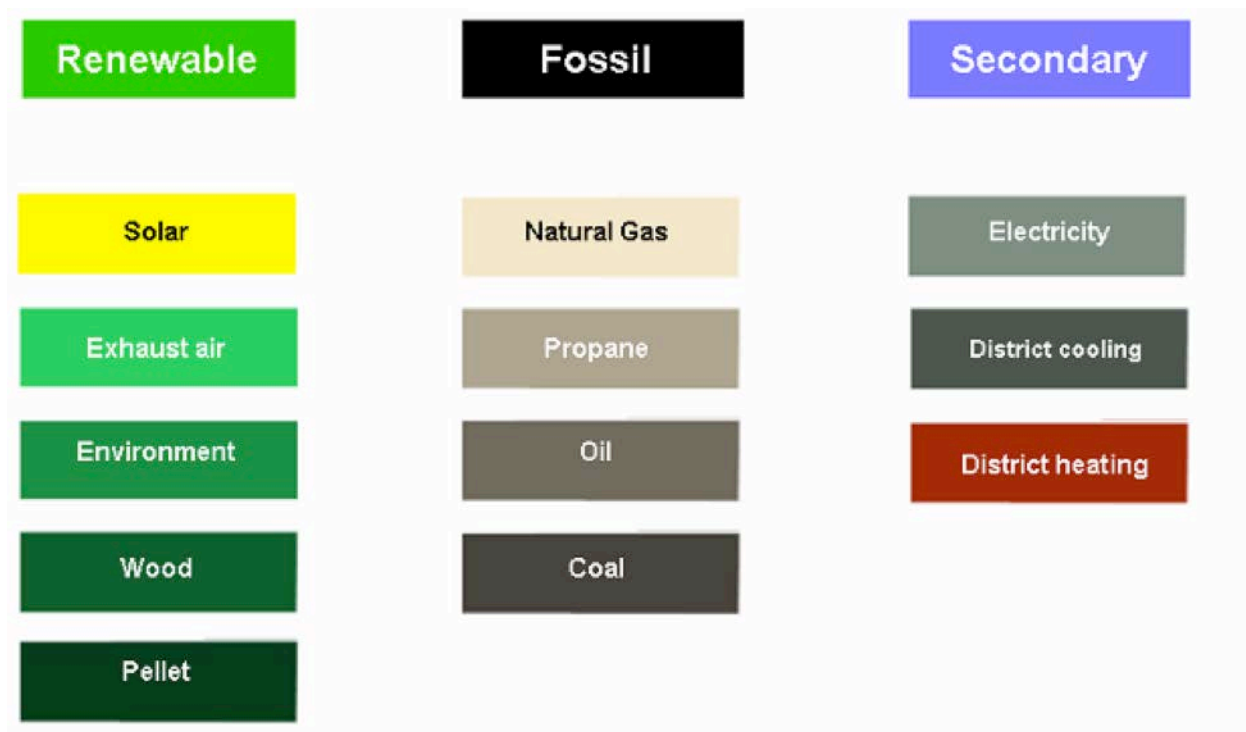
In the appearing Energy Source dialog box, click on a listed Source item and use the dropdown to specify energy sources (Natural gas, Propane, Oil, Wood, Coal, Electricity or Pellet) for the Boiler/Furnace or Cooling System.

If you use more than one energy source for a function, click the **+ (plus)** button, then choose an additional energy source.

Define the proportion of each source used so that the Total adds up to 100%.

To remove a row, click the **- (minus)** button.

In the table below, the energy sources used by Energy Evaluation are listed according to their type and with their colors as displayed on the Energy Balance Evaluation report.



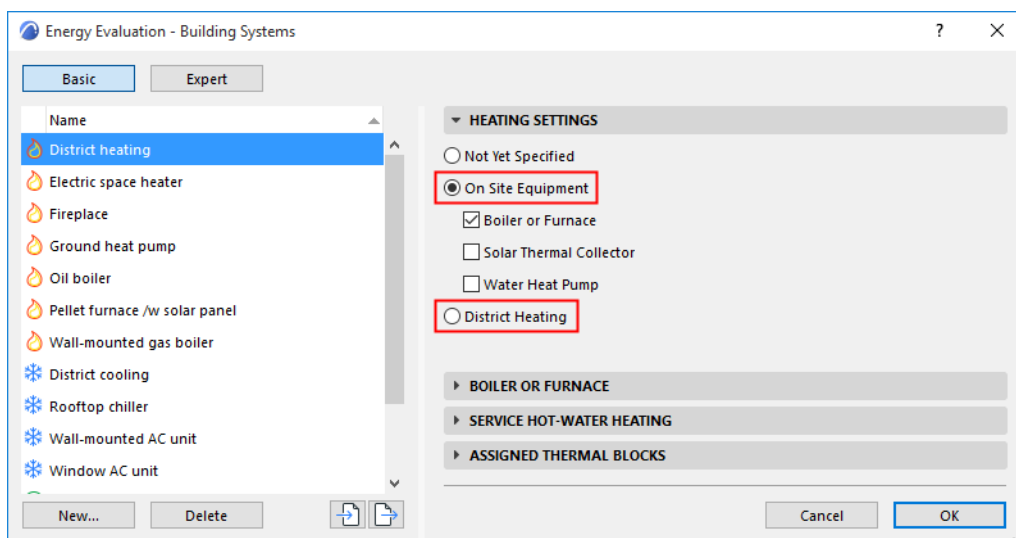
Heating

Choose either **District Heating** or **On Site Equipment** to describe the building's heating system.

District heating means that the evaluated building's heating and/or hot water needs are supplied by an external plant in the form of hot water or steam through a pipeline.

Use the Energy Source button to define the source or multiple sources of energy consumed by the heating system, as applicable.

See [Energy Source dialog box](#).



On Site Equipment

Choose one or more equipment types corresponding to the heat source to be used:

- Boiler or Furnace
- Solar Thermal Collector
- Water Heat Pump

Each option's settings are located on the corresponding dialog box panel below.

Boiler or Furnace

Fine-tune heater performance data such as Capacity and System Heat Loss or use the smart defaults provided by the program.

Specify whether or not to Include Service Hot-water Generation

▼ Boiler or Furnace

Capacity: W

System Heat Loss: %

Include Service Hot-Water Heating

Use the **Energy Source** button to define the source or multiple sources of energy consumed by the heating system, as applicable.

[See *Energy Source dialog box*.](#)

Solar Thermal Collector Settings

Solar thermal collector panels are designed to collect heat by absorbing sunlight and converting the energy in solar radiation into a more usable form.

▼ HEATING SETTINGS

On Site Equipment District Heating

Boiler or Furnace Not Yet Specified

Solar Thermal Collector

Water Heat Pump

▼ SOLAR THERMAL COLLECTOR SETTINGS

Type: ▼

Target: ▼

Panel Area: m²

Angle to South: °

Tilt Angle: °

- **Type:** Choose either Flat Plate or Evacuated tube.
- **Target:** Set the target for the recovered solar energy: Heating, Hot Water, or both.

Note: If solar collectors are used for multiple targets, then Hot water generation always has the highest priority. This means that if the solar collector is unable to generate enough energy to feed all targets, then it is going to feed the water heater first and use the remaining available energy for the Space heating.

Specify the collector geometry data required for the calculations:

- Panel area
- Orientation angle to the South (or to the North, for southern hemisphere locations)
- Tilt Angle

Heat Pump Settings

Specify system target and priority (drag-and drop the list items in the desired order) as well as source.

System parameters can also be fine-tuned, or use the smart defaults provided by the program.

Service Hot Water Heating

Enter Cold and Hot water temperature target values. The calculation engine uses this data to calculate the energy consumption related to hot water generation.

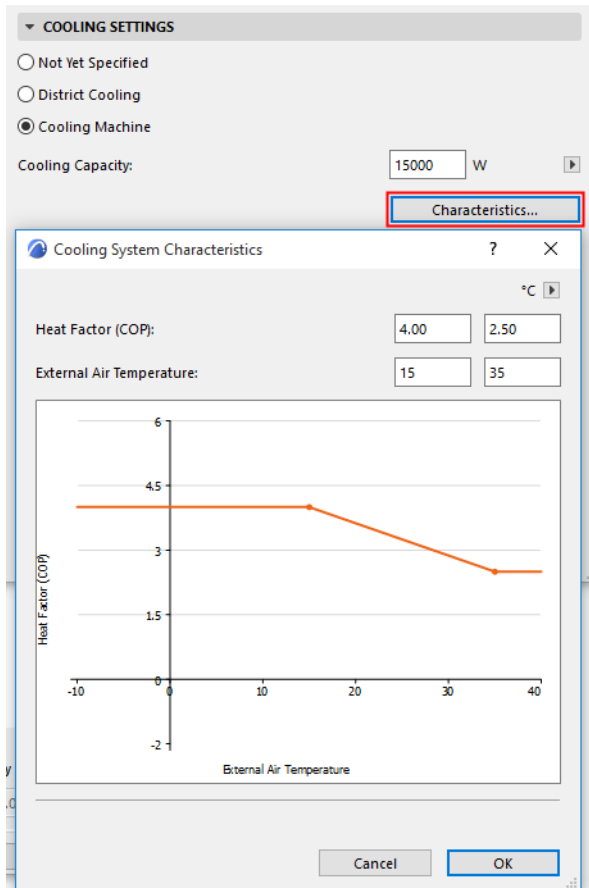
To change the temperature scale (Celsius or Fahrenheit), click the arrow button next to the input field.

Note: For other units (apart from temperature scale), Energy Evaluation uses the units defined in Archicad, at **Options > Project Preferences > Calculation Units**.

See [Calculation Units Preferences](#).

Cooling

Use this dialog to describe the building's cooling system, or use the smart defaults provided by the program.

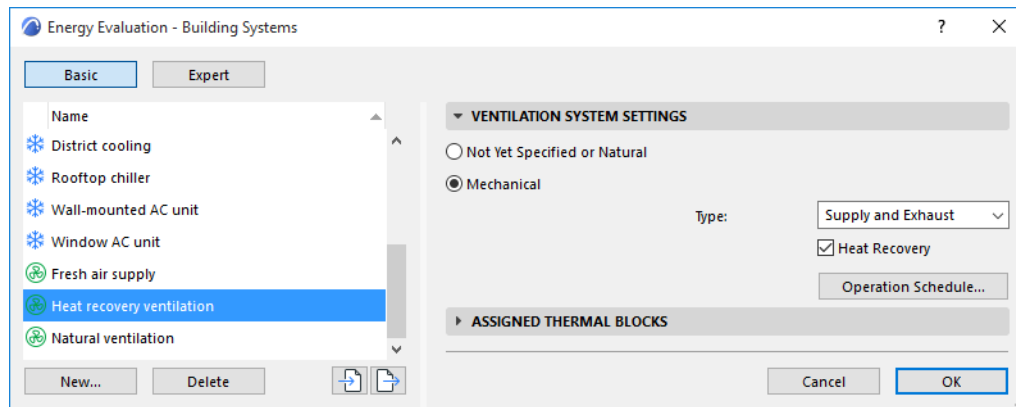


- **Cooling Machine** means that some type of cooling system is to be installed in the building. Click **Characteristics** to see and edit the performance graph of the cooling system.
- **District cooling:** In certain countries, cooled air may be obtained from an external source. If the building will be connected to such an external cooling system, select District cooling and use the **Energy Source** button.

See [Energy Source](#) to define the source or multiple sources of energy used by the district cooling plant.

Ventilation

Specify the type of ventilation used in the building: “Not Yet Specified Or Natural” or Mechanical Ventilation. Both of these ventilation types can be defined with Steady Air Change Rate as well as Time Scheduled Air Change Rate.



Not Yet Specified Or Natural Ventilation

Natural ventilation involves no MEP systems: natural air currents drive fresh air into the building and used air out of it. From an ecological point of view, natural ventilation is preferable to mechanical solutions, but due to its limited controllability its use is largely limited to residential buildings.

Mechanical

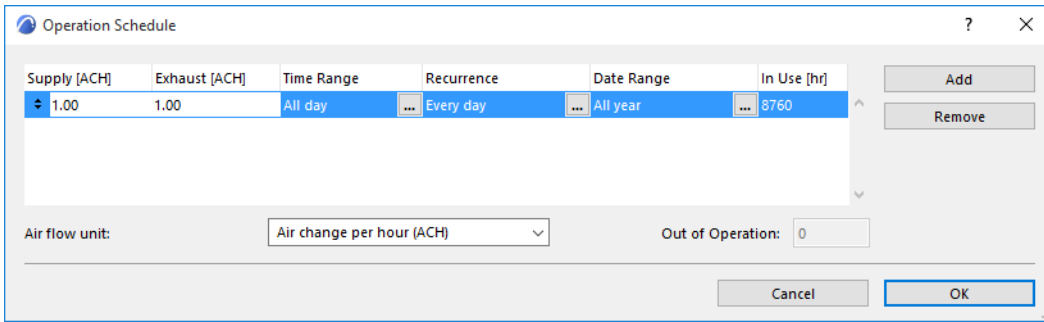
For building functions other than residential, the standards usually prescribe strict target air exchange values, which necessitate a mechanical ventilation system of some sort:

- **Supply-only** systems rely on fans to drive fresh air into the thermal block.
- **Exhaust-only** systems rely on fans to extract used air from the thermal block.
- **Supply and exhaust** systems mechanically control both the air intake and outlet procedure.
- **Heat Recovery:** Mechanical ventilation systems may or may not have air-to-air heat recovery units. Use the dedicated checkbox to specify this setting.

Note: The Air to air recovery controls are available only if you have chosen a ventilation system that involves mechanical exhaust. If your ventilation type is set to Natural, these controls are inactive.

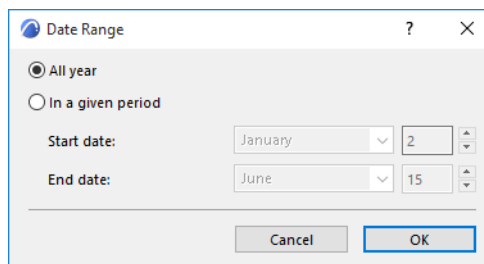
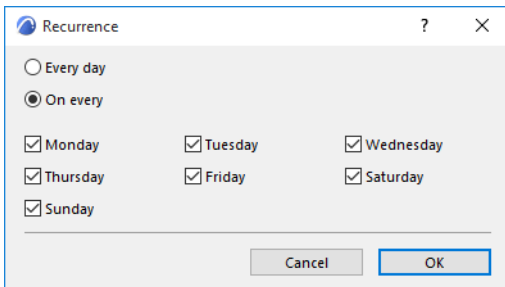
An air to air energy recovery system can regain a percentage of the heat content of mechanically expelled ventilation air.

- **Steady Air Change Rate:** Enter a value for the hourly air change rate. This target value depends on national standards, and will vary depending on the building's function and the local climate.
- **Time Scheduled Air Change Rate:** Supply and exhaust data may vary by day of the week and over the course of the year. The ventilation Operation Schedule can consist of any number of daily schedules, which correspond to specific date ranges and days of the week.



For each daily schedule, you can define the following characteristics:

- **Recurrence:** days of the week when the particular schedule is in effect
- **Date Range:** periods of the simulation year during which the schedule occurs



Recurrence and Date Range define the **In use hours** (the total number of hours during which the selected Daily Schedule is in effect).

If the **Out of Operation** hours counter on the bottom right corner of the Yearly Operation Scheduler shows any number except 0, it means that further modifications in Recurrence and Date Range are necessary because there are some hours of the reference year without operation schedule assigned to them.

If the **Overlapped** hours counter on the bottom right corner of the Yearly Operation Scheduler shows any number except 0, it means that further modifications in Recurrence and Date Range are necessary because there are some hours of the reference year with more than one operation schedules assigned to them.

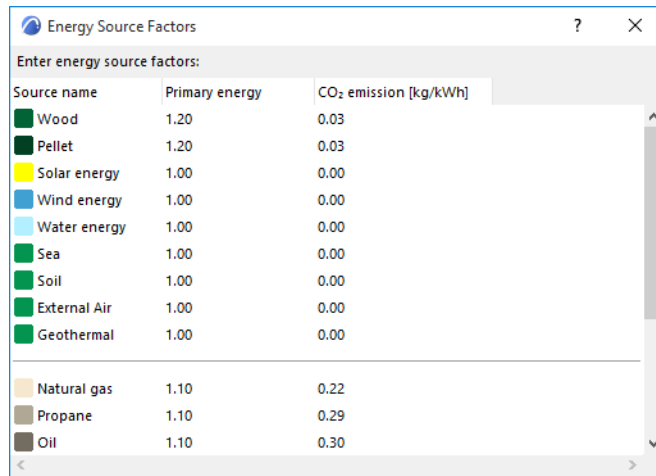
Energy Source Factors (Energy Evaluation)

Open this dialog box from:

- the Model Review Palette's pop-up at the top right, or
- **Design > Energy Evaluation > Energy Source Factors.**

Primary Energy And CO2 Emission Factors

For each energy source, default values are shown for primary energy and CO2 emission factors. These default values, which vary according to Project Location, are based on standard DINV-18599.

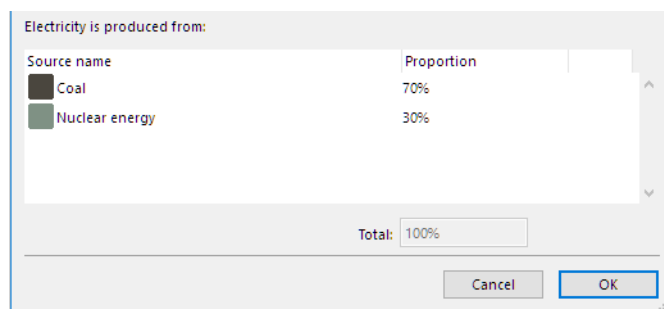


Source name	Primary energy	CO ₂ emission [kg/kWh]
Wood	1.20	0.03
Pellet	1.20	0.03
Solar energy	1.00	0.00
Wind energy	1.00	0.00
Water energy	1.00	0.00
Sea	1.00	0.00
Soil	1.00	0.00
External Air	1.00	0.00
Geothermal	1.00	0.00
Natural gas	1.10	0.22
Propane	1.10	0.29
Oil	1.10	0.30

Use these default values, or enter other region-specific data if available.

Sources of Electricity Production

At the bottom of the Energy Source Factors dialog, define the building's source or multiple sources of electricity. Click on a listed Source item and use the dropdown to specify the source of electricity (Natural gas, Propane, Oil, Wood, Coal, Nuclear energy, Wind energy, Solar energy, Water energy or Unknown).



Source name	Proportion
Coal	70%
Nuclear energy	30%

Total: 100%

Buttons: Cancel, OK

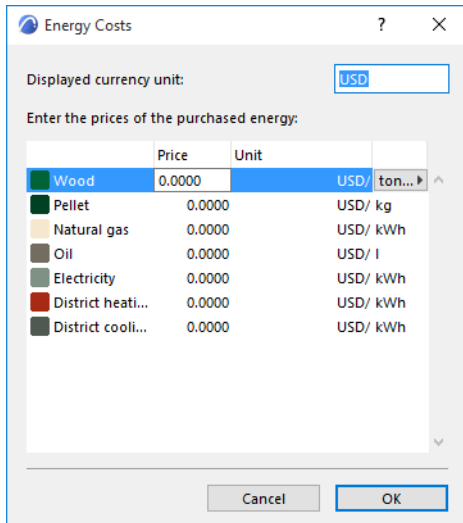
If you use more than one energy source, click the **+** (plus) button, then choose an additional source. Define the Proportion of each source used so that they add up to 100%. To remove a row, click the **-** (minus) button.

Energy Costs (Energy Evaluation)

Open this dialog box from

- the Model Review Palette's pop-up at the top right, or
- **Design > Energy Evaluation > Environment Settings.**

In the top part of the panel, enter the currency name or its abbreviation in the Displayed currency unit field.



Use the bottom part of the panel to define the Prices of purchased energy. These prices, of course, vary by location and therefore must be entered numerically by the user. Select the Unit of energy (e.g. kWh) from the dropdown list. Next, select a row representing a source of energy, and then enter the price, in the currency you chose.

Energy Performance Evaluation

The following topics detail the evaluation process and the resulting reports of Archicad's Energy Evaluation feature.

[Start Energy Simulation \(Energy Evaluation\)](#)

[Interoperability with Green Software \(Energy Evaluation\)](#)

[Advisory Messages \(Energy Evaluation\)](#)

[Errors And Recommended Solutions](#)

[VIP-Core Dynamic Simulation Engine \(Energy Evaluation\)](#)

[Energy Evaluation Report Dialog](#)

[Reading the Energy Performance Evaluation - PDF Report](#)

[Reading the Energy Performance Evaluation - XLS Report](#)

Start Energy Simulation (Energy Evaluation)

Once all necessary input data are obtained by means of geometry analysis and user input, you can start the data input into the VIP-Core calculation engine built into Archicad.

See [Energy Performance Evaluation](#).

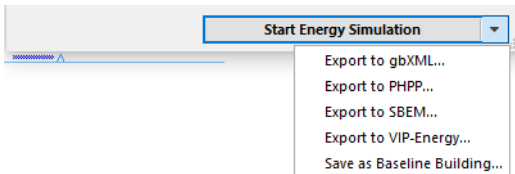
Select the **Start Energy Simulation** button from the Energy Model Review Palette to launch the VIP-Core Calculation Engine, which performs the dynamic energy analysis. Energy Evaluation calculates the energy balance of a building during a period of one year, and displays the results on the Energy Performance Evaluation Report.

For information on exporting data to 3rd party energy calculation software, see [Interoperability with Green Software \(Energy Evaluation\)](#).

Interoperability with Green Software (Energy Evaluation)

Software interoperability increases the design team's work efficiency and removes a significant cost barrier from the design project. Energy Evaluation enables the export of project geometry and property data derived from the Archicad model. Use these communication platforms to transfer architectural building model data directly to building performance analysis tools, eliminating the need for time consuming plan takeoffs and building geometry remodeling.

Click the arrow next to the Start Energy Simulation button to display the available data export file formats.

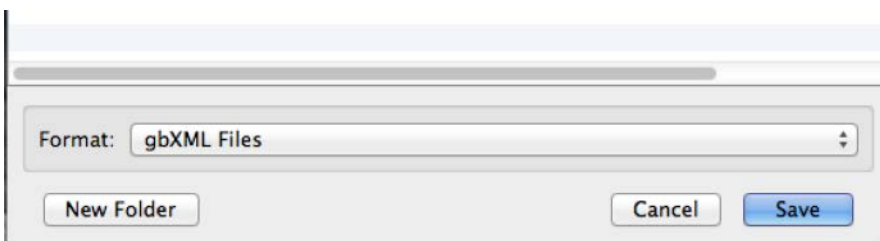
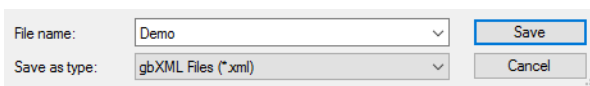


Note: Five different export formats are listed. The first two (Export to gbXML and Export to PHPP) are available in every Archicad as standard features. The latter three (Export to SBEM, Export to VIP-Energy and Save as Baseline Building) are functions exclusive to the EcoDesigner STAR add-on, available for SSA/Forward customers and Education license holders.

Export to gbXML

XML - extensible markup language - is a computer language that allows software programs to communicate information. The Green Building XML (gbXML) open schema helps facilitate the transfer of BIM data to building energy analysis tools. Supported by the leading 3D BIM applications and major building performance analysis tools, gbXML has become the industry standard for BIM to SIM interoperability.

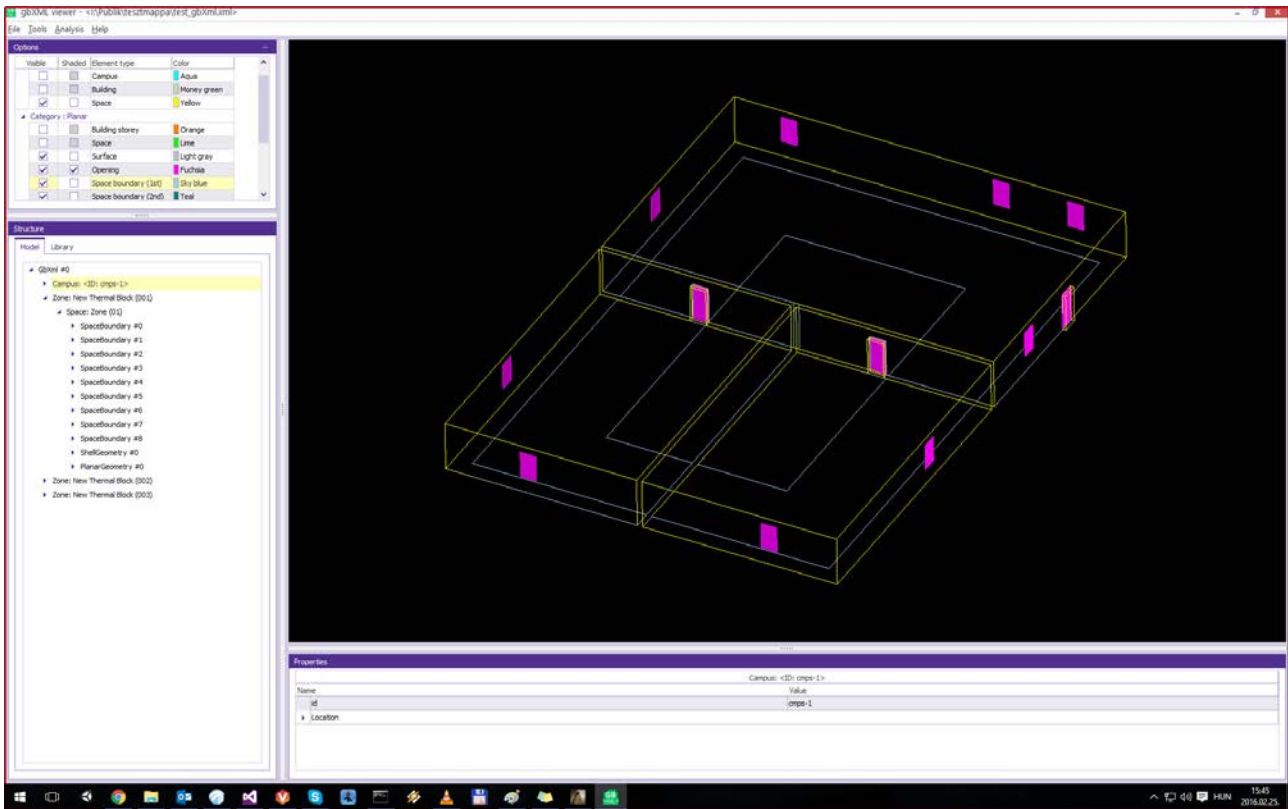
Use the dedicated export panel to save project geometry and property data derived from the Archicad model in gbXML file format.



The following project data are exported from Archicad via gbXML:

EcoDesigner STAR Data		gbXML Data
Type	Label	Location in file
Thermal Blocks (= IFC Zone)	Thermal Block	Zone
	General identifier	Zone/id
	Thermal Block title	Zone/Name
	Area	Area is listed under Spaces
	Volume	Volume is listed under Spaces
	Thermal Block id	Zone/Description
	Zones contained	Campus/Building/Space (connection: Zone/id = zoneIdRef)
Zones (= IFC Space)	Zone	Campus/Building/Space
	ID	Space/Description
	Name	Space/Name
	Story	Space/buildingStoreyIdRef
	Thermal Block	Space/zoneIdRef
	Category	Space/TypeCode
	Area is listed under Thermal Blocks	Space/Area
	Volume is listed under Thermal Blocks	Space/Volume
Polygons	Defined in the BIM model	Space/ShellGeometry, Space/SpaceBoundary/PlanarGeometry, Campus/Surface/PlanarGeometry
Material Properties	Building Material	Material (connection: Layer/MaterialId, Construction/LayerId)
	Material name	Material/Name
	Thermal Conductivity [W/mK]	Material/Conductivity
	Density [kg/m ³]	Material/Density
	Heat Capacity [J/kgK]	Material/Specificheat
	Uvalue [W/m ² K]	Material/Rvalue [m ² K/W] (Reciprocal of U value)
	Thickness (Composite Skin or Element property)	Material/Thickness
Opaque 2nd Level Space Boundary Properties	Structure	Construction, Campus/Surface, Campus/Building/Space/SpaceBoundary
	Type	Campus/Surface/surfaceType
	Orientation	Surface/RectangularGeometry/Azimuth, Surface/RectangularGeometry/Tilt
	Category	Campus/Surface/surfaceType
	Thermal Block	Surface/AdjacentSpaceId spaceIdRef
	Name	Construction/Name, Layer/Name
	Thickness (Composite Skin or Element property)	Material/Thickness (connection: Layer/MaterialId, Construction/LayerId, Surface/constructionIdRef)
	UValue	Construction/Uvalue
Transparent 2nd Level Space Boundary Properties	Openings	Campus/Surface/Opening
	Type	Campus/Surface/Opening/openingType
	Orientation	Tilt and Azimuth are defined at the polygons
	Thermal Block	Inherits AdjacentSpaceIds of the parent Surface: Campus/Surface/AdjacentSpaceId/spaceIdRef
	Overall Uvalue	Surface/Opening/Uvalue
Location	Project Location	Campus/Location
	Longitude	Longitude
	Latitude	Latitude
	AngleToProjectNorth	CADModelAzimuth

The gbXML protocol describes the 3D building model in a way quite similar to IFC. The gbXML model can be visualized using dedicated viewers or the graphical user interfaces of building performance simulation software that are compatible with the gbXML format.



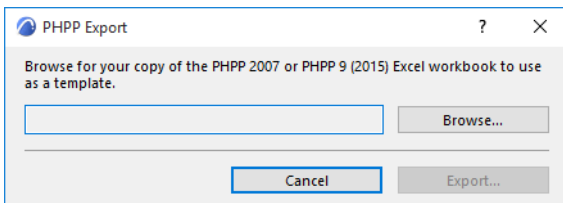
Export to PHPP

Certified Passivhaus buildings (mainly small scale residential) are designed according to a strict yet straightforward directive in order to be extremely energy efficient. PHPP is the Excel workbook created by the Darmstadt Passivhaus Institute, to calculate and document the energy performance of projects that aspire for Passivhaus accreditation.

Archicad supports both PHPP versions 7 and 9.

Note: The PHPP Excel workbook is an independent product of the Darmstadt Passivhaus Institute and is not a part of Graphisoft Archicad. For more information on PHPP and the Passivhaus Institute visit the official website at <http://www.passiv.de/>

The Export to PHPP button activates the PHPP Export dialog box, which enables you to send building information model data to the PHPP Excel workbook you specify, for post processing.



Use the PHPP Export panel to browse for the official PHPP Excel workbook to be used as a template, then save the project-specific PHPP Excel file under a different name.

The following 2nd level space boundary and material property data are exported from Archicad to the respective fields of the project-specific PHPP workbook:

Data location in EcoDesigner	Data location in PHPP
Structures	Areas
Building shell elements	Area Input entries
Orientation	Building Element Description
Building structure	Building Assembly Description
Area	Area Input / User Determined
Thermal Property Assignment	Corresponding Building Element Assembly
U-value Calculator	U-Values
Skin Name	Area Section
External heat transfer coefficient	Heat Transfer Resistance exterior (reciprocal of the External heat transfer coefficient)
Internal heat transfer coefficient	Heat Transfer Resistance interior (reciprocal of the Internal heat transfer coefficient)
Thermal Conductivity	Thermal Conductivity
Thickness	Thickness
Thermal Conductivity	Thermal Conductivity
Openings	Windows
Openings on building shell entry (combined*)	Windows list entry (each ArchiCAD opening individually)
Orientation	Description; Deviation from North; Angle of Inclination from the Horizontal
(From ArchiCAD building model)	Window Rough Openings / Width and Height
(From ArchiCAD building model)	Installed in Area in the Areas worksheet

***Note:** Openings are not listed one by one in EcoDesigner; instead, the data are totaled for the opening type and orientation. However, they are exported to PHPP individually.

The Passivhaus directive sets stringent threshold values for the Specific annual net heating and cooling demands (15 kilo watt hours per square meter, year maximum), total annual specific primary energy demand (less than 120 kilo watt hours per square meter, year), and average infiltration (maximum 0.6 1/h).

Note: The Key Values section of the Building Energy Evaluation Report displays these performance data, as calculated by The VIP-Core Calculation Engine.

Key Values

General project data

Location: Stockholm
 Activity Type: Office
 Evaluation Date: 5/23/2011 2:27 PM

Building geometry data

Treated floor area: 14014 m²
 Building shell area: 9562 m²
 Ventilated volume: 46246 m³
 Glazing ratio: 22 %

Building shell performance data

Air leakage: 0.45 ACH
 Outer heat capacity: - J/m²K

Heat transfer coefficients

U value [W/m²K]
 Building shell average: 0.63
 Roofs: 0.25 - 0.25
 External walls: 0.45 - 0.45
 Basement walls: -
 Openings: 1.60 - 1.60

Specific annual demands

Net heating energy: 37.26 kWh/m²a
 Net cooling energy: 18.31 kWh/m²a
 Energy consumption: 196.47 kWh/m²a
 Primary energy: 238.77 kWh/m²a
 Operation cost: 10.54 EUR/m²a
 CO₂ emission: 24.43 kg/m²a

Energy Evaluation in Archicad can be used to design a building as energy efficient as a Passivhaus. However, in order to apply for official Passivhaus accreditation, a PHPP Excel workbook must also be completed and submitted to the Passivhaus Institute.

Advisory Messages (Energy Evaluation)

The “Advisory Messages” dialog box appears if:

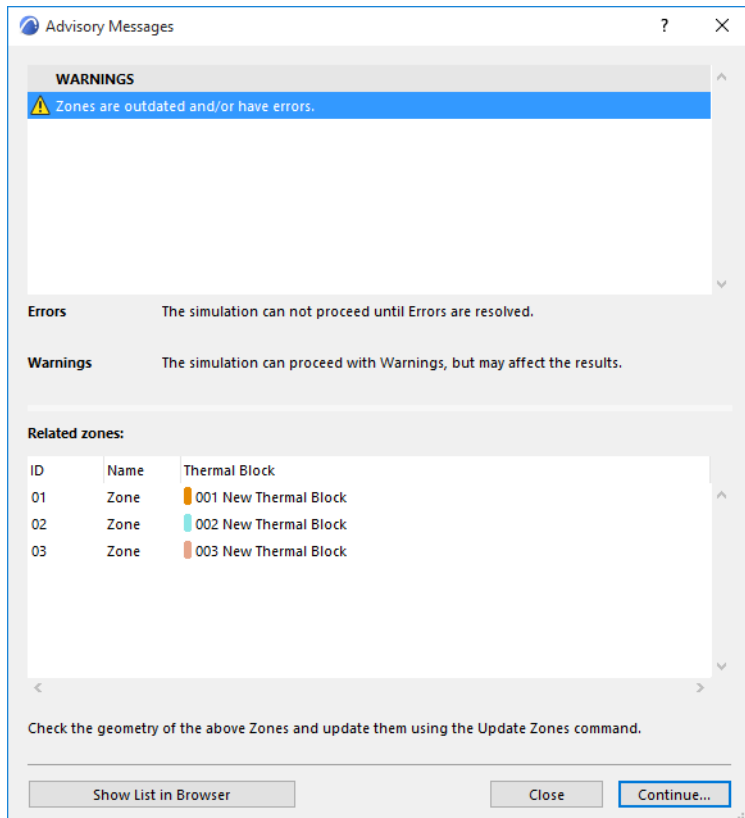
- the required input data are insufficient

See [Additional Data Assignment and Input](#).

- there are simulation errors

See [VIP-Core Dynamic Simulation Engine \(Energy Evaluation\)](#).

Errors will be required to be resolved before continuing. However, resolving Warnings is optional.



Errors And Recommended Solutions

(mandatory to resolve before continuing)

- Physical properties of a used materials are not defined.
 - Error icon: “Building Materials with missing properties”
 - List of missing Building Materials
 - To-do: “Select the relevant Building Material from the dedicated dialog box and enter the missing values (values are required to be more than zero) in the Physical Properties panel.”
- “Climate data is not available.” (missing or the server is not accessible)
 - Error icon, “Climate Data is not available.”
 - “To edit the Climate Data, open the Environment Settings dialog box from the Model Review palette.”
- “One or more used Building Material/Composite Structure is missing.”
 - Error icon, “Structures with missing attributes.”

- List of structures with missing Building Material
- “Select the relevant model elements and edit its Building Material attribute.”
- “Energy simulation runtime error.” (engine errors during the hourly energy simulation)
 - Error icon, “Simulation engine errors.”
 - List of <Engine error code + description>.
 - To-do: “Contact technical support.”
- “No valid Operation Profile selected.”
 - Error icon, “Operation Profile is missing or not defined.”
 - To-do: “Select an available profile from the Operation Profile dialog.”
- “Energy model analysis error.”
 - Error icon, “Energy model analysis did not found any structure or openings.”
 - To-do: “Check the geometry of the Zones and update them using the Update Zones command.”

Warnings And Recommended Solutions

(optional to resolve before continuing)

- “Zones are outdated and/or have errors.” (i.e. not up to date, overlapped zones, visible zones which are not assigned to any TB) - zones on hidden layers will not appear
 - “Outdated zones:”
 - List of Zones - Zone ID, zone name, thermal block name
 - To-do: “Check the geometry of the above Zones and update them using the Update Zones command.”
- “Location data varies between project location and climate data.”
 - Warning icon, “Location data does not match between project location and climate data.”
 - To-do: “Either modify the project location from the Location Settings dialog box or modify the climate settings from the Environment Settings dialog box accessed via the Model Review palette.”
- Simplified building system is attached to a Thermal Block.

“Thermal Blocks with simplified building system:”

 - List of Thermal Blocks - color swatch, number, name
 - To-do: “Choose detailed settings from the Building Systems dialog and add the relevant details of HVAC systems used within building.”
- “Building shell contains one or more empty opening(s).”
 - Warning icon, “Building shell is not closed because external wall contains empty opening.”
 - To-do: “Review the Opening list and modify empty opening if necessary.”

On the Structures and Openings lists, a yellow warning exclamation mark appears next to those entries which contain faulty or missing data. (The problematic list items are also highlighted in pink.)

[See *Additional Data Assignment and Input*.](#)

Show List in Browser

- Press the Show List in Browser button of the Advisory Messages dialog to display the list of errors and warnings in a browser window. This way the list can remain open for reference while you fix the building model in Archicad following the instructions of the Suggestions / Notes field.

Warnings

Warning	Related items	Suggestions / Notes
Zones are outdated and/or have errors.	001 Flat 001 Flat 002 Flat 002 Flat	Check the geometry of the above Zones and update them using the Update Zones command.

VIP-Core Dynamic Simulation Engine (Energy Evaluation)

Specification

The VIP energy simulation software module integrated in Archicad (a.k.a. engine) is a result of more than 20 years of research and commercial use. It relies entirely on dynamic models, in which every model is broken down to a level where facts and behavior are known. Each component group is analyzed separately. The calculation is repeated every hour. The accuracy of each model has been validated and verified against real buildings under real usage. In calculating a building's energy consumption, the program utilizes known or measured facts about all parts of the energy flow.

VIP-Energy is produced by Structural Design Software in Europe AB <http://www.strusoft.com/>

For technical information about VIP-Energy and for a more detailed explanation of the different calculating models, see <https://strusoft.com/software/vip-energy-calculation-software/>.

Validation of Calculation Accuracy

Energy Evaluation uses the same simulation kernel as EcoDesigner STAR add-on for Archicad. This kernel is validated with ANSI/ASHRAE Standard 140-2007: Standard Method of Test for the Evaluation of Building Energy Analysis Computer Programs.

Note: To produce code-compliant energy simulation results, the expert user interface of EcoDesigner STAR is necessary. EcoDesigner STAR is available for SSA/Forward customers and Education license holders.

The ASHRAE Standard 140 test documentation for EcoDesigner STAR will be available on the official product website, following the add-on's release.

The stand-alone VIP Energy product, which also uses the same simulation kernel as Energy Evaluation in Archicad and EcoDesigner STAR, is also validated with the following tests:

- EN-15265
- IEA-BESTEST
- ASHRAE-BESTEST (ANSI/ASHRAE Standard 140-2001)
- StruSoft-BESTEST

Energy Evaluation Report Dialog

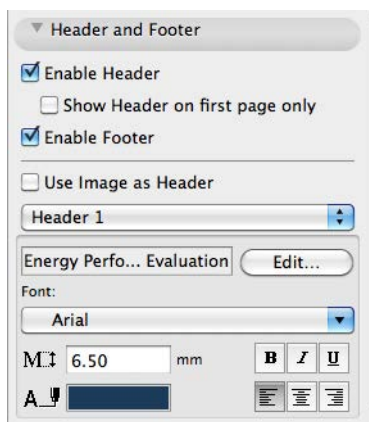
After the dynamic energy simulation is finished, the Evaluation Report dialog appears. Use the controls on the left side to customize certain aspects of the Energy Performance Evaluation report's content before saving it.



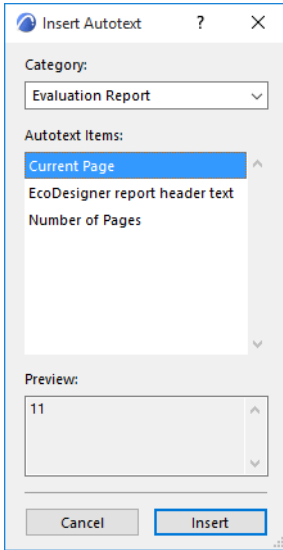
The right side of the dialog contains a preview of the Energy Performance Evaluation report PDF document.

Header and Footer Settings

Use the controls of the Header and Footer Settings dialog to fill the two Header rows as well as the Footer of the PDF Energy Performance Report with custom content.

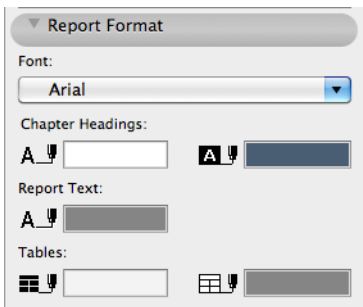


This content can also be an Image or even Autotext, which allows for a wide range of smart report title customization.



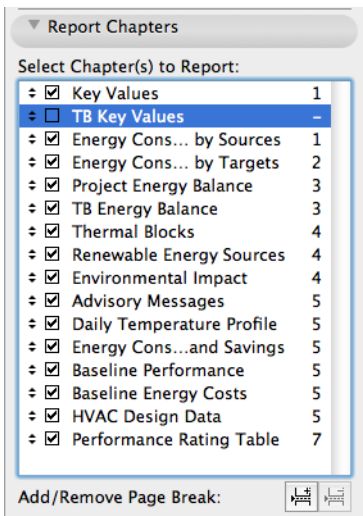
Report Format

This dialog defines font format and pen colors used for the Energy Performance Evaluation report PDF.



Report Chapter Settings

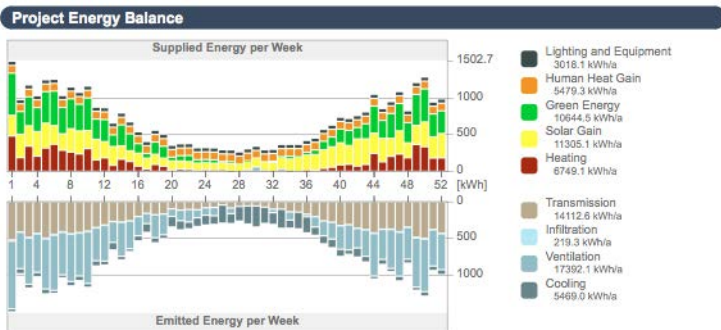
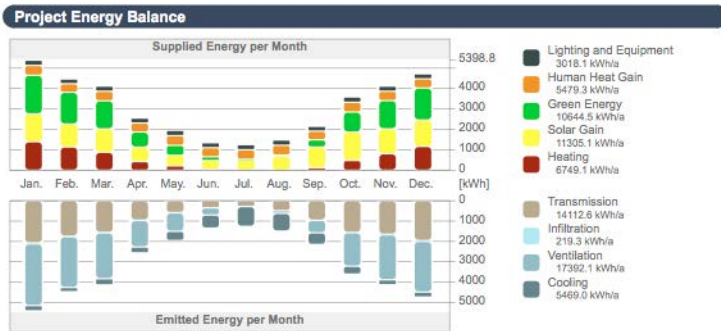
In the Report Chapters section the content of the Energy Performance Evaluation report PDF is listed.



Change the order of the chapters by grabbing them with the arrows displayed before their names and dragging them. Use the buttons below the list to Add/Remove Page Breaks.

Some of the report chapters can be further customized by applying certain chapter-specific settings:

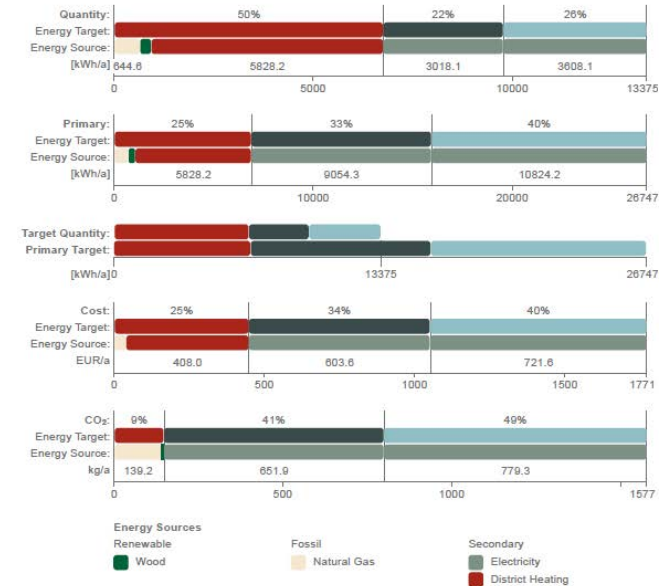
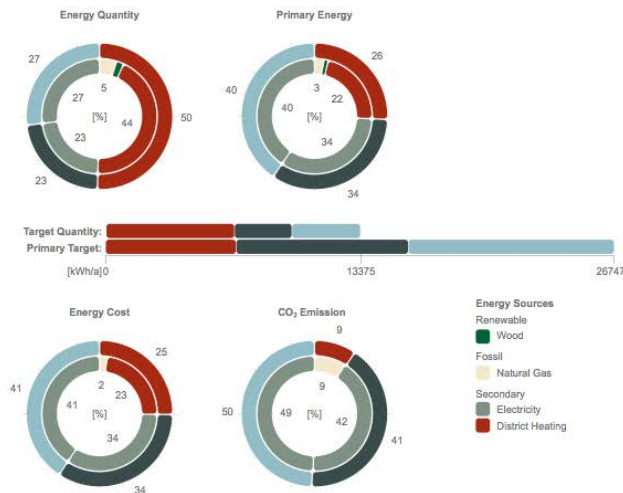
- Key Values Chapter Settings: Choose one of the three available Energy unit options: kWh, kBtu, MJ
- Project Energy Balance Chapter Settings: Choose Weekly or Monthly for Time interval, and specify the Energy unit (kWh, kBtu, MJ). It is also possible to customize the colors of the bar chart.



- Energy Consumption by Targets and by Sources Chapters - Settings: Toggle between Pie or Bar Chart Display

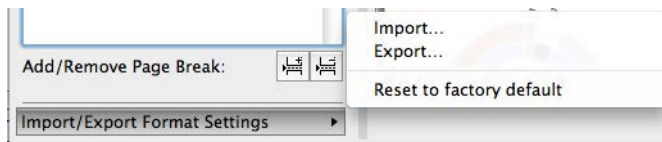
Target Name	Quantity kWh/a	Primary kWh/a	Cost EUR/a	CO ₂ Emission kg/a
Heating	6749	6868	446	146
Cooling	0	0	0	0
Service Hot-Water	0	0	0	0
Ventilation Fans	3608	10824	721	779
Lighting & Appliances	3018	9054	603	651
Total:	13375	26747	1771	1577

Target Name	Quantity kWh/a	Primary kWh/a	Cost EUR/a	CO ₂ Emission kg/a
Heating	6749	6868	446	146
Cooling	0	0	0	0
Service Hot-Water	0	0	0	0
Ventilation Fans	3608	10824	721	779
Lighting & Appliances	3018	9054	603	651
Total:	13375	26747	1771	1577



Format Settings Import/Export

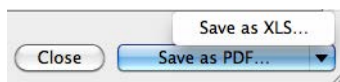
This button on the bottom left corner of the Evaluation Report dialog makes it possible to migrate PDF report formats between projects (e.g. to apply a company template format to the Energy Performance Evaluation report PDFs of every project).



There's also an option to reset the format of the Energy Performance Evaluation report to factory default.

Close and Save as Commands

At the bottom of the Evaluation Report:



- The Close button takes you back to the Energy Model Review palette, allowing you to further adjust the input data.
 - Save as PDF lets you specify a location to which to save the Evaluation Report in PDF format
- [See Reading the Energy Performance Evaluation - PDF Report.](#)
- Click the button with the triangle in the bottom right corner to save the detailed monthly energy balance data in Microsoft Excel file format.

[See Reading the Energy Performance Evaluation - XLS Report.](#)

Reading the Energy Performance Evaluation - PDF Report

PDF Header

Energy Performance Evaluation

[Project Number] [Project Name]

Key Values

General Project Data		Heat Transfer Coefficients		U value	[W/m ² K]
Project Name:	for Documentation Illustr...	Building Shell Average:	0.61		
City Location:		Floors:	0.04 - 0.04		

The PDF header displays custom project information as defined on the Header / Footer Settings dialog.

Key Values

The Key Values section of the PDF Evaluation Report displays the most important project data.

Under General Project Data, basic information, such as Location and Operation Profile, as specified by the user ([see Additional Data Assignment and Input](#)). The date of the evaluation is also shown.

Key Values

General Project Data			Heat Transfer Coefficients		U value	[W/m ² K]
Project Name:	for Documentation Illustr...		Building Shell Average:	0.61		
City Location:			Floors:	0.04 - 0.04		
Climate Data Source:	DRYCOLD.TMY		External:	0.36 - 0.62		
Evaluation Date:	Mar 1, 2013 3:57:01 PM		Underground:	-		
			Openings:	3.00 - 3.00		
Building Geometry Data			Specific Annual Demands			
Gross Floor Area:	99.31	m ²	Net Heating Energy:	70.30		kWh/m ² a
Treated Floor Area:	96.00	m ²	Net Cooling Energy:	0.00		kWh/m ² a
External Envelope Area:	202.80	m ²	Total Net Energy:	70.30		kWh/m ² a
Ventilated Volume:	259.20	m ³	Energy Consumption:	139.33		kWh/m ² a
Glazing Ratio:	6	%	Fuel Consumption:	139.33		kWh/m ² a
Building Shell Performance Data			Primary Energy:	278.62		kWh/m ² a
Infiltration at 50Pa:	2.19	ACH	Fuel Cost:	18.46		EUR/m ² a
Outer Heat Capacity:	11.36	J/m ² K	CO ₂ Emission:	16.43		kg/m ² a

The Building Geometry chapter contains result of the model geometry analysis such as areas, volume and glazing ratio.

[See Automatic Model Geometry and Material Property Analysis in Energy Evaluation.](#)

The Building Shell Performance Data section displays overall Air Leakage in air change per hour (also called air exchange rate - the number of interior volume air changes that occur per hour in 1/h). Outer heat capacity (measures the capacity of building structures to store heat against changing outside air temperature) is also an important building envelope performance metric.

Also in the Key Values section, the minimum and maximum values of the Calculated heat transfer coefficients are listed for the entire building, for every Building structure group and for the openings on the building shell.

Under Specific annual demands, the most important energy performance data (net demand and gross consumption-related data) - projected to a unit area of the building - are listed, making the comparison of different sized projects possible.

Thermal Blocks

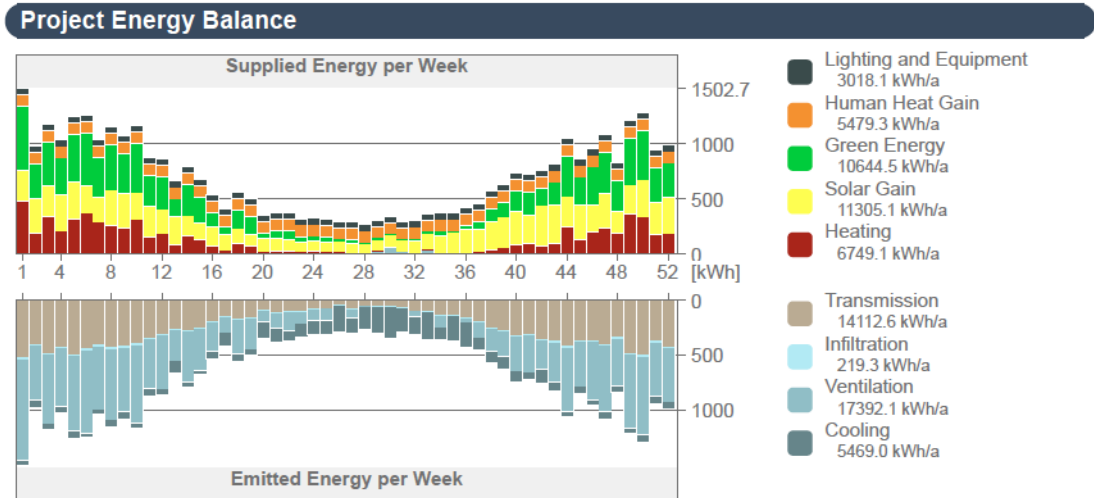
The table in this chapter tabulates the thermal blocks with their geometry data and Operation Profiles.

Thermal Blocks

ID	Name	Zones Assigned	Operation Profile	Gross Floor Area m ²	Volume m ³
001	Storage basement	8	unconditioned	1219.52	3142.72
002	Staircases	32	Circulation and tra...	788.24	2222.08
003	Retail 1	2	Retail shop/depart...	459.83	1373.18
004	Retail 2	1	Retail shop/depart...	117.90	409.79
005	Retail 3	1	Retail shop/depart...	119.00	413.90
006	Retail 4	2	Retail shop/depart...	467.35	1366.05
007	Office 1	1	Personal office	147.08	385.95
008	Office 2	1	Personal office	148.50	388.98
009	Office 3	1	Personal office	273.40	716.69
010	Office 4	1	Personal office	147.61	385.95
011	Office 5	1	Personal office	149.03	388.98
012	Office 6	1	Personal office	277.74	713.66
013	Office 7	1	Personal office	273.40	716.69
014	Office 8	1	Personal office	147.61	385.95
015	Office 9	1	Personal office	149.03	388.98
016	Office 10	1	Personal office	277.74	713.66
017	Flat 1	1	Residential	191.60	486.19
018	Flat 2	2	Residential	172.10	437.81
019	Flat 3	1	Residential	62.89	162.74
020	Flat 4	1	Residential	187.16	486.19
021	Flat 5	1	Residential	62.65	162.74
022	Flat 6	1	Residential	191.60	486.19
023	Flat 7	1	Residential	62.89	162.74
028	Flat 8	1	Residential	187.16	486.19
029	Flat 9	2	Residential	171.71	443.86
030	Flat 10	2	Residential	169.34	437.81
031	Flat 11	2	Residential	172.23	443.86
032	Flat 12	1	Residential	62.65	162.74
033	Attic	3	unconditioned	955.20	2609.40
	Total:	75		7812.13	21081.64

Monthly Energy Balance Bar Chart

The Monthly Energy Balance bar chart is a graphical display of the amount of energy the building emits (bottom part of chart), as well as the building’s Supplied energy: the amount of energy it absorbs from the environment and its own internal heat sources (top part of chart), by month or week (depending on Energy Performance Evaluation Report Chapter Settings).



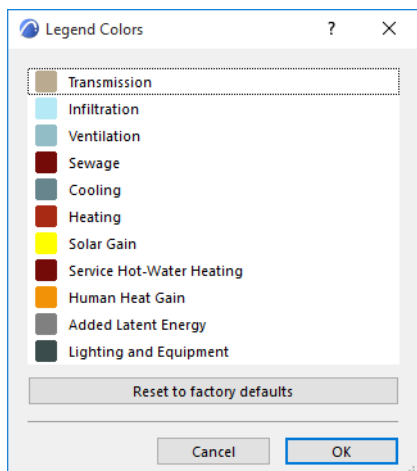
According to the energy balance equation - which is the foundation of building physics - the Emitted energy and Supplied energy bars must be equal every month. The vertical axis of the chart shows an energy scale. Along the horizontal axis, the twelve months of the year are shown.

The Monthly Energy Balance Bar Chart shows the cumulative results of the hourly energy balance calculations executed by The VIPCore Calculation Engine.

Energy Supply and Emission Types

To the right of the Monthly Energy Balance diagram, the energy flows that make up the bars of the chart are listed. The number and type of these energy balance components that appear on a particular Energy Evaluation Report depend on the settings on the evaluated building’s Building Systems dialog. Total annual energy flow quantities are displayed in kWh/a.

The table below lists all energy supply and emission types with their default Energy Balance bar chart display color.



These energy supply and emission categories, besides being used to visualize the energy balance, also provide additional information on the MEP systems (cooling, heating, ventilation type) and green energy solutions of the evaluated project.

The left column shows the energy flows that appear on the Emitted energy side of the energy balance bar chart, while in the right column the Supplied energy types are listed. The column in the middle contains energy flows that may appear as either emitted or supplied, depending on the particular situation.

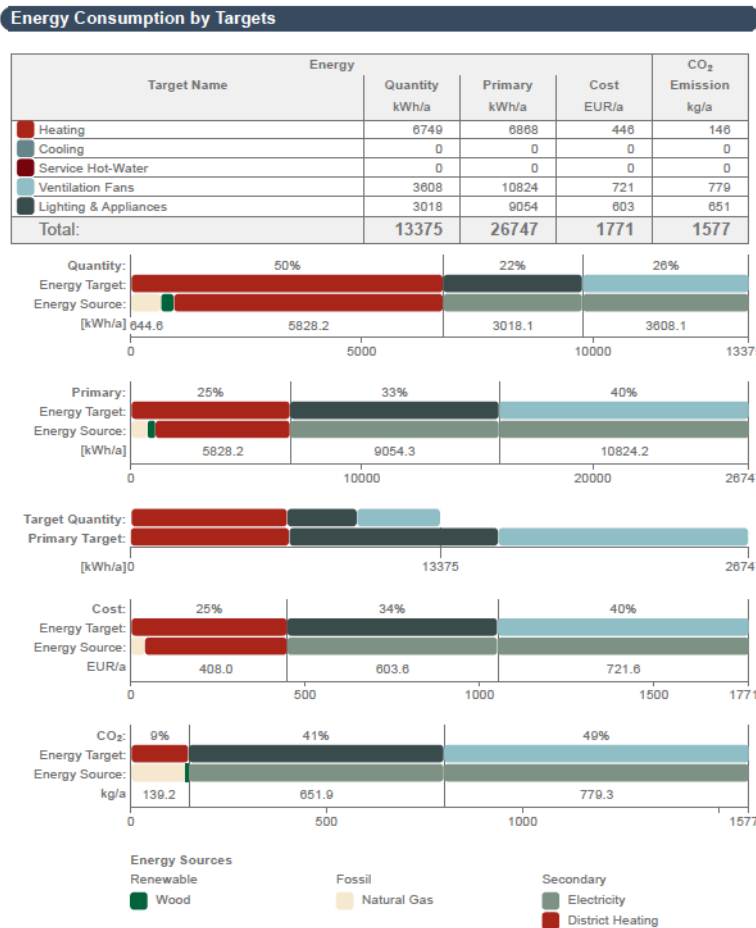
Energy flows that are framed with solid lines in the table above require the input of one or more Energy Sources.

[See Energy Source.](#)

Energy Consumption by Targets

This section of the Evaluation Report contains two tables and several charts.

The table's far-left column lists energy Targets by name, plus their color codes used in the pie chart. The Quantity column lists the magnitude [e.g. kWh/a] while the Cost column shows the price [currency/a] of energy spent on each target in one year. Next to the Primary energy breakdown, the table's far-right column shows the carbon footprints associated with the listed energy target magnitudes.



The Distribution by Targets charts graphically display the percentages of the

- Energy quantities
- Primary energy
- Costs

- Carbon footprints

associated with each energy target.

The Target Energy Quantities by Primary Targets bar chart helps users to compare the magnitude of energy consumption with primary energy consumption.

It is possible to display the Energy Consumption chart in either bar or pie chart format.

[See Report Chapter Settings.](#)

Primary Energy

The primary energy value is the 'common denominator' among different energy source consumption types, when determining the building's total energy consumption. Not only does it indicate the net energy source consumed, but it also incorporates the energy needed for the manufacturing, transportation and the raw material processing of the energy source, as well as its transportation to the place of use. Minimizing the specific primary energy demand is a great way to improve the designed buildings' overall performance.

The primary energy factors assigned to the energy sources differ according to building location.

[See Location Settings.](#)

Use the Energy Source Factors dialog to enter the region-specific data if available, or evaluate the project using the default assignments (based on standard DINV-18599).

Energy Consumption by Sources

This section of the Evaluation Report contains one table and several charts.

Energy Consumption by Sources						
Source Type	Energy		Quantity kWh/a	Primary kWh/a	Cost EUR/a	CO ₂ Emission kg/a
	Source Name					
Renewable		Wood	276	331	0	6
Fossil		Natural Gas	644	709	38	139
Secondary		Electricity	6626	19878	1325	1431
		District Heating	5828	5828	407	0
Total:			13375	26747	1771	1577*

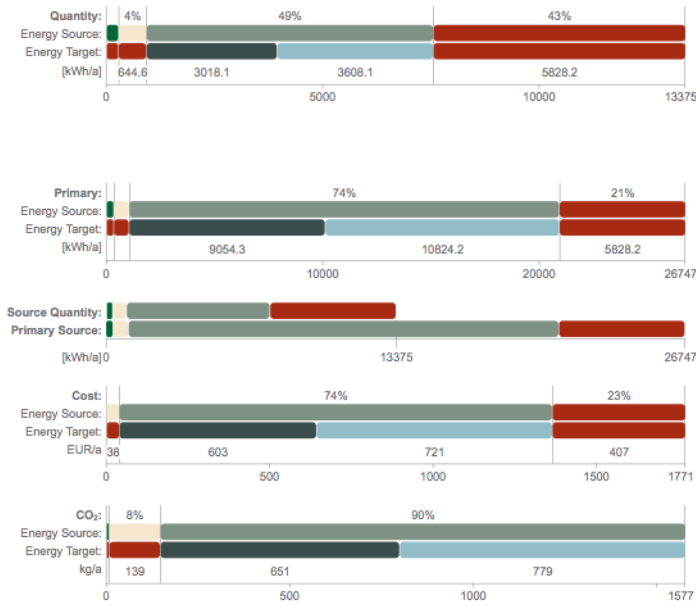
The table's far-left column lists energy Sources by type (Renewable, Fossil and Secondary) and name, plus their color codes used in the pie chart. The Quantity column lists the magnitude [e.g. kWh/a] while the Cost column shows the price [currency/a] of each energy source consumed in one year.

The table's far-right column shows the carbon footprints associated with the listed energy source magnitudes.

The Distribution by Sources charts graphically display the percentage distributions of the

- Energy quantities
- Primary energy
- Costs
- Carbon footprints

of the used energy sources.



The Source Energy Quantities by Primary Sources bar chart helps users to compare the magnitude of energy consumption with primary energy consumption.

It is possible to display the Energy Consumption chart in either bar or pie chart format.

[See Report Chapter Settings.](#)

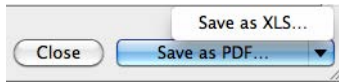
Environmental Impact

This chapter summarizes the environmental impact of the building's operation, by displaying Carbon Footprint and Primary Energy, according to Energy Sources.

Environmental Impact			
Source Type	Source Name	Primary Energy kWh/a	CO ₂ emission kg/a
Renewable	Wood	331	0
Fossil	Natural Gas	709	139
Secondary	Electricity	19878	1431
	District Heating	5828	0
Total:		26747	1577

Reading the Energy Performance Evaluation - XLS Report

Click the button with the triangle in the bottom right corner of the Energy Model Review palette to save the detailed monthly energy balance data in Microsoft Excel file format.



Project Key Values Page

The Project Key Values section of the XLS Energy Performance Report displays basic information regarding the project. Its content is rather similar to the Key Values section of the Energy Performance Evaluation PDF report, yet it contains extra information about Climate Data, Design Loads and Advisory Messages (Errors and Warnings).

Below the Evaluation Date, Project Data such as Name and Location (with coordinates) is displayed, according to the user input on the Location Settings dialog.

Key geometry and shell performance metrics like Gross floor area, Ventilated volume and Outer heat capacity come from the Automatic Model Geometry Analysis.

Calculated heat transfer coefficients ([see U-Value \(R-Value\) Calculator](#)) are listed for the entire building, for every Building structure group ([see Structures Tab Page](#)) and for the openings on the building shell ([see Openings Tab Page](#)).

Heating and cooling system Design Loads are simulation results that indicate gross system magnitudes.

Note: Energy Evaluation in Archicad is not directly capable of providing information for sizing systems and equipment. The design loads under Project Key Values are only displayed for your information. System sizing is only possible with the expert functions of EcoDesigner STAR, available for SSA/Forward customers.

Advisory Messages display the errors and warnings that occurred before and during the simulation.

Project Energy Report Page

The level of data detail displayed on the Project Detailed Results tab page of the XLS Energy Performance Report exceeds the level of detail displayed on the Monthly Energy Balance bar chart of the PDF Energy Performance Report:

Monthly Values	MEP System type	Target	January	December	Annual Total [kWh]
Energy flows					
<i>All numbers are in [kWh]</i>					
Transmission			-2069.83	-1955.56	-14112.61
Infiltration			43.51	-37.05	-219.31
Human heat gain			478.49	446.97	9479.26
Solar gain			1364.97	2292.31	11305.12
Heating			1400.85	1170.49	6749.08
	Central		1400.85	1170.49	6749.08
		Boiler	206.31	209.78	920.86
		Space heating	206.31	209.78	920.86
		Circulation pump	0.00	0.00	0.00
		Solar thermal collector	0.00	0.00	0.00
		Space heating	0.00	0.00	0.00
		Circulation pump	0.00	0.00	0.00
		Heat pump	0.00	0.00	0.00
		Space heating	0.00	0.00	0.00
		Auxiliary systems	0.00	0.00	0.00
		District	1194.53	960.71	5828.22
		Space heating	1194.53	960.71	5828.22
		Circulation pump	0.00	0.00	0.00
	Local		0.00	0.00	0.00
		Boiler	0.00	0.00	0.00
		Space heating	0.00	0.00	0.00
		Circulation pump	0.00	0.00	0.00
		Solar thermal collector	0.00	0.00	0.00
		Space heating	0.00	0.00	0.00
		Circulation pump	0.00	0.00	0.00
		Heat pump	0.00	0.00	0.00
		Space heating	0.00	0.00	0.00
		Auxiliary systems	0.00	0.00	0.00
		Dx heater	0.00	0.00	0.00
		Space heating	0.00	0.00	0.00
		Auxiliary systems	0.00	0.00	0.00
		Not yet specified	0.00	0.00	0.00
Service hot-water heating			0.00	0.00	0.00
		Boiler	0.00	0.00	0.00
		Service hot-water heating	0.00	0.00	0.00
		Circulation pump	0.00	0.00	0.00
		Solar thermal collector	0.00	0.00	0.00
		Service hot-water heating	0.00	0.00	0.00
		Circulation pump	0.00	0.00	0.00
		Heat pump	0.00	0.00	0.00
		Service hot-water heating	0.00	0.00	0.00
		Auxiliary systems	0.00	0.00	0.00
		District	0.00	0.00	0.00
		Service hot-water heating	0.00	0.00	0.00
		Circulation pump	0.00	0.00	0.00
Cooling			-263.37	-247.46	-5468.98
	Central		0.00	0.00	0.00
		Mechanical	0.00	0.00	0.00
		District	0.00	0.00	0.00
	Local		0.00	0.00	0.00
		Mechanical	0.00	0.00	0.00
		DX Cooling	0.00	0.00	0.00
		Not yet specified	263.37	247.46	5468.98
Ventilation			-3029.50	-2484.14	-17350.77
		Mechanical	-3024.24	-2484.14	-17350.77
		Not yet specified or Natural	-5.26	-5.00	-41.36
Lighting and Equipment			262.61	247.54	3018.10
		Lighting	103.92	96.05	1184.98
		Equipments	158.69	151.49	1833.12
Service hot-water			0.00	0.00	0.00
Supplied energy			3506.31	3157.31	26571.86
Emitted energy			-5406.22	-4729.21	-37213.33
Difference			-1899.91	-1571.90	-10641.47
Auxiliary systems			317.95	290.30	3608.06
On-site electricity			0.00	0.00	0.00
		Solar photovoltaic	0.00	0.00	0.00
		Wind energy	0.00	0.00	0.00
		Potential production	0.00	0.00	0.00
		Solar photovoltaic	0.00	0.00	0.00
		Wind energy	0.00	0.00	0.00

- Heating and Cooling are further broken down according to systems and specific target (Space heating, Hot water generation). Auxiliary energies necessary to operate the systems appear separately.
- Lighting and Appliance electricity consumptions are listed separately as opposed to combined.

Note: It is only possible to define so-called Local Heating and Cooling systems with EcoDesigner STAR, available for SSA/Forward customers. Local systems data will therefore always appear as 0 in reports generated with Energy Evaluation in Archicad. The same is true for On-site electricity

Detailed Inputs Page

This tab page displays building geometry information as it appears on the Structures and Openings tab pages. Use these data as input for third-party Excel-based stationary energy calculation software.

Interoperability

- [File Types Opened by Archicad](#)
- [File Types Saved by Archicad](#)
- [Merging Files to Archicad](#)
- [Import/Export Property Data via Spreadsheet](#)
- [Import BIM Content](#)
- [Working with PDF Documents](#)
- [Working with DWG/DXF Files](#)
- [Working with XREFs](#)
- [Working with Point Clouds](#)
- [Working with Rhino 3D Models](#)
- [Working with RFA & RVT Files](#)
- [Working with FBX files](#)
- [Export to FRILO](#)
- [Export to Artlantis](#)
- [Interoperability with MEP Applications](#)
- [Interoperability with Energy Analysis](#)
- [Working with IFC](#)

File Types Opened by Archicad

The file types opened by Archicad are listed here, in the following categories:

[Archicad File Types \(Open\)](#)

[CAD Formats Opened by Archicad](#)

[Image Formats Opened by Archicad](#)

[Model/3D Formats Opened by Archicad](#)

[Other Formats Opened by Archicad](#)

[Read More](#)

Archicad File Types (Open)

Archicad Projects (.pln, .pla, .bpn)

Archicad can open projects from version 17 and up.

You cannot open project files from earlier versions directly in Archicad 27; you have to convert them first.

To convert, open and save the project in Archicad version 17-26. Then you can open the project in Archicad 27.

[See also *Archicad file and protection compatibility*.](#)

Solo Projects (.pln)

These are the main native document type of Archicad. A .pln file includes all model data and the views generated for the project, as well as the default settings, attributes and library references.

Archive (.pla)

Archive files are similar to Solo Projects, but these include all the library parts, background images and linked textures used in the Project, not just references to them.

[For more information, see *Archive Projects*.](#)

Backup Project file (.bpn)

Opening a Backup Project file (.bpn) allows you to recover the last saved version of a damaged project file, provided that the Make Backup Copy checkbox is active in **Options > Work Environment > Data Safety & Integrity**.

To open a backup file, do one of the following:

- Browse and double-click its name in the file system
- From the Archicad Open dialog or other system dialogs: use the dialog's "All file types" filter to include BPN files in the available list

Archicad Project Template (.tpl)

[For details, see *Template Files*.](#)

Archicad 2D Lines (.2dl)

These files are vectorial drawings saved from Archicad's 3D Window (using the Vectorial 3D Engine). These drawings contain no model information and can be touched up in the Floor Plan with 2D tools.

Important note: In the 3D Window, this option is only available with the Vectorial 3D Engine - not available in Hardware acceleration mode.

Module File (.mod)

[For details, see *Create Module \(.mod\) File*.](#)

PlotMaker Layoutbook (.lbk)

PlotMaker Drawing (.pmk)

CAD Formats Opened by Archicad

DWF File (.dwf)

Drawing Web Format, a 2D vector graphics format created by Autodesk.

The DWF file will contain all drawing elements with some exceptions and limitations:

- pictures: due to format limitations, only straight pictures can be written to a DWF file,
- cameras will not be included.

DXF/DWG File (.dxf, .dwg)

Archicad opens all DWG file formats starting with AutoCAD 2000.

If you choose this file type, the Settings button appears in the Open dialog box, which opens the DXF-DWG Translation Setup dialog box.

For details, see [Opening DWG/DXF Files](#).

However, in most cases you need not adjust these translation settings, because the default setup will give you the desired result when opening a DXF/DWG file.

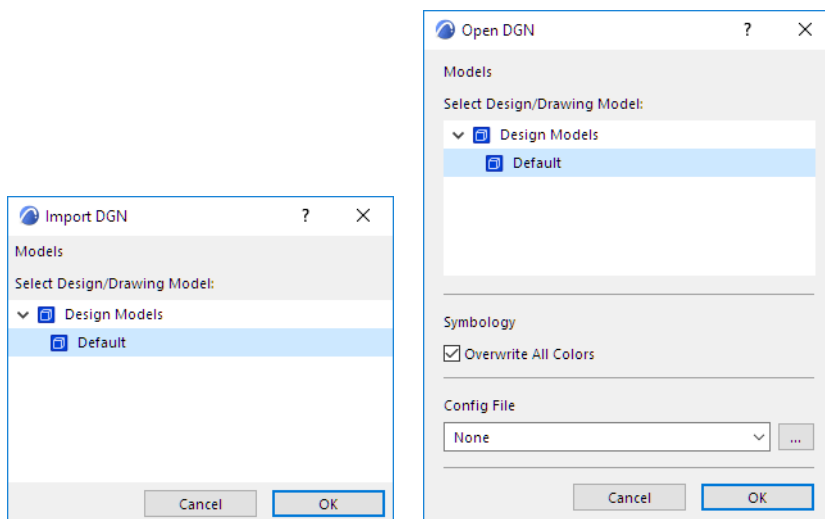
Note: If you are working in AutoCAD and open a DWG that was originally saved from Archicad, an Alert may appear. However, you should go ahead and work with the saved DWG in AutoCAD - no stability problems should result.

MicroStation Design File (.dgn) - Version 7 or 8

Do one of the following:

- Use **File > External Content > Place External Drawing** and select a MicroStation Design File (.dgn) file (version 7 or 8)
- Use **File > Open** and select a MicroStation Design File (.dgn) file (version 7 or 8)
- Drag and drop a .dgn file to import into the current Archicad project.

Set options in the **Open DGN** dialog or **Import DGN** dialog, as applicable:



Design/Drawing Model

Use the pop-up to select one of the models in the DGN model (if it contains multiple models).

Symbology

Overwrite All Colors: The Archicad Pen Set is overwritten using the DGN color table.

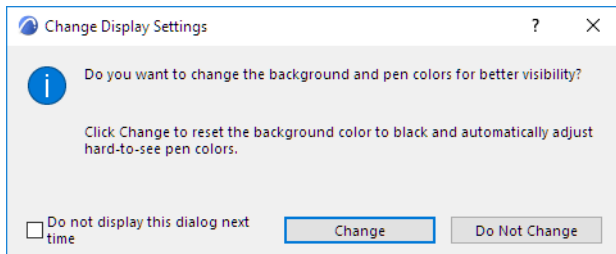
Note: If you check this box, and if you use a configuration file, its definitions will overwrite the Archicad Pen Set.

Config File

Optional. A default configuration file (txt) is available in the Archicad Defaults folder. Use or customize a config file to ensure company-standard conversions.

Background Colors Visibility

If you used **File > Open** to open the DGN file: a dialog will appear with the option of changing the background color to black.



See also [Save as MicroStation Design File \(.dgn\) - Version 7.](#)

Image Formats Opened by Archicad

All Images (.bmp, .dib, .rle, .jpg, .jpeg, .jpe, .jfif, .gif, .tif, .tiff, .png, .hdr)

Various types of picture files can be opened as model pictures, but they will appear in their own separate windows, not the Archicad main window.

Model/3D Formats Opened by Archicad

3D Studio File (.3ds)

3D files can be saved in 3D Studio format. The Add-on required for this is part of the Archicad package. 3D files in 3D Studio format can be opened by Archicad only if you install the Goodies for your language version of Archicad.

[See Goodies.](#)

IFC (.ifc), IFCXML (.ifcxml), IFCzip

IFC stands for Industry Foundation Classes.

Archicad supports the latest and most widely used IFC formats.

[To read more, see Working with IFC.](#)

Rhino (.3dm) - Import Model or Import as Object

Open or merge a **Rhino 3D Model** file (.3dm) directly in Archicad with one of these commands:

- **File > Open**
- **File > Interoperability > Merge** (from File or BIMcloud)

To import a Rhino file as a single Archicad Object:

- **File > Libraries and Objects > Import Rhino 3DM file as Object**

[For further information, see Working with Rhino 3D Models.](#)

SketchUp (.skp)

To open as a new Archicad project:

- Use **File > Open**, then browse for the .skp file.

To add to a running Archicad project:

- **Drag-and-drop** the Sketchup file into Archicad, or
- use **File > Interoperability > Merge**, then browse for the .skp file.

The Sketchup model will be saved as a GDL object to the Embedded Library.

[See also SketchUp: Export and Import Limitations.](#)

Google Earth (.kmz)

Archicad imports a Google Earth model as follows:

- .kmz objects become GDL objects, placed on the Archicad Layer
- .kmz polygons become Fills, placed on the 2D Drafting General Layer
- Placemarks become GDL objects, placed on the Archicad Layer

Important: Make sure that the merged Google Earth model to be opened

- contains only a single location pin
- uses the Absolute method to define its altitude (see below).

To open a .kmz file directly in Archicad:

1. Use the File > Open command
2. Choose the .kmz file type to open
3. Click Settings to open KMZ Import Options.

- Check the box to import any 'look at viewpoints' as cameras, if the .kmz file contains them.
- Create Hotspots for Objects: Use these checkboxes to control where Hotspots should appear on the imported model.

KMZ Import Optimization

Use these two checkboxes if importing the Google Earth model is taking too long.

- **Collect 3D geometry placemarks into a single Library Part:** If you check this, Archicad will generate a single GDL object for multi-geometry placemarks (instead of separate ones). This can reduce the time it takes to import the .kmz file.
- **Display 3D geometry placemarks without fills:** This option can speed up navigation in large models.

Altitude Settings

In Google Earth, there are several ways to export altitude data, but Archicad supports only the "Absolute altitude" method.

If your .kmz file contains alternative altitude settings (either "Clamped to ground" or "Relative to ground"), an information dialog appears: you should reexport the model from Google Earth using the Absolute altitude method.

See also [Google Earth: Export and Import Limitations](#).

Stereolithography (.stl)

Use the **File > Open** command or **File > Interoperability > Merge** to open or merge an .stl file directly in Archicad. An opened or merged STL model will be opened as a Morph in Archicad.

STL files only carry the 3D model's surface geometry, divided into triangles, without any additional information such as color, surfaces etc.

The format is usually used for rapid prototyping, 3D printing, and computer-aided manufacturing. STL format has two kind of representation: ASCII and Binary

- ASCII STL
An ASCII STL file lists the triangles with the coordinates of their nodes in a readable format for any text editor application.
- Binary STL
In a Binary STL file the data is compressed but only readable for the appropriate STL application/machine. Binary representation is more commonly used due to its smaller file size.

More about STL format: [http://en.wikipedia.org/wiki/STL_\(file_format\)](http://en.wikipedia.org/wiki/STL_(file_format))

For information about saving an Archicad model to STL format, see [Stereolithography \(.stl\)](#).

Open STL file

1. Go to **File > Open > Open file**
2. Select Stereolithography File as file type
3. Browse for the STL file
4. Click Open

Merge STL file into existing model

1. Go to **File > Interoperability > Merge**
2. Select Stereolithography File as file type
3. Browse for the STL file

4. Click Open

Limitations

1. Archicad does not check the exported STL file for errors, e.g. in terms of solidity. These are usually done by the 3D printer.
2. STL file does not store the scale and unit of the model, these have to be set in the viewer/printer. Archicad always exports 1 m as 1 STL unit.
3. If the imported geometry is very complex, it may affect the performance of Archicad. In this case, you will see a warning dialog.
4. Note that if the geometry of the model is overly complex, there is a chance that the import will fail and no object will be generated.

FBX (.fbx)

Open or merge an FBX file (.fbx) in Archicad with one of these commands:

- File > Open (opens a new model in Archicad)
- File > Interoperability > Merge (adds the model to a running Archicad project)
- File > Libraries and Objects > Import FBX as GDL Object (imports the model without placing it in the project)
- Drag-and-drop

At import, the FBX model is saved as a GDL Object to the Embedded Library.

To read more, see [Working with FBX files](#).

Other Formats Opened by Archicad

HPGL File (.plt)

For details, see [Plot](#).

Windows Enhanced Metafile (.emf)

Windows Metafile (.wmf)

Point Cloud Data File (.e57)

See [Working with Point Clouds](#).

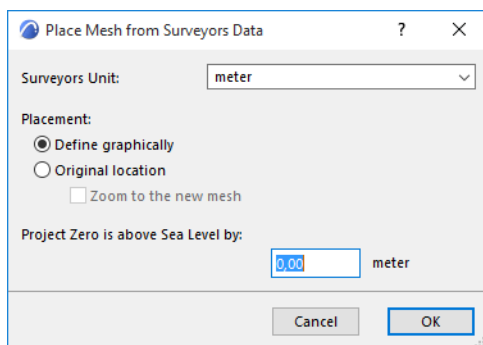
Surveyors' data (.xyz)

Import any text file containing xyz data (e.g. surveyor's data), or similar data, into Archicad, creating a Mesh element automatically.

Note: To convert .xyz files to Point Cloud objects placed in Archicad, see [Working with Point Clouds](#).

Note: The text file must consist of rows of data, with each row containing three numerical entries. The row can be optionally preceded by an additional numerical ID. Use separators between the data entries. Separators must be used consistently. The separator can be a semi-colon or a white space; you can also use commas or periods, provided that you do not also use the same character as a decimal marker in this file. Any row in which the 3 data (or 3+1, in case of ID's) is preceded or followed by a non-numerical data entry is considered a "comment" and will not be calculated. If the data in the text file is entered in an inconsistent format, a warning will appear that the file format cannot be read.

1. Click **File > Interoperability > Place Mesh from Surveyors Data**.
2. In the appearing directory dialog box, browse for a .txt or .xyz file containing the data you need.
3. The **Place Mesh from Surveyors Data** dialog box appears:



- **Surveyors unit:** Enter the unit for the survey data.

Placement:

- **Define graphically:** You will place the Mesh with a mouse-click in the desired location.
- **Use coordinates of data file:** The new Mesh will be placed according to the zero-point as defined in the surveyors data file.

In this case, you can optionally **Zoom to the new mesh** as soon as you place it.

Project Zero is above Sea Level by: Enter the elevation of your Project Zero above Sea Level (in the Surveyors Unit you have defined). Archicad will take this into account when converting the z-values of the surveyors data file into the Mesh.

File Types Saved by Archicad

The file types saved by Archicad are listed here, in the following categories:

[Archicad File Types Saved by Archicad](#)

[CAD Formats Saved by Archicad](#)

[Image Formats Saved by Archicad](#)

[Model/3D Formats Saved by Archicad](#)

[Document and Other Formats Saved by Archicad](#)

Archicad File Types Saved by Archicad

Archicad Projects (.pln)

Archicad supports unicode file names of essentially unlimited length.

Archive (.pla)

For details, see [Archive Projects](#).

Archicad Project Template (.tpl)

Choose this option if you want to save the current project settings for future use as a template.

For details, see [Template Files](#).

Archive Options

For details, see [Archive Projects](#).

Project (.pln) in an earlier Archicad version format

Archicad 27 can back-save project files in Archicad 26 format only (among previous Archicad file formats).

To back-save:

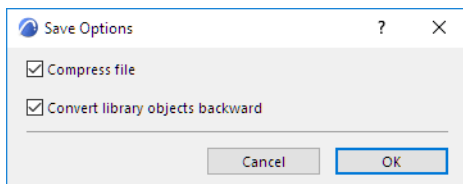
1. Use **File > Save as...**
2. In the **Save** dialog box, choose **Archicad 26 Project** from the **Save as type** list.

Backward Compatibility Supported for Library Parts

When back-saving your Archicad project, any library parts that were revised for the latest version will be automatically replaced by their counterparts in the previous version.

However, if the library part in the latest-version project is using new parameters or functions that did not exist in the previous version, that library part will not be replaced when you back-save the project: it will be a missing library part.

In the **Options** of the Save as dialog box, you can uncheck the **Convert library parts backward** checkbox. This way, the library parts revised for the latest version will not be replaced by their original counterparts in the previous version, and will be shown as “missing”.



BIMx File Format/BIMx Hyper-model

For details, see [Graphisoft BIMx](#).

PlotMaker Drawing (.pmk)

Use the Publisher to save a model view in PMK file format.

For details, see [Publisher File Formats](#).

Module File; Module File from Clipboard (.mod)

For details, see [Create Module \(.mod\) File](#).

GDL Script (.gdl)

This option will save your model in GDL script.

For details, see [GDL Object Editor](#).

Archicad 2D Lines (.2dl)

These files are vectorial drawings saved from Archicad's 3D Window (using the Vectorial 3D Engine). These drawings contain no model information and can be touched up in the Floor Plan with 2D tools.

Important note: In the 3D Window, this option is only available with the Vectorial 3D Engine - not available in Hardware acceleration mode.

CAD Formats Saved by Archicad

DWF File (.dwf)

You can save Floor Plan and Section/Elevation/IE or 3D Document views in DWF (Drawing Web Format):

Activate the required story or Section/Elevation/IE or 3D Document window and choose the **Save as** command in the **File** menu. Select the DWF file format from the list, click the **Save** button and choose one of the following options:

- ASCII - readable and editable ASCII text file. May be rather long.
- Binary - binary file with a smaller size than the ASCII version.
- Compressed Binary - smallest, most easily transportable compressed binary file.
- You can view the saved DWF file with any web browser that supports the DWF Plug-In. You can download a DWF viewer from: <https://www.autodesk.com/products/design-review/download>. A Java-based public domain viewer - with some limitations - can be found at: <http://www.cadviewer.com>.

DXF/DWG Files (.dxf, .dwg)

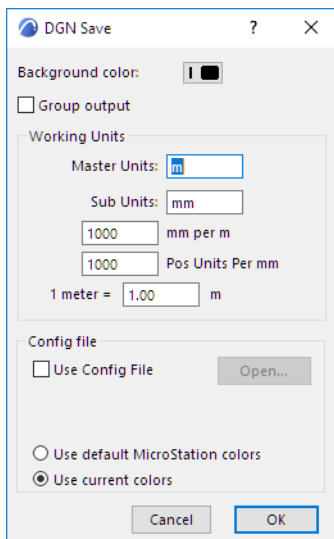
For details, see [Saving DWG/DXF Files](#).

MicroStation Design File (.dgn) - Version 7

Use **File > Save as...**, and choose the MicroStation Design File (.dgn) file format.

The Archicad to MicroStation converter exports 2D views (plan, elevation and section windows) from Archicad into 2D MicroStation format (version 7 only).

In the appearing dialog:



Specify the background color and Working Units for the exported file.

All elements are exported as drawing primitives (points, lines, arcs, texts, polylines, triangles, and polygons).

Config File

During the output, you are prompted to designate an optional configuration file (.txt), which controls the layer assignment, line type, font and color conversion.

- The DGN v7 format is limited to 64 layers. If the Archicad file contains more, then it rolls around. The configuration file can control the correspondence between layers. In the absence of the configuration file, the default order is alphabetical.
- Pen color conversion: Defined in the configuration file. Otherwise, the Add-on will use the default assignments.
- Font conversion. If the configuration file is missing, the add-on will default to the international vector font (always present in MicroStation).
 - If you define a Chicago-Chicago translation, make sure that the Chicago TrueType font is loaded into MicroStation (with the **Element > Text > View > File > Import** command). If it is not loaded, it will revert to the default.

If you would like the text to look the same in both files, use monospaced fonts (e.g., Courier) on the plan.

Symbol hatches and linetypes are not exported in this version.

After the DGN file is loaded into MicroStation, you must use the **Full View** command to see the entire drawing; it will not adjust the scale automatically.

[See also Open *MicroStation Design File \(.dgn\) - Version 7 or 8.*](#)

Image Formats Saved by Archicad

Windows BMP Image (.bmp)

Click Options to set output preferences.

JPEG Image (.jpg)

Click Options to set output preferences.

GIF Image (.gif)

Click Options to set output preferences.

TIFF Image (.tif)

Click Options to set output preferences.

PNG Image (.png)

Click Options to set output preferences.

Model/3D Formats Saved by Archicad

IFC (.ifc), IFCXML (.ifcxml), IFCzip

IFC stands for Industry Foundation Classes.

To read more, see [Working with IFC](#).

Structural Analytical Format (.xlsx)

Use this to export the model to structural analysis programs.

See [Collaboration with Structural Analysis Software](#).

Rhino (.3dm) - Save

From the 3D window, use **Save as Rhino 3D Model** (.3dm) to export part or all of the Archicad model to Rhino.

See also [Working with Rhino 3D Models](#).

Revit 3D Model (.3dm) - Save

From the 3D window, use **Save as Revit Project file** (.rvt).

See also [Export 3D Model as Revit 3D Geometry](#).

WaveFront File (.obj)

You can save 3D models and their surfaces in the WaveFront format, a photorealistic rendering/animation system.

Two output files are generated: one for the geometry and one for the surface material definitions, according to the WaveFront standards. The .mat file is automatically created in the same folder that was selected in the Save as dialog box for the .obj file.

The document you save will include the following information:

- 3D model geometry (.obj file): All geometric surfaces. No edges in the WaveFront format.
- Archicad surfaces (.mat file): All surface attributes used in the model, identified by name, with full characteristics.

After choosing the .obj file format, click Save. The WaveFront dialog box opens:

- **Layers-Surfaces:** Click this radio button to create groups in the WaveFront file for each Archicad layer, and within these groups, create internal groups for each surface.
- **Elements-Surfaces:** Click this radio button to create a group in the WaveFront file for each Archicad construction element, and within these groups, create internal groups for each surface.

Set drawing unit to: In this field, you can define the distance in the WaveFront project that will be translated into a drawing unit in Archicad or vice versa. As Archicad works with metric data internally, setting the drawing unit to 1 meter (=1000 mm) will result in a 1:1 model after the conversion.

3D Studio File (.3ds)

3D files can be *saved* in 3D Studio format. The Add-on required for this is part of the Archicad package.

3D files in 3D Studio format can be *opened* by Archicad only if you install the Goodies for your language version of Archicad.

See [Goodies](#).

Saving in .3ds format

With the 3D window in front you can save 3D models of your projects in Autodesk's 3D Studio 3DS photorealistic rendering and animation file format.

The resulting document will include the following information:

- **3D model geometry:** All surfaces and all edges.
- **Surfaces:** All surfaces used in the project, including the name and the characteristics.
- **Light sources:** Both the sun and the additional light sources are included. Sun will be handled as a spotlight very far away from the project.
- **3D projection:** The current perspective settings and Fly Through paths are included.
- **Shadow casting information:** The current shadow casting setup in the PhotoRendering Settings/Effects dialog box of Archicad is included.
- **Smooth (welding) information:** The smoothing setup in the PhotoRendering Settings/Effects dialog box of Archicad is included.

After choosing the .3ds file format, click Save. The Save 3D as 3D Studio object dialog box opens:

- **Construct 3D Studio objects according to Archicad objects:** Click this button to create a 3D Studio file without grouping.
- **Construct 3D Studio objects according to Element types and Surfaces:** Click this button to create a group in the 3D Studio file for each Archicad construction element, and within these groups, create internal groups for each surface.
- **Construct 3D Studio objects according to Layers and Surfaces:** Click this button to create groups in the 3D Studio file for each Archicad layer, and within these groups, create internal groups for each surface.
- **Construct 3D Studio objects according to Surfaces:** Click this button to create a group in the 3D Studio file for each surface.
- **Drawing Unit:** In this field, you can define the distance in the Archicad project that will be translated into a drawing unit in 3D Studio. As Archicad works with metric data internally, setting the drawing unit to 1 meter (=1000 mm) will result in a 1:1 model after the conversion.
- **Colors:** You can choose either the Archicad pen colors, or match the Archicad colors to a similar 3D Studio color.
- **Archicad texture information:** You can select whether to include surface texture information in the 3D Studio file.
- **Object = Group Name:** You can select whether the object's name should be a generated unique ID or the group's name.

Note that 3D Studio works with names up to 10 characters. For names generated by Archicad the convention is the following:

- **Chars 1 to 7:** unique ID or clipped layer/surface name.
- **Chars 8 to 9:** used for a distinctive number to avoid identical names generated by clipping. Up to 100 variations of the same name are allowed this way.
- **Char 10:** used for slicing information. 3D Studio can handle up to 65,536 polygons by object. If an object has multiple polygons, Archicad will split the object into several objects and put the segment number into Char 10.

ElectricImage File (.fact)

You can save 3D models of your projects in the FACT format for ElectricImage Animation System (EIAS), a photorealistic rendering/animation system.

The document you save will include the following information:

- **3D model geometry:** All surfaces. (No edges in the FACT format.)
- **Surfaces:** All surfaces used in the project, including the name and characteristics.
- **Light sources:** Both the sun and the additional light sources are included.
- **3D projection:** The current perspective settings are included.

With the 3D window in front, choose **File > Save as**, or **Document > Creative Imaging > Create Fly-Through**. The Save 3D FACT Document dialog box will appear:

- **Grouping hierarchy according to Archicad Layers:** Click this button to create groups in the FACT file for each Archicad layer.
- **Grouping hierarchy according to Archicad Elements:** Click this button to create a group in the FACT file for each Archicad construction element. Do NOT use this grouping option for large models. EIAS is not very efficient when working with a large number of groups.
- **Grouping hierarchy according to Archicad Surfaces:** Click this button to create groups in the FACT file for each Archicad surface.
- **Color Blend, Super Sample, Self Shadowing:** Of the numerous FACT group attributes, these three can be controlled from within Archicad. To learn more about these and other group attributes, see the EIAS documentation.
- **Output Format:** You can choose to save your file in either ElectricImage 2.1 or 2.9 format.
- **Set Drawing Unit to:** In this field, you can define the distance in the Archicad project that will be translated into a drawing unit in ElectricImage. As Archicad works with metric data internally, setting the drawing unit to 1 meter (=1000 mm) will result in a 1:1 model after the conversion.

Note: You do not need to use the Transporter application required for previous versions. Archicad automatically transforms the coordinate system to match the EIAS specification.

VRML File (.vrl)

You can save 3D models of your projects in the VRML format. VRML is an acronym for the Virtual Reality Modeling Language. Using VRML you can build your own three-dimensional virtual worlds and link them together on the World Wide Web. To view VRML worlds, you need a VRML helper application (or plug-in) called VRML browser. Check both your HTML and VRML browsers manual to see how to configure them.

The document you save will include the following information:

- **3D model geometry:** All surfaces as IndexedFaceSet elements including surface attributes.
- **Surfaces:** All surfaces used in the project. Texture data is included; texture coordinates will be computed by the VRML browser. Check your browser manual to see what texture image file formats it supports.
- **Light sources:** Additional light sources are included. If no additional lights are placed you have to use your VRML browser's default headlight.
- **3D projection/Animation path:** The current perspective settings are included.

Set up a 3D view and with the 3D window at the front, choose **Save as** from the **File** menu and choose the VRML format, or place any number of cameras on the Floor Plan and choose **Create Fly-Through** and set the Result to VRML File, then click Save. The Save VRML 3D Document dialog box will appear:

- **Copy textures to destination folder:** Check this box to export the surface textures used by the Archicad model.
- **Write vertex normals:** Check this box to export normal vectors.

With these options active, you can export a sophisticated and complex model. If the viewer application does not display the model correctly, try exporting again with one or more checkboxes inactive.

Lightscape File (.lp)

You can save 3D models of your projects in the native file format of Lightscape, a visualization system for generating lighting simulations of three-dimensional models using a radiosity algorithm.

The document you save will include the following information:

- **3D model geometry:** All surfaces.
- **Surfaces:** All surfaces used in the project, including the name and characteristics.
- **Textures:** Exported surfaces will refer to the appropriate texture according to the current settings of the surface dialog. Texture images are not included in the output file: however, the path of the first texture file is included. If all the image files are collected into a single Archicad library folder, the visualization system will automatically locate them. Otherwise, you will have to collect and copy these image files into the appropriate folder of the visualization system.

Note: Use DOS filename conventions for naming texture image files when working with Lightscape.

Before saving the Lightscape model, open the Surfaces dialog box, check the image names and make the necessary changes.

- **Light sources:** All light sources used in the project, except the sun. Shadow casting information is also exported according to the current settings of the PhotoRendering effects dialog box.
- **Projection information:** 3D projection data exported as the default view in Lightscape. Only perspective views can be used. If you choose an axonometric view, the saving process will report an error.

Set up a 3D perspective view in Archicad, generate the view, then choose **File > Save as**. In the list, choose the Lightscape document format, and click the **Save** button. The Save as 3D Lightscape Object dialog box appears:

- **Construct Lightscape objects according to Layers:** Click this radio button to create groups for each Archicad layer in the Lightscape file.
- **Surfaces:** Click this radio button to create groups for each Archicad surface in the Lightscape file.

Decompose elongated triangles: Check this box if you want to decompose elongated triangles into smaller triangles. This will generate a more detailed mesh, which is useful for Lightscape's radiosity algorithm.

Note that in the case of very large scenes, it may excessively increase the number of polygons (and the file size).

U3D File (Windows Only)

[See 3D Content in PDF.](#)

SketchUp (.skp)

From the 3D window, use **File > Save as** to save the Archicad model in .skp format.

In the appearing Sketchup Export Options dialog box:

- Check which elements and layers to export; use these options to optimize the size of the exported model
- Choose the particular Sketchup format you want to save to.

The resulting SketchUp file's global location coordinates will correspond to Archicad's project location coordinates.

The pin in the SketchUp file (the location reference point) corresponds to the Archicad project origin.

[See also *SketchUp: Export and Import Limitations*.](#)

Google Earth (.kmz)

There are two ways to save an Archicad model to Google Earth:

1. Use **File > Interoperability > Send Model to Google Earth**.

The model is saved as a .kmz file and opened in Google Earth in one step. You must have Google Earth 4 or newer installed on the computer.

Note on Altitude: Use "Set Altitude as" options in the **3D Model Export Options** in Google Options to determine how your Archicad model will be placed into Google Earth.

2. From the 3D window, use **File > Save as** to save the Archicad model in .kmz format.

In the Save as dialog box, click **Options** to see export options specific to .kmz format:

- Choose whether to export the active Archicad camera as the 'look at viewpoint' in the kmz file.
- Set altitude using one of two methods: Absolute Altitude (above sea level) or Clamped to Ground. "Absolute" means that your model will be placed at its exact altitude, as defined in Archicad. "Clamped to ground" means that your model will be placed on the ground at the defined location (similar to Archicad's "Gravity" function).

[See also *Google Earth: Export and Import Limitations*.](#)

Stereolithography (.stl)

Use the **File > Save as** dialog to save an Archicad model to STL format.

STL files only carry the surface geometry of a 3D model without any additional information such as color, surface etc. It contains the surface divided into triangles.

The format is usually used for rapid prototyping, 3D printing, and computer-aided manufacturing.

Save to STL format

1. Open 3D window with only those elements visible that you wish to export.
2. Go to File > Save as
3. Select Stereolithography File as file type
4. Browse for the destination folder
5. Enter the file name
6. Click Save
7. In the Save 3D as STL dialog, choose from Binary or ASCII representation.

- ASCII STL

An ASCII STL file lists the triangles with the coordinates of their nodes in a readable format for any text editor application.

- Binary STL

In a Binary STL file the data is compressed but only readable for the appropriate STL application/machine.

Binary representation is more commonly used due to its smaller file size.

More about STL format: [http://en.wikipedia.org/wiki/STL_\(file_format\)](http://en.wikipedia.org/wiki/STL_(file_format))

For information about opening or merging an STL file to Archicad, see [Stereolithography \(.stl\)](#).

Limitations

1. Archicad does not check the exported STL file for errors, e.g. in terms of solidity. These are usually done by the 3D printer.
2. STL file does not store the scale and unit of the model, these have to be set in the viewer/printer. Archicad always exports 1 m as 1 STL unit.

Twinmotion

Save the Archicad model to Twinmotion, using the Save As... Twinmotion (.tma) command from the 3D window.

Artlantis

Export the 3D model to Artlantis version 2020 or later (*.atl).

See [Export to Artlantis](#).

FBX (.fbx)

Saving the Archicad model to FBX results in a single FBX file containing geometry, surface and texture information.

1. Activate the 3D window.
2. Use File > Save as.
3. Choose the FBX file format, then click Save.

To read more, see [Working with FBX files](#).

Document and Other Formats Saved by Archicad

PDF (.pdf)

Click Page Options and Document Options to set output preferences. These options will vary depending on the active window from which you are saving.

For details, see [Save Document in PDF Format](#).

Windows Enhanced Metafile (.emf)

Click Options to set output preferences.

Windows Metafile (.wmf)

Click Options to set output preferences.

Merging Files to Archicad

Use the following commands:

- **File > Interoperability > Merge from file**
- **File > Interoperability > Merge from BIMcloud**

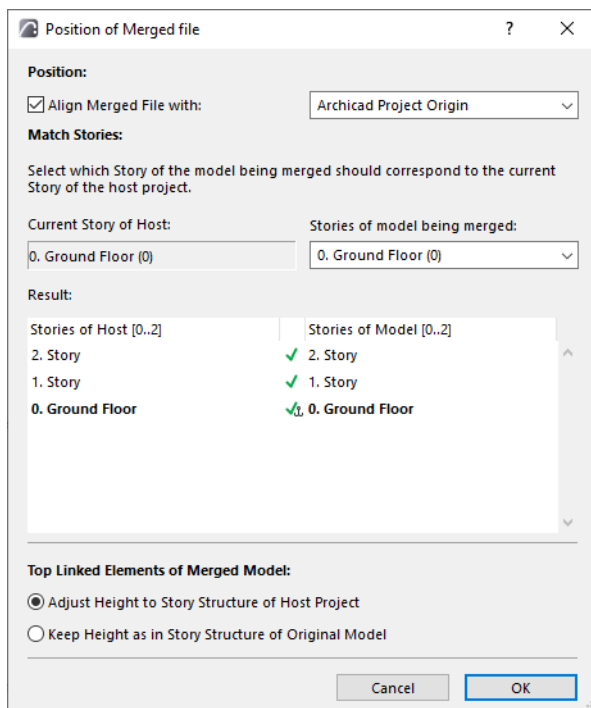
If needed, log in to BIMcloud. [See *Log in to BIMcloud*](#).

Merge the elements of another Archicad Project (version 17 and up), a Module, DWF/DWG/DXF, HPGL (.plt) file, IFC file, Rhino (.3dm) or other image or model-type file into Archicad.

- Files can be merged into either the current Floor Plan, the currently open Section/Elevation/IE, 3D Document or Detail/Worksheet window.
- When merging a Module into a Section/Elevation/IE or 3D Document window, only 2D Elements will be pasted.
- When merging a Project or Module, you can drag, rotate and mirror it before placing it to the final location.

Merging and Story Structure

When merging a Project or Module file, Archicad matches the stories of the merged (imported) file to the current (open) one. When you merge a multistory building into your current Floor Plan, Archicad will ask you to define which story from the imported file will match the story you are working on, and it will also suggest a possible match.



If you do not have enough stories in your current plan to accommodate all the merged ones, Archicad will automatically create the missing stories.

Note: You cannot merge multistory elements or modules into 2D windows other than the Floor Plan.

Note: There is a difference between *merging* a multistory project, and placing a multistory *Hotlink Module*. If the host project contains fewer stories than the Hotlink Module you are placing, the module stories

which do not “fit” in the host project will not be placed. To resolve this, create additional stories as needed in the Host project.

Top Linked Elements of Merged Model

- **Adjust Height to Story Structure of Host Project:** Top Linked elements will stretch or shrink to match the corresponding story structure of the host file. (Doors/Windows whose sill or header is linked to a particular story will adjust to the current file’s story structure.)
- **Keep Height as in Story Structure of Original Model:** Top Linked elements will not stretch (they retain their original heights). (Doors/Windows whose sill or header was linked to a particular story in the merged file will lose the story link, and instead will be linked to the Wall base.)

Merging and Attributes

Since attributes (Layers, Surfaces, Lines, Fills, etc.) are identified by their names, the following rules are observed:

- If the name of the attribute is the same in the two Archicad Projects, the merged (imported) elements will inherit the attributes of the *current* Project.
- If the name of an attribute for the merged Project is not present in the current one, this attribute is *appended* to the current Project’s attribute set to accommodate the new names.

Merge DXF/DWG File

See [Merge DXF-DWG](#).

Merge IFC File

For details, see [Import IFC Model to Archicad](#).

Merge Stereolithography (.stl) File

See [Stereolithography \(.stl\)](#).

Merge SketchUp (.skp) File

Use File > Interoperability > Merge to merge content from an .skp file into an existing Archicad model. The Floor Plan must be active in Archicad.

Since the Archicad model already contains project location information, you are asked in a dialog how to handle the two sets of coordinates (Archicad and SketchUp):

- Overwrite the Archicad information with the SketchUp information
- Ignore the SketchUp information. Your Project Location information remains as-is, and you can place the imported object anywhere you wish: drag the appearing boundary box and click to place.
- Place the incoming information relative to the existing coordinates. The imported object will be placed in its true location relative to your project origin.

Step by Step Guide for Merging a Google Earth Snapshot into Archicad using SketchUp

1. Open a project in Archicad
2. Open the Location Settings dialog, check that it has the desired longitude and latitude set
3. Press the ‘Export’ button
4. Set the format to ‘SketchUp file’
5. Give a new file name and press ‘Save’
6. Close the Location Settings dialog.

7. Using SketchUp (downloadable from <http://www.sketchup.com>) open the file saved in step 5
8. Open up the 'Model Info' dialog and check that the 'Geo-location' is set
9. If not (if using Mac), click the 'Set Manual Location...' button: the correct longitude and latitude should be displayed; press OK
10. Use SketchUp > File > Geo-Location > Add More Imagery... to open the 'Add Location' dialog box; the Google Earth image should be centered at the Archicad origin
11. Zoom in or out as desired
12. 'Select 'Region' and 'Grab' it
13. The same region should now be visible in the SketchUp file viewer
14. Save the changes to the .skp file
15. Return to your Archicad project file
16. With the 'Ground floor' view open, select File > Interoperability > Merge
17. Select the .skp file you just saved in step 14
18. A dialog box containing three toggle buttons should appear. Choose the 'Place the new geometry relative to existing coordinates' option and press OK
19. The Google Earth snapshot should now appear in the Archicad project
20. Open a 3D view and fit to see the snapshot in detail

See also [SketchUp: Export and Import Limitations](#).

Merge Google Earth (.kmz) File

Use File > Interoperability > Merge to merge content from a .kmz file into an existing Archicad model. The options are similar to the Open Google Earth file process.

See [Google Earth \(.kmz\)](#).

Location Information

Since the Archicad model already contains project location information, you are asked in a dialog how to handle the two sets of coordinates (Archicad and Google Earth):

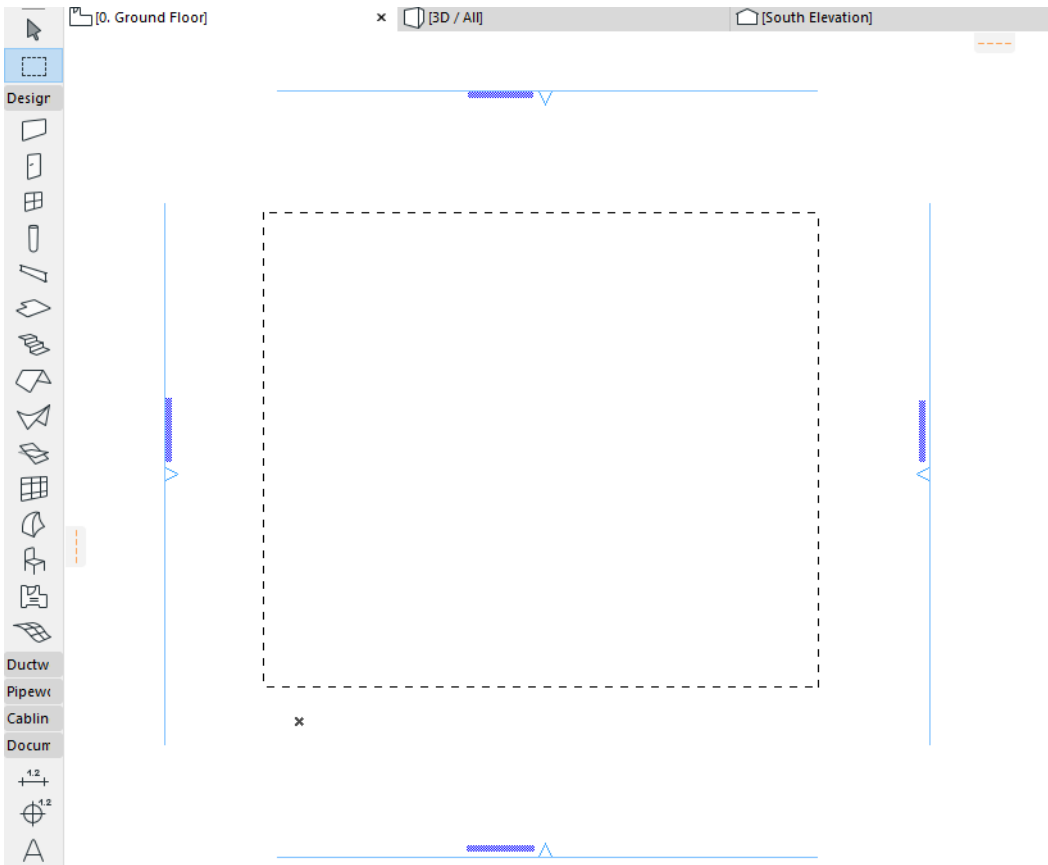
- **Overwrite** the Archicad information with the Google Earth information
- **Ignore** the Google Earth information. Your Project Location information remains as-is, and you can place the imported object anywhere you wish: drag the appearing boundary box and click to place.
- **Place** the incoming information relative to the existing coordinates. The imported object will be placed in its true location relative to your project origin.

See also [Google Earth: Export and Import Limitations](#).

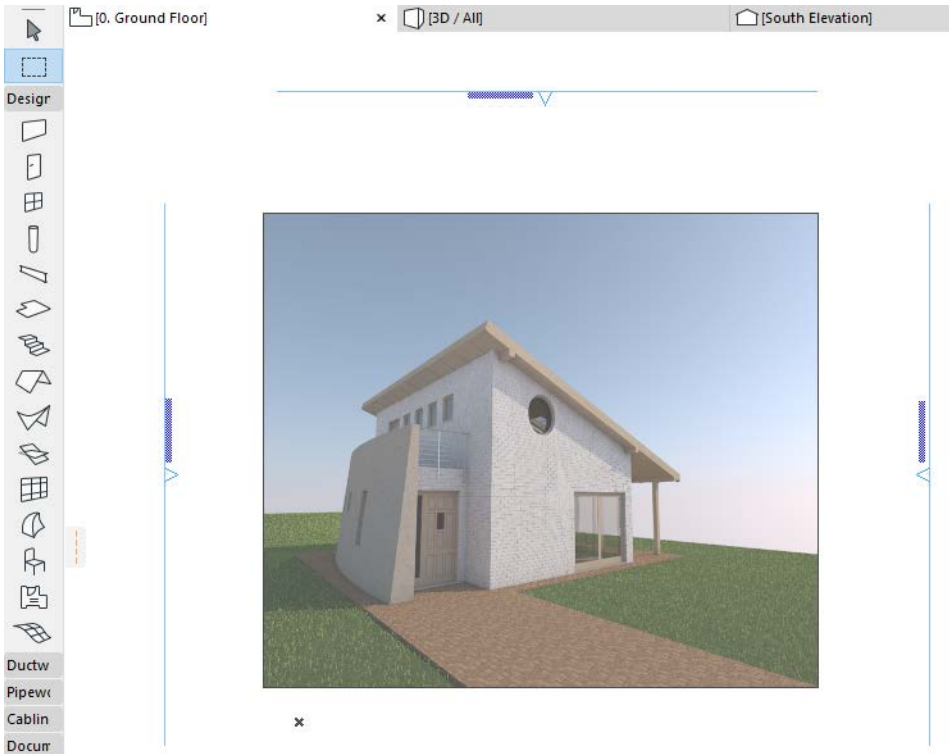
Merging Image-Type Files

You can merge an image-type file into your Archicad Project. (These include documents created in drawing or painting programs; 3D views and elevations saved in picture format; PhotoRenderings saved only in picture format.) Before choosing **Merge** for an image-type file, you can define its size on the plan in advance: Choose the **Marquee** tool and draw a Marquee of the desired size and position.

See also [Marquee Area](#).



The merged image will fit into this rectangle.



Note: Be careful to draw the Marquee with the same proportions as the original picture, otherwise the figure will be distorted.

SketchUp: Export and Import Limitations

- SketchUp models can contain two types of object faces and edges. Currently, standalone SketchUp edges are not imported into Archicad.
- In SketchUp a single face can have different front and back colors and textures. In Archicad a GDL face can only have one color and texture defined. Archicad's import function defaults to the front face's properties unless the back face has texture defined and the front face doesn't.
- SketchUp models containing faces that are very irregular (outside boundary crossing itself, i.e. figure of eight) or faces that contain holes may be imperfectly imported into Archicad
- Some SketchUp faces get imported into Archicad but lose their texture
- There is a flag in the .skp format to state whether the file is Geo-referenced or not. The Mac version of Archicad is unable to correctly set this flag and the exported .skp files may require that the flag be manually turned on (the longitude and latitude are always correctly set in the .skp file).

Google Earth: Export and Import Limitations

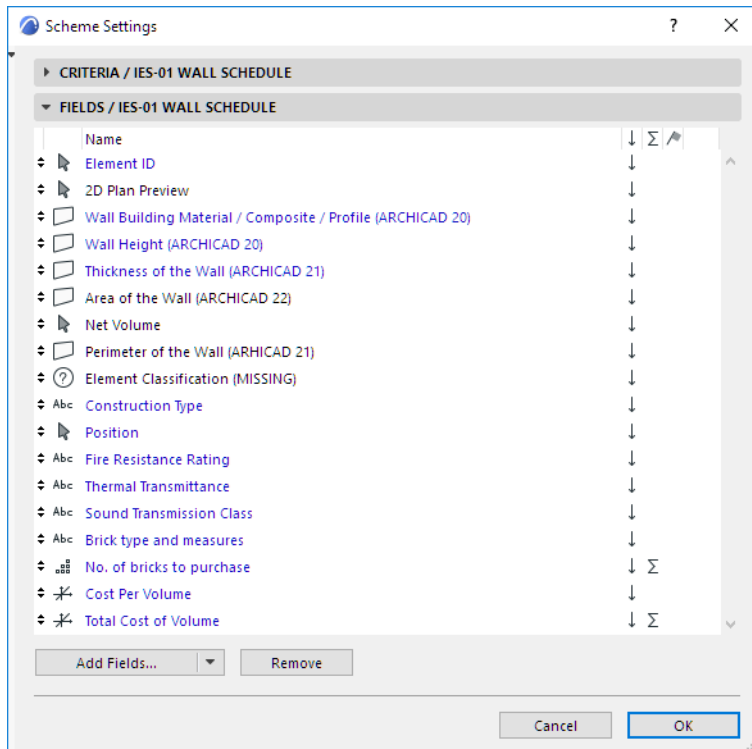
- Occasionally Google Earth .kmz files contain triangular faces with very small angles that are not imported into Archicad
- If the Archicad 3D view is such that the camera is positioned underground, then the view shown in Google Earth isn't the same as in Archicad.

Import/Export Property Data via Spreadsheet

Use this workflow to enable outside consultants to fill out an automatically configured Excel spreadsheet with Property data, for easy import to the corresponding elements/Building Materials in your Archicad model.

For an overview of Properties in Archicad, see also [Properties and Classification Systems](#).

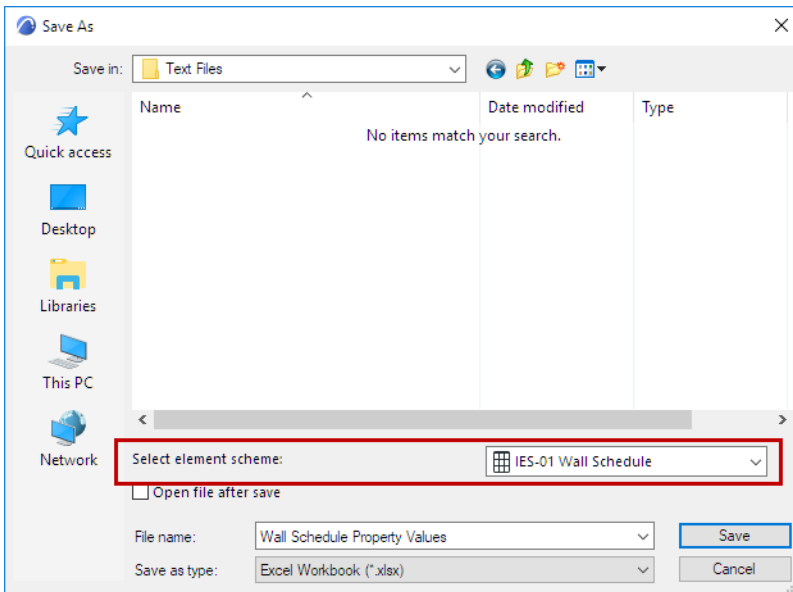
1. In Archicad, save an Interactive Element Schedule that you will use for the data exchange. The fields of the Scheme Settings should include parameters to identify the items (e.g. Element ID) and selected Properties which require data from the consultant. It is recommended to set these to “Undefined”, so the consultant knows which data are needed.



See [Assign Classifications/Properties in Element Schedule](#).

2. Use the **File > Interoperability > Classifications and Properties > Export Property Values from Schedule** command.

- In the appearing dialog box, select the Element Schedule Scheme that you prepared in Step 1 above. This scheme will serve as the basis for the data exchange.



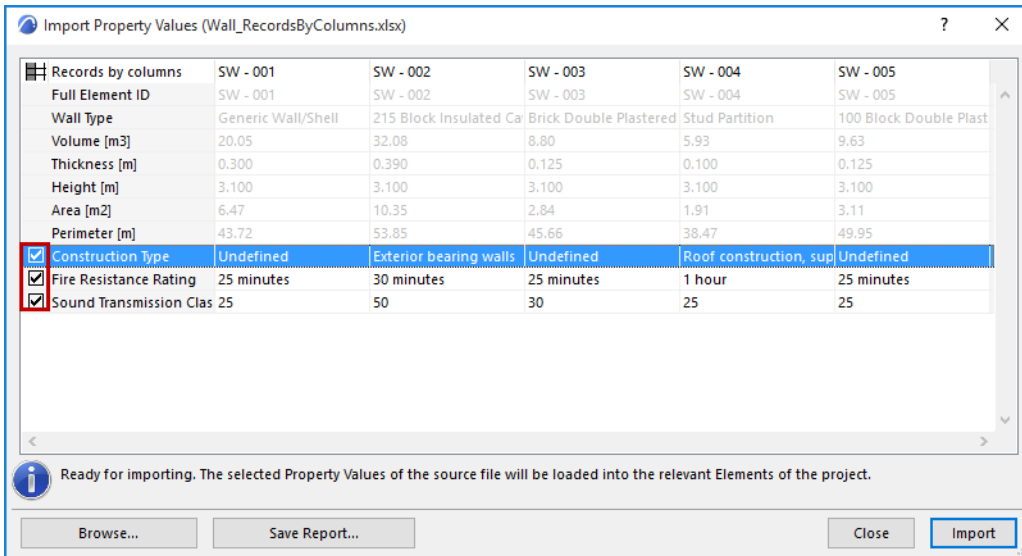
- Enter a file name, then click **Save** to save it as a spreadsheet in Excel format (XLS or XLSX).
- Send the spreadsheet to the consultant (he/she does not need Archicad or any other BIM program).
- In the Excel file, the consultant fills out the cells that require values, saves the file and sends it back to you.

Note: In the spreadsheet, any cell showing a black dashed line means that the property is not available for that item, and can be ignored by the consultant.

	B	C	D	E	
	Compact Element ID	Type	Net Volume	Thermal Transmittance	Construction Type
2	CRE - 001	Column	0.12	Undefined	Structural frame
3	CRE - 001	Column	0.12	Undefined	Structural frame
4	CRE - 001	Column	0.12	Undefined	Structural frame
5	CRE - 001	Column	0.12	Undefined	Structural frame
6	CRE - 001	Column	0.12	Undefined	Structural frame
7	CRE - 001	Column	0.20	Undefined	Structural frame
8	CRE - 002	Column	0.21	Undefined	Structural frame
9	CRE - 003	Column	0.21	Undefined	Structural frame
10	CRE - 004	Column	0.21	Undefined	Structural frame
11	CRE - 005	Column	0.21	Undefined	Structural frame
12	SW - 001	Wall	9.15	Undefined	Structural frame
13	SW - 002	Wall	11.16	Undefined	Structural frame
14	SW - 003	Wall	0.97	Undefined	Structural frame
15	SW - 003	Wall	1.77	Undefined	Structural frame
16	SW - 003	Wall	3.14	Undefined	Structural frame
17	SW - 003	Wall	22.80	Undefined	Structural frame
18	SW - 004	Wall	0.30	Undefined	Structural frame
19	SW - 004	Wall	0.50	Undefined	Structural frame
20	SW - 004	Wall	4.63	Undefined	Structural frame
21	SW - 004	Wall	10.67	Undefined	Structural frame
22	SW - 005	Wall	2.87	Undefined	Structural frame
23	SW - 005	Wall	15.66	Undefined	Structural frame
24	SW - 006	Wall	4.19	Undefined	Structural frame
25	SW - 007	Wall	0.50	Undefined	Structural frame
26	SW - 007	Wall	0.84	Undefined	Structural frame
27	SW - 009	Wall	7.42	Undefined	Structural frame
28	SW - 010	Wall	0.05	---	---
29	SW - 011	Wall	0.03	---	---
30	SW - 011	Wall	1.18	Undefined	Structural frame

7. In Archicad, import the Excel file using **File > Interoperability > Classifications and Properties > Import Property Values into Elements**.
8. The Import Property Values dialog box shows a preview of the data to be imported. Here you can filter the imported properties: uncheck any properties that you don't want to import.

Note: Only Element Properties can be imported to Archicad. Other parameters are ignored in the data exchange (they are shown in grey in this preview).



9. The preview highlights any errors or inconsistencies: [see Manage Data Errors](#), below. You cannot import data if there are any errors present.
10. If the data is free of errors, click **Import** to automatically add the element-specific property values from the spreadsheet to the Archicad model elements.

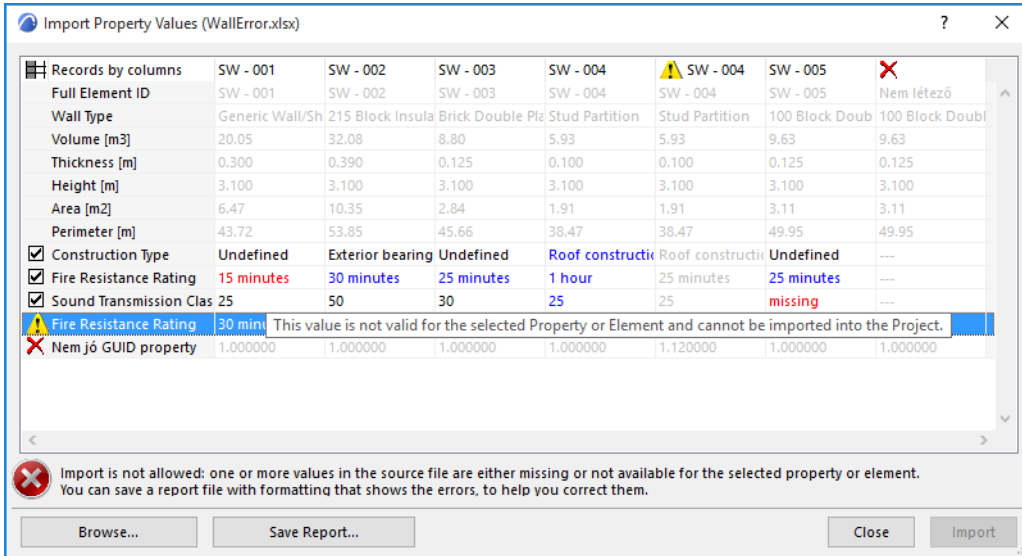
Undo will reverse the entire data import process.

Manage Data Errors

When you import data to Archicad using **Import Property Values**, the preview highlights any errors or inconsistencies. You cannot import data if there are any errors present.

Archicad gives you help in fixing them.

In the preview window, hover over such fields to see information on the issue.



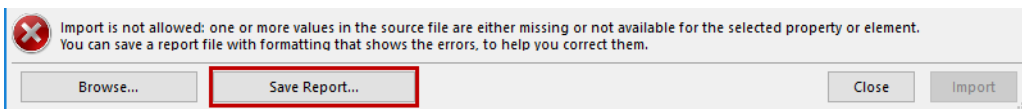
Note the codes:

- **Black** data items: Valid data, will be imported (provided that the property’s box is checked. If unchecked, such properties are grey and will not be imported.)
- **Grey** data items: These are not element properties and will not be imported.
- **Red** data items: Invalid or missing data; cannot be imported. Possible reasons:
 - Data format is wrong (e.g. the entered data does not correspond to one of the property’s defined options in an Option Set)
 - Data is missing (the cell says “missing” in red)
 - The property is not available for the element (the cell has a line in red)
 - The model element that the property applies to is not available, because it is on a locked layer, not reserved in Teamwork, or has been deleted
- **Blue** data items belong to duplicate properties or duplicate elements. Only the values of the first instance of this property will be imported. The duplicate items themselves are marked with an information icon.
- **Orange** data items are hidden in Archicad due to an Add-on (e.g. the freeBIM Manager).
- **Warning triangle:** The property does not exist in the target project.

To proceed, you can either uncheck the properties that are problematic (these values will be skipped entirely), or you can correct the data before import.

To correct the data:

1. Click **Save Report**. This creates an Excel spreadsheet which highlights data errors and provides further information on them.



- Open the Report spreadsheet and fix the errors as needed. (Add missing data, delete duplicate properties, fix invalid formats, provide data for Undefined properties.)

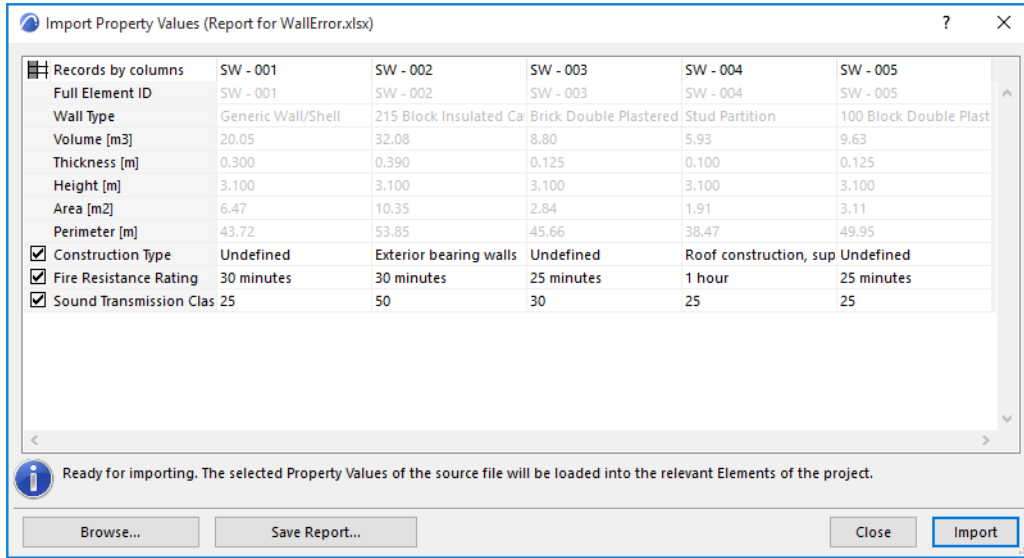
	B	C	D	E	F
2	Full Element ID	SW - 001	SW - 002	SW - 003	SW - 004
3	Wall Type	Generic Wall/SI	215 Block Insulated Cavity	Brick Double Plastered	Stud Partition
4	Volume [m3]	20.05	32.08	8.80	5.93
5	Thickness [m]	0.300	0.390	0.125	0.100
6	Height [m]	3.100	3.100	3.100	3.100
7	Area [m2]	6.47	10.35	2.84	1.91
8	Perimeter [m]	43.72	53.85	45.66	38.47
9	Construction Type	Undefined	Exterior bearing walls	Undefined	Roof construction, supp
10	Fire Resistance Rating	15 minutes	30 minutes	25 minutes	1 hour
11	STC	25	50	30	25
12	Fire Resistance Rating	30 minutes	1 hour	25 minutes	25 minutes
13	Invalid GUID property	1.000000	1.000000	1.000000	1.000000
14					

Note that the spreadsheet's Legend sheet provides additional help for understanding errors/color codes.

	A	B	C
1	Legend for Element Property Import Report		
2	Formatting	Status	Description
3	Black headers and fields	Valid data	Valid data, will be imported provided that the property's box is checked at importing.
4	"<?>" in a data field	Undefined value	The Property value is undefined for the element. If you enter a value of the appropriate data type into this field, the element will get this custom value. If you leave the "<?>" field unchanged, this property value will be omitted at import.
5	"---" in a data field	Not available	The property is not available for that element, and can be ignored by the consultant.
6	Grey headers and fields	Identifying data	These are not element properties and will not be imported. Including these at export serves for Element identification.
7	Orange fields	Hidden data	Data items belong to Properties set as invisible in the Project by an Add-On.
8	Blue headers and fields	Duplicates	Data items belong to duplicate properties or duplicate elements. Only the values of the first instance of this property will be imported.
9	Red headers and fields	Invalid data	Invalid or missing data; cannot be imported. Possible reasons:
10			- The data does not correspond to the referred Property's Data type
11			- Data is missing
12			- The model element that the property applies to is not available, because it is locked, placed on a locked layer, not reserved in Teamwork, or has been deleted
13			- The Hotlinked Module that the referred element is located in is not available, not editable, or has been deleted.

- Once the errors are fixed, return to Archicad and import the Report spreadsheet.
 - If the Import Property Values dialog box is open, click **Browse** to open the newly saved Report spreadsheet.
 - Or use **Import Property Values** to browse for the spreadsheet.

4. If all data is OK, click **Import**.



Update Properties in Hotlink Modules - Experimental Feature

The standard Archicad workflow supports the automated import and export of classification/property data. The **Update Properties in Hotlink Modules** feature extends this workflow to include properties of Hotlink module elements (and not just elements belonging to the host file).

Enable this feature

Go to Options > Work Environment > More Options > Experimental Features.

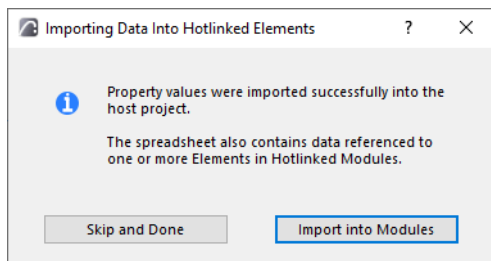
Check the box to **Enable Property Update in Hotlink Modules**.

This is an experimental feature, which may change or disappear in a future Archicad release. Please try it out!

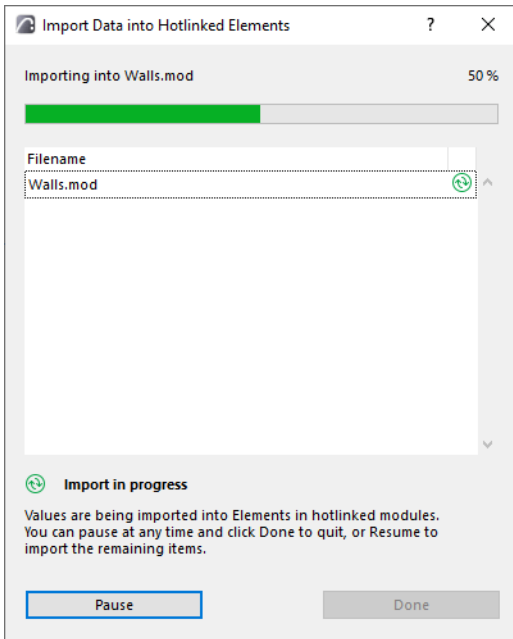
See [Experimental Features](#).

Suggested workflow

1. In Archicad, open the host file and update all Hotlink Modules. If you are in Teamwork: do Send and Receive.
2. Save an Interactive Element Schedule that you will use for the data exchange.
3. Use the “Export Property Values from Schedule” command (File > Interoperability > Classifications and Properties) to export the properties to Excel.
4. The consultant updates the Excel file with the required new or changed property data.
5. Use the “Import Property Values into Elements” command to import the revised property data from the Excel file.
 - Property values are first updated in the elements of the host file.
 - If the host file contains Hotlink modules whose property data should be updated: in the appearing dialog, choose **Import into Modules**.



The progress feedback allows you to pause or resume the import process.



Click Done when finished.

Import BIM Content

Archicad enables the one-step import of BIM data that have been predefined in an external Archicad file (PLN, PLA or TPL format). With this function, you import the following BIM Content to your Archicad project:

- A Classification System
- Any Properties belonging to that Classification System
- The IFC Translator “Mapping” presets which refer to that Classification System:
 - Type Mapping for IFC Import
 - Type Mapping for IFC Export
 - Property Mapping for IFC Export
 - Property Mapping for IFC Import

Import BIM Content from Archicad File

Use the **Import BIM Content** command to browse for any Archicad file that you have prepared or downloaded, which contains the needed Classification System and other definitions.

1. Go to **File > Interoperability > Classifications and Properties > Import BIM Content**.
2. From the appearing **Merge BIM Data** dialog, browse for any Archicad file (PLN, PLA or TPL format) which contains the Classification/Property/IFC Translator Mapping definitions that you need.
3. Click **Open** to import these data.

See also [Conditions for Downloading or Importing BIM Content](#), below.

Download BIM Data File from Website

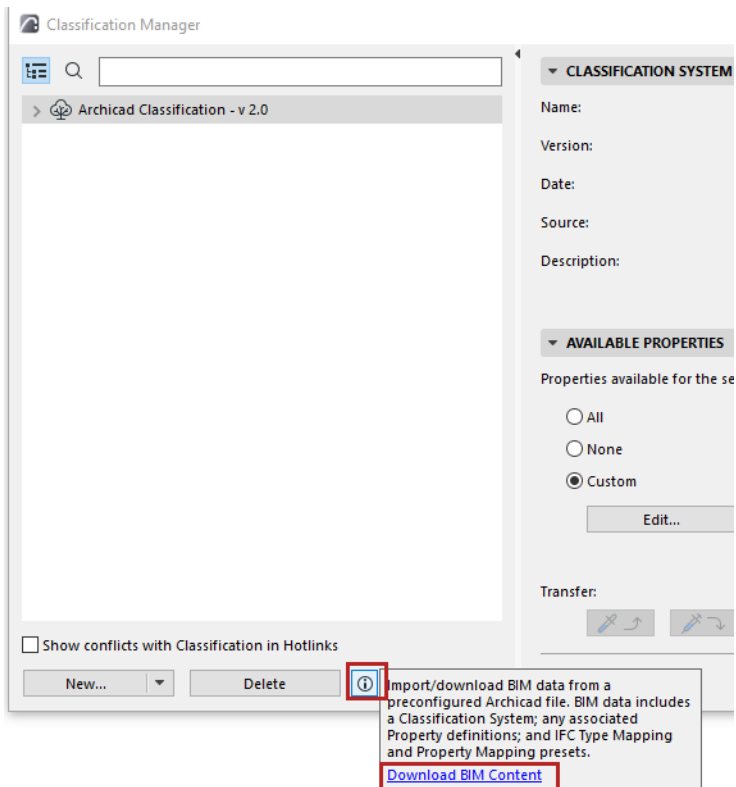
With this command, you browse for and download a preconfigured Archicad file.

These files, predefined for various international standards, are available at Graphisoft’s website: https://www.graphisoft.com/downloads/archicad/BIM_Data.html

1. Go to **File > Interoperability > Classifications and Properties > Download BIM Content**.

The same link is available from:

- Classification Manager
- Property Manager
- IFC Translators dialog box



2. This takes you to a Graphisoft website, where you can choose an Archicad file containing standard Classification/Property/Mapping definitions.
3. Download the file to any location.
4. Use **File > Interoperability > Classifications and Properties > Import BIM Content** to browse for and import the downloaded file.

Conditions for Downloading or Importing BIM Content

The import will work only if none of the imported data already exist in the receiving Archicad project. If data already exist, you get a warning that the import failed.

Consult the Report Window in Archicad (scroll to the bottom of the window as needed) to see which data are in conflict.

In this case, delete the duplicate data from your project, then use **Import BIM Content** again.

Working with PDF Documents

The following sections describe how to output PDF documents from Archicad.

[Create PDF Output Using the Publisher](#)

[Save Document in PDF Format](#)

[3D Content in PDF](#)

[Explode into Current View \(PDF Document Source\)](#)

[Export/Import Issues Using PDF Markup](#)

To place a PDF drawing in the project, see:

[Place Drawing in Model View](#)

[Place Drawing on Layout](#)

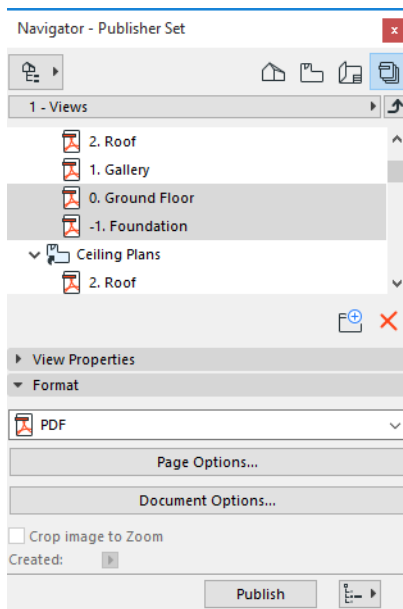
Create PDF Output Using the Publisher

From the Publisher, you can save Publisher Sets, or one or more Publisher items (views or layouts), in Acrobat PDF (Portable Document Format). (Publisher items are set to PDF format by default.)

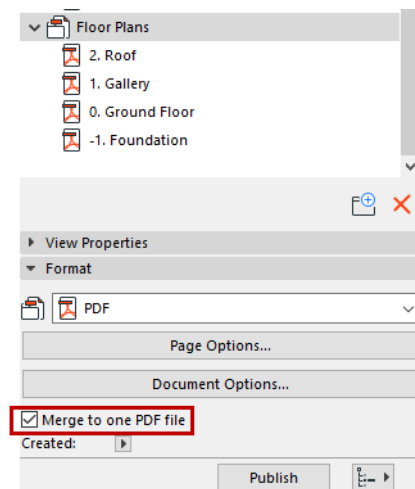
1. Go to the desired Publisher Set in the Navigator's Publisher mode.
2. Make sure that "Save" is selected from "Publishing Properties."

See [Define Publishing Method: Print, Save, Plot, BIMx](#).

3. Choose one or more Publisher item(s) from the Publisher Set.
4. In the Format section of the Publisher, set Format to "PDF."



If the contents of a **folder** are all in PDF format, a checkbox gives you the option to **Merge to one PDF file**. (Make sure you *select* the folder that contains the PDF format items.) If you leave this checkbox unmarked, you will publish each file in the folder as a separate PDF file.



Note: Once you have checked the "Merge to one PDF file" for one of the Publisher map folders (or for the entire Layout Book), the option is no longer available for any of its subfolders. To change your PDF merge

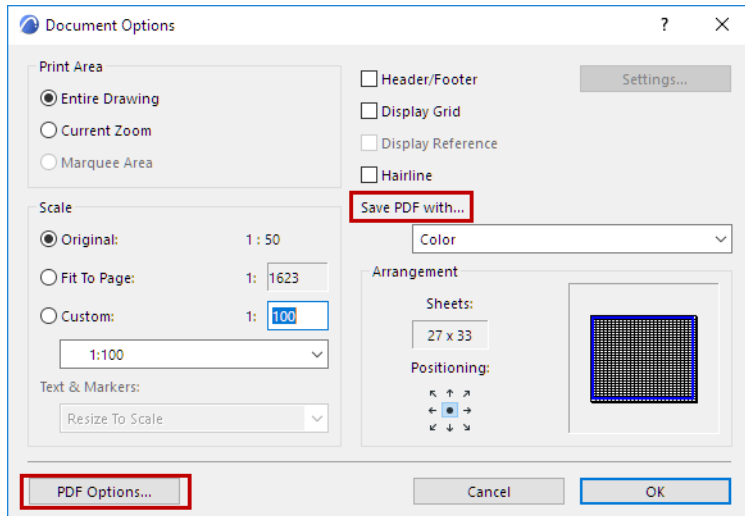
preference to a lower-level folder, first uncheck this preference for the higher-level folder (or at the Layout Book level).

5. Click **Page Options** to set the Paper Size and Orientation of the resulting PDF Document.

Note: The Page Options button is available if any of the selected Publisher items are *views*. If the Publisher items are *layouts*, their page options are taken from the layout settings; in this case, the Page Options button is not visible.

6. Click **Document Options** to define your preferred settings for the content of the resulting PDF Document. These are similar to the options in the Print 2D/Print 3D dialog boxes.

Here, you can save the contents in **Color**, **Black & White**, or **Grayscale**.



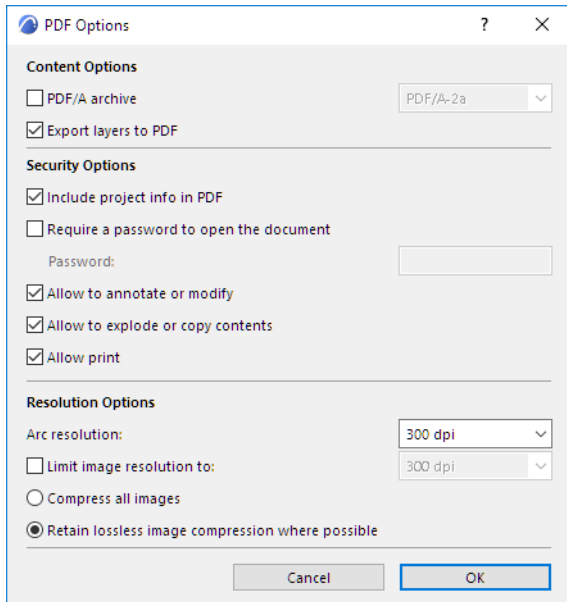
7. At the bottom left of the Document Options dialog box, click **PDF Options** and set the options as desired.

See [PDF Options](#) for details.

8. Click **Publish**.

9. Browse for the PDF output at the Path you defined in Publisher Set Properties.

PDF Options



Content Options

PDF/A archive format

The default is PDF/A-2a.

Notes:

- If you choose one of the PDF/A formats, you cannot also set a password.
- If you choose one of the PDF/A formats, the U3D file embedding option is not available!

[See 3D Content in PDF.](#)

- The PDF/A-1a and PDF/A1b formats do not support transparency effects. (These features are available with the PDF/A-2 formats.)

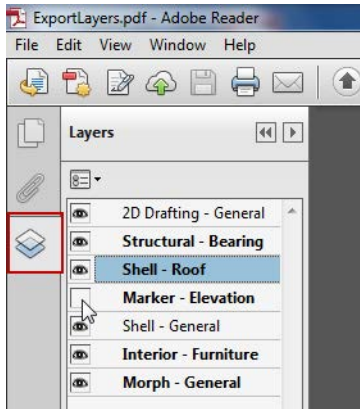
[For more information on the PDF/A standards, see: ISO 19005-1:2005 Part 1: Use of PDF 1.4 \(PDF/A-1\) and ISO 19005/2:2011 Part 2: Use of ISO 32000-1 \(PDF/A-2\).](#)

[Watch the Video](#)

Export Layers to PDF

When saving a view or layout in PDF format, information on the visible layers of that view/layout - if they contain elements - will also be saved to the PDF document. The **Export Layers to PDF** checkbox is checked by default.

When you open the resulting PDF document, you can show or hide the content by layer. (Click the Layers panel on the Navigation Pane. If you do not see the Layers panel, or use View > Show/Hide > Navigation Panes > Layers.) The layer names are the same as in Archicad.



Restrictions on Exported Layers:

- Items placed directly on Layouts are exported without layer information, so their visibility cannot be controlled in the PDF.
- Layers that do not contain any elements are not exported
- Hidden layers are not exported
- Embedded layers are not exported

[Watch the Video](#)

Embed 3D Content

[See 3D Content in PDF.](#)

Security Options

Include Project Info in PDF

This checkbox is active by default: when you save a project in PDF format, the PDF retains the Archicad Project Info data (entered at **File > Info > Project Info**). This can be read in the PDF's Document Properties (**File > Properties**).

It can be useful to *uncheck* this box if you do not want the PDF to display e.g. the Architect Name (as the Author shown in the PDF Document Properties) - for example, if the PDF is to be submitted as part of an anonymous competition.

Require Password

Check the box if you want to password-protect the document.

Additional Security Options

Check any of the following to allow the PDF's recipient to access or modify the contents:

- Allow to annotate or modify
- Allow to explode or copy contents
- Allow print

Resolution Options

Arc Resolution

The **Arc resolution** setting reflects the fact that the PDF format does not recognize arcs/circles and must approximate them by converting them to polygons. The Arc resolution setting defines the accuracy of these polygons.

Limit image resolution to:

The maximum image resolution of the images saved to PDF. Images whose resolution is less than this limit will remain as-is.

Choose a compression option:

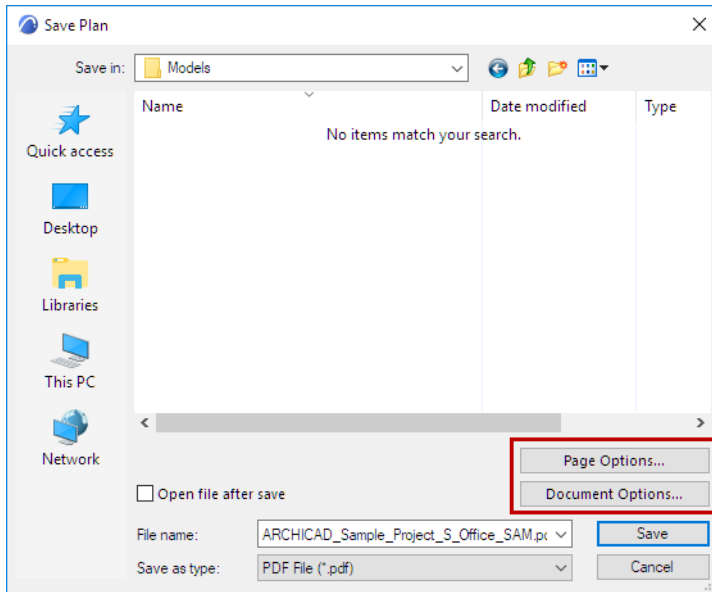
- **Compress all images:** All images will be converted to JPG format, which entails a compression to 80%.
- **Retain lossless image compression where possible:** Images will be saved in PNG format (and remain lossless). If the original image was in JPG format, it will remain JPG.

Save Document in PDF Format

Go to **File > Save As**, and choose **PDF** as the file format.

PDFs created with Archicad can include standard PDF features such as layers, thumbnails, and bookmarks.

Note: To limit the window contents to be saved, use the Marquee tool.



Click **Page Options** to set the Paper Size and Orientation of the resulting PDF Document.

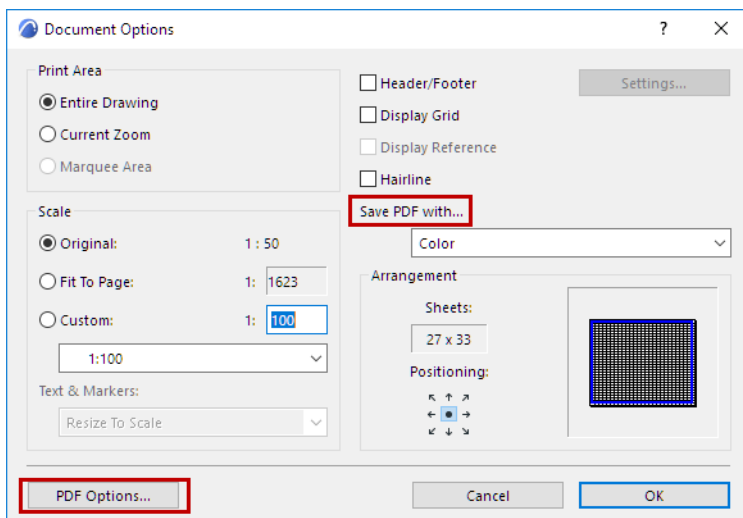
Note: No Page Options button is available if you are saving to PDF from a Layout window.

Click **Document Options** to define your preferred settings for the content of the resulting PDF Document. These are similar to the options in the Print 2D/Print 3D dialog boxes.

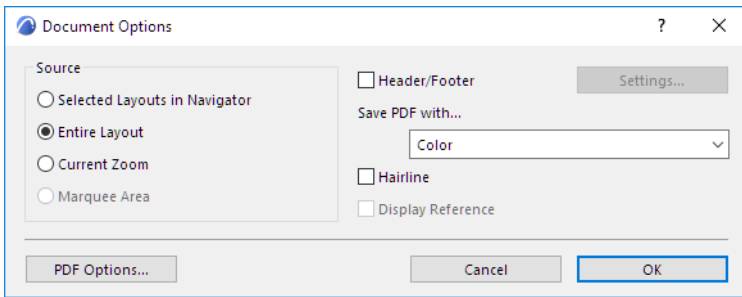
Note: No Document Options button is available when saving to PDF from a formatted list window.

For more information, see [Print 2D Document](#).

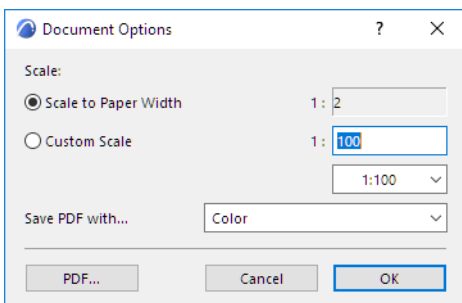
When saving to PDF, you can save the contents in **Color**, **Black & White**, or **Grayscale**.



- For a **Layout**, you can choose to output multiple Layouts (the ones currently selected in the Navigator or Organizer) into PDF files, or else just the current Layout (at its current zoom or in its entirety).



- For a **formatted list**, you have Scaling options and the ability to place each zone or page of the list onto a new sheet.



- **PDF Options:** At the bottom left of the Document Options dialog box, click **PDF Options**.

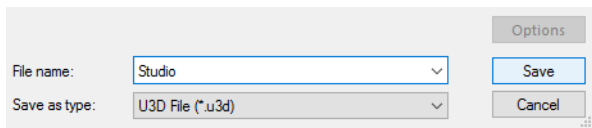
See [PDF Options](#) for details.

3D Content in PDF

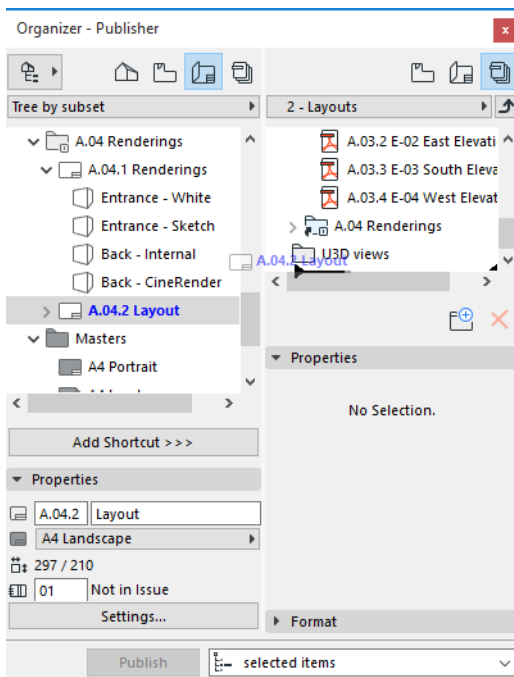
When creating a PDF, Archicad allows you to take advantage of the U3D file format.

When publishing a PDF document from Archicad, you can embed a U3D file into the PDF, then “navigate” the view within the PDF as if it were a 3D model.

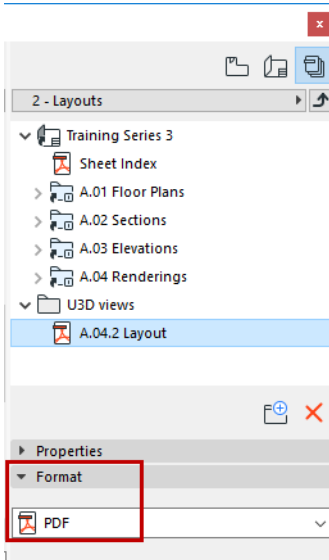
1. From the 3D window, set up the desired model view. It is recommended to clear unnecessary detail from the 3D model as much as possible, in order to optimize the navigation speed in the resulting PDF document.
2. Save the 3D window contents as a file using the “**Save as**” command. Choose the “U3D file” format from the list of available formats.



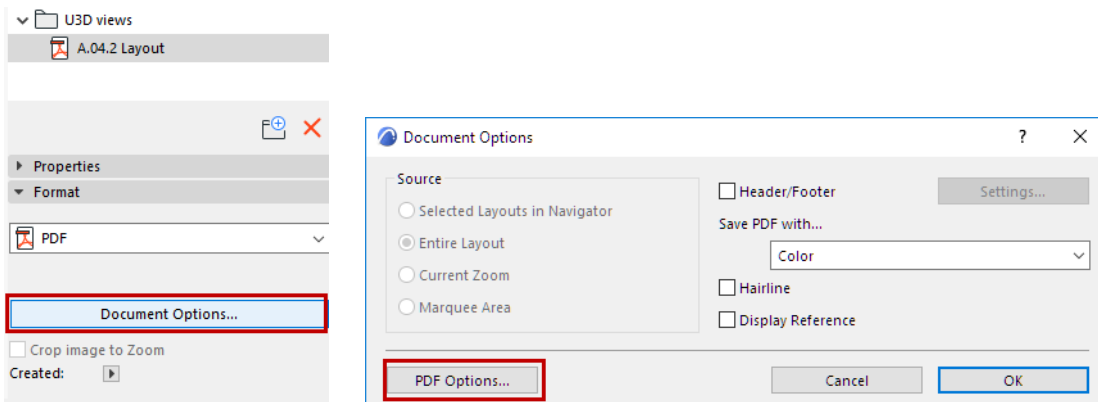
3. In the Navigator or Organizer Palette, save the 3D model as a view, then add the 3D view to a Layout.
4. Next, add this Layout to a Publisher set. Select the Layout in the Publisher Set.



5. Set the Publisher Set Properties to PDF format.



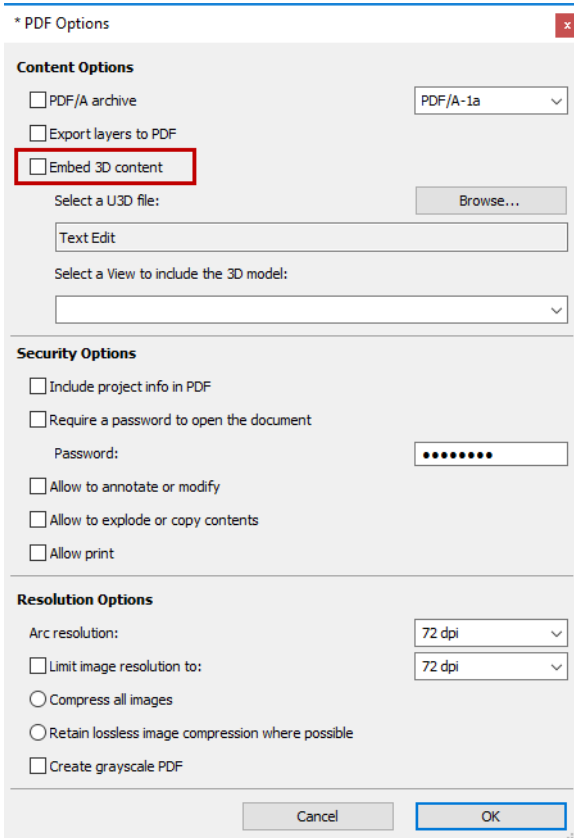
6. In the Format tab page at the bottom of the Publisher palette, choose **Document Options**, then **PDF Options**.



7. In the appearing **PDF Options** dialog box, check the “Embed 3D content” checkbox.

Note: If you choose one of the PDF/A formats in PDF Options, the U3D file embedding option is not available!

8. Click the “Browse” button to select the file you saved in U3D format (in Step 1 above).



The screenshot shows the "PDF Options" dialog box with the following sections:

- Content Options**
 - PDF/A archive (dropdown: PDF/A-1a)
 - Export layers to PDF
 - Embed 3D content (highlighted with a red box)
 - Select a U3D file: (Browse... button)
 - Text Edit (input field)
 - Select a View to include the 3D model: (dropdown menu)
- Security Options**
 - Include project info in PDF
 - Require a password to open the document (Password: [masked])
 - Allow to annotate or modify
 - Allow to explode or copy contents
 - Allow print
- Resolution Options**
 - Arc resolution: (dropdown: 72 dpi)
 - Limit image resolution to: (dropdown: 72 dpi)
 - Compress all images
 - Retain lossless image compression where possible
 - Create grayscale PDF

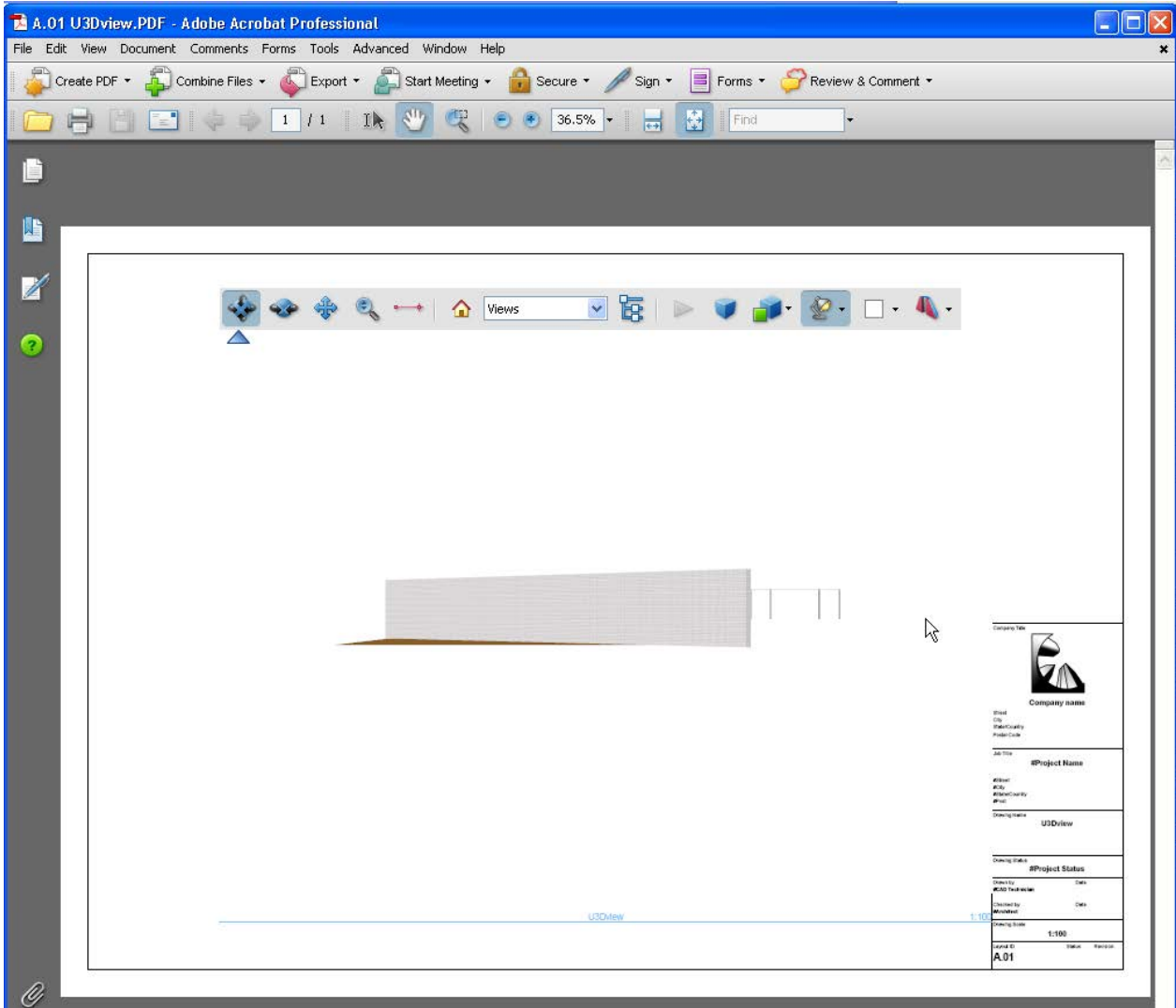
Buttons: Cancel, OK

9. Click **OK**.

10. In the Publisher, select the desired publisher item and publish it in PDF format.

The resulting PDF file will contain an embedded, navigable U3D file.

The U3D model embedded in the PDF can be navigated only by using Adobe Reader 7.0 or later.



Working with DWG/DXF Files

The DXF/DWG Add-on is installed together with the Archicad package and is smoothly integrated in the Archicad user interface. It allows you to:

- Open AutoCAD drawings as Archicad Projects or Archicad Library Parts.
- Merge AutoCAD drawings with your Archicad Project file.
- Place AutoCAD drawings as Archicad Drawings to Layouts or Model Views.
- Add AutoCAD drawings as external references (XREFs) to the Archicad Project.
- Import BLOCKs from an AutoCAD drawing and create Archicad Library Parts out of them within their own newly created Library.
- Save or Publish your Archicad Floor Plans, Sections/Elevations, Detail Drawings and 3D Views in AutoCAD formats.
- Save your Archicad Layouts in AutoCAD formats.

Use or edit **Translators** to make it easier to apply a customized set of conversion rules to the equivalent elements and functions of Archicad and DXF/DWG files.

For details on Translator settings, see [DXF/DWG Translation Setup](#).

Opening DWG/DXF Files

Saving DWG/DXF Files

Merge DXF-DWG

Working with XREFs

Opening DWG/DXF Files

You can open DXF and DWG format files in Archicad as Drawings or Layouts.

- Choose **File > Open** and then select the file you need in the directory dialog box.
- Conversion rules for data exchange between Archicad and AutoCAD are stored in a file called a Translator (extension XML).
- The Translator field in the Open dialog box lists default translators you will use in most cases (such as “for as is output”).
- If you don’t know which default translator to use, choose Settings to open the DXF-DWG Translation Setup dialog box. Each default translator listed at the top is briefly described in the Description field.

For more information on Translator settings, see [DXF/DWG Translation Setup](#).

For placed Drawings, you can customize the size, scale, title and framing options for your drawings in the Drawing Settings dialog box.

See [Drawing Settings](#).

[Place external drawing file \(multiple methods\)](#)

Opening DXF/DWG file as Library Part

Use the **File > Libraries and Objects > Open Object** command.

This will open the GDL Master Window, where you can configure the new object obtained from the entire file.

For more information, see [GDL Object Editor](#).

Opening with Drag-and-Drop

You can import DXF/DWG drawings using Drag-and-Drop:

Left-click the AutoCAD drawing, dragging it over the desired location and dropping it. The file will be opened using the last-used DXF/DWG Translator. With this method, the DWG/DXF file is simply merged into the Archicad Project file. The 3D content of the file is not imported, only its 2D appearance.

If several DXF/DWG files are drag-and-dropped, only one of them will be dropped; since you cannot control which file will be dropped, it is recommended to drag-and-drop only one such file at a time

There are several ways to use Drag-and-Drop techniques while opening/merging AutoCAD drawings:

- Drag-and-dropping a DXF/DWG file onto the background of the Archicad application window, the Archicad menu bar or the Archicad icon will result in opening the file.
- Drag-and-dropping a DXF/DWG file onto the Archicad Floor Plan, Section/Elevation/IE or 3D Document window or Detail/Worksheet window will place the file as a Drawing.
- Drag-and-dropping a DXF/DWG file onto a Layout will place the Drawing onto the Layout.

Opening Password-Protected Files

In AutoCAD 2004 **and later versions**, it is possible to protect drawings with a password.

Currently, on the Mac platform it is not possible to open password-protected AutoCAD 2004 and later DXF/DWG files because the operating system does not support the encryption method used by these password mechanisms. In such a case an alert will come up on your Mac.

On Windows the operating system may be such that it does not support the password decoding routine. In this case you will be alerted that the file cannot be opened.

If the system supports the decoding routine, a dialog appears where you can enter the password you received from the person sending you the file.

For XREF files, the program will require the password the first time the XREF is reloaded in Archicad. If several XREF files are being reloaded, only the password-protected file will not be reloaded if its password is not entered correctly.

Opening Blocks as Library Parts

The closest analogy of an AutoCAD block is an Archicad Library Part. Block references (insert entities) match the library part instances. The difference: while a library part is a separate file, the block is stored within the AutoCAD file itself.

Converting blocks into library parts is the only way to preserve the blocks' 3D structure.

Drawbacks of converting blocks into library parts:

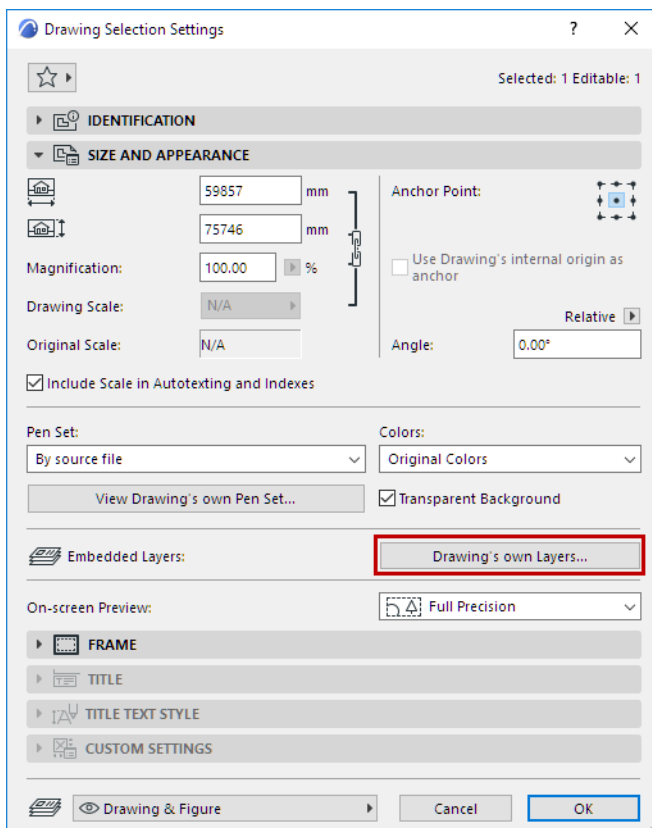
- you must regularly weed through the object libraries created during DXF/DWG import, to eliminate duplicate and unnecessary items,
- some editing functions available for details stored in blocks will not be available for the library part instance.

Import Selected Layers of DWG/DXF/DWF Files

See [“Enable Partial Open” at Open Options \(DXF/DWG Translation Setup\)](#).

Show/Hide Layers of Imported DWG/DXF/DWF Files

Use the Embedded Layers control in the Properties panel of Drawing Settings to control the display of an imported DWG, DXF or DWF file, by showing or hiding its own layers (i.e. the layers in the source file).



This control is available if the source of the selected Drawing is a DWG, DXF or DWF file.

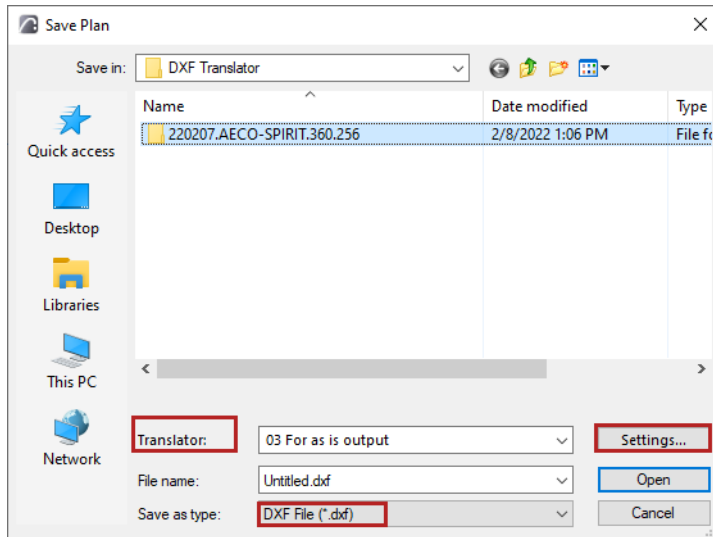
For more information, see [Embedded Layers](#).

Saving DWG/DXF Files

You can save or publish a project view or layout in Archicad in DXF and DWG formats. (No DWG save is possible from a schedule, index or list view.)

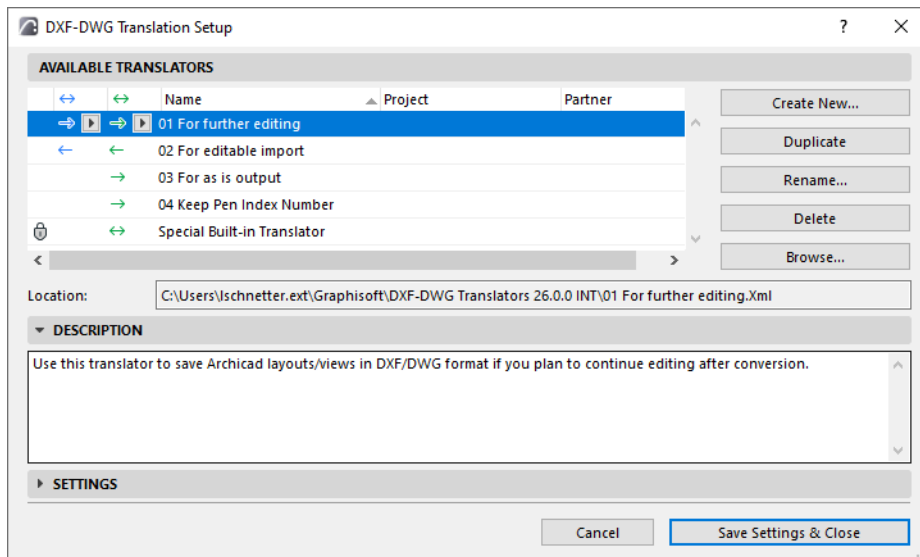
1. Choose **File > Save as**, then the desired file type (DXF or DWG).

For information setting up Publisher Sets for output in DWG/DXF Format, see [Publisher](#).



- To save only part of the view, use the Marquee to delimit the project. Save As will save only the part that falls within the Marquee.
 - If you are working in AutoCAD and open a DWG that was originally saved from Archicad, an Alert may appear. However, you should go ahead and work with the saved DWG in AutoCAD - no stability problems should result.
2. The Translator field in the Save dialog box lists default translators you will use in most cases (such as “for as is output”).
 3. If you don’t know which default translator to use, choose Settings to open the DXF-DWG Translation Setup dialog box. Each default translator listed at the top is briefly described in the Description field.

For details, see [DXF/DWG Translation Setup](#).



Entity Conversion

Simple Archicad elements get converted into the corresponding AutoCAD entities:

- Hotspots to points,
- Lines (without arrowheads) to lines,
- Circles to circles,
- Arcs (without arrowheads) to arcs,
- Ellipses to ellipses,
- Elliptic arcs (without arrowheads) to elliptic arcs,
- Lines and arcs with arrowheads are “complex elements”.
- Texts to multiline texts,
- Fills to hatches,
- Pictures to images,
- Dimensions (all kinds) to dimensions.

Note: Fills in walls, columns and library parts are treated the same way.

Note: Archicad and AutoCAD dimensions have different aspects of appearance to be configured, so your dimensions will often appear differently. The possible locations of the dimension text relative to the dimension line is the most apparent difference.

Complex elements: Drawing details are made up of more than one AutoCAD entity. Unless you choose “Explode complex Archicad Elements,” they are drawn into a block each and an insert is placed into the entity section.

- **Walls:** they are drawn into blocks named WALL_<n> where n is the index of the wall element. The block contains visible contour lines, fills (see notes to the polygon fills), block references to windows and doors and an attached label. The window and door references may be followed by their window dimension block references and attached label references if they have any.

Note: The contours of an arc wall are polylines made up of straight segments.

- **Columns** are drawn in blocks named COLUMN_<n> where n is the index of the column element. The block contains the visible contour lines, the fills (see notes to the polygon fills) and the block references to the attached label.

- **Windows** are drawn in blocks named WINDOW_<n> where n is the index of the window element.
- **Doors** are drawn in blocks named DOOR_<n> where n is the index of the door element.
- **Objects** are drawn in blocks named OBJECT_<n> where n is the index of the object element.
- **Lights** are drawn in blocks named LIGHT_<n> where n is the index of the light element.

Note on windows, doors, objects and lights: When the library part doesn't have any 2D script, it may have up to 8 different looks, depending on "Uniform Pen," "Use Symbol Linetypes" and the possible mirrored state. So one library part of this kind cannot have more than 8 blocks exported. Others (with 2D scripts) may have any number of different looks and they will have exactly as many blocks as looks occur on the exported floor.

Since more than one instance may share the same detail block, the attached labels cannot be put into these blocks – they follow the block references.

- **Slabs** are drawn in blocks named SLAB_<n> where n is the index of the slab element. The block contains the contour and hole polygons and the block reference to the attached label.
- **Roofs** are drawn in blocks named ROOF_<n> where n is the index of the roof element. The block contains the contour and hole polygons and the block reference to the attached label.
- **Beams** are drawn in blocks named BEAM_<n> where n is the index of the beam element. The block contains the contour and hole polygons and the block reference to the attached label.
- **Meshes** are drawn in blocks named MESH_<n> where n is the index of the mesh element. The block contains the contour and hole polygons, the mesh level lines and the block reference to the attached label.
- **Zones** are drawn in blocks named ZONE_<n> where n is the index of the zone element. The block contains the foreground fill, the zone stamp and the block reference to the attached label.
Note: since the default zone-fill colors are light pastel colors and the AutoCAD color set does not contain an equivalent, they all will be RGB-matched to a light gray. If you prefer, pick a saturated color (e.g. red or yellow) for zone fill color and blue or black for zone stamp color.
- **Cameras** and camera sets are not exported.

Using SHX Fonts in the Conversion Process

The DXF-DWG Add-on relies on the availability of AutoCAD fonts – including the SHX fonts. The Add-on will display a message asking for these fonts during the course of a conversion.

If you do not have these fonts available, you can expect problems in text formatting (such as incorrect line breaks). It is recommended that you acquire the SHX font files, which are part of the AutoCAD package, from an AutoCAD user. (Although the fonts are not available with Archicad for proprietary reasons, AutoCAD users are allowed, even encouraged, to distribute them.) True-type version of all SHX fonts are also available in the AutoCAD package.

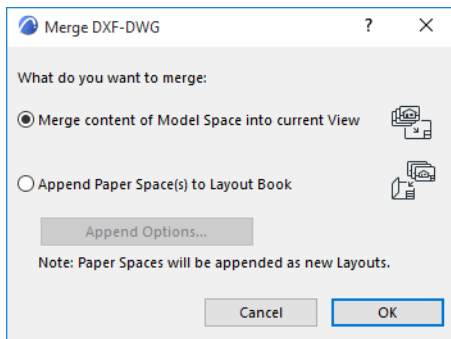
Once you get the fonts, create a separate folder for them. When the Add-on asks for an SHX font file, select it from this collection. If you don't have the requested one, you may select another one. (In this case the conversion results may differ.) Once you have shown the folder to the Add-on, it will henceforth automatically search in that folder (but not in any newly created sub-folders) for the required SHX font files.

Note on TrueType fonts: You can use TrueType fonts in Archicad without any problem on either Windows or Mac. (Mac users should copy the TrueType font files into the system drive's Library/Font folder.)

Note on Fonts vs. Shapes: AutoCAD uses SHX files not only as fonts but for other purposes as well. Complex lines insert shapes into their patterns. The shapes may be added to the drawings as simple symbols. It may cause problems if you select an SHX of this kind instead of a font – so when the Add-on requests you to locate an SHX file, use the “Skip” option if you are not sure an SHX file you found is the right kind. The requested filename should give you a clue about which kind is needed – font or shape.

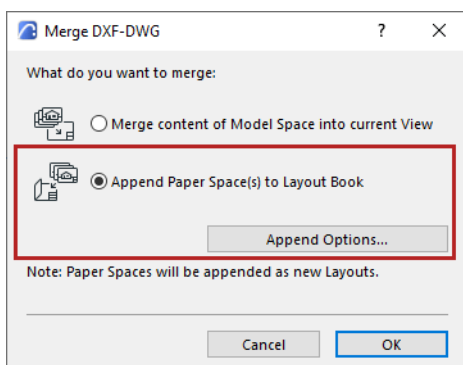
Merge DXF-DWG

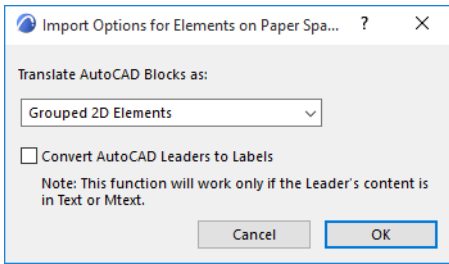
1. Use **File > Interoperability > Merge**, then choose where to browse the file: either **from File** or **from BIMcloud**
2. In the **Merge file** dialog, browse for the desired DXF or DWG file.
3. Click **Open**.
4. The **Merge DXF-DWG** dialog box appears. You have two choices:



- **Merge content of Model space into current View:** Choose this to merge the Model Space portion of the DXF-DWG file into the Archicad project. The Top View of AutoCAD elements in the DXF-DWG Model Space will be generated and placed into as 2D Archicad elements.
Set further options in the appearing dialog: [Merge DXF-DWG Model Space Dialog Box](#).
- **Append Paper Space(s) to Layout Book:** Choose this to append the Paper Space(s) of the DXF-DWG file merged to the Archicad Layout Book.
 - Paper Space definitions will be converted to Archicad Layouts.
 - A new Master Layout will be created if the DXF-DWG drawing contains Paper Space size definitions not already available in Archicad.
 - DXF-DWG Viewport content will be converted into Drawings and placed onto Layouts. These Drawings are not editable; they will appear in the Archicad Drawing Manager with “embedded” status.
 - Elements drawn in Paper Space will be converted to Archicad elements and placed directly onto Layouts.

Click **Append Options** to set further options:.



**Translate AutoCAD Blocks as:**

Choose an option:

- 2D Elements
- Grouped 2D Elements
- Library Parts

[See *Open Options \(DXF/DWG Translation Setup\)*.](#)

Convert AutoCAD Leaders to Labels: Check to perform this conversion during the Merge process.

If you leave this box unchecked, AutoCAD Leaders will be converted to simple Archicad 2D elements.

Note: This checkbox is also available in the Translation Setup Dialog, at **Custom Functions > Open Extras**. [See *Custom Functions \(DXF/DWG Translation Setup\)*.](#)

Related Topics

[Round-Trip Conversion \(Smart Merge\)](#)

[Working with DWG/DXF Files](#)

Round-Trip Conversion (Smart Merge)

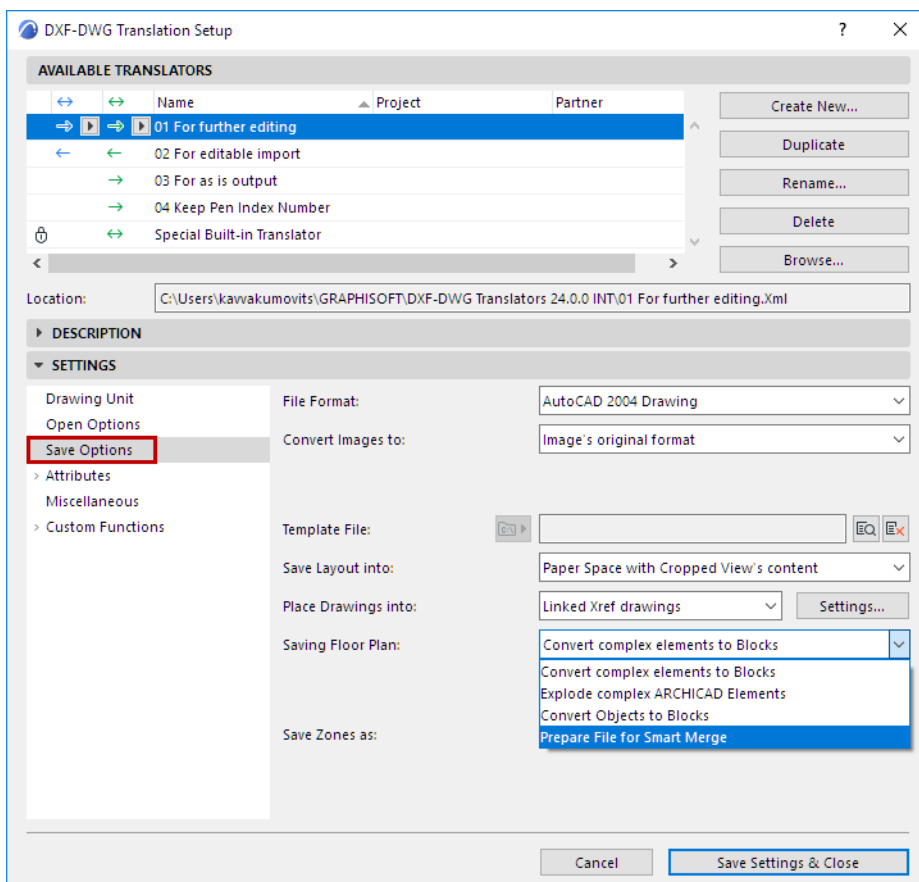
Use Smart Merge (round-trip conversion) if you are merging the DXF or DWG file back into the same Archicad file from which you saved it. Smart Merge allows you to work on the same project in Archicad and AutoCAD simultaneously, and exchange the additional information with minimal loss.

Smart Merge handles:

- New elements added in Archicad or AutoCAD.
- Elements created in Archicad or AutoCAD and modified (stretched, dragged, rotated, mirrored) in the other application or both applications.
- Elements created in Archicad or AutoCAD and deleted in either or both applications.

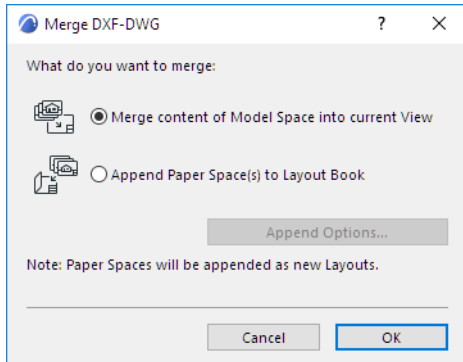
If you cooperate with someone using AutoCAD (or any other CAD program that can communicate via DXF or DWG format), use the following process:

1. You work out the architectural aspects of the whole plan to a point where it can present an overall layout for your partner to add details (at this point, you need not be completely finished with your part).
2. Save the Archicad PLN file.
3. Then you save a DXF or DWG file for your partner:
 - Use **File > Save as**, and set the file type to DXF or DWG.
 - Next to the Translator, click Settings to open the DXF-DWG Translation Setup dialog box.
 - In the Settings panel, use the **Saving Floor Plan** option: Prepare File for Smart Merge.

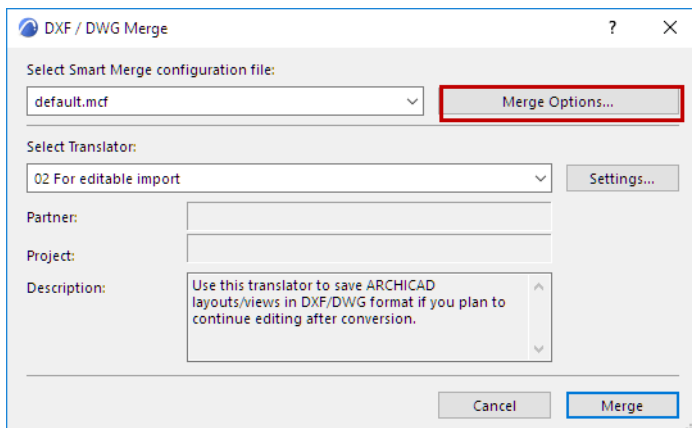


For more information, see [Saving Floor Plan](#).

4. You continue your work on the original plan, while s/he works on the DXF/DWG file.
5. When it is time to merge:
 - **File > Interoperability > Merge > Merge from File**
 - Browse for the file, and click Open.
 - From the Merge DXF-DWG dialog box, choose the **Merge Content of Model Space into Current View** option, and click OK.



- From the **DXF/DWG Merge** dialog box, choose Merge Options,



- The **Smart Merge Options** dialog box appears, allowing you to modify the Smart Merge configuration file and the translator options (if desired).

Note: You can also set up these options in advance at **File > Interoperability > DXF-DWG > Smart Merge Options**.

See [Smart Merge Options](#).

6. Repeat as needed.

Related Topics

[Merge DXF-DWG](#)

[Working with DWG/DXF Files](#)

Working with XREFs

External reference files (XREF) are similar to Hotlinked Modules, except that they are DXF or DWG files and not Archicad files.

[See also *Hotlink Modules and XREFs in Teamwork*.](#)

External referencing is a way of saving disk space and work. You can attach a DWG or DXF file to your Floor Plan or to a Detail; the file will be loaded into the project and displayed, using the settings of the DXF/DWG Translator you have selected.

You can snap to all drawing elements within the XREF and print or plot them.

A common use for XREFs is to draw elements that are common to several kinds of drawings within a project (e.g., the outlines of exterior and interior walls). If you have a file containing this data, you can attach it to other drawings that can use the geometric information contained in the XREF file without the Project file actually having to contain the data. Also, if the source of the data in the external reference (the DWG or DXF) is changed, then these changes can be updated immediately so the latest data is always available.

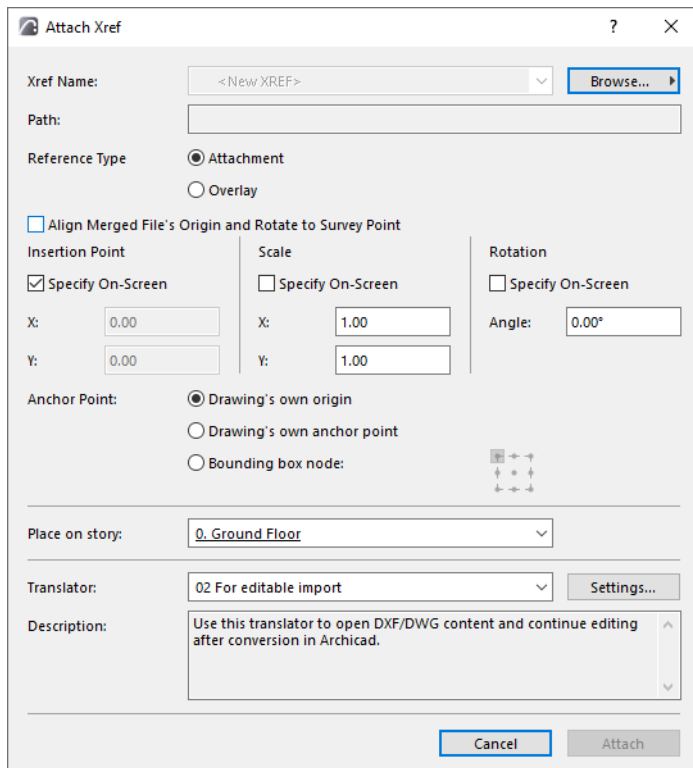
XREF is available in both Archicad and AutoCAD - both applications can detect circular references.

In Archicad, external reference files work much as they do in AutoCAD. Archicad XREFs can display 2D elements from DWG files, but three-dimensional AutoCAD elements (such as Solid elements) are not visible in Archicad's 3D window.

Note: If any of the functions described do not work properly, the DXF/DWG Add-on may be missing, or the DXF/DWG Translator is not set. You can check this easily by choosing **File > Open** and looking for the appropriate file types in the Files of type field. The missing Add-on can be loaded with the **Add-on Manager** command in the **Options** menu.

Attach XREF

1. From the Floor Plan, Detail or Worksheet window, use **File > External Content > Attach XREF**.
2. The Attach XREF dialog box appears.



3. From the XREF Name pop-up:
 - choose from the list of XREFs already loaded in the project
 - or Browse for a DWG or DXF file in a local file system or from BIMcloud
4. Set insertion and rotation options as needed.
[See *Attach XREF for details*.](#)
5. Choose a DWG/DXF Translator which will determine conversion options for the attached file (or just use the default translator shown). Click Settings to view the options for the current translator.
6. Click **Attach**.

You can attach and detach XREF files in Archicad, unload and reload them, bind them to the project file and view information about them. These functions are available in the XREF Management Dialog box (**File > External Content > XREF Manager** and **File > Attach XREF**).

Note: You should not attach XREFs from different locations if they have the same file name, because AutoCAD cannot open such files. (If they have the same name but different file extensions - DXF vs. DWG - this can still cause problems.)

Manage XREFs

To manage XREFs attached to the project, use **File > External Content > XREF Manager**.

The XREF Management dialog box displays information about the attached files, including the name of the reference (usually the same as the attached file), its status (Loaded, Detached, Reloaded, Unloaded or

Bound), size, type (Attach or Overlay), Date and the number of instances, as well as information on the translator used to open each XREF.

For details, see [XREF Management Dialog Box](#).

Attributes of XREF Files

Using XREFs in Round-Trip Communication

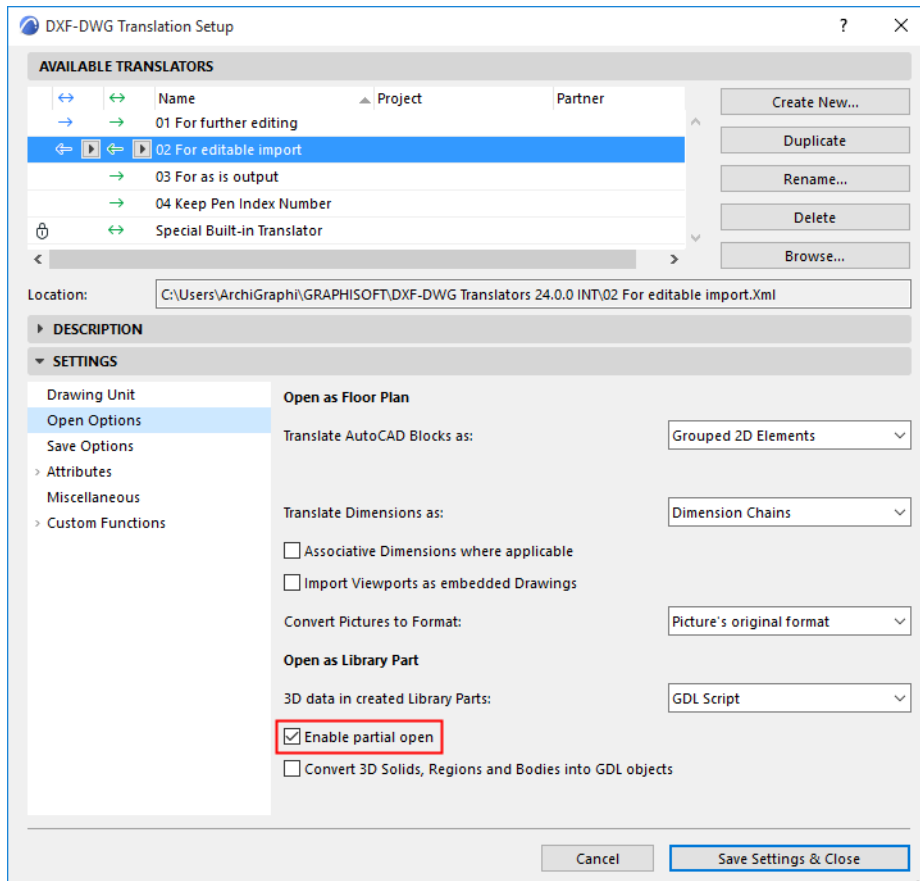
Attributes of XREF Files

Filter Layers of XREF Files - Enable Partial Open

When you attach an XREF to your project, Archicad will follow the settings of the DXF/DWG Translator used. This Translator can be selected and its settings viewed in the Attach XREF dialog box.

See [Attach XREF](#).

One such Translator setting is the “Enable Partial Open” checkbox in the Open Options part of Translator Settings:



If this setting is enabled, then every time you attach an XREF file, the Partial Open dialog box will appear on screen, letting choose which layers you wish to attach along with the drawing.

See [“Enable Partial Open” at Open Options \(DXF/DWG Translation Setup\)](#).

Layer Settings of Attached XREF Files

When you attach an external reference file to an Archicad Project, a layer will be created for it.

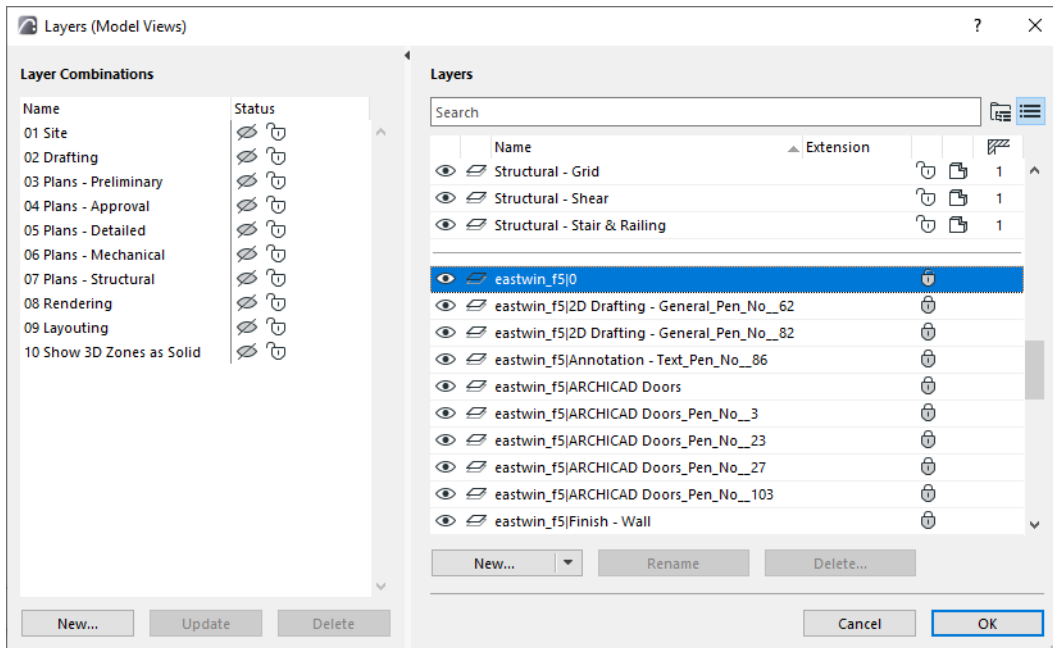
Exception: If you have enabled the Partial Open setting (see above) and unchecked that layer name in the Partial Open dialog box, that layer will not be created in your Archicad project.

The resulting layer names will have the following format:

‘XREF_NAME | LAYER_NAME’.

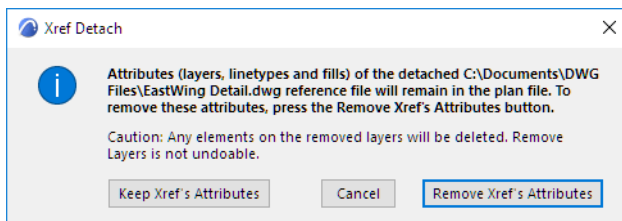
For example, the PEN_56 layer of a DWG file that was attached to the project with the XREF name 3D_STUFF will become the layer 3D_STUFF|PEN_56.

In Archicad Layer Settings, the layers of any attached XREFs are listed separately, at the bottom of the layer list.



XREF layers are not available from layer selection pop-ups in other parts of the user interface (e.g. Tool Settings dialog boxes and Info Box). However, if an XREF element is included in the last selection, its XREF layer does appear - for information purposes only (in italics) - in the Layer chooser pop-up.

When the XREF is detached, a dialog lets you choose between retaining or removing the detached XREF's attributes, including layers, linetypes and fills.



Using XREFs in Round-Trip Communication

In this scenario, you are assigned to work on one part of a larger project. You get the whole project (or the part of it relevant to your job), extend it with your part and send it back. In this case we recommend that you use the XREF (External Reference) model. Using this model you can see all parts of the project, but modify only the part assigned to you.

The XREF model is based on a “master” AutoCAD file containing general information (e.g. a site map).

- First, attach this “master” file as an XREF to your Archicad plan to see the current state of the project.
- Detail plans, including your file, are added as XREF’s. Make your modifications in your external file, using Archicad. You will be able to modify only your file.
- Meanwhile, other members of the project team are working on their files in parallel. To receive the changes made by the others, reload the “master” file. The updated file contains all the modifications submitted by the other parties.
- If you want the others to see the work you have done so far, you can send your changes by exporting your work in AutoCAD format.

The **XREF model** preserves the “master” AutoCAD file: each team member works only on his/her own part without being able to modify others’ work, because externally referenced files don’t have to be sent back to their original sources.

Due to essential differences between Archicad and AutoCAD, perfect round-trips are not possible. (For example, if you import and then export something, polylines and structures are lost.) In practice, however, you can rely on the suggested XREFs, which do not involve overwriting the other party’s original file, so that you don’t need to rely on a perfect round-trip. If you work in round-trip communication, your file never gets overwritten. The specialist gets new DXF/DWG files from you from time to time, which contain imperfect reproductions of his modifications. You can avoid these imperfections if you keep his additions in a separate XREF file, attached as an external reference to the file you send back to him.

Working with Point Clouds

Archicad opens .xyz and .e57 file formats and converts them into objects, which can be placed on the Floor Plan or in the 3D window. The user can then align the building model to point clouds, on the Floor Plans, sections/ elevations and the 3D window.

Point Clouds are not visible on any published output!

Import and place Point Cloud files for the following workflows:

- Modeling existing buildings prior to renovation: The point cloud serves as the initial basis for the architectural model of the rebuilt/renovated building.
- Modeling the surroundings prior to creating a new building: Use point cloud import to create a rough model of the surroundings and neighbor buildings, as an aid to visualization.
- Modeling the “as-built” building for detecting the deviations from the planned state.

Topics in this section:

[Import Point Clouds](#)

[Trace Point Cloud to Create Model Elements](#)

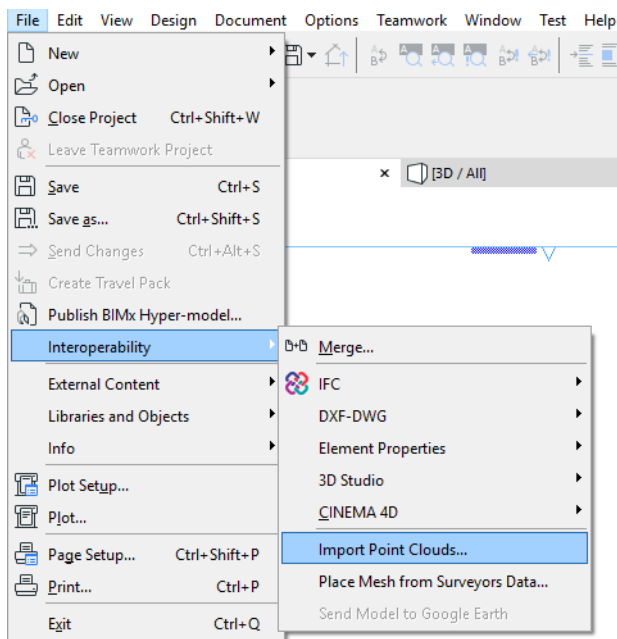
[Optimize Display of Point Cloud Object](#)

[Redefine Origin of Point Cloud Object](#)

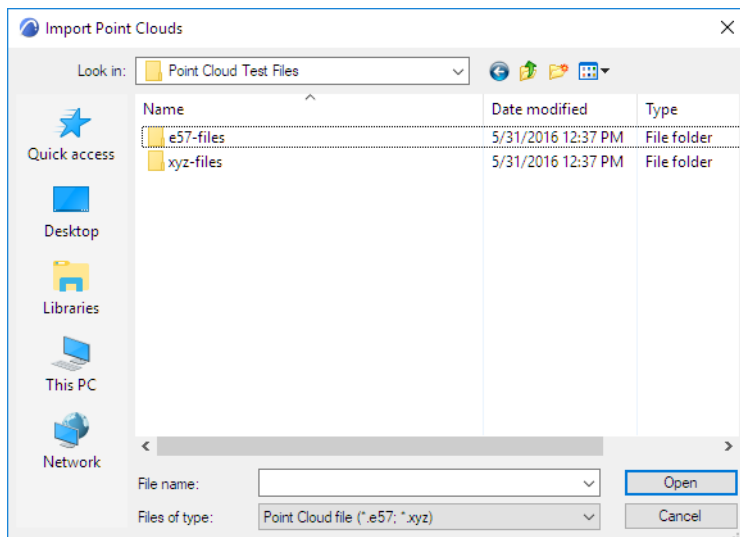
[Place Point Cloud Object in Teamwork Project](#)

Import Point Clouds

1. Use File > Interoperability > Import Point Clouds.



2. In the appearing dialog box, browse for the location of the point cloud file(s) and select one or more files. The files must have a format of either .e57 or .xyz.

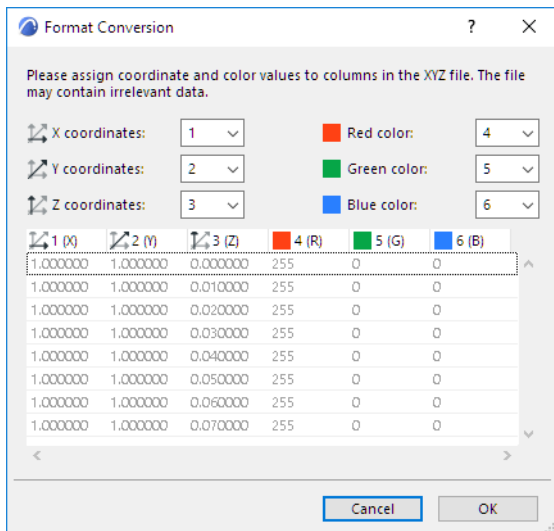


3. Click **Open**.

4. For XYZ format files:

If any of the selected files has the extension .xyz, the following **Format Conversion** dialog box appears:

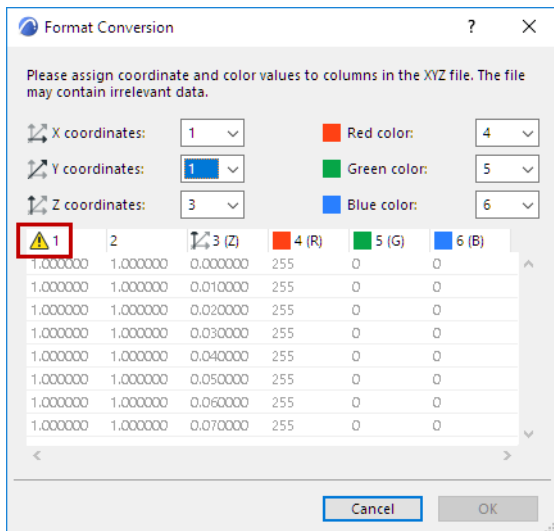
Note: E57 format files are standardized and require no format conversion. You will skip the Format Conversion dialog box and continue with the next step.



This dialog box gives you the opportunity to correct (if needed) how the X, Y and Z coordinates and RGB colors of the imported file should be interpreted in Archicad.

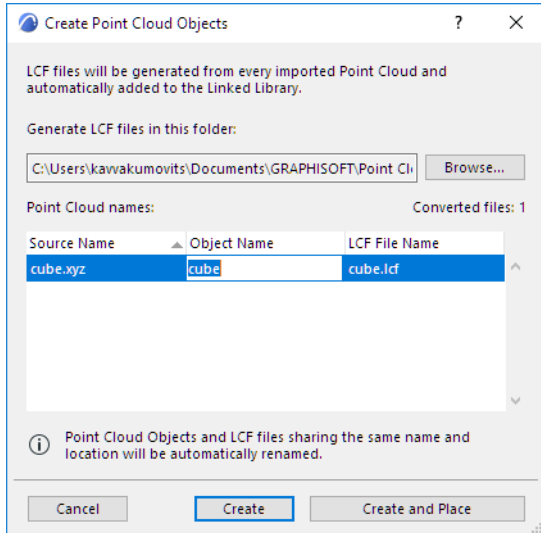
The only editable data here are the six fields at the top of the dialog box. For each field (X, Y and Z coordinates, as well as Red, Green and Blue color), assign one of the columns shown in the bottom dialog box. (By default, the first six columns are numbered, but some of the columns may contain unneeded data.)

If the data assignment becomes invalid - for example, if the same column is assigned to two fields - the OK button is grayed, and you will see an exclamation point at the top of that column.



For more information on how to set up Format Conversion (if needed) for the .xyz file you are importing, consult the surveyor who created the file.

5. The **Create Point Cloud Objects** dialog box appears.



The new Point Cloud objects (listed in the dialog box) will be created as LCF files and saved in the linked library shown in the path field. If needed, browse for a different folder path.

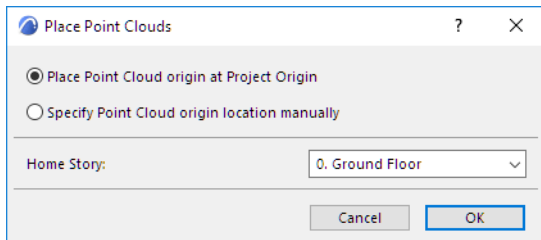
6. **Object Name:** This is the name of the newly created object. If needed, edit the object name.

7. Do one of the following:

- Click **Create** to create the Point Cloud file(s) and object without placing them.
- Click **Create and Place** if you want to place the objects now (Solo projects only. [For Teamwork projects, see below: Place Point Cloud Object in Teamwork Project.](#))

Conversion of the Point Cloud files may take a few moments.

8. If you choose Create and Place, the **Place Point Clouds** dialog box appears.



Here, you can:

- Specify the placement method (either place it at the Project Origin, or click anywhere to place it manually)
- Define the Home Story for the Point Cloud objects

See also [Redefine Origin of Point Cloud Object.](#)

Click **OK** to place the Point Cloud objects. You can snap to any point on the resulting object.

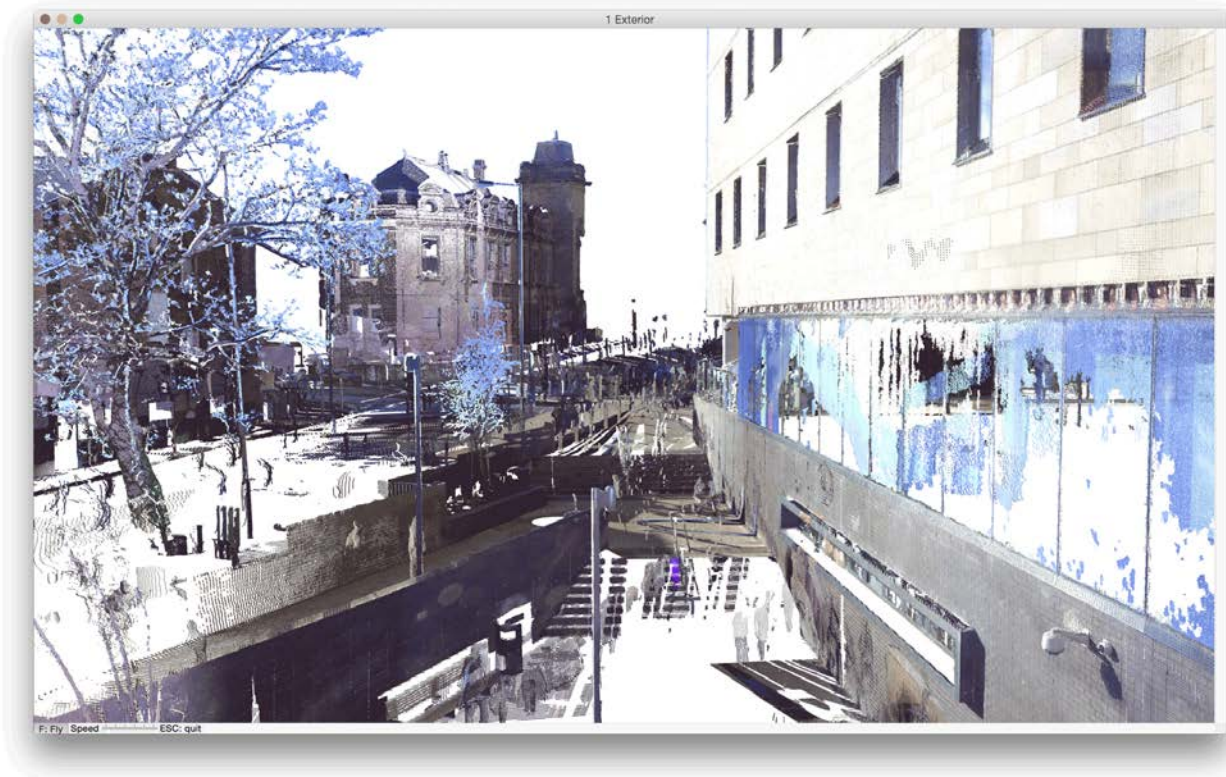
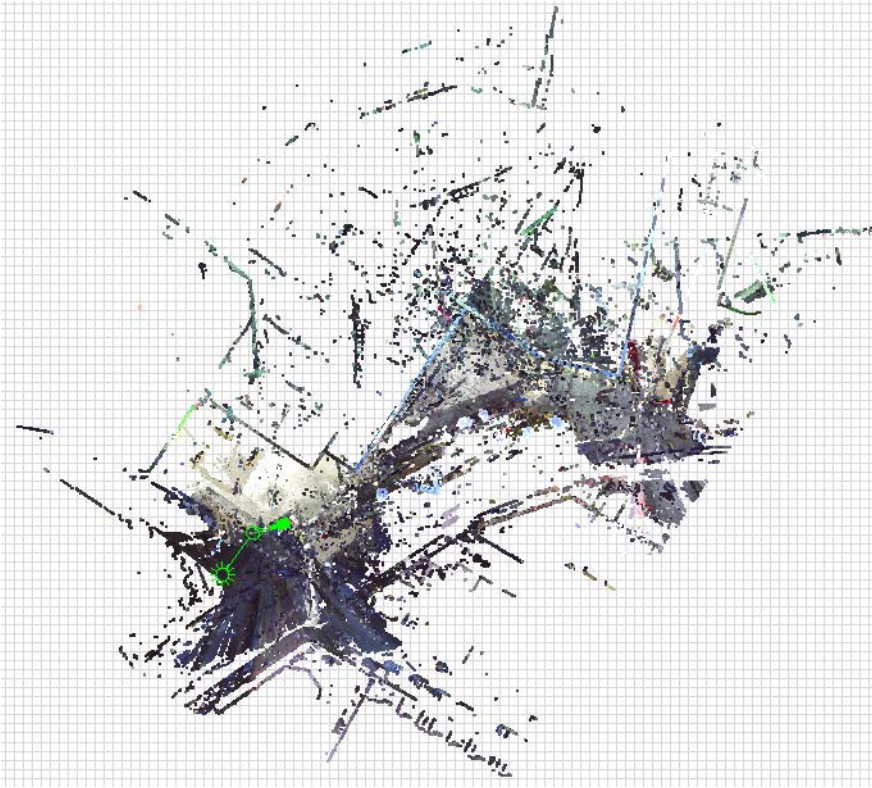
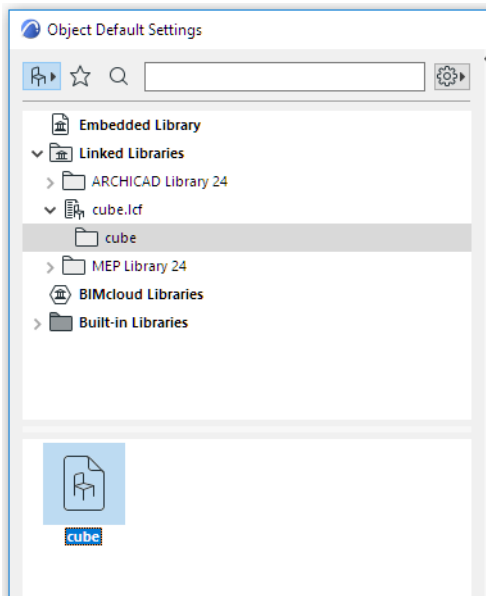


Image Credit: BAM Construct UK Ltd, www.bam.co.uk | FARO Technologies UK Ltd, www.faro.com

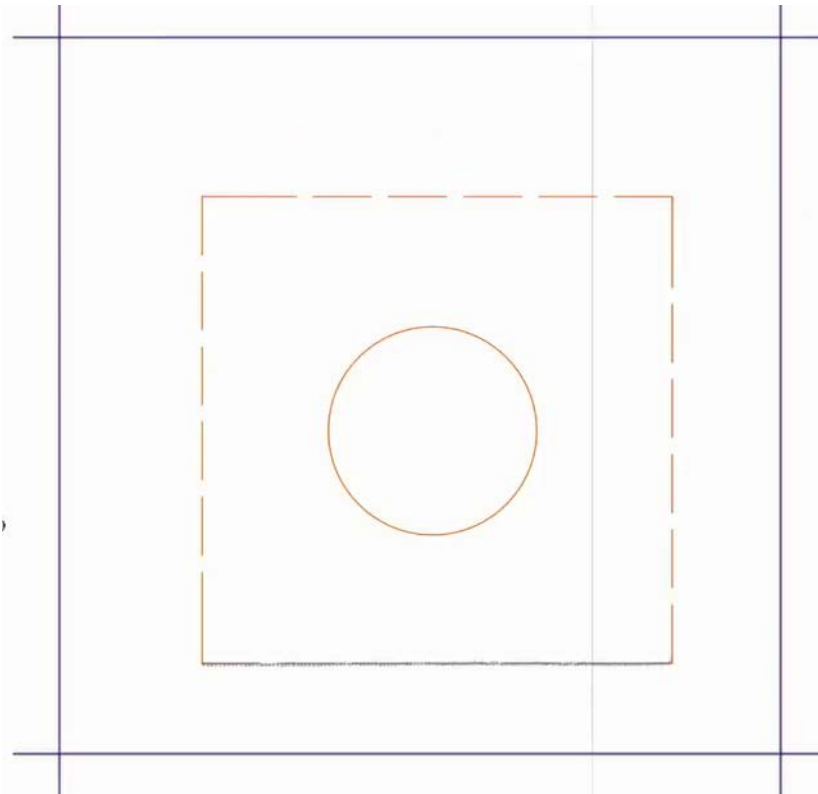
Each time you import a particular Point Cloud file, a new LCF file is created as a linked library. You can place the new LCF file any number of times from the Object Settings dialog box.



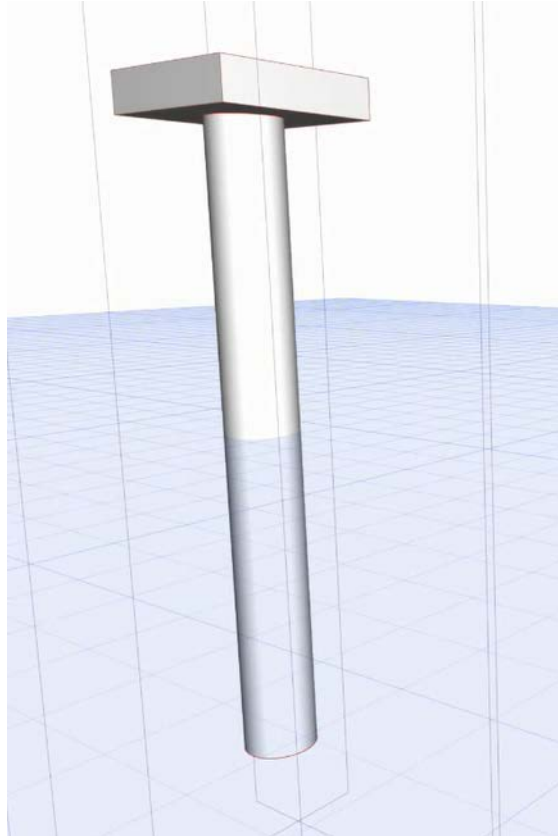
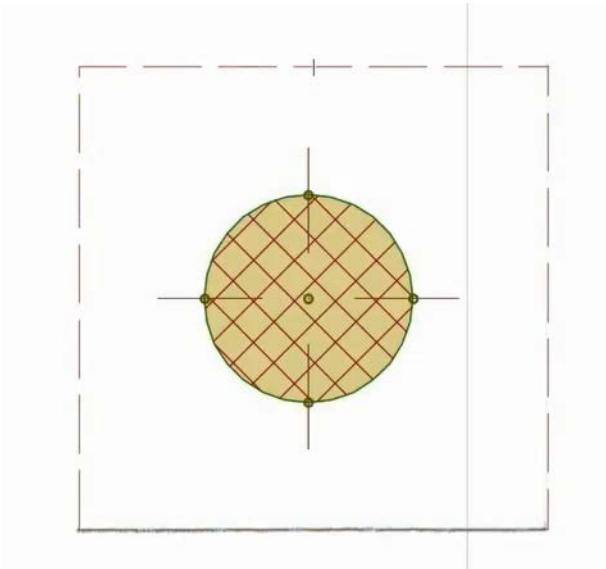
[Watch the Video](#)

Trace Point Cloud to Create Model Elements

Here, we have used line tools to trace the contours of Point Cloud shapes on the Floor Plan.

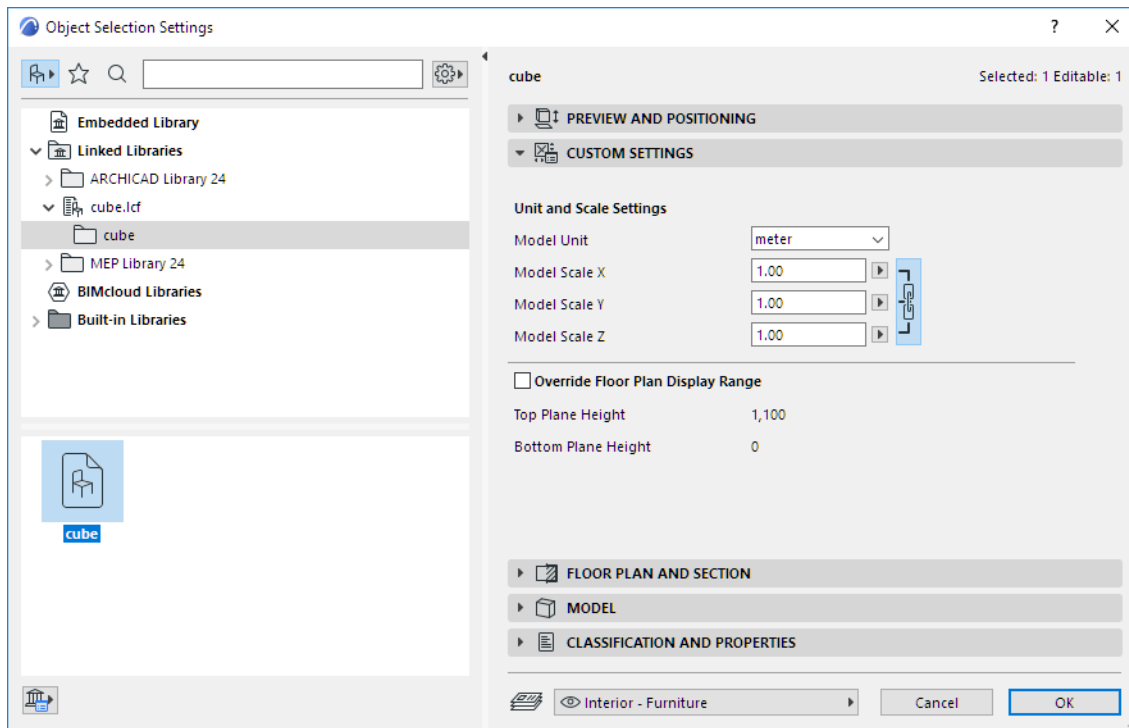


They can be transformed into model elements using the Magic Wand.



Optimize Display of Point Cloud Object

Use the Custom Settings panel of a Point Cloud object to change its display, to make it easier to work with.



Unit and Scale Settings: As needed, stretch or resize the object using the Model Scale settings.

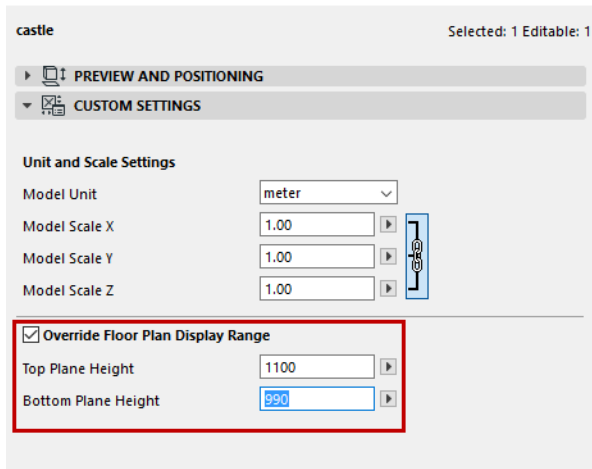
[Watch the Video](#)

Override Floor Plan Display Range: By default, the object will be displayed according to the current Floor Plan Cut Plane height. If needed, you can override this display limit for the point cloud object, if it is hard to see contours.

For example, we wish to trace the column shape in this garage object, but at the current Floor Plan Display range, the contour is too blurred.



You can narrow the display range values to make the contour easier to trace:



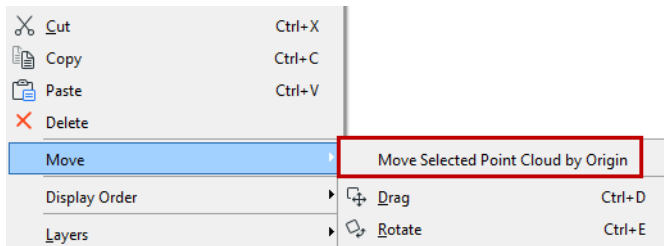
[Watch the Video](#)

[Watch the Video](#)

[Watch the Video](#)

Redefine Origin of Point Cloud Object

If your selection includes a Point Cloud object, the context menu contains the command: **Move selected Point Cloud by origin**.



Click this command, then click in the project to place the Point Cloud's new origin.

Place Point Cloud Object in Teamwork Project

After importing a Point Cloud file, the **Create Point Cloud Objects** dialog box appears.

[See workflow above, at *Import Point Clouds*.](#)

Click **Create**. (The Create and Place command is not available in a Teamwork project.)

The new Point Cloud objects (listed in the dialog box) will be created as LCF files and saved to the location specified in the dialog box.

You must add the new objects to the Teamwork project before you can place them.

[See *Manage BIMcloud Libraries*.](#)

Working with Rhino 3D Models

Topics in this Section:

[Open or Merge Rhino Model](#)

[Import Rhino 3DM File as Object](#)

[Hotlink Rhino File](#)

[Export Model to Rhino](#)

Open or Merge Rhino Model

Use these workflows to open or merge a **Rhino 3D Model** file (.3dm) directly in Archicad

- **File > Open**
- **File > Interoperability > Merge**

A Rhino model imported to Archicad will appear as a non-editable GDL object, maintaining the logical structure/buildup of the original Rhino model. It is placed into Archicad project's Embedded Library. You have the option to adjust the model segmentation before and/or after import to Archicad, depending on your preferred settings (described below).

Topics in this section:

[Import Options for Rhino 3D Model](#)

[Rhino Object Parameters in Archicad](#)

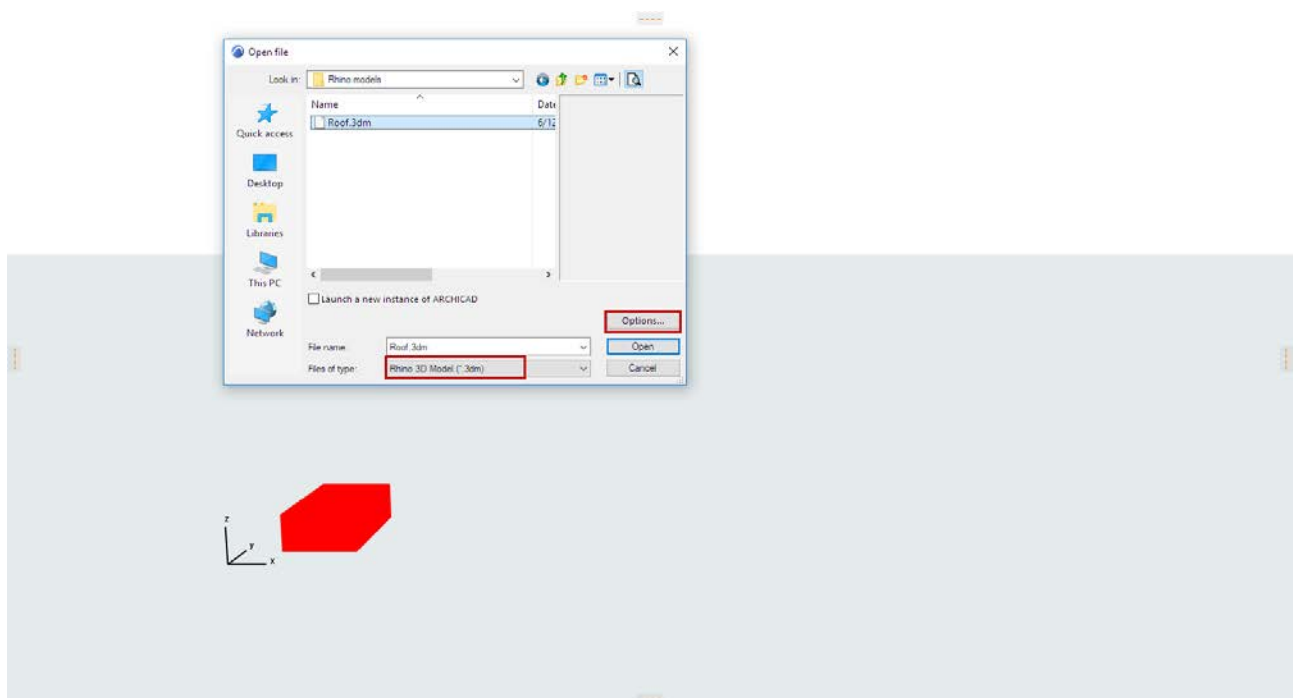
[Rhino Properties in Archicad](#)

[Rhino View Display in Archicad](#)

Import Options for Rhino 3D Model

The Import Options dialog is available when importing a Rhino model in any of the following workflows:

- **File > Open**
- **File > Interoperability > Merge**
- **File > External Content > Place Hotlink**, from the **New Hotlink Module** dialog box.



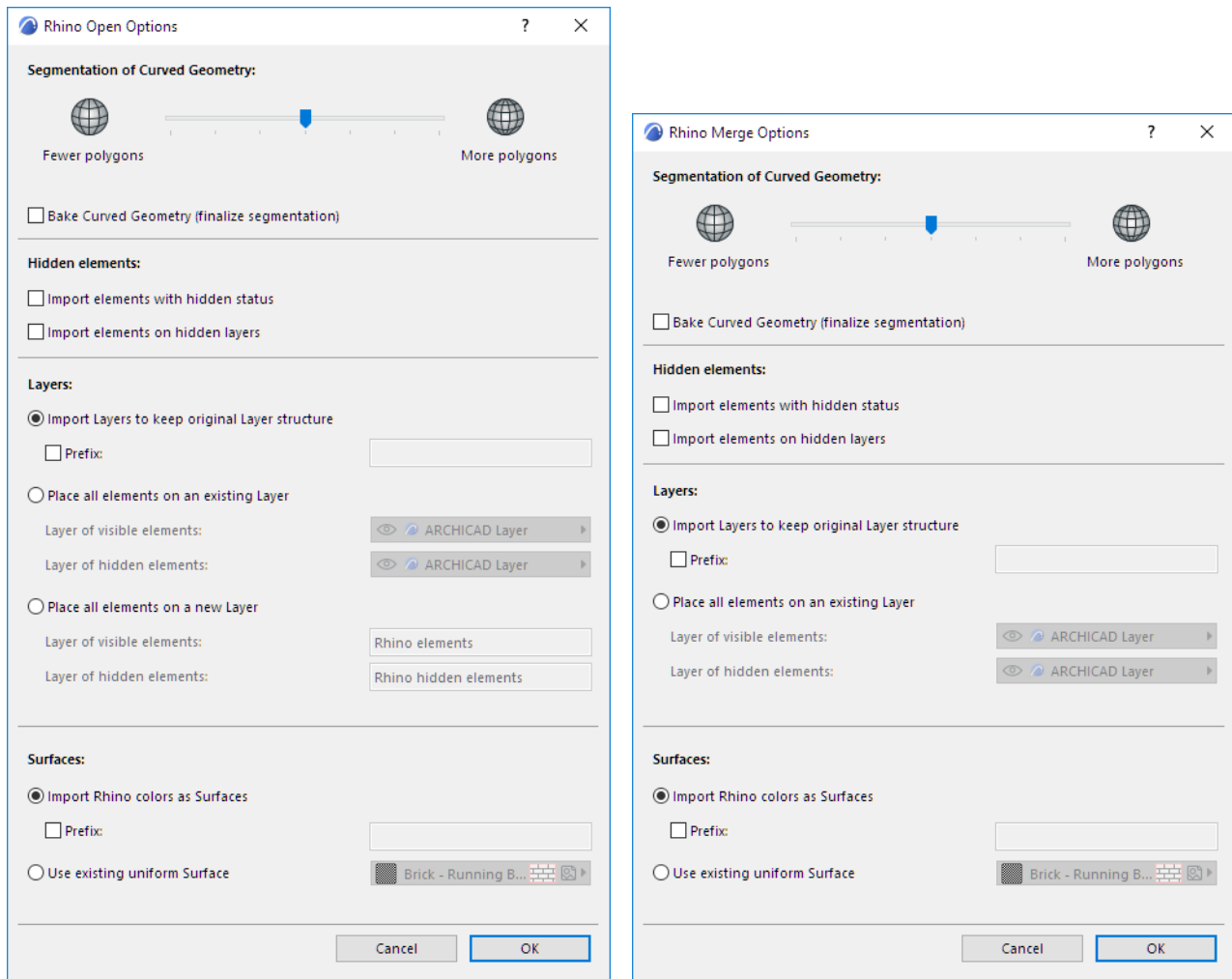
In each workflow, choose the Rhino 3D Model (.3dm) file type, and click **Options** to open the dialog.

The name of the dialog box varies depending on the command:

- **Rhino Open Options**
- **Rhino Merge Options**

- **Rhino Import Options** (when Hotlinking a Rhino model)

See [Hotlink Rhino File and Import Rhino 3DM File as Object for these workflows.](#)



Segmentation of Curved Geometry

Use the slider to adjust the segmentation of curved surfaces imported from Rhino.

Notes:

- The segmentation slider affects curved geometry only, not planar elements
- If you increase segmentation (resulting in a higher number of polygons for smoother surfaces), remember that a high polygon count affects both file size and performance

Baked Curved Geometry

Optionally, check the **Bake Curved Geometry** box to *finalize segmentation* before import. As a result:

- You will not be able to further adjust segmentation once the model is imported to Archicad
- On the other hand, element snapping will be available
- In general, navigation is faster in models if this option is checked

Hidden Elements

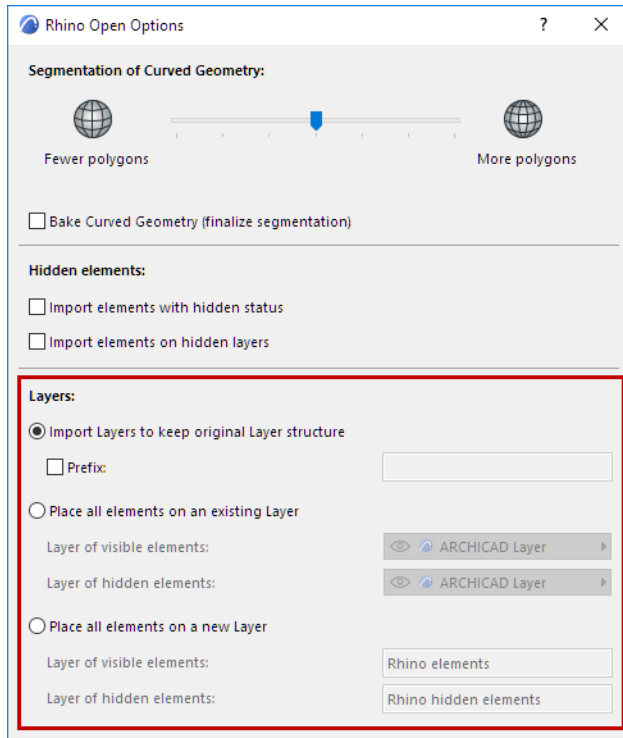
Use these controls to define how to interpret Rhino’s “hidden” elements.

In Rhino (unlike in Archicad), it is possible to hide an individual element, whether its Layer is visible or not. By default, such hidden elements are NOT imported to Archicad.

To import them, check the **Import hidden elements** box.

Layers

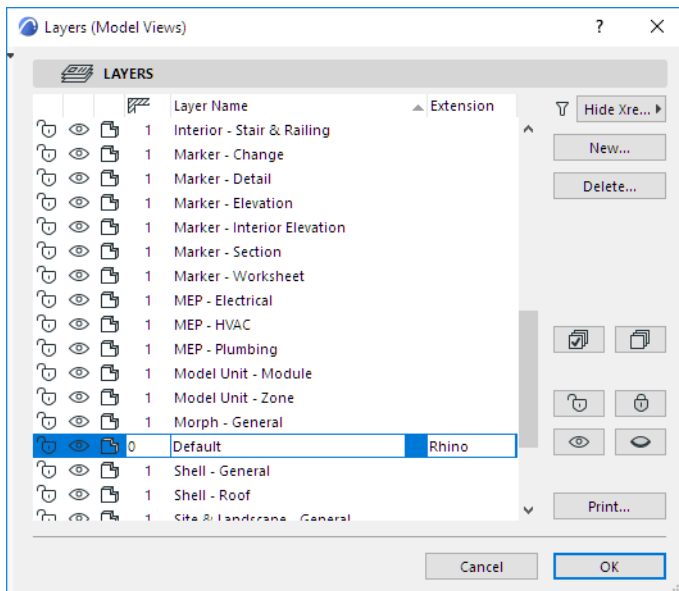
Use these controls to define the Layers onto which to place imported Rhino elements (visible and hidden, if any).



Import Layers to Keep Original Layer Structure (Default)

Each Rhino Layer is recreated as a separate Archicad Layer, named after the Rhino Layer's folder path: folder_subfolder_layer.

- These Layer names automatically get the extension "Rhino".
- Any identical Layer names are differentiated with a number added onto the extension.
- You can add an optional **Prefix** to the Rhino Layers.



Place all elements on an existing Layer

Use the pop-up to choose Archicad project Layers for the visible and for the hidden elements imported with the Rhino model.

Place all elements on a new Layer (Available only when using Open. Not available with Merge.)

Create new Layers in Archicad for the imported visible and hidden elements.

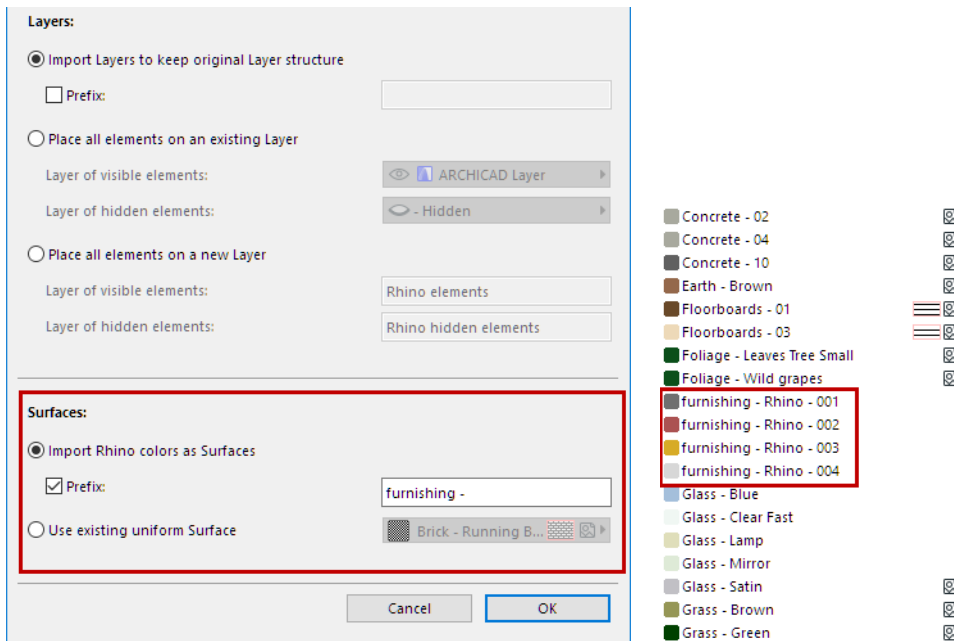
Surfaces

Use these controls to define how to handle Rhino colors, which are handled as Surface attributes in Archicad.

Import Rhino colors as Surfaces

With this option, each imported Rhino color

- is recreated as a separate Archicad Surface.
- is automatically assigned the name of “Rhino” plus a number
- can be assigned an additional, optional **Prefix**

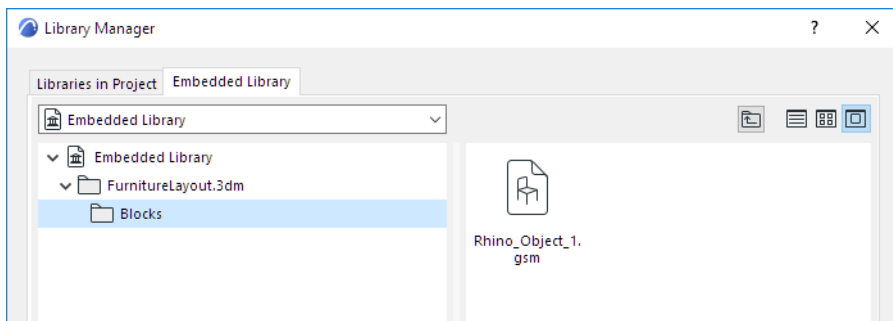


Use existing uniform Surface

Use the pop-up to choose a single Archicad Surface to use for all colors imported with the Rhino model.

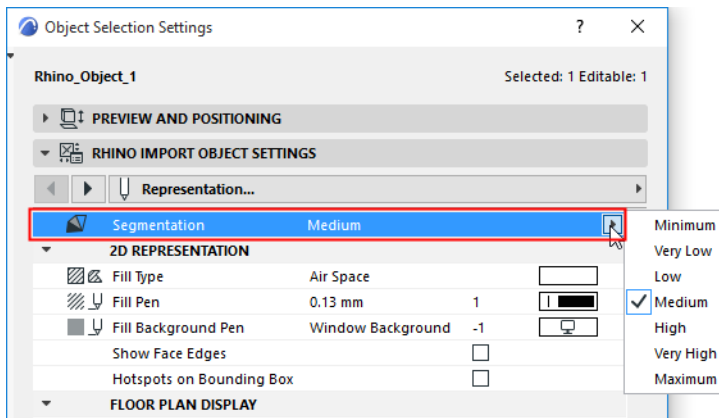
Rhino Object Parameters in Archicad

Each imported Rhino element becomes a separate Archicad object (.gsm), with a file name prefix of "Rhino". These are stored in the Archicad Embedded Library, in a separate folder for each imported Rhino File, and a separate sub-folder for each layer.



Use the **Rhino Import Object Settings** panel of Object Settings to set its appearance.

Note: Object Settings of placed, Rhino-based hotlinks cannot be edited.



Note the following Rhino-specific parameters:

Segmentation

This control is available for objects imported from Rhino, provided that you did *not* check the **Bake Curved Geometry** option at import. (See [Baked Curved Geometry](#), above.)

Note: If you placed the selected object in Archicad using the Object tool (instead of importing it from Rhino), Segmentation is not available.

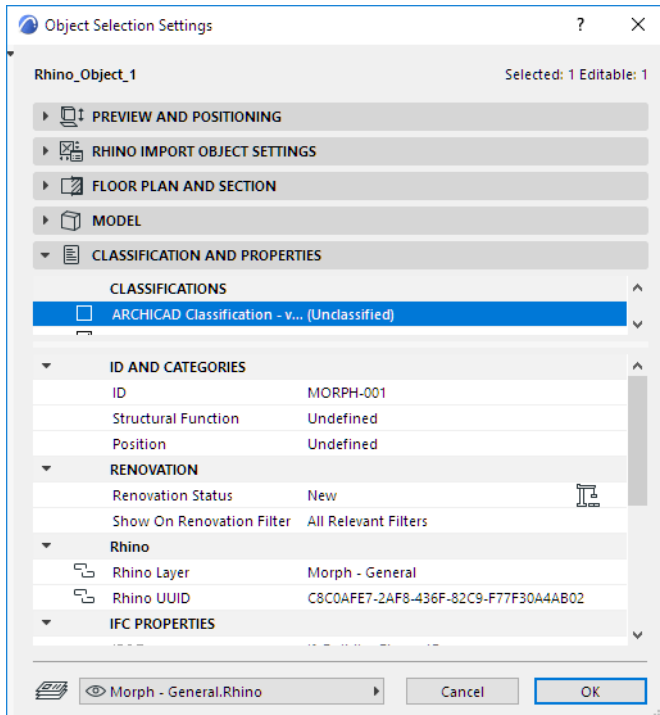
Use the Segmentation pop-up to adjust the smoothness of a selected curved object. Remember that a high polygon count may affect file size and performance.

2D Representation

- **Show Face Edges:** Show them if you want to see the internal surface edges in 2D.
- **Hotspots on bounding box:** Turn this on if you want to see additional hotspots on the object's bounding box in 2D. This can be useful, for example, for dimensioning an orthogonal object such as a curtain wall.

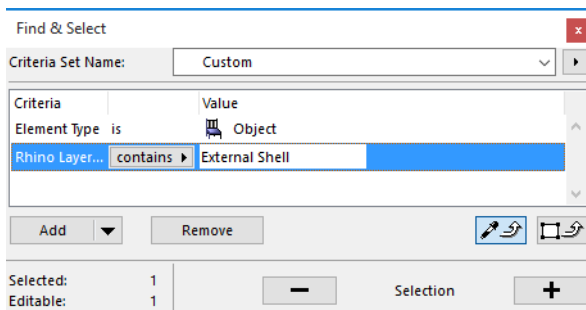
Rhino Properties in Archicad

Every element imported from Rhino and placed in Archicad is assigned an Element Property Group called Rhino, in the Object Settings Classification and Properties panel.



These properties store the element's original Layer and original ID in Rhino. (This is true even if you chose to place all Rhino elements onto a single Layer in Archicad.)

Use these properties to find and select Rhino elements by Layer, even after modifying the Archicad Layer structure.



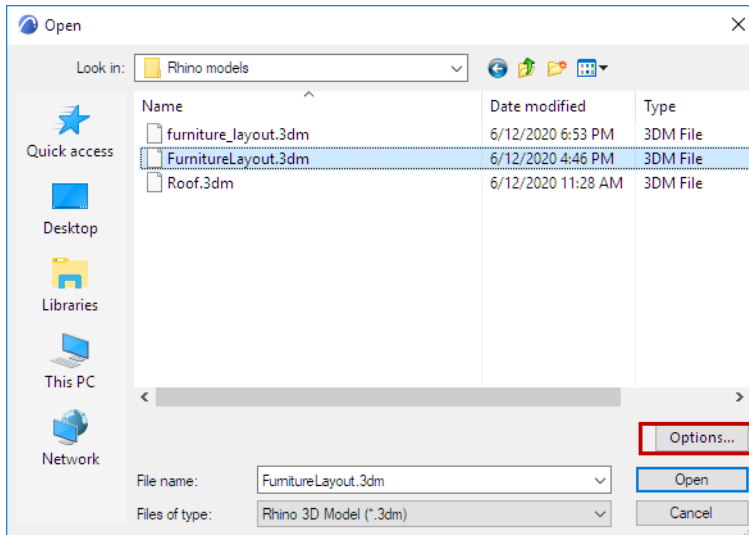
Rhino View Display in Archicad

After **Open**, the views of the imported Rhino model are now 3D axonometric views listed in Archicad's View Map. The view's display mode (a 3D Window Setting) depends on the original display mode in Rhino.

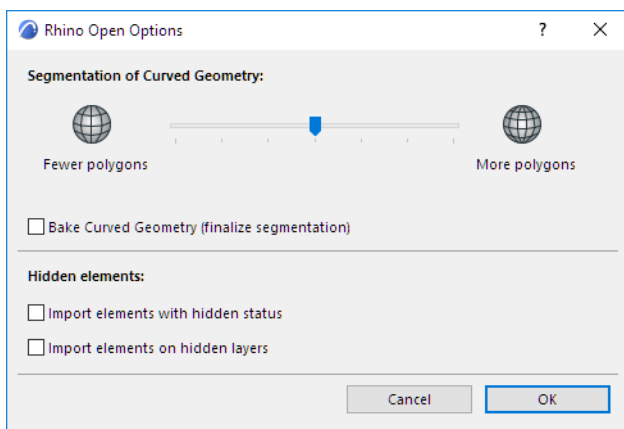
Display Mode of View in Rhino	Display Mode of View in Archicad
Wireframe, Ghosted, X-Ray, Technical	Wireframe
Shaded, Rendered, Artistic, Pen	Shaded

Import Rhino 3DM File as Object

1. Use this command to import a Rhino file as a single object in Archicad:
File > Libraries and Objects > Import Rhino 3DM file as Object
2. Click **Options** for Rhino Open Options:



3. Set your preferences for Segmentation and Hidden elements:



- **Segmentation of Curved Geometry**

Use the slider to adjust the segmentation of curved surfaces imported from Rhino.

Notes:

- The segmentation slider affects curved geometry only, not planar elements
- If you increase segmentation (resulting in a higher number of polygons for smoother surfaces), remember that a high polygon count affects both file size and performance

- **Baked Curved Geometry**

- Optionally, check the **Bake Curved Geometry** box to *finalize segmentation* before import. As a result:
 - You will not be able to further adjust segmentation once the model is imported to Archicad
 - On the other hand, element snapping will be available
 - In general, navigation is faster in models if this option is checked

- **Hidden Elements**

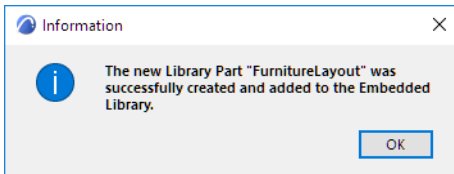
Use these controls to define how to interpret Rhino's "hidden" elements.

In Rhino (unlike in Archicad), it is possible to hide an individual element, whether its Layer is visible or not. By default, such hidden elements are NOT imported to Archicad.

To import them, check the **Import hidden elements** box.

4. Click **OK**.

5. You are alerted that the import was successful:

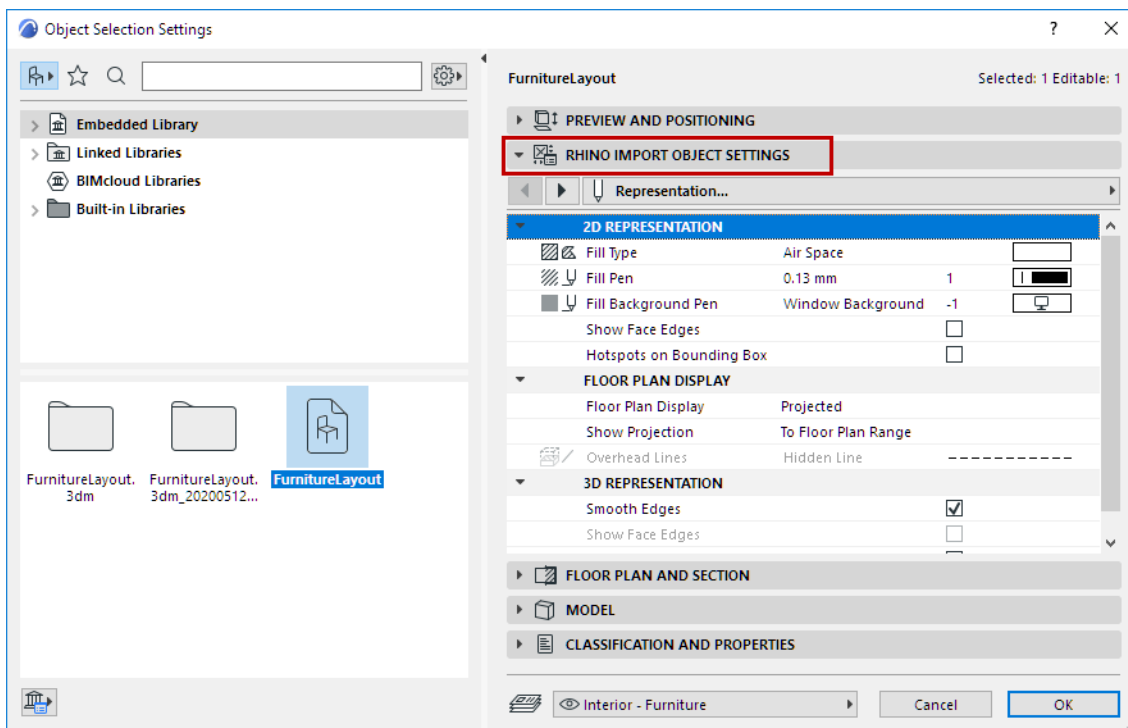


6. Use the Object tool to place it into the project.

Rhino Object Settings in Archicad

A Rhino file placed as a single object in Archicad is stored in the Archicad Embedded Library.

Use the **Rhino Import Object Settings** panel of Object Settings to set its appearance.



Note the following Rhino-specific parameters:

Segmentation

This control is available for objects imported from Rhino, provided that you did *not* check the **Bake Curved Geometry** option at import.

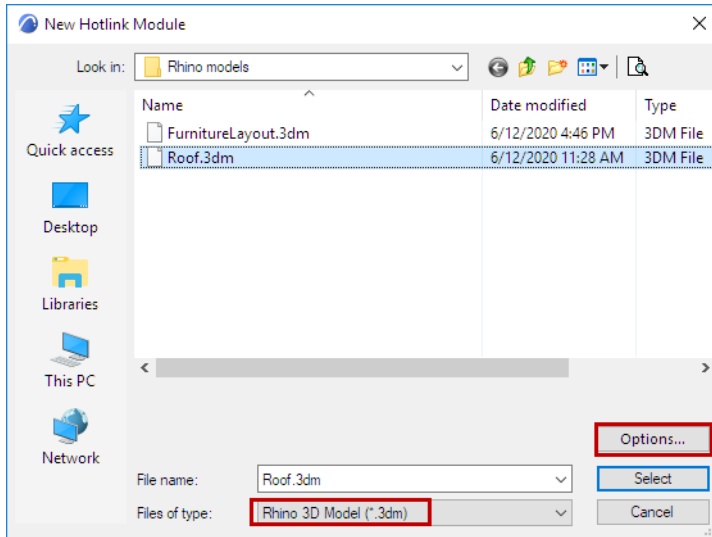
Use the Segmentation pop-up to adjust the smoothness of a selected curved object. Remember that a high polygon count may affect file size and performance.

2D Representation

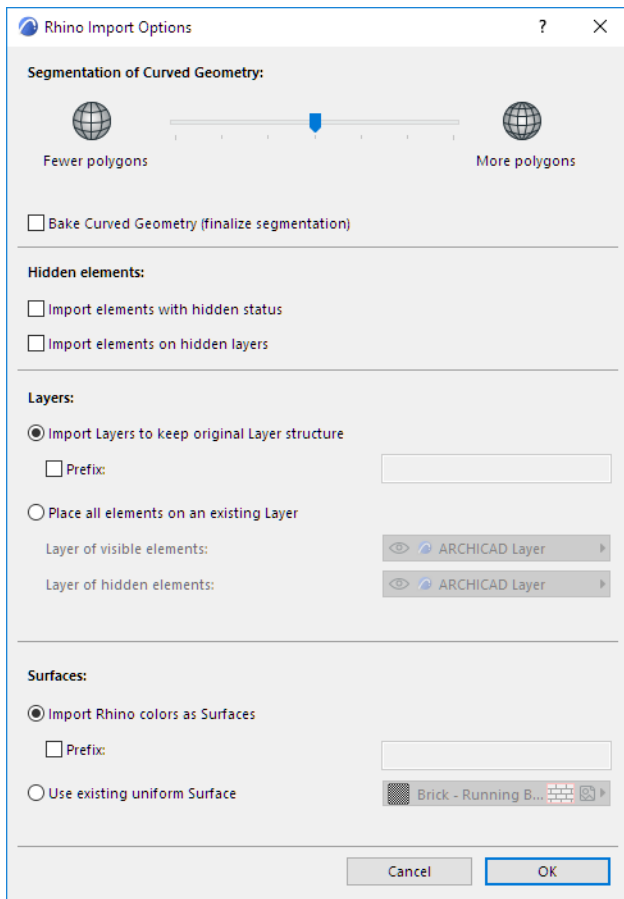
- **Show Face Edges:** Show them if you want to see the internal surface edges in 2D.
- **Hotspots on bounding box:** Turn this on if you want to see additional hotspots on the object's bounding box in 2D. This can be useful, for example, for dimensioning an orthogonal object such as a curtain wall.

Hotlink Rhino File

1. As when placing any other Hotlink, use **File > External Content > Place Hotlink**.
2. From the **New Hotlink Module** dialog, choose the Rhino 3D Model (*.3dm) format, then click **Options**.



3. The **Rhino Import Options** dialog appears.



These are the same options as when opening/merging a Rhino model:

[See Import Options for Rhino 3D Model.](#)

Note: The Segmentation settings cannot be changed after you place a Rhino-based hotlink.

4. After setting the Import Options, click **OK** to return to the New Hotlink Module dialog.
5. Select the newly created Rhino-based hotlink, and click **Select** to return to the **Place Hotlink** dialog box.
6. Follow the usual procedure for placing and managing this Hotlink. The Hotlink will be placed on the currently opened story in Archicad.

[See Place Hotlink for an overview of this process.](#)

Check Import Settings of Hotlink Module

Each Rhino-based Hotlink Module can have its own import settings. To see the hotlink's settings (or Conversion Info):

1. Open Hotlink Module Manager (File > External Content > Hotlink Module Manager)
2. Select the desired Rhino-based module from the list of placed modules.
3. At the right, check the **Conversion Info** section to see that module's information.



Export Model to Rhino

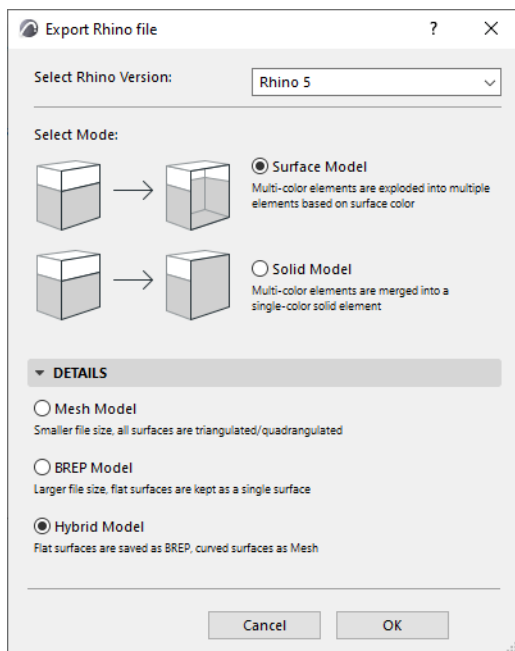
An Archicad model exported to Rhino transfers geometrical information only, because Rhino handles only this type of data.

In Rhino, Archicad elements are distinct, selectable elements. Relevant element attributes (surface color, texture) and the original ARCHICAD layer structure and grouped elements are retained.

Non-geometrical data (e.g. properties, tags, schedule data, quantities, etc.) are not transferred. Mapping of AC textures is not supported.

Export Archicad Model to Rhino

1. From the 3D window, use **File > Save as Rhino 3D Model (.3dm)** to export part or all of the Archicad model to Rhino.
2. Click **Options** in the Save as dialog box to access additional options:



Select Rhino Version

Choose the version of Rhino to export to.

Surface Model

Choose this if you want the Archicad element surfaces to retain their original colors in Rhino. Each multi-color Archicad elements must be exploded into several Rhino elements.

Note: Although a single Archicad element can have a different color for each surface, a single Rhino element can have only one color for all surfaces. Thus, to retain all the surface colors of a multi-color Archicad element, it will be exploded into multiple elements – each with a different color – in Rhino.

Use the Surface Model option if you are using the exported Archicad model for reference-model purposes (e.g. you are modeling part of the project in Rhino, and you want to see the surrounding environment – created in Archicad - in their correct colors.)

Solid Model

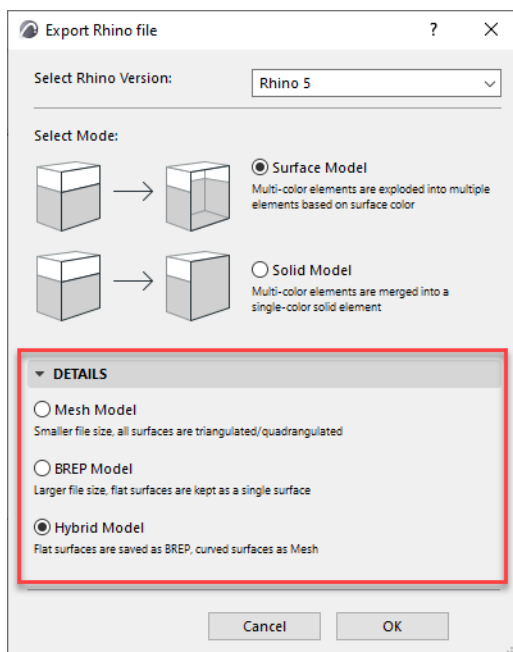
With this option, Archicad solid elements are interpreted in Rhino as a single Mesh or Polysurface element. Consequently, it will have a single color in Rhino (even if the original Archicad element used different colors on different surfaces).

Use the Solid Model option, for example, if you use the Rhino model in a simulation, where colors are less important.

Detailed Options (Advanced)

Note: The default options will work for most users. However, you can click the **Details** tab to see and change export settings if needed.

Given the differences in the way Archicad and Rhino work with curved surfaces and edges, Archicad seeks to optimize the way each element is interpreted. This is especially relevant for Archicad's composite elements and Morphs, which might have curved segments and/or multiple surfaces.



The default setting is **Hybrid** – a smart method that interprets each element as either BREP or Mesh depending on its surface geometry – so you end up with both kinds of elements in Rhino.

- If the element consists of all flat surfaces, it is automatically exported to Rhino as **BREP**
- If the element has curved surfaces, it is automatically exported to Rhino as a **Mesh**

However, you may wish to ensure that all model elements in Rhino are uniformly interpreted as either Mesh or BREP (for example, for simplified further editing or for export to a third program that does not recognize Meshes.)

- BREP Model (Boundary Representation)
 - Larger file size than Mesh Model
 - Can interpret holes and many-sided polygons
 - Internal triangulation of curved surfaces is not visible (only the boundary curve is shown)
 - No user control over hard vs. soft shading on edges
- Mesh Model

- Smaller file size than BREP Model
- All surfaces are triangulated or quadrangulated. Thus, a more complex element (e.g. many-sided polygon or element with a hole) is also triangulated, with resulting additional edges
- The show/hide status element edges cannot be individually controlled; the display of all element edges is uniformly controlled with the Mesh Wires option

For each edge, you can set whether its shading is displayed as hard or soft.

Working with RFA & RVT Files

Use the following workflows to import and export 3D models via native Revit file formats: .RVT and .RFA.

Archicad uses the ODA Teigha BIM Engine, which imports RFA file versions 2015-2023.

[Import RFA as GDL Object](#)

[Hotlink 3D RVT Model](#)

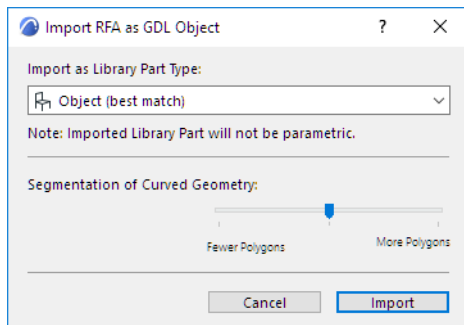
[Export 3D Model as Revit 3D Geometry](#)

Import RFA as GDL Object

This command imports a Revit Family (RFA) object (mostly manufacturer content) and generates a GDL Object, Door or Window having the same precise geometry and all parameters.

The resulting object will lose its editable parametric capabilities.

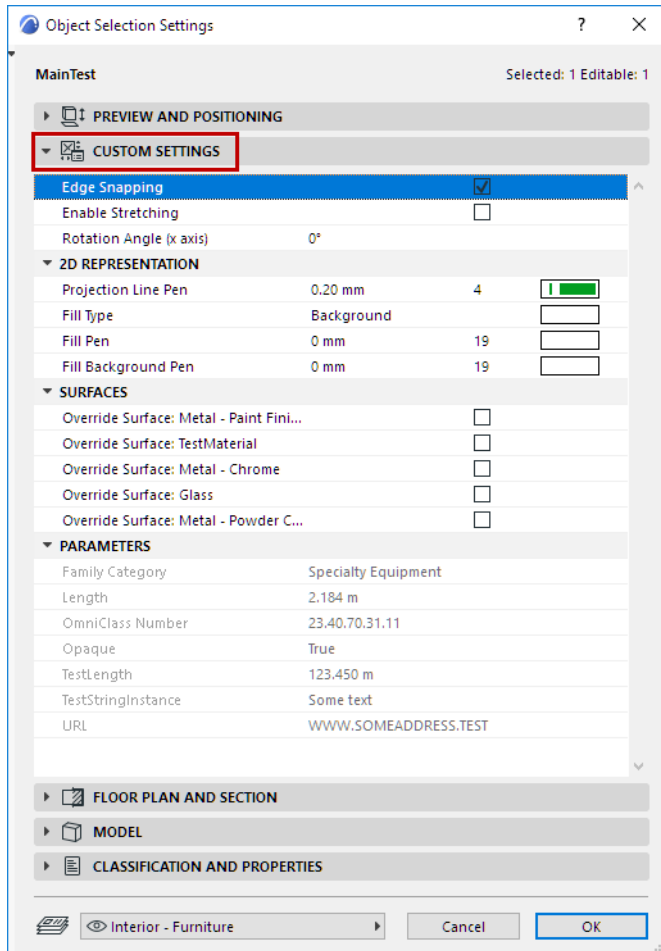
1. Use **File > Libraries and Objects > Import RFA as GDL Object**.
2. Browse for an RFA file to import, then click **Open**.
3. In the appearing **Import RFA as GDL Object** dialog, customize the import options:



- **Import as Library Part Type:** Based on the Revit Family Category and Host Object Category, the Library Part type will automatically select the best match: Object, Door, Window, or MEP Equipment. You may override the type here.
 - Imported MEP Equipment inherits the data and connections from the original RFA file, so you can connect them to MEP elements in your Archicad model.
 - **Segmentation of Curved Geometry:** Use the slider to set the level of geometric detail (polygon segmentation) at one of three levels. The more polygons, the larger the resulting object. It is recommended to keep this setting in the middle. If the object is too large, use Fewer Polygons. If the result is too segmented and rough, then use More Polygons. To check the Polygon count, download and install the Polycount add-on which is part of the Goodies package available at <http://www.graphisoft.com/downloads/goodies/index.html>
4. Click **Import**.
 - The imported family object is converted to a GDL Object, Door or Window in the Archicad project from which you issued the Import command.
 - The generated GDL Object, Door or Window is placed into the project's Embedded Library. Use the corresponding tool to place it into the project.

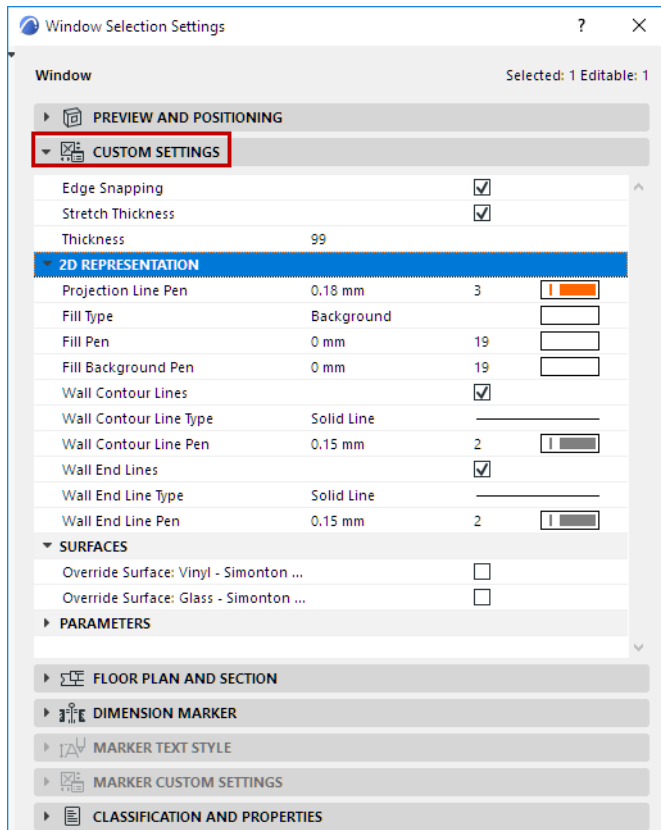
Imported Object: Custom Parameters in Archicad

With the “Import as Object” option, the following parameters are created:



- Edge Snapping, on or off.
- Enable Stretching: When on, Dimension 1, 2 and Height become enabled in the Preview and Positioning panel and can be graphically edited in the model. Stretching will distort the geometry.
- Rotation Angle (x axis): Rotate the object 0/90/180/270 degrees or a custom angle around the x axis.
- 2D Representation: Projection Line Pen, Fill Type and Fill Pens on Floor Plan.
- Surfaces: Allows override of individual embedded Surfaces.
- Parts Visibility by Category: Control the visibility of parts of the Object as defined by its RFA Categories
- Parameters: data as defined in the RFA file. This data is locked as read-only, and has no effect on the object's geometry.

Imported Door or Window: Custom Parameters in Archicad



- Edge Snapping, on or off.
- Stretch Thickness: When on, enables you to modify the Thickness of the Door/Window (will stretch the geometry).
- 2D Representation: Projection Line Pen, Fill Type and Fill Pens on Floor Plan; Wall Contour Lines: can be turned on or off and set line type and pen; Wall End Lines can be turned on or off and set line type and pen.
- Surfaces: Allows override of individual embedded Surfaces.
- Parts Visibility by Category: Control the visibility of parts of the Door/Window based on their Revit Category. This option only appears if there are multiple Categories defined in the RFA.
- Parameters: Data as defined in the RFA file. This data is locked as read-only, and has no effect on the object's geometry.

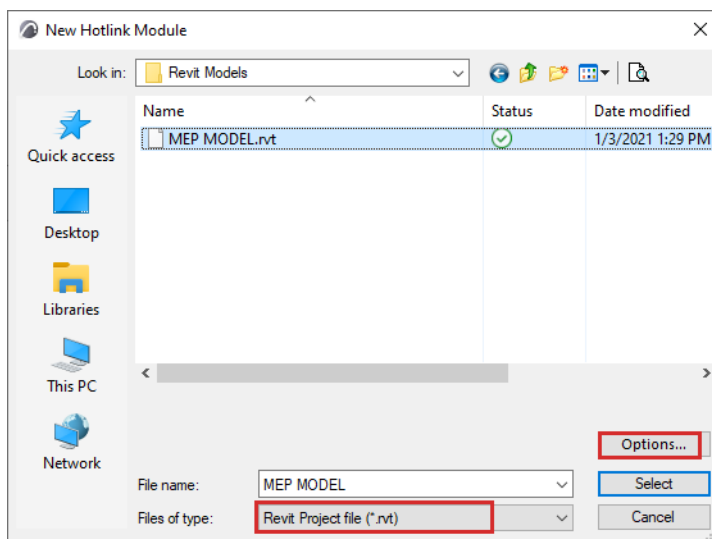
Note: The 'Host Object' will be automatically filtered out during Import.

Hotlink 3D RVT Model

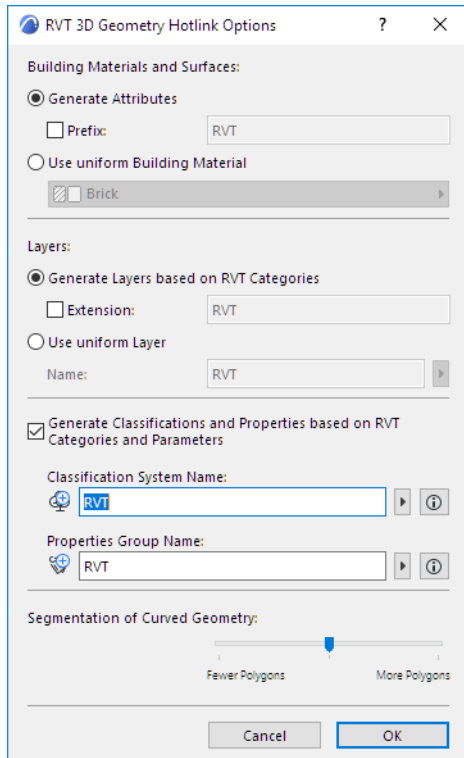
Use this workflow to import 3D model elements from Revit RVT files, placing them as a Hotlink in Archicad. The resulting GDL objects use the precise geometries and properties, including true surface colors, of the imported elements. The “Discipline” view filter set in Revit is also correctly recognized in Archicad.

Use the same workflow to place an RVT Hotlink as any other hotlink in Archicad.

1. In the Archicad Floor Plan: Use **File > External Content > Place Hotlink**.
2. In the appearing Place Hotlink dialog box, click **Select Module**.
3. From the **New Module** pop-up button, click **From File** or **From BIMcloud**
4. The **New Hotlink Module** dialog appears: browse for an RVT type file.
5. Click **Options**.



6. The **RVT 3D Geometry Hotlink Options** dialog appears. Here, customize the import options:



Building Materials and Surfaces

Choose one of these two options for handling imported Revit Building Materials and Surfaces.

- **Generate Attributes:** Adds each Revit surface and material to the Archicad project as Surface and Building Material attributes, with an optional prefix to distinguish them in the Archicad attribute lists.
- **Use uniform Building Material:** Converts all Revit surfaces and materials to a single Building Material in Archicad. In this case, click the pop-up to choose a Building Material from the Archicad project. All generated Library Parts will use the selected Building Material. (Choose this if you want to avoid increasing the number of Archicad project Attributes.)

Layers

Choose one of these options for creating Layers for imported Revit Objects:

- **Generate Layers based on RVT Categories:** Places Objects onto Archicad Layers that correspond to Revit Categories. Use an optional extension to distinguish these Layers in the Archicad Layer list.
- **Use uniform Layer:** Places all imported Objects onto a single Archicad Layer. Enter a Layer name in the field. (If such a Layer already exists in the project, that Layer will be used; otherwise, a new Layer is created.)

Classifications and Properties

Check the box if you want to **Generate Classifications and Properties based on RVT Categories and Parameters**.

Revit Parameters (textual, numeric, and True/False only) are imported as Archicad Properties, in a Property Group using the name you define below.

- **Classification System Name:** Revit Categories are imported to Archicad as a Classification System, using the name you define here. (All Categories of the Revit project are recreated in Archicad - not just the ones assigned to the hotlinked content.)

The Classification System name will also use the Properties Group name you define below (e.g. RVT). Its Version is the version number of the RVT file format you have placed. (After import, you can see this data in Archicad's Classification Manager.)

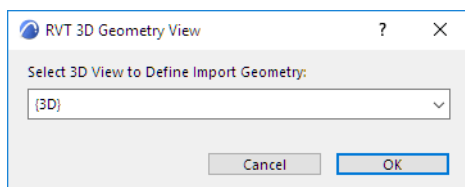
- **Properties Group Name:** The imported parameters will be added to the Archicad project as a Property Group, using the Properties Group name you enter here (e.g. RVT). (After import, you can see this data in Archicad's Property Manager.)

Note: After hotlink, it is recommended to merge (copy) the newly added Classification System and its Properties into the Archicad host file, in Classification Manager and Property Manager.

Segmentation of Curved Geometry

Use the slider to set the level of geometric detail (polygon segmentation) at one of three levels. The more polygons, the larger the resulting object. It is recommended to keep this setting in the middle. If the object is too large, use Fewer Polygons. If the result is too segmented and rough, use More Polygons.

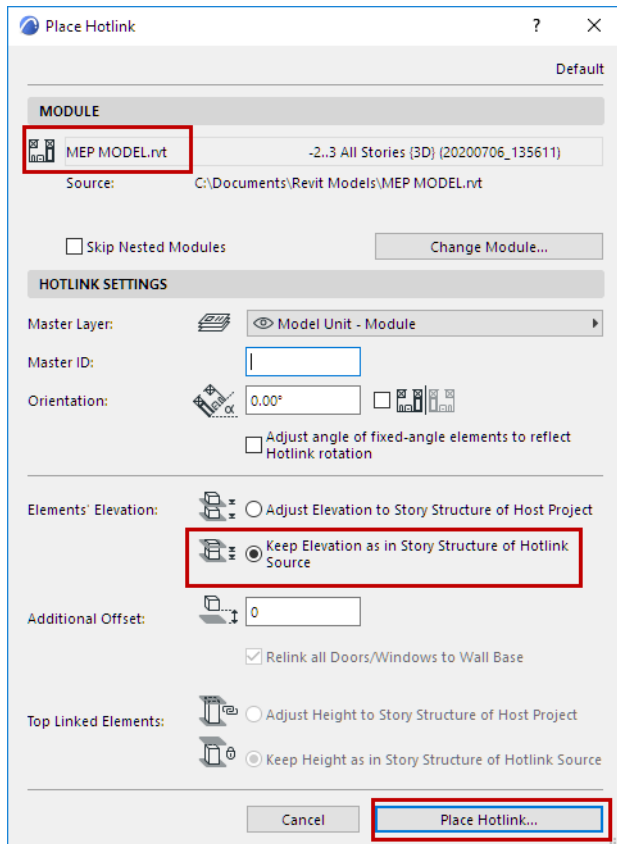
7. Click **OK** to return to the New Hotlink Module, then click **Select**.
8. The RVT 3D Geometry View dialog appears.



Choose a Revit 3D view (if there is more than one) from which to import the model elements. Archicad will create GDL objects based on the content of the selected view. Only Revit objects created in a 3D view of Revit can be imported to Archicad.

9. Click **Select** to return to the Place Hotlink dialog. The RVT file you selected is shown as the module to be placed.

10. For the Elements' Elevation option, make sure you choose **Keep Elevation as in Story Structure of Hotlink Source** (as shown below). Otherwise, the imported model may “fall apart” when its elements are placed in a different story structure.



11. Click **Place Hotlink**.

- Elements of the imported Revit 3D view are converted to GDL Objects in the Archicad project.
- The objects are placed into the project's Embedded Library.

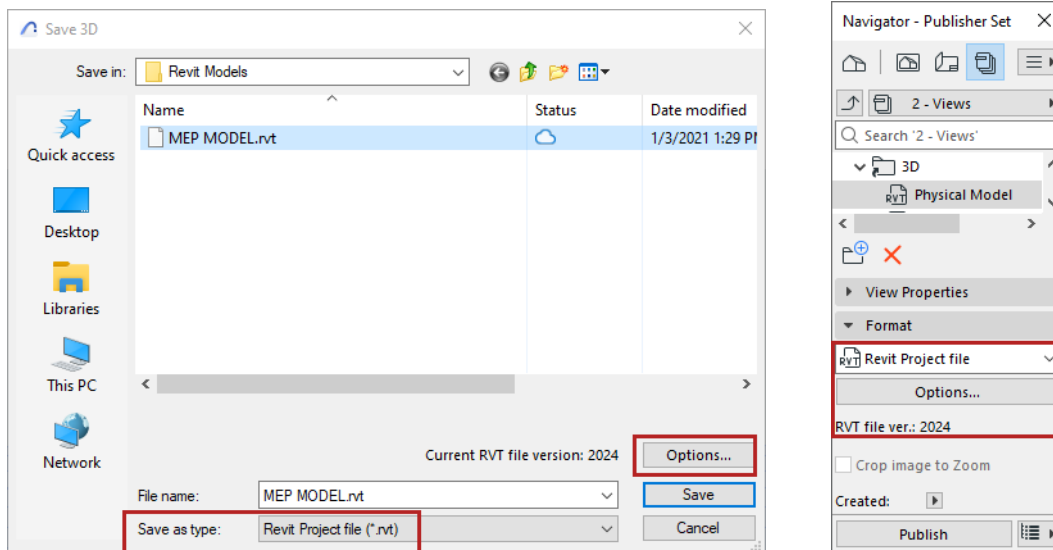
Hint: To view the RVT model file on its own, hotlink the file into an empty Archicad project.

For general information on managing Hotlinks in Archicad, see [Hotlink Module Manager](#).

Export 3D Model as Revit 3D Geometry

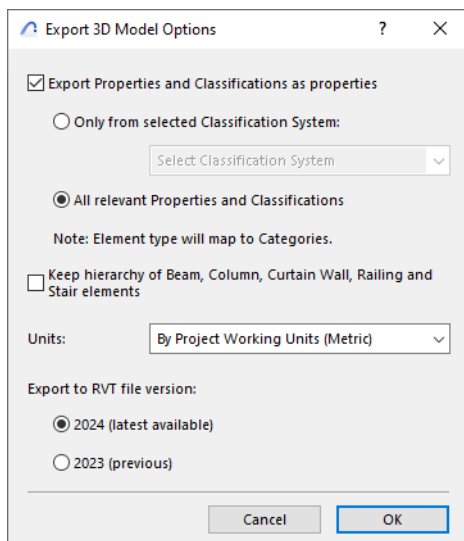
This command exports Archicad 3D model content to Revit, generating RVT objects with precise geometries (including true colors) and properties.

1. In Archicad, open the 3D window containing the view to export. The View Settings, including Partial Structure Display, affect the content of the exported object. Graphic Overrides are ignored.
2. Do one of the following:
 - Use **File > Save As**, and choose the RVT file format.
 - Add the 3D View to a Publisher Set and set its format to Revit Project file.



Save the file as RVT, or Publish the view in RVT format

3. Click **Options** to bring up the **Export 3D Model Options** dialog.



- **Export Properties and Classifications as properties:** Check this to include Classification and Properties data in the exported model. Since Revit does not support multiple classifications, use the options below to determine which properties (or classifications) to export.

- **Only from selected Classification System:** Choose one Classification System to export its Properties with exported elements. Properties from other Classification Systems will not be exported.
- **All relevant Properties and Classifications:** All Properties in the exported elements will be included. All Classification Systems from the exported elements will also be included, but handled as properties in Revit.

Note: Exported elements are mapped to Categories in Revit, based on their Archicad Element type.

- **Keep hierarchy of Beam, Column, Curtain Wall, Railing and Stair elements:** Check this to export these element types as single elements. Uncheck it to export them as hierarchical sets of elements (for example: an exported Curtain Wall will include separate Frame and Panel elements).
- **Units:** Choose **By Project Working Units** to automatically determine Metric or Imperial for the exported model elements, or manually set the units.
- **File version:** Choose the preferred Revit version to save to.

Notes:

- 2D data such as Drawings and Layouts are not exported.
- Parameters of GDL objects are not exported
- Archicad Project and Story data are not exported.
- Interference Checking (collision detection) in Revit can be applied to the exported RVT file.
- Archicad Zones are exported with their 3D representation only; therefore Revit Rooms are not generated from them automatically.
- The elements are exported as so-called Revit DirectShape (interoperability) elements. They can be used as reference model elements. Therefore, an exported Archicad Wall is not a Wall with full native functionality - it is just a referenced Wall element.
- Exported geometries are frozen and locked; they cannot be modified.
- The DirectShape elements do not contain materials in Revit, they have only colors (including transparency); therefore, only Archicad Surfaces are exported (Building Materials are not).

Working with FBX files

Archicad can open and save the FBX file format.

Use FBX to transfer your 3D models to and from other 3D modeling applications, typically for rendering.

You can also open objects in FBX format (e.g. from object repositories) directly in Archicad.

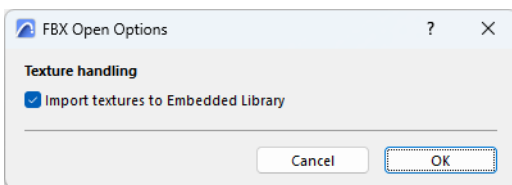
Open

Open or merge an FBX file (.fbx) in Archicad with one of these commands:

- **File > Open** (opens a new model in Archicad)
- **File > Interoperability > Merge** (adds the model to a running Archicad project)
- **File > Libraries and Objects > Import FBX as GDL Object** (imports the model without placing it in the project)
- Drag-and-drop

At import, the FBX model is saved as a GDL Object to the Embedded Library.

The FBX Open Options dialog box opens, where you can decide how to handle textures: check the box if you want to import textures and save them to the Archicad project's Embedded Library.



Save

Saving the Archicad model to FBX results in a single FBX file containing geometry, surface and texture information.

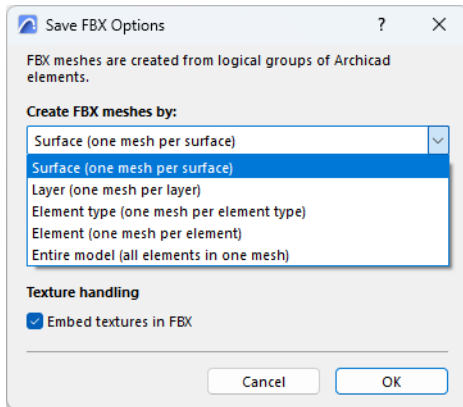
1. Activate the 3D window.
2. Use File > Save as.
3. Choose the FBX file format, then click Save.

The Save FBX Options dialog box opens:

Create FBX meshes by:

When you export the Archicad model, the resulting FBX file consists of meshes, which separate the model into smaller chunks. These FBX meshes are constructed from logical groups of Archicad elements.

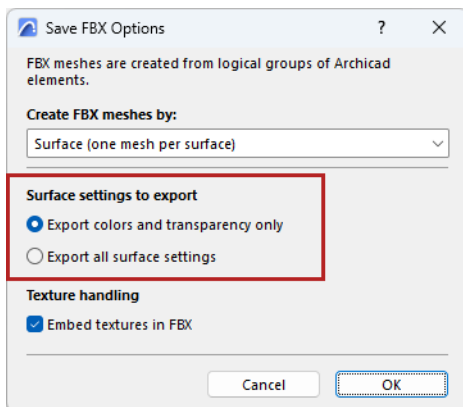
Choose a grouping logic:



- **Surface (one mesh per surface):** Create a mesh for elements having the same Surface.
- **Layer (one mesh per layer):** Create a mesh for elements on the same Layer.
- **Element type (one mesh per element type):** Create a mesh for elements of the same type (one mesh for all Walls, one mesh for all Beams, etc.)
- **Element (one mesh per element):** Create a separate mesh for each element. Each hierarchical element (like Curtain Wall) is exported as a single mesh.
- **Entire model (all elements in one mesh):** Create a single FBX mesh out of all elements.

Surface settings to export

Archicad surfaces include a variety of light-related parameters. Choose which surface parameters to export.



- **Export colors and transparency only:** For each surface, export only the color and transparency values - skip the intensity values. This mode maximizes compatibility with other applications.
- **Export all surface settings:** All parameters are preserved - this can be useful if you plan to reimport the model to Archicad.

Texture handling

Embed textures in FBX: Check to export textures and embed them in the resulting FBX file.

Export to FRILO

Export a SAF file from Archicad directly to FRILO.

To use this function, you must have FRILO installed on the same computer as Archicad.

1. Use File > Interoperability > FRILO Connection > Open in FRILO.

If you do not see the command: Use the Work Environment dialog to add this command to the File > Interoperability menu, and/or to a toolbar.



2. FRILO starts up and opens the model.

The SAF file is exported using the settings of the chosen SAF translator.

To choose a different SAF translator: use **File > Interoperability > FRILO Connection > FRILO Export Settings.**

Export to Artlantis

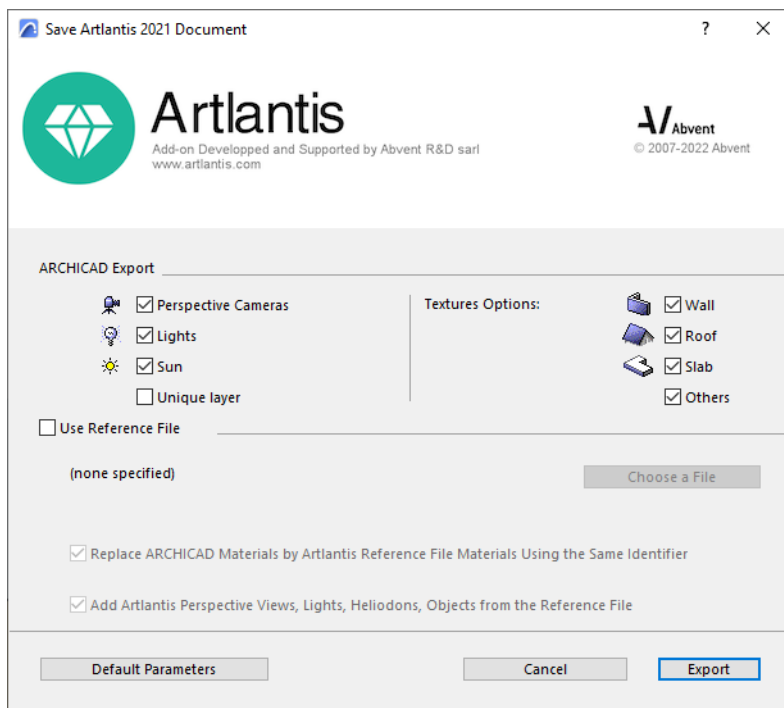
Use this add-on to export the Archicad model into Artlantis versions 2020 or later. Although this add-on is installed by default with Archicad, the add-on remains the property of Abvent R&D, which ensures its development and maintenance.

For more information, please visit: <http://www.artlantis.com>.

How to Export

1. Display the perspective 3D view in Archicad.
2. Select the **File > Save as...** command.
3. In the file format list, choose the **Artlantis** (*.atl) version you need.
4. Click **Save** button, and the Export dialog box will be displayed.

From this dialog, you can either export a new Artlantis file, or update an .atl file already sent into Artlantis.



Creating a New File

Check the items to export.

- **Perspectives Cameras:** All cameras created in Archicad will appear on Artlantis's Perspectives Inspector List.
- **Lights*:** All lamps inserted into the Archicad project will appear on Artlantis's Lights Inspector List.
- **Sun*:** The heliodon (geographical position, date and time) will appear on Artlantis's Heliodons Inspector List.

- **Unique Layer:** If this is checked, all the Archicad layers are exported into a single layer to Artlantis. If unchecked, all the Archicad layers will be recovered into the Layers list of the Artlantis Objects Inspector.
***Note:** If the Lights or Sun controls are disabled on the export dialog, open the Archicad PhotoRendering Settings dialog box (**Document > Creative Imaging**), activate the Basic Rendering Engine and switch **ON** the appropriate light source.
- **Texture Options Wall, Roof, Slab, Others*:** Textures associated with the Archicad Surfaces will be exported.
***Note:** The Others option includes all Archicad tools such as Column, Beam, Stair, Door, Window, as well as Mesh and Object.

Update an .atf File Already Sent into Artlantis

All controls described in the [Creating a New File](#) section are also valid here!

1. Select the **Use Reference File** option
2. Click **Choose a File**.
3. Select the .atf reference file.

Options

- Replace Archicad Materials with Artlantis Reference File Materials Using the Same Identifier
- Add Artlantis Perspective Views, Lights, Heliodons, Objects from the Reference File

If either of these options is activated:

New elements in the Archicad project (geometry, viewpoints, lamps, heliodon) will be exported only if they are not part of the reference file.

Note on Exporting Curved Elements

By default, curved Archicad elements exported into Artlantis are never smoothed. To smooth the curves in Artlantis, go to the Shaders inspector tab **Advanced Parameters** area, select the needed material, and use the **Smoothness** slider.

Interoperability with MEP Applications

This section describes the recommended workflow for data exchange with MEP applications.

[Export Archicad Files to MEP Software](#)

[Import MEP Data to Archicad](#)

Export Archicad Files to MEP Software

Preparing the Archicad Model for Export

Before handing an Archicad file over to an MEP engineer using MEP-specific software, certain simplifications have to be made to the architectural model. Like structural engineers, MEP experts need only certain parts of the project - those which are relevant to their work.

Relevant Data for Export to MEP

Typically, the most relevant 3D elements for export to MEP are:

- load bearing structures,
- elements of the building envelope,
- the boundaries of interior spaces.

Also important:

- structural grid of the load bearing structure,
- planned placement of lighting objects, bathroom and kitchen equipment,
- positioning of built-in furniture,
- all space functions and numberings, must also be clearly indicated,
- levels, geometry and structural grid of suspended ceilings and false floors (this will assist the engineer to correctly design HVAC equipment (air inlets, sprinklers etc.).

Layers containing elements of the architectural model that are *irrelevant* to the MEP design should be switched off and thus excluded from the export data. Irrelevant elements are usually these:

- floor claddings,
- door and fenestration details,
- furniture.

It is possible to define separate layer combinations that specify content suited for the electrical, plumbing, lighting, acoustic, low current system engineer or elevator design specialist.

[See *Layer Combinations*.](#)

The two most commonly used file formats to export Archicad data to MEP applications are DWG and IFC.

Save Views to DWG

The majority of MEP engineers work in a 2D design environment (mostly Autodesk AutoCAD) and therefore wish to receive AutoCAD DWG files.

In Archicad, the architect creates DWG files by defining floor plan, section or elevation views, and exporting these, rather than exporting the 3D model itself. It is advisable to create a Publisher Set with pre-configured settings in Archicad to enable easy creation of the documentation to be handed over to the MEP expert. This way, the preset views representing the most up-to-date state of the project may be saved in the specified destination folder in DWG format, by a touch of a button.

[See *Working with DWG/DXF Files*.](#)

Save Models to IFC

Some MEP engineers design directly in a 3D work environment (AutoCAD MEP or Revit MEP). To fit this workflow, Graphisoft has developed a translator-based IFC data exchange feature.

[See *Export IFC Model from Archicad*.](#)

This feature translates the Archicad model to IFC, which is currently the most advanced BIM model exchange format. When creating IFC content for the MEP specialist, it is advisable to use an IFC Translator preconfigured for MEP exchange (e.g. “DDS-CAD MEP Export”), or to set up an IFC translator customized for your needs.

[See *IFC Translators: Overview*.](#)

Many MEP applications support BIM Collaboration Format (BCF)-based communication as an enhancement to IFC-based data exchange. The BCF workflow enables participants to add comments to the overlapping issues (e.g. collisions). BCF comments include the referenced IFC elements' GUIDs and an attached preview (snapshot) picture; these enable users to easily locate the “problematic” model elements in the referenced view, using the same exact camera position as the snapshot. Archicad's Issue Management supports BCF commenting, import and export.

[See *Issue Handling*.](#)

IFC Model Exchange with Archicad for Revit

This Graphisoft Add-In is developed for MEP engineers using Autodesk Revit.

This add-in optimizes the IFC data import from any IFC data source to Revit used by MEP engineers.

It is available from Graphisoft's website at:

<https://graphisoft.com/downloads/interoperability>

Import MEP Data to Archicad

The two most commonly used file formats to import data from MEP applications to Archicad are DWG and IFC.

Importing MEP Data via DWG

MEP data from AutoCAD arrive in DWG format. The best practice is to import only the MEP content, plus the structural grid added for reference.

Import the received MEP content in Archicad either as an XREF or place as a Drawing. Archicad's Drawing Tool includes an Embedded Layer Visibility Setting, which enables the switching of layers on and off within the drawing content.

For further information, see [Show/Hide Layers of Imported DWG/DXF/DWF Files](#).

Importing MEP Data via IFC

Archicad's IFC import function automatically translates the imported 3D MEP elements into native MEP Modeler objects.

See [MEP Modeler](#).

IFC Model Exchange with Archicad for Revit

This Graphisoft Add-In is developed for MEP engineers using Autodesk Revit.

This add-in optimizes the IFC data import from any IFC data source to Revit used by MEP engineers.

It is available from Graphisoft's website at:

<https://graphisoft.com/downloads/interoperability>

Collision Detection in Archicad

Archicad's Collision Detection function can detect collisions between MEP elements and architectural elements.

See also [Collision Detection](#).

Interoperability with Energy Analysis

Archicad users have several ways of co-operating with energy specialists.

[Integrated Energy Evaluation](#)

[Export Files from Archicad to Energy Analysis Software](#)

[Import Energy Analysis Results Into Archicad](#)

Integrated Energy Evaluation

Archicad's Energy Evaluation function executes an automatic model geometry and material property analysis on the building model. The resultant data is then used as input by the VIP-Core dynamic energy simulation program module integrated in Archicad. VIP-Core carries out the hourly energy-flow analysis to produce the Energy Performance Evaluation report. The energy performance of the design's current state is easily evaluated within Archicad, and the effects of changes made on the architectural model in order to improve the energy household may be reviewed instantly. Energy Evaluation can be applied at all phases of architectural design, it is therefore not necessary to exit Archicad and exchange data with external energy analysis applications for building energy performance optimization.

In order to produce local energy standard compliance documentation, however, the use of third party Energy Analysis Software may still be necessary. Archicad therefore supports the export of building model data produced by the automatic geometry and material property analysis.

To find out more about VIP-Energy analysis software, please visit [StruSoft's official product website](#).

For details on Archicad's Energy Evaluation function, see [Energy Evaluation](#).

See also the section on [Interoperability with Green Software \(Energy Evaluation\)](#).

Export Files from Archicad to Energy Analysis Software

Before the architectural model is exported to an energy analysis application, a 3D model view with simplified content must be defined. The Archicad Building Energy Model view should be simplified, by hiding unnecessary data. Volumes of internal spaces and their relative positions should be properly modeled.

For detailed information on building energy model creation with Archicad, please consult the following Archicad Help chapters (under Energy Evaluation Workflow: Overview):

- [Prepare the Architectural BIM for Energy Evaluation](#)
- [Define Thermal Blocks](#)
- [Automatic Model Geometry and Material Property Analysis](#)

There are two file formats available to export data from Archicad to energy analysis applications: IFC and Excel.

Using IFC

Energy specialists who work with applications offering the IFC-based file import option should use an IFC Translator preconfigured in Archicad for energy applications, or create a custom translator setup. For example, the exported architectural model must contain relation information between the Zones and their boundary elements.

See [Export IFC Model from Archicad](#).

Using Excel

Use the Geometry Data page of the result Excel table as an input for other energy calculation software, such as third-party excel spreadsheets that utilize simplified stationary energy calculation algorithms.

See [Reading the Energy Performance Evaluation - XLS Report](#).

Import Energy Analysis Results Into Archicad

Energy specialists use their energy analysis software to process the architectural data received via Archicad file export (detailed above). However, if the energy specialist makes changes in the architectural model, you will not be able to import those changes back into Archicad. Even though building energy applications are capable of changing certain model attributes (e.g. shading settings, building element properties), automatically implementing these modifications in the BIM model is not possible, due to the lack of data exchange functionality between the energy analysis software and CAD applications.

Unlike other design activities (e.g. structural, MEP), the energy expert produces written documentation, and does not modify the BIM model. Results of the energy analysis are typically in text format, enriched with tables, charts and graphs that display building element and envelope performance, energy consumption, energy balance overview and breakdown, HVAC system parameters and various additional results that depend on the specific program's additional functionality.

Thus, alteration requests by the energy specialist still have to be communicated to the architect in the traditional way, and eventually the architect has to implement the changes on the Archicad model based on instructions provided via sketches, email or telephone.

Note: Graphisoft offers an add-on to Archicad called EcoDesigner STAR. This add-on provides extra functionality that turns Archicad's built-in Energy Evaluation function into a high-end building energy analysis tool.

For details, see <https://community.graphisoft.com/t5/Data-Exchange-articles/EcoDesigner-STAR/ta-p/304044>

Working with IFC

IFC (Industry Foundation Classes) is an open, neutral data exchange format. It is optimized for OPEN BIM- and model-based interoperability among disciplines, primarily in the AEC industry.

BIM, or “Building Information Modeling,” is one of the biggest advances in the building industry’s working methods since the introduction of CAD software. BIM is not synonymous with 3D projects. Three-dimensional geometric representation is only one part of the digital deliverables. A project includes non-graphical information, such as calculations, that are used in surveying, facility management and energy calculation. A prerequisite for a successful BIM project is that intelligent information can be exchanged between different software and even operating systems, throughout the stages of the building process. This interoperability demands a neutral file format with an open standard that supports different systems. IFC is such a system, enabling us to synchronize building models between the disciplines much more easily.

The IFC format is used for the efficient import and export of 3D elements and related non-geometric data, regardless of the application used to create the model. The IFC format is ISO-certified and can be integrated into any existing quality assurance policies your office may have. IFC is developed in part by buildingSMART (formerly the IAI - the International Alliance for Interoperability).

Graphisoft has played an active role within the buildingSMART organization since 1996 and supports the IFC standard. Through IFC, Archicad can communicate with other disciplines within the context of the building model. The building model can be exported back to literally hundreds of other systems that support IFC.

The list of registered IFC supporter applications (by functional category/design disciplines) is available [here](#).

Archicad supports the import and export of both IFC2x3 and IFC4 versions. IFC data are managed according to the worldwide-certified IFC standard, IFC2x3 TC1 (Technical Corrigendum 1).

For more information, see the [buildingSMART website](#).

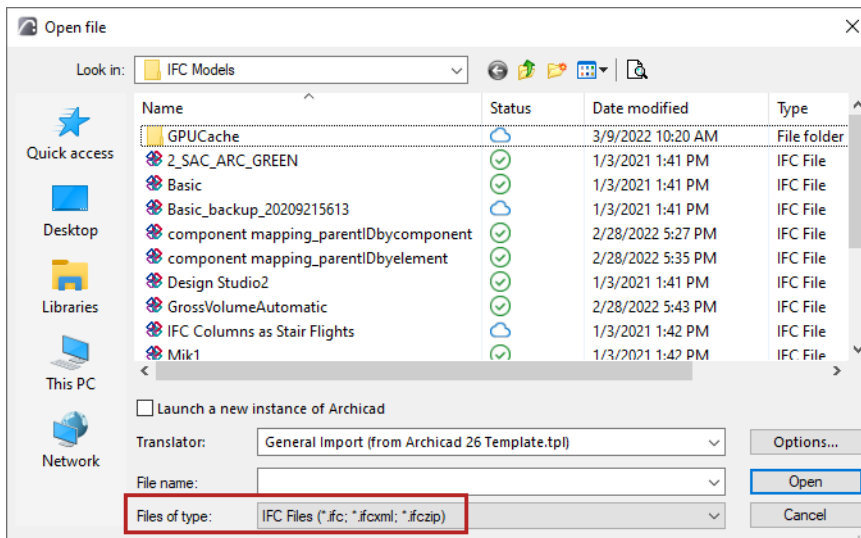
This chapter provides an insight into the IFC standard and how it works in Archicad.

Related Topics

- [IFC File Types Supported by Archicad](#)
- [Archicad Classifications and Categories in IFC](#)
- [Export IFC Model from Archicad](#)
- [Import IFC Model to Archicad](#)
- [Hotlink IFC File](#)
- [Update with IFC Model](#)
- [IFC Project Manager](#)
- [Manage IFC Data at Element Level](#)
- [IFC Local Preferences](#)
- [Define Element Criteria using IFC Data](#)
- [IFC Translators: Overview](#)
- [Translators for Import \(Detailed Settings\)](#)
- [Translators for Export \(Detailed Settings\)](#)
- [Filter Model at Import \(Secondary Filter\)](#)
- [Filter Model at Export](#)
- [Appendix: IFC Terms and Concepts](#)
- [Appendix: IFC Data Mapping in Archicad](#)

IFC File Types Supported by Archicad

Archicad supports the export and import of the following IFC data file formats:



- .ifc: the default IFC exchange format using the STEP physical file structure
- .ifcXML: IFC data file using the XML document structure. This format is recommended for architects whose partners' applications cannot read the original .ifc format, but can manage XML databases (such as budget, energy calculations, etc.). This format delivers the same model information as the plain .ifc format, but the elements and their properties are stored in a more informative structure. An .ifcXML file is normally 300-400% larger than an .ifc file.
- .ifcZIP: IFC data file using the ZIP compression algorithm. It is the compressed version of .ifc or .ifcXML. An .ifcZIP file usually compresses an .ifc down by 60-80% and an .ifcXML file by 90-95%.

Note: .ifcZIP is compatible with e.g. Windows compressed folders, WinZip, etc. So if the recipient cannot read .ifcZIP, then a ZIP extractor application can open the file into .ifc or .ifcXML format.

Archicad Classifications and Categories in IFC

Element Classifications for IFC

Structural Function Category for IFC

Position Category for IFC

Element Classifications for IFC

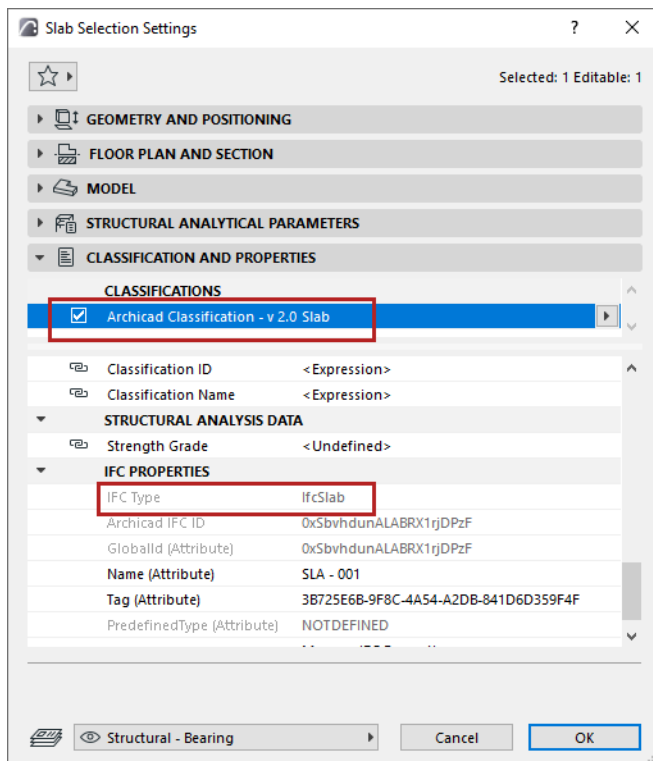
In Archicad, you can optionally assign one or more classifications to any project element. A classification makes parameters and functions available to these elements for use by other applications: to organize project data, to make models transparent, to locate elements, for listing purposes, to share exact data, etc.

For each construction element, classifications can be set in the Classification and Properties panel of its Settings dialog box.

Archicad provides a flexible way to classify elements according to any national or company standard classification system. You can assign one Classification value per Classification System in the Elements' Settings dialog, or from the Interactive Schedule.

Element classification affects the IFC export process: each element is saved to the IFC model according to an IFC Type.

This is displayed in the "IFC Type" item in the Classification and Properties panel (under IFC Properties).



Note: The IFC Type Mapping of Archicad elements by classification depends on the Type Mapping for Export setting of the current Preview Translator. See [IFC Translators: Overview](#).

For more details, see [Properties and Classification Systems](#).

Structural Function Category for IFC

“Load-Bearing Element” or “Non-Load-Bearing Element”

In collaborating with partners who are using structural programs, you can assign the “load-bearing” classification to elements in Archicad and export this information with the IFC file.

Note: By default, Revit Structure will display only walls that have the “load-bearing” property in the “Structural” discipline views. But, if you forget to classify required walls as “Load-Bearing Elements” in Archicad, the structural engineers can also change the bearing function of walls, or they can apply another display discipline (e.g. “Architectural”) in order to show all walls exported from Archicad.

The current IFC standard supports the export of the “load-bearing” IFC property only for the following elements: Wall, Slab, Column, Beam, Roof; elements classified as such in Archicad; and objects having these sub-types.

If you have classified Archicad elements as “load-bearing”, then you can use the Partial Structure Display “Core of Load-Bearing Elements Only” option to display those elements only. This is particularly useful if you wish to export only load-bearing elements to a structural engineer: in this case, just use the “visible elements” export mode on a view that uses this Partial Structure Display option.

[See *Partial Structure Display*.](#)

This category is useful for listing purposes - for example, you can calculate the materials needed for all load-bearing walls or columns, separately from materials of non-load-bearing elements.

Criteria in the **Find & Select** dialog box include: search for those elements that have been defined as load-bearing or non-load-bearing, or whose Structural Function category is “Undefined”. Since IFC export can be limited to selected elements only, you can - for example - export all selected elements defined load-bearing (not just their Core only, as described in the example above).

[See also *Find and Select Elements*.](#)

The load-bearing data is also a standard property of exported elements. Moreover, if other applications (such as a structural program) also categorize elements according to their load-bearing function, Archicad can filter these elements upon importing them, and such elements will show up in the Archicad model with that classification.

Position Category for IFC

“Interior” or “Exterior”

This category is useful if you plan to send an IFC file to partners (e.g. energy analyzers) who are able to differentiate elements by their location in the building. When you export the Archicad model to IFC format, your model elements that are classified as “Interior” or “Exterior” will carry this information to the partner’s application via IFC.

Naturally, the opposite is also true. Since Position (IsExternal) is a standard IFC property for certain building elements (e.g. Wall, Slab, Column, or Beam), if an imported IFC model contains such data, then the appropriate category is automatically added to the respective elements.

Criteria in the **Find & Select** dialog box include: search for elements categorized as either “Exterior” or “Interior”, or whose position is “Undefined”.

[See also *Find and Select Elements*.](#)

Interactive Schedule Criteria are also available to differentiate listed elements by Position.

Note: Within Archicad, the Position category is fully usable as an Archicad property (e.g. in **Find & Select** and **Interactive Schedule**). However, it will not be exported as an IFC property for certain element types (e.g. Ceiling (IfcCovering)) which are not supported by the IFC standard.

Export IFC Model from Archicad

There are three export functions:

- **Save as (File menu):** exports the entire current Archicad project or just its filtered content to an entirely new IFC file.
- **Merge to IFC Model (File > Interoperability > IFC):** merges only certain elements or, alternatively, the entire content of the current Archicad project into an existing (unopened) IFC file. For example, use this command if you want to add selected content from an Archicad model to an IFC file from another discipline.
- **Publisher (Document menu):** publishes views to any of the IFC file formats. For each Publisher item, you can vary the Translator, so that the view content can be customized to the needs of the recipient, creating the desired output with a single click.

[See IFC Format.](#)

Regardless of the method, the export process is controlled by an IFC Translator for Export, which defines how Archicad elements should be interpreted and displayed in IFC. As part of the export process (see steps below), you will choose an IFC Translator for Export.

Save as IFC/Merge to IFC Model

Both of these export functions (Save as and Merge) follow essentially the same steps.

1. Choose the IFC export command you need.

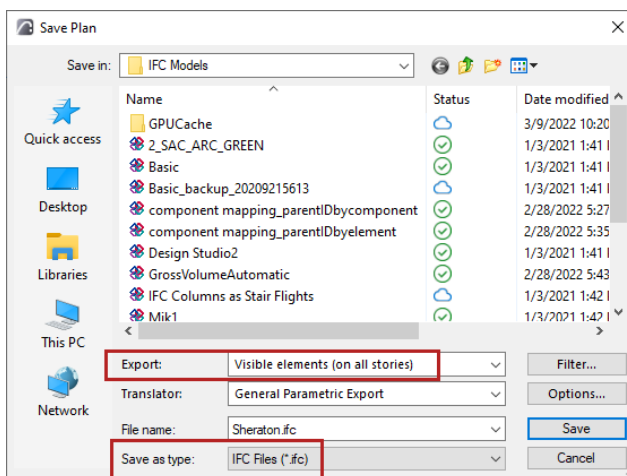
File > Save as...

File > Interoperability > IFC > Merge to IFC Model

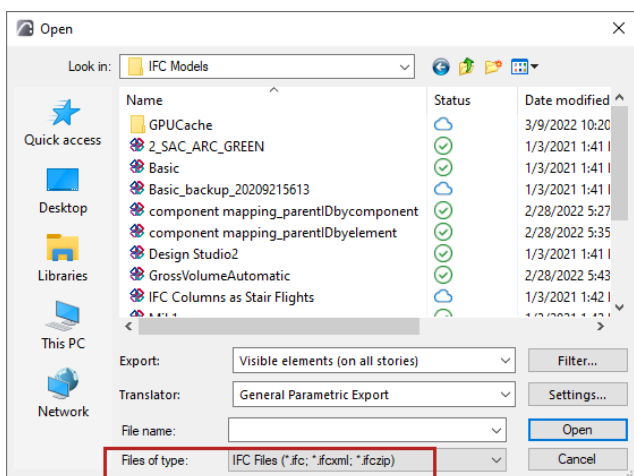
2. File type

Save as IFC: From the “Save as type” choose any of the IFC file formats.

Merge to IFC Model: Only the IFC file types are available.



Save as IFC



Merge to IFC Model

[See IFC File Types Supported by Archicad.](#)

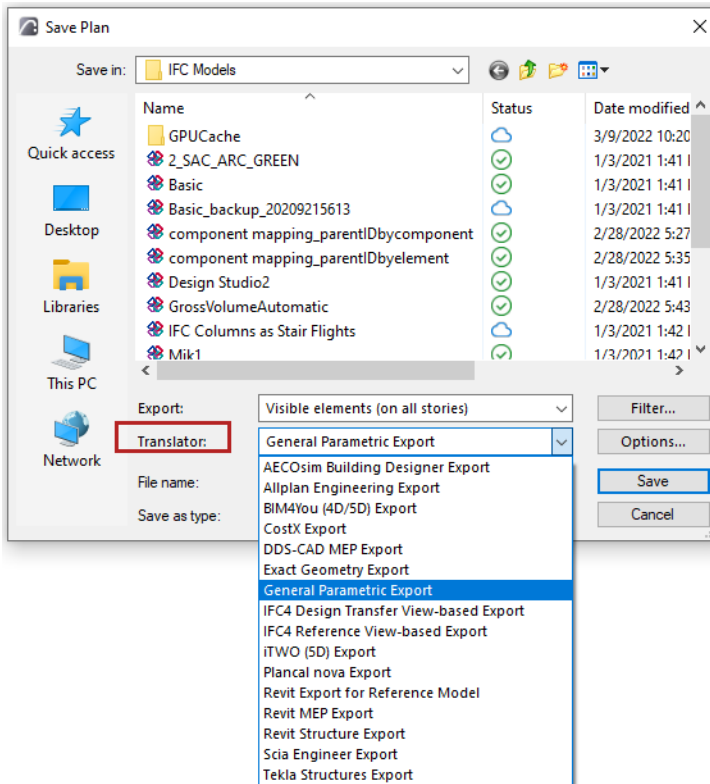
3. Define File Name

Use the “File name” field to define a file name (Save as), or browse for an existing IFC model (Merge to IFC Model).

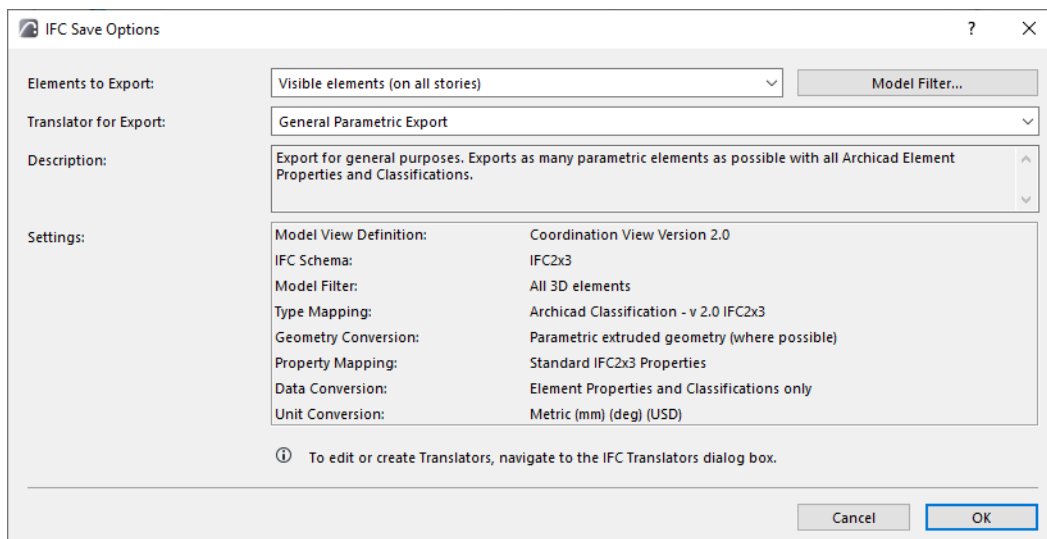
4. Choose a Translator for Export

Use the drop-down list to choose among the predefined or user-defined translators.

[See also Translators for Export \(Detailed Settings\).](#)



Click **Options** to open **IFC Save Options**, with a summary of the selected Translator settings.



5. Filter Model at Export

As needed, further filter the set of elements to export, using the **Elements to Export** dropdown and/or the Model Filter dialog box, both available from IFC Save Options.

See [Filter Model at Export](#).

6. Export

Click “Save” (Save as) or “Open” (Merge to IFC Model) to start the export process.

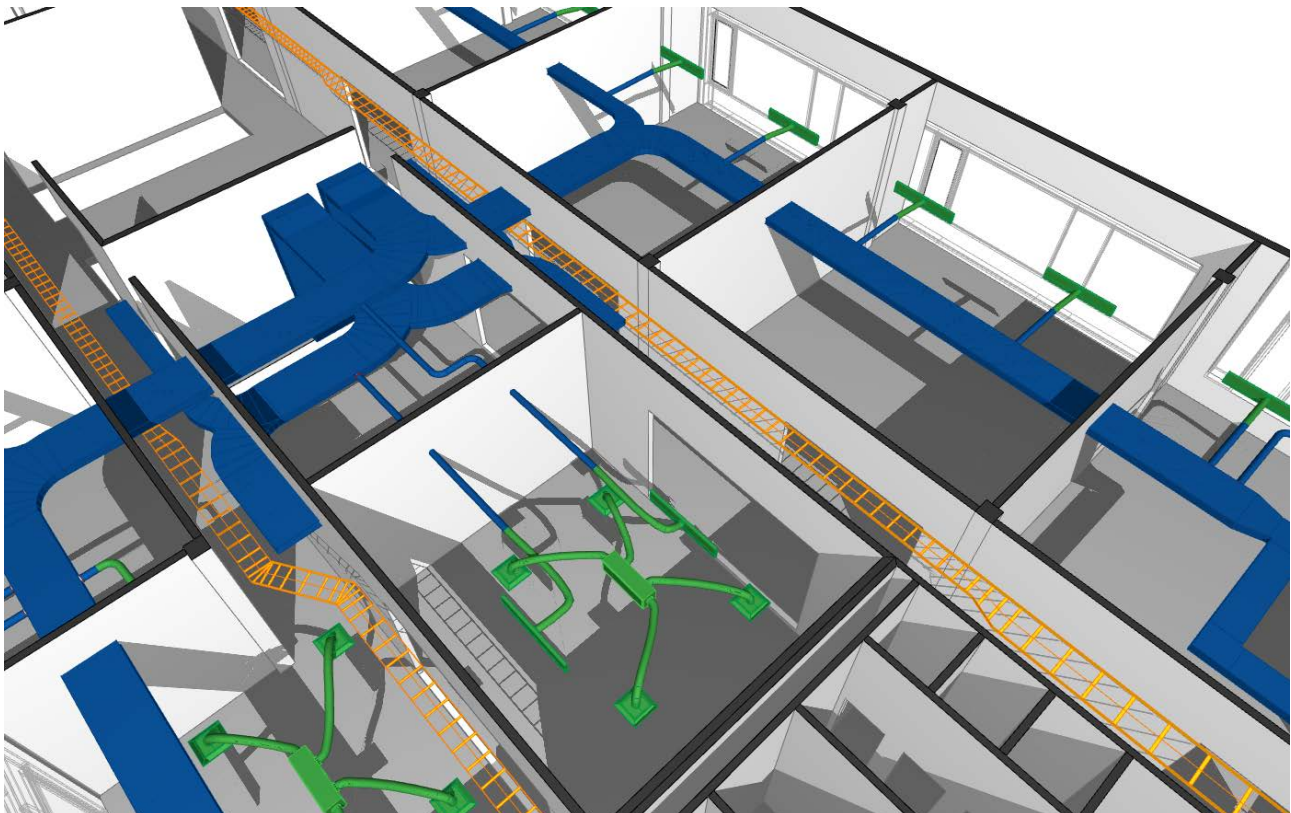
Hint: After exporting an IFC model, it is good practice to check the IFC model in an IFC viewer (or in Archicad by reopening the IFC file). There are many free IFC viewers on the market that can be readily downloaded, including:

- DDS-CAD Viewer: <http://www.dds-cad.net>
- Solibri Model Viewer: <http://www.solibri.com>
- Tekla BIMsight: <http://www.teklabimsight.com>

Merge to IFC Model: Protected Content

In case of Merge to IFC Model, the content of the existing IFC file is protected: merging a file to IFC will only add to it, without replacing any of its elements. This means that if a merged element has the same IFC GlobalId Attribute as an existing element in the IFC file, then the element will be merged using a new IFC GlobalId Attribute. Elements will never be overwritten as a result of a merge. This also means that the Merge to IFC Model command is not suitable for merging changes from one version of an IFC file to another version of the same file!

This illustration shows the result of merging an Archicad model into an existing file that contains an MEP model.



Import IFC Model to Archicad

Archicad provides three methods for importing an IFC model to Archicad: Open, Merge, and Hotlink.

Open versus Merge

- **Open:** Opens the entire model or filtered parts of an IFC file as a new project. Archicad will transform the imported elements into corresponding Archicad elements, based on settings in the chosen IFC Translator for Import.
- **Merge** (File > Interoperability > Merge): Available only in Floor Plan view. Inserts an IFC model in whole or in part into a running Archicad project. Merge preserves your running project, without overwriting its data (this is the “Reference model” concept). However, even in this “Reference model”, the merged elements are real Archicad elements based on the IFC-Archicad mapping convention.

In the Merge process, Archicad automatically generates new IFC GlobalId (Archicad IFC ID) values for all imported elements to avoid overwriting the IFC GlobalIDs of the host project. For the purposes of listing and searching, the elements’ original GlobalId values are also retained, in the form of a parameter called “External IFC ID”.

Overview of IFC Import: Defining how IFC Data are Converted to Archicad

When you open (or merge or hotlink) an IFC file in Archicad, the program carries out the following steps for each imported IFC element, using the settings in the chosen IFC **Translator for Import**:

1. Does the IFC element need to be imported to Archicad? (based on the Translator’s “Model Filter for IFC Import” preset)
2. Which Archicad tool (e.g. Wall, Object) should be used to place the IFC element in Archicad? (This is based on the Translator’s “Geometry Conversion for IFC Import” preset.)
3. Which Archicad layer should the element be placed on, and using which attributes? (This is based on the Translator’s “Material and Surface Conversion for IFC Import” preset.)
4. Which Archicad Classification should be assigned to the element? (This is based on the Translator’s “Type Mapping for IFC Import” preset.)
5. Based on the Classification: which Properties are available for the element? (This uses the Property Availability definitions, defined in the Classification Manager of the template file used at import.)
6. Each of these Element Properties is populated with its default Property value. (This uses Property definitions of the Property Manager, in the template file used at import.)
7. The program examines the IFC data assigned to the incoming element, and attempts to generate the equivalent property value for the element in Archicad (This is based on the Translator’s “Property Mapping for IFC Import” preset.)

Important Note on Translators and Template Files

- When you import an IFC project using **Open** or **Hotlink IFC**: You will choose an IFC Translator for Import from an *external* Template file, selected by you.
- When you import an IFC project using the **Merge** command: You will choose an IFC Translator for Import from the current host project.

The following sections describe these processes.

[Open IFC Model](#)

[Merge IFC Project](#)

[Hotlink IFC File](#)

Related Topics:

[IFC Translators: Overview](#)

[Translators for Import \(Detailed Settings\)](#)

[Filter Model at Import \(Secondary Filter\)](#)

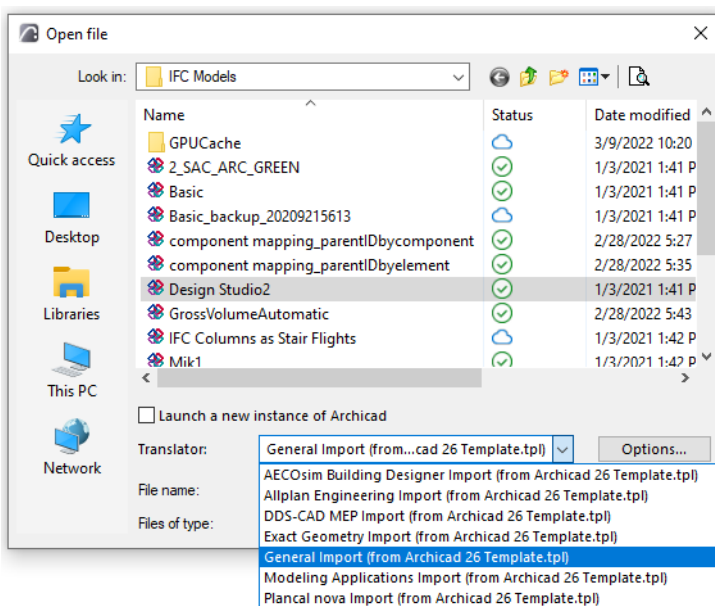
Open IFC Model

Follow these steps to import IFC data using the Open process.

1. Use **File > Open**.
2. Select the IFC File Format.
3. Choose the IFC file to import.
4. Use the drop-down list to choose among the IFC Translators.

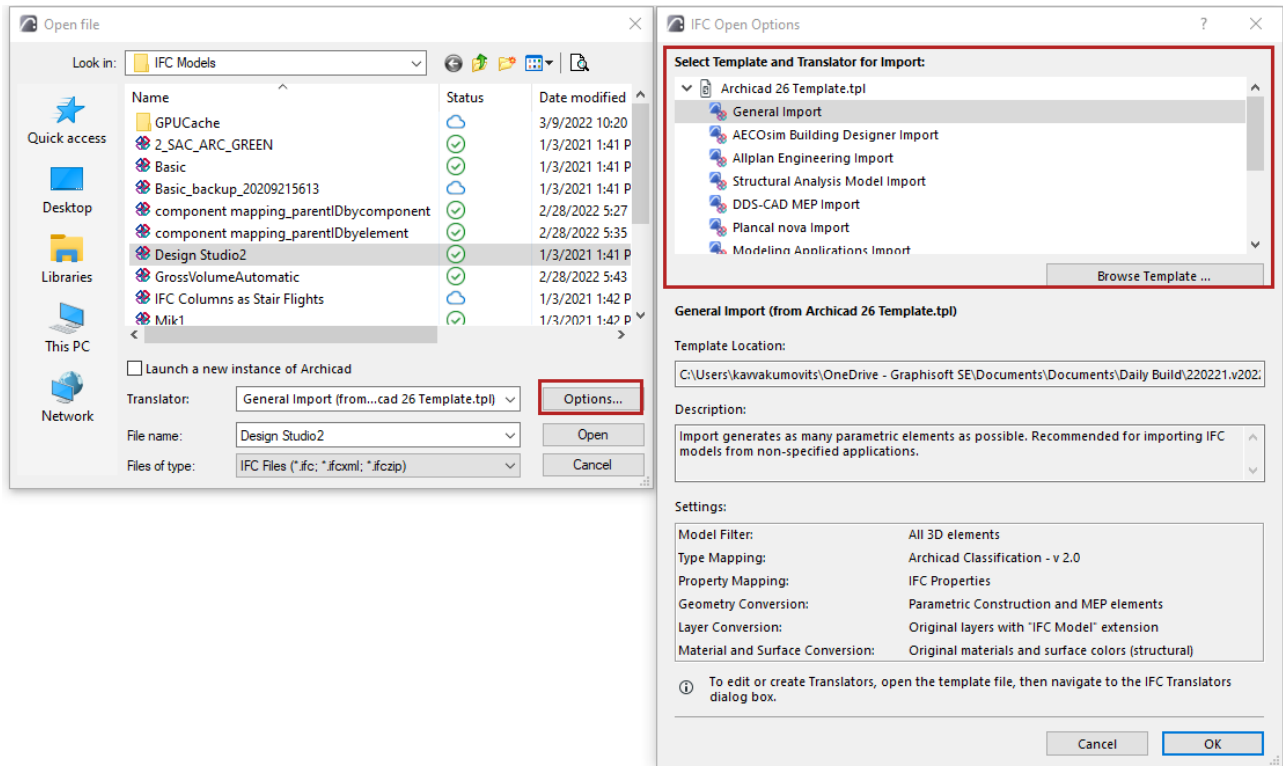
Note: In the Open IFC process (just like the Place IFC Hotlink process), the file is imported based on an IFC Translator for Import, *from a Template file* selected by you.

For each listed Translator, its parent Template is shown in parentheses. If the Translator you need is not shown, you may need to search under a different template (see next step).



5. From the Open dialog, click **Options** to see **IFC Open Options**.

Here, each loaded template is shown together with its Translators, location, and presets. Choose a Translator from one of the templates and click OK, or click **Browse Template** to load another one.



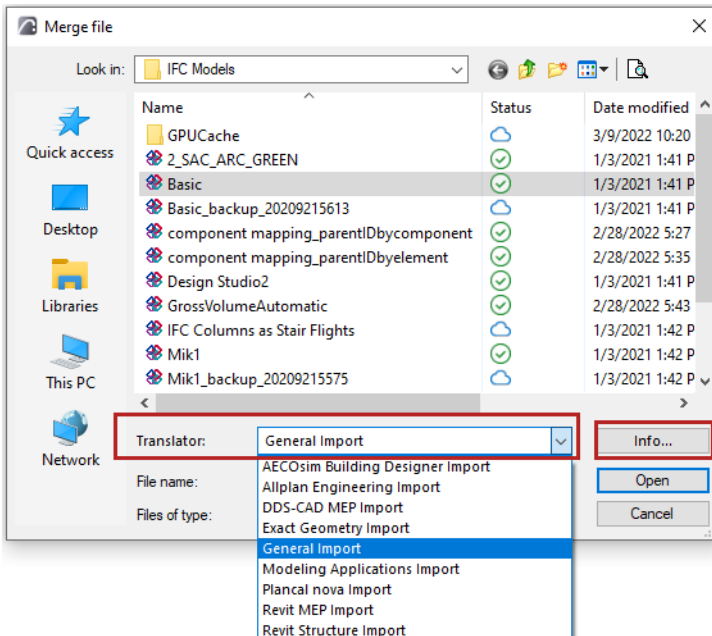
6. Click “Open” to import the IFC model based on the selected Translator. Additional steps follow, depending on the type of imported model.

[See Additional Options at IFC Import.](#)

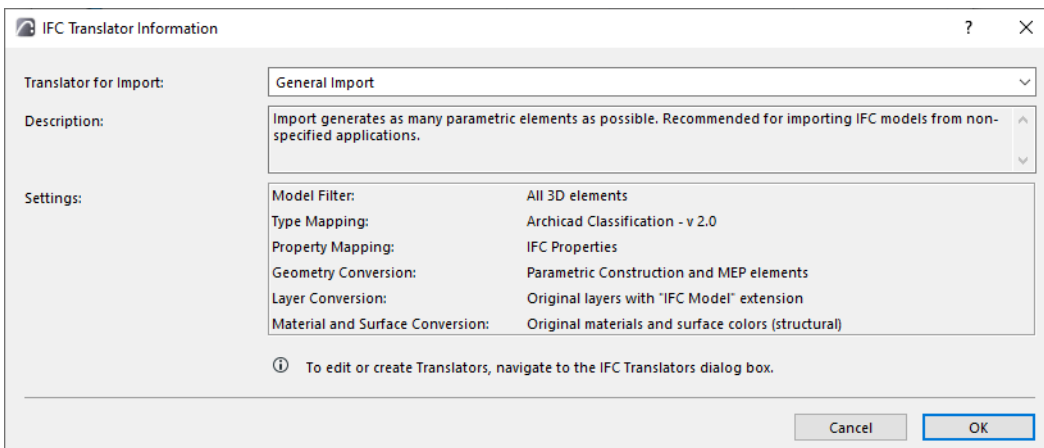
Merge IFC Project

1. Use **File > Interoperability > Merge**.
2. Select the IFC file format.
3. Choose the IFC file to merge.
4. Choose a Translator

Use the drop-down list to choose among the IFC Translators. These are the IFC Translators in the current host project.



5. Choose **Info** from the Merge dialog to see a summary of the selected Translator settings.



Note: In the Merge process, Archicad uses an IFC Translator for Import that you choose *from the current host project*.

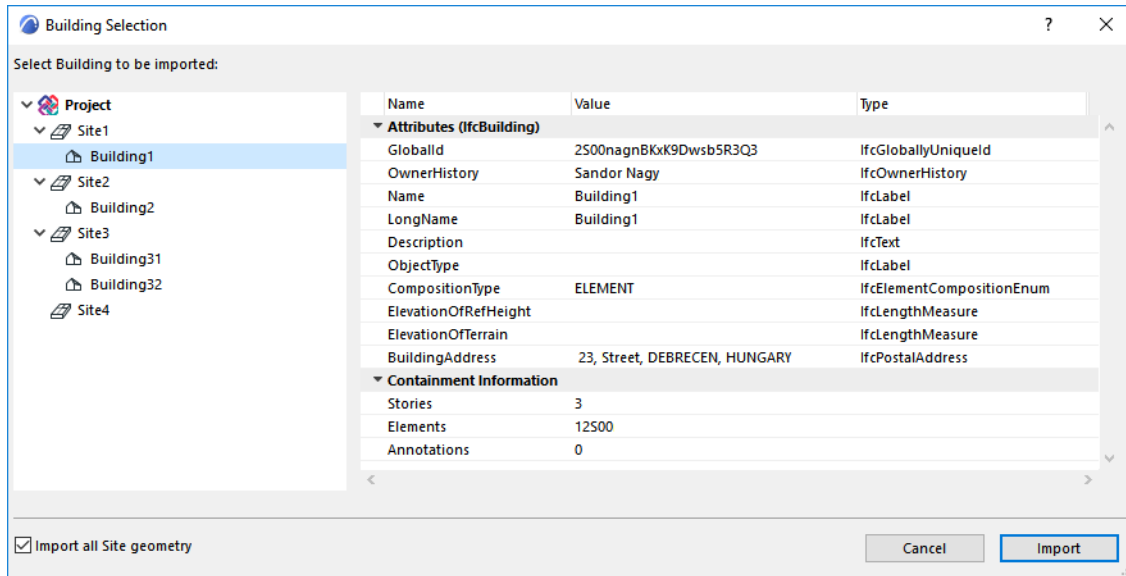
6. Click **Open** to merge the IFC Project to Archicad.
7. Additional steps follow, depending on the type of imported model.

[See next section.](#)

Additional Options at IFC Import

Building and Site Selection

If the IFC file contains multiple IfcBuildings and/or IfcSites, then a Building Selection dialog appears, so you can select one Building. The import process can import only one building at a time.



On the left of this selection dialog, view the site and building hierarchy in the tree structure. At right, information is shown on each selected building.

If the “Import all Site geometry” checkbox (at the bottom left of the dialog box) is not enabled, then the selected IfcBuilding will be imported with only the geometry of the specific IfcSite to which the building is assigned.

To import the entire site model, regardless of the chosen building, activate the “Import all Site geometry” option.

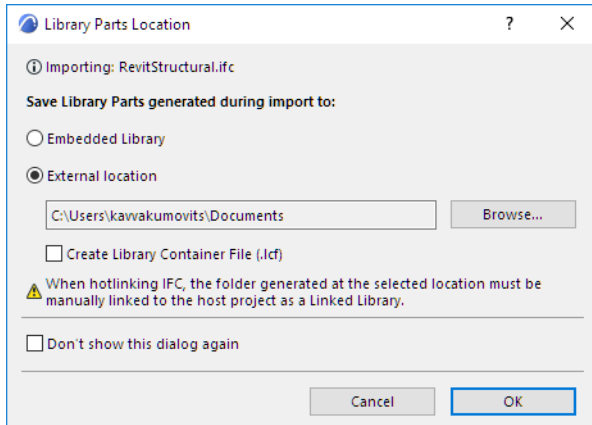
Filter Model at Import

As needed, further filter the set of elements to import, using the Model Filter dialog box.

See [Filter Model at Import \(Secondary Filter\)](#).

Library Parts Location

Choose where to save Library Parts generated during IFC import:

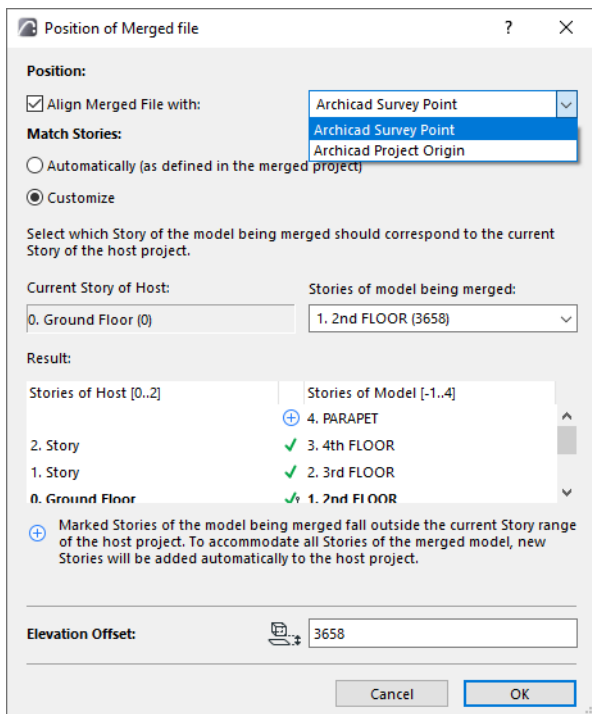


The External location is recommended. Make sure you manually link this folder to the project as a Linked Library.

Notes:

- When updating a Hotlinked Module, the Linked Library will not be updated! If Linked Library contents have changed, you must update the library manually.
- Make sure that the target folder is write-enabled at the time of import. If it is not, the import process will fail.
- In Teamwork, the Linked Library option is not available; generated Library Parts are saved to the Embedded Library.

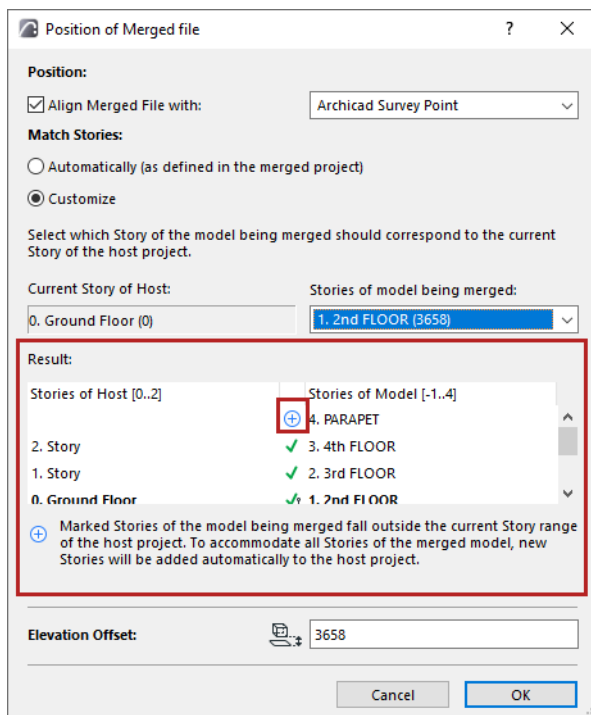
Position of Merged file



- Align merged file with either the Project Origin or Survey Point of the Archicad model.
 - Match Stories:
 - Automatically: Stories are placed as they are in the incoming merged file.
 - Customize: Choose which story of the incoming IFC project which should be placed onto the current story in Archicad. In the list, each story of the IFC model is shown with its own elevation in parentheses.
- Note:** In either case (**Automatic** or **Custom** stories), new stories will be created in the host project if needed, to accommodate extra stories in the merged project.
- **Elevation Offset:** To adjust the vertical position of the imported model, enter a positive or negative value in the Elevation Offset at the bottom of the dialog. By default (and this is recommended), the Elevation offset value is the absolute difference between the elevation values of the chosen IFC model story and the current Archicad story.

Result

- The Result shows which merged stories will be placed on which stories of the host.
- New stories (if any) are marked with a plus sign. These stories will be created in the Host when you click OK to close the dialog box.



Hotlink IFC File

Place IFC files as hotlinks directly into the currently open Project as protected reference content. You can place this hotlink as many times as needed. Thus, in a project using repeated building blocks/parts, you do not need to request a huge IFC file based on the entire project - a smaller IFC file that contains just the module is enough.

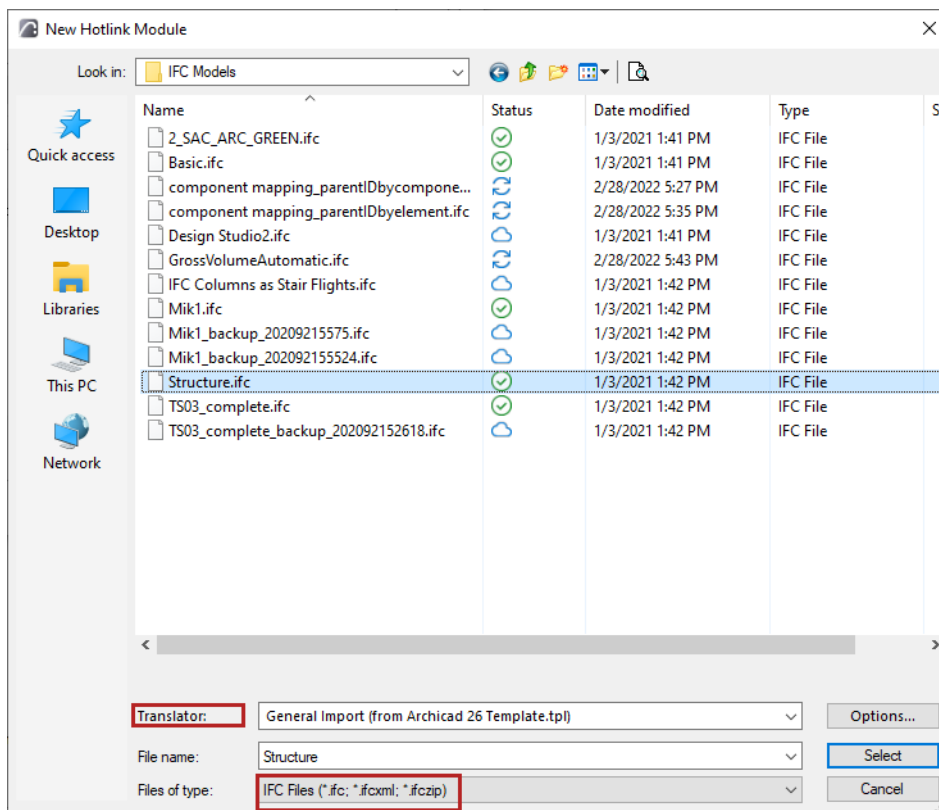
Place IFC Hotlink

The steps for placing an IFC hotlink are the same as when placing a PLN-based hotlink, with some additional options described here.

1. As when placing any other Hotlink, use **File > External Content > Place Hotlink**.

[See Place Hotlink for an overview of this process.](#)

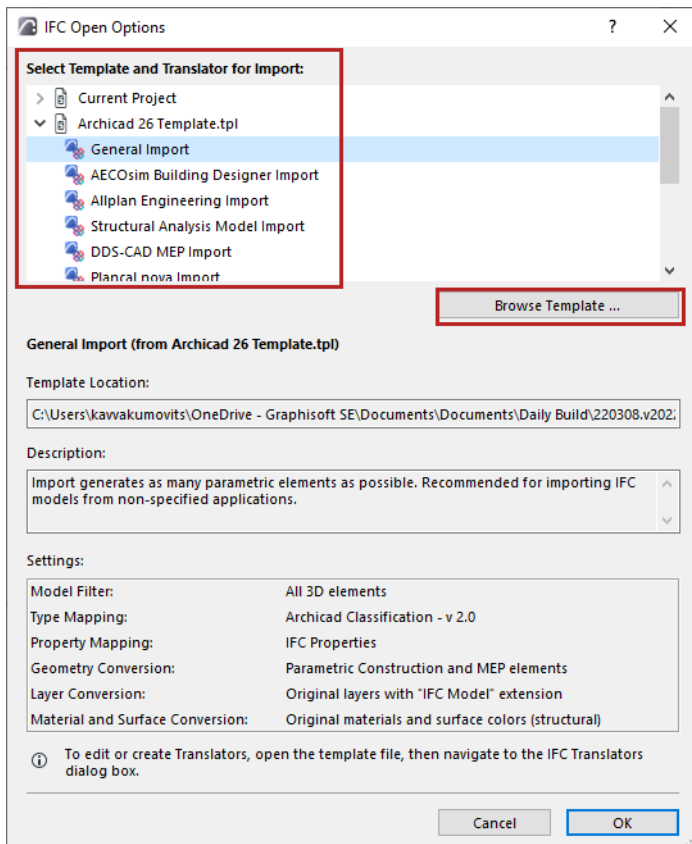
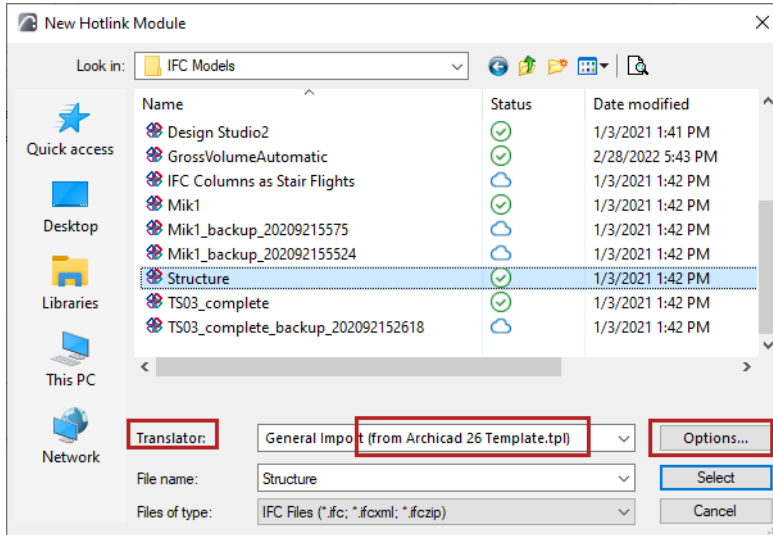
2. From the New Hotlink Module dialog, choose an IFC format file and a Translator for Import.



Note: In the Place IFC Hotlink process (just like Open IFC), the hotlink elements are imported based on an IFC Translator for Import, *from a Template file selected by you*.

For each listed Translator, its parent Template is shown in parentheses. If the Translator you need is not shown, you may need to search under a different template (see next step).

3. From the Open dialog, click **Options** to see **IFC Open Options**.



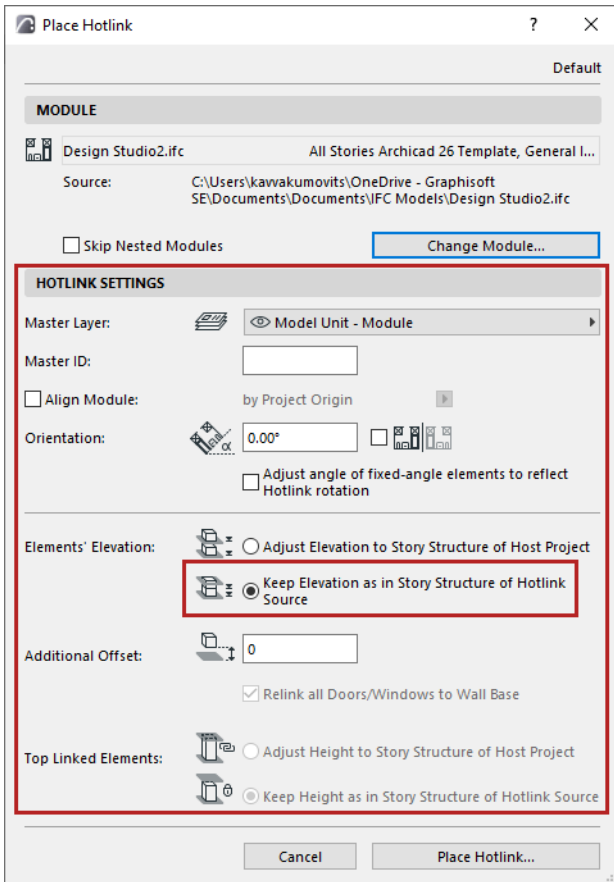
Here, each loaded template is shown together with its Translators, location and presets. Choose a Translator from one of the templates and click OK, or click **Browse Template** to load another one.

See [IFC Translators: Overview](#) for more information.

4. Click **Select** to return to the **Place Hotlink** dialog box.

For details, see [Hotlink Settings](#).

Note: When hotlinking an IFC file, make sure you set the Elements' Elevation option to Keep Elevation! Otherwise, the imported model may "fall apart" when its elements are placed in a different story structure



5. Click **Place Hotlink**.

6. You are prompted to choose where to store the library parts.

See [Library Parts Location](#).

7. If the IFC Translator for Import requires it, the **Model Filter for Import** also appears, to allow for further element filtering before placing the hotlink.

See [Filter Model at Import \(Secondary Filter\)](#).

Layers of IFC Hotlink Elements

When placing an IFC file as a Hotlink, remember that the imported elements' own Layers depend on the Layer Conversion settings of the Translator for Import that you are using.

See [Layer Conversion for IFC Import](#).

Avoid Model Element Intersections with Hotlinked IFC Elements

Like other IFC content imported to Archicad, IFC-based Hotlink content depends on the settings of the IFC Translator used for import.

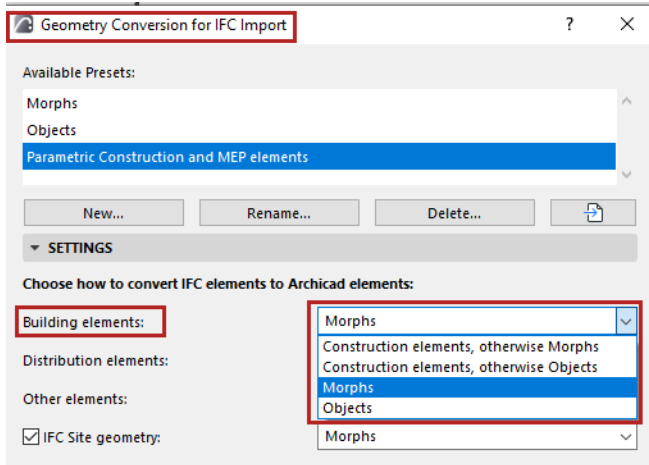
If you are importing elements as native AC construction elements, unwanted intersections can be a problem.

To avoid intersections, do one of the following:

- Make sure the imported IFC elements are placed onto a separate Archicad layer. (Check the Layer Conversion setting of the IFC Translator used for import.) (See [Layer Conversion for IFC Import](#).)

Note: Archicad will automatically apply a unique Layer Intersection Group setting to this new Layer. Elements whose layers have differing intersection numbers never intersect. (See [Use layers to prevent intersections.](#))

- Use the Translator's Geometry Conversion settings to create Objects or Morphs out of the imported elements. Such elements do not participate in automatic element intersections.



[Watch the Video](#)

Update with IFC Model

This command (**File > Interoperability > IFC > Update with IFC Model**) updates the current Archicad project content with an IFC model.

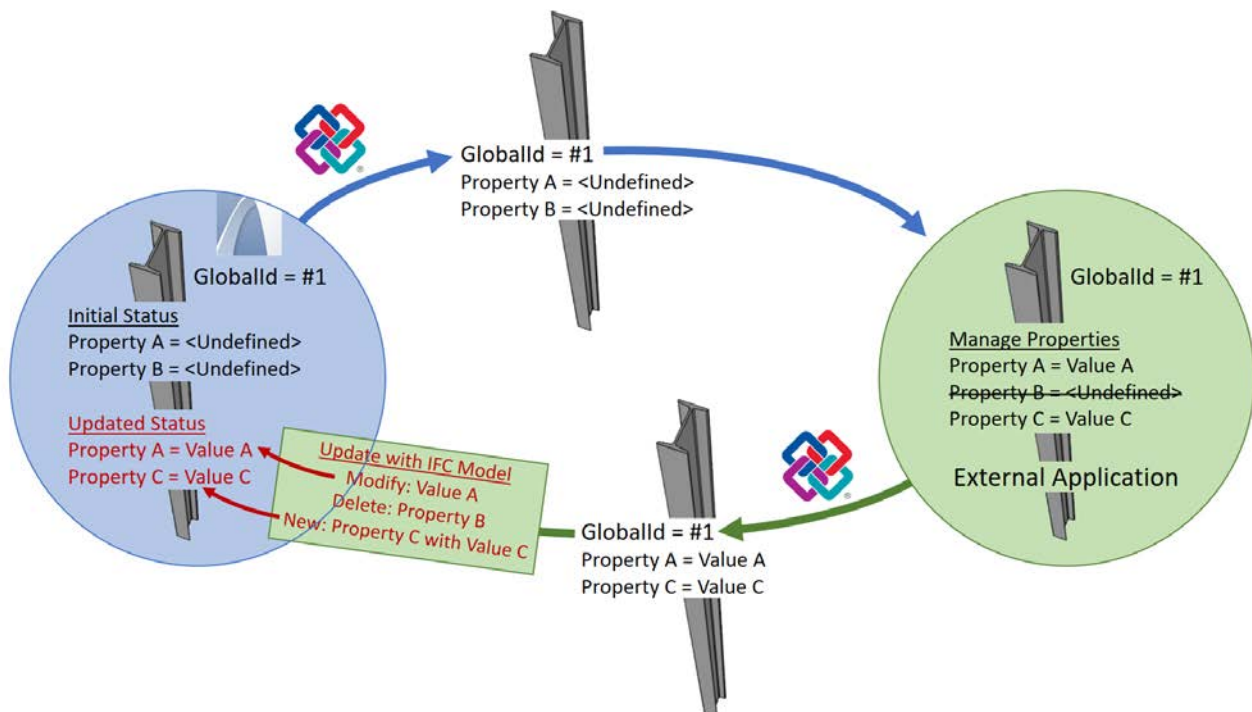
Use this if the Archicad project and the IFC file contain elements in common, and if you want to add to or modify the project elements' data using IFC data.

- Exported Archicad properties that are mapped to IFC properties are updated, provided that the correct Translator settings are used at both export and import. (Archicad 22 and later)
- “Elements in common” are those whose GlobalId Attribute is the same in both the host project and the imported IFC model.
- “IFC data” include IFC Attributes, IFC Properties and IFC Classification References

Thus, this command is optimized for tracking and updating changes to property data. This command is *not* suitable for updating model element changes.

Notes:

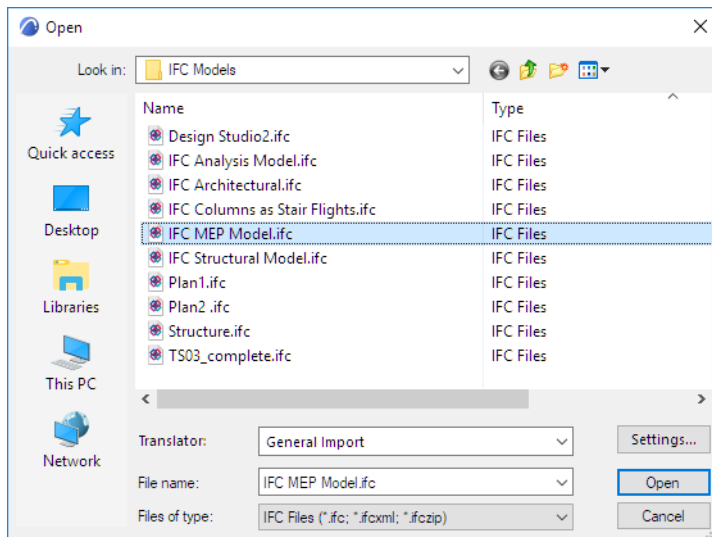
- Property update using this command works properly only if Property and Type Mapping definitions are used correctly and consistently, both at export and at import
- Deleted elements and modified elements are ignored!
- Geometry data (size, color, layer) will not be updated or modified.
- Properties of locked project elements will not be updated or modified.



Workflow for Update

1. Go to **File > Interoperability > IFC > Update with IFC Model**.

In the Open dialog:



- Browse for an IFC model.
 - Choose an appropriate Translator for Import. The Translator should use Property and Type Mapping settings that are analogous to those used when the original Archicad model was exported to IFC.
2. Click Open.
 3. In the appearing dialog, set the update options as needed (see [Update Options](#), below).
 4. Click **OK**.

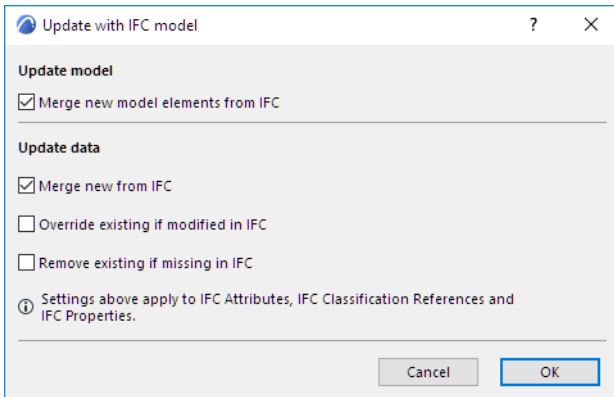
Example Workflow: Property Mapping at Export and Import

1. The Archicad model is exported to IFC, using the **File > Save As IFC** command.
Archicad elements are exported together with their IFC data. Based on the **Translator for Export - Property Mapping for Export** preset (which also depends on the Type Mapping for Export preset), Archicad properties are mapped to IFC properties.
 2. An external consultant works on the IFC model, deleting some properties, modifying others, and adding some new ones.
 3. The edited model is imported back to Archicad using **File > Interoperability > IFC > Update with IFC Model**.
 4. New and modified IFC properties are added and updated in the Archicad model, based on the mapping definitions of the **Translator for Import - Property Mapping for Import** preset.
- **Important:** To map the properties back to existing Archicad properties, make sure you activate the option to **Import as Archicad Properties**.

Update Options

Check the options to specify which data to update in the Archicad model.

Note: These update data options are only relevant for incoming IFC Attributes and IFC Classification References. *Properties* are updated based on Translator-based property mapping definitions at export and import.

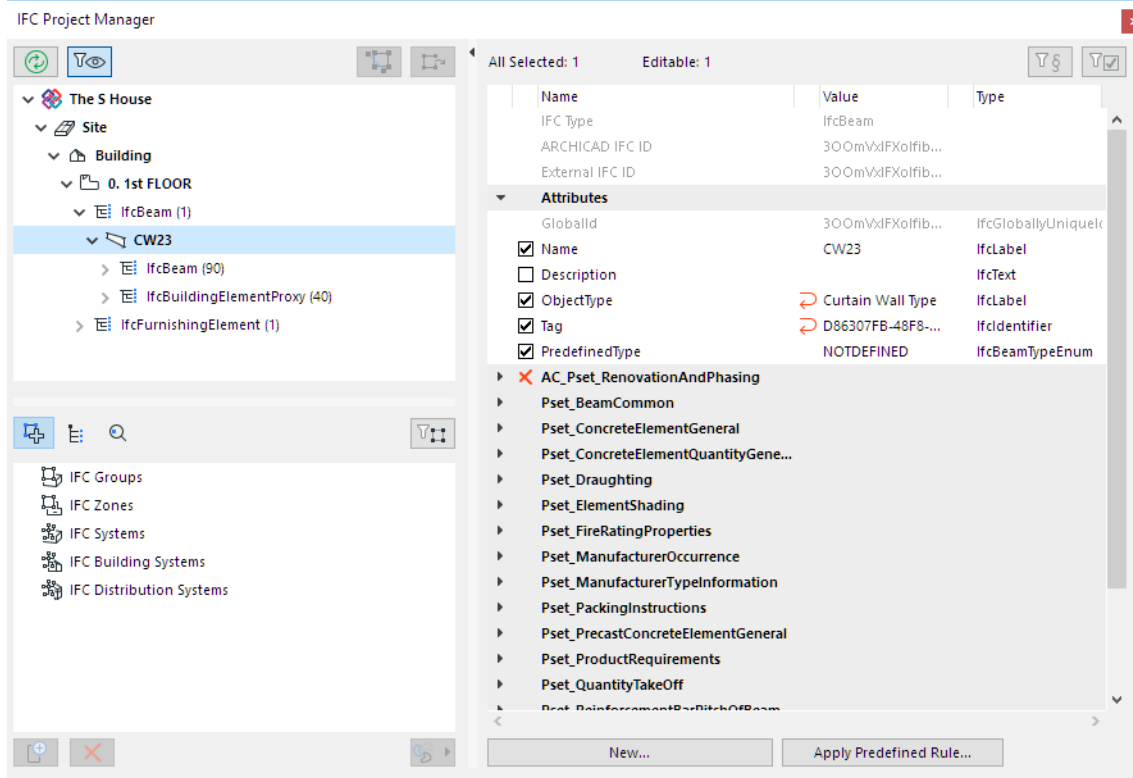


- **Merge new model elements from IFC**
- **Merge new from IFC:** IFC Attributes and IFC Classification References which do not yet exist in the host project will be added to elements in common.
- **Override existing if modified in IFC:** For elements in common, the project data will be modified by the corresponding data values from the merged IFC file. Corresponding data: Attribute and Classification Reference must have the same name.
Note: This is a full override: all values for data items having the same name will be modified.
- **Remove existing if missing in IFC:** For elements in common, project data which do not exist in the merged IFC file will be removed.

IFC Project Manager

The IFC Project Manager (**File > Interoperability > IFC**) provides a hierarchical overview of the *current project's* IFC model database. Each model element is listed with its IFC attributes and properties.

Most of these are inherited from import/export IFC standards set elsewhere (e.g. Type Mapping and Property Mapping presets of the project's IFC Preview Translator.) However, you can edit values in IFC Project Manager for any specific element, and/or create custom IFC properties or Property sets as needed.



Related Topics:

[Functions of the IFC Project Manager](#)

[Navigating in the IFC Project Manager: Tree Structure](#)

[Navigating in IFC Project Manager: Attributes and Properties](#)

[Assign or Edit IFC Property/Attribute Values](#)

[Create New, Custom IFC Property](#)

[Apply Predefined Rules](#)

[Using Assignments in IFC Project Manager](#)

[Using Type Products in IFC Project Manager](#)

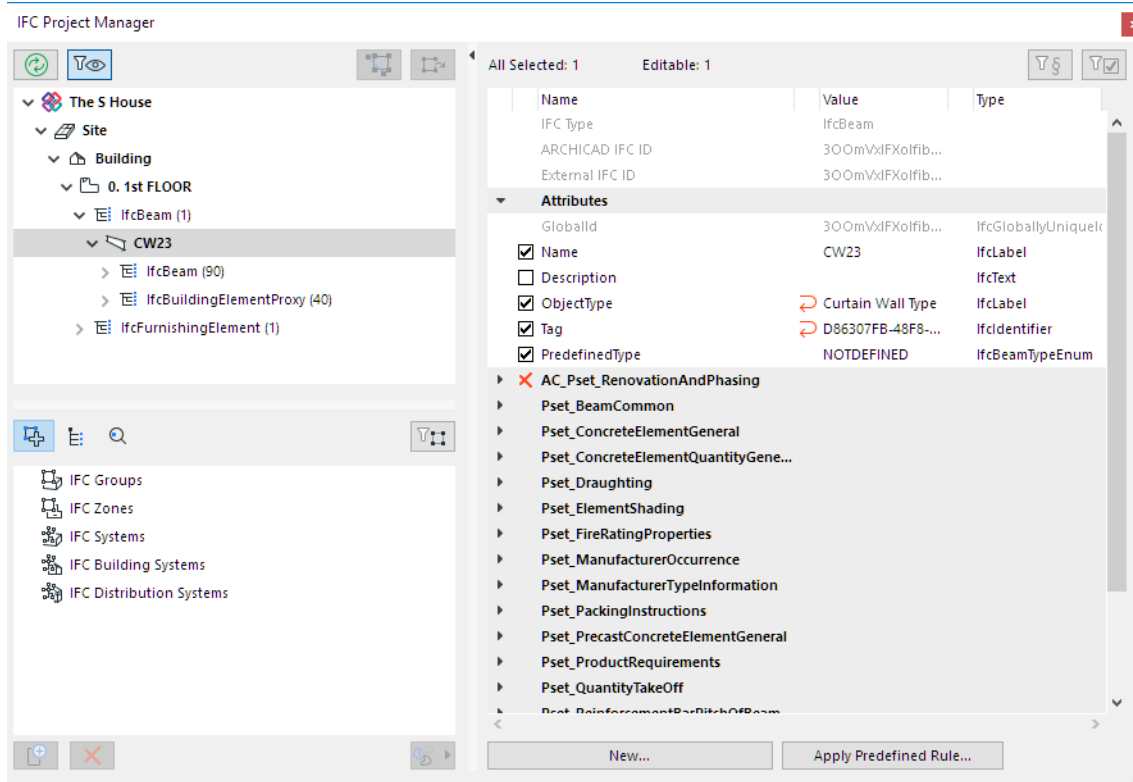
Functions of the IFC Project Manager

- View all IFC model entities generated from the Archicad project (including IFC import content) in a hierarchy
- View all IFC data available for project elements
- Create new IFC data (custom IFC Properties and Classification Reference) for any model element
- Group elements or assign entities to elements (called IFC Assignments) and manage their IFC data
- Manage IFC Type Products assigned to elements with their with their IFC data
- Assign elements directly to the IfcSite or IfcBuilding. For example, the neighboring buildings and site context elements (trees, fences, roads, vehicles etc.) can be assigned to the IfcSite; thus, their default direct relation to their home story is removed.
- Update the IFC model with the modifications of the Archicad project
- Navigate easily between the Archicad model and the IFC model
- Filter the IFC model by elements visible in the current view
- Find elements both in the IFC and Archicad models by their IFC ID's
- In Teamwork, reserve the IFC Project Manager to modify data for the following elements: IfcProject, IfcBuilding, IfcSite, Assignments, and Type Products. To modify data for model elements (e.g. IfcWall, IfcColumn, IfcBeam) in IFC Project Manager, you must also reserve the elements themselves.

The IFC Project Manager palette is a modeless dialog box, so you can keep it open while working in the Archicad project.

Navigating in the IFC Project Manager: Tree Structure

The tree structure (upper left side of the IFC Project Manager) displays the IFC model hierarchy and lists IFC Entities of the current project according to their “IFC Type” classification.

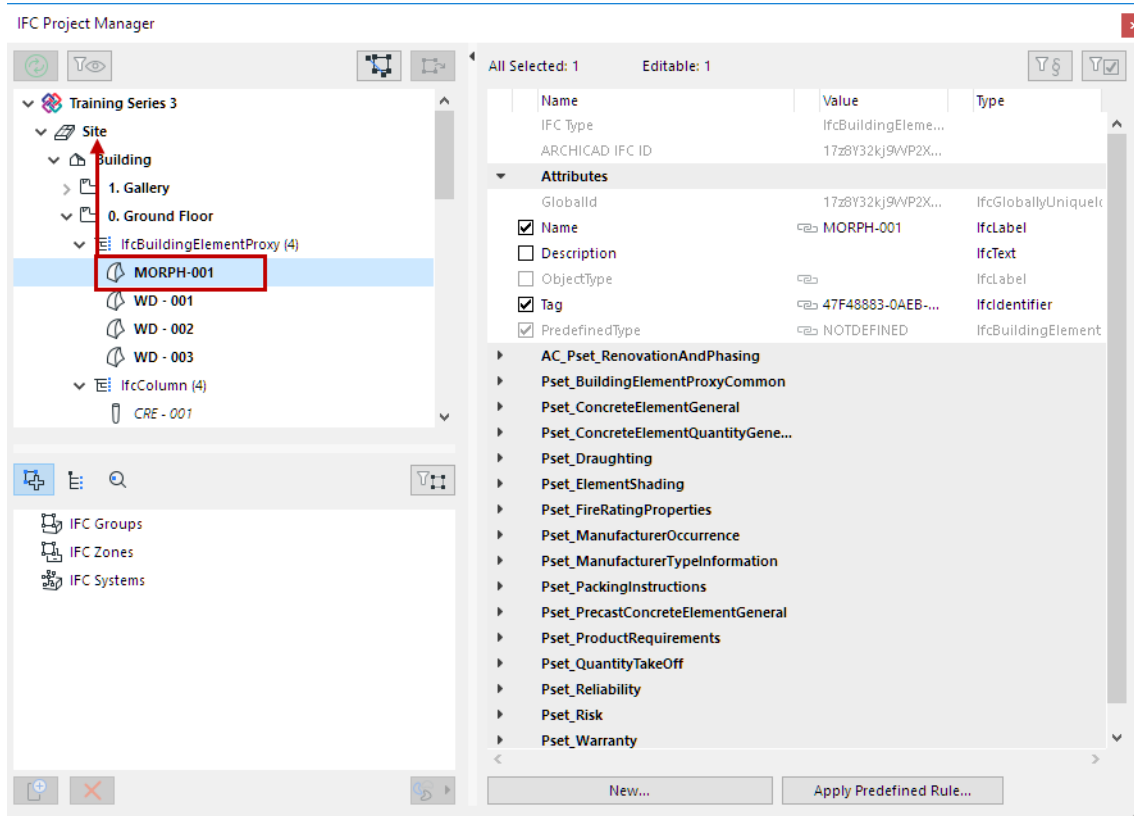


See [IFC Model Hierarchy](#).

Archicad Elements in IFC Story Structure

In Archicad, all elements and objects are linked to their Home Story. Therefore, by default, elements are listed in the IFC model hierarchy under the IfcBuildingStorey that corresponds to their Home Story.

You can break this link (to the IfcBuildingStorey) by dragging the element out of its IfcBuildingStorey location into the IfcSite. For example, a neighboring building modeled with a Morph can be placed into the Site folder.

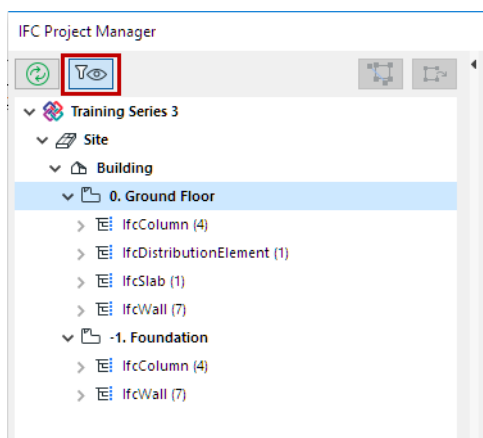


You can also place elements into the Building (such as an element that is logically tied directly to a particular building rather than a story).

To return an element to its automatic IfcBuildingStorey classification (corresponding to its Home Story), just drag it out of the IFC Project Manager.

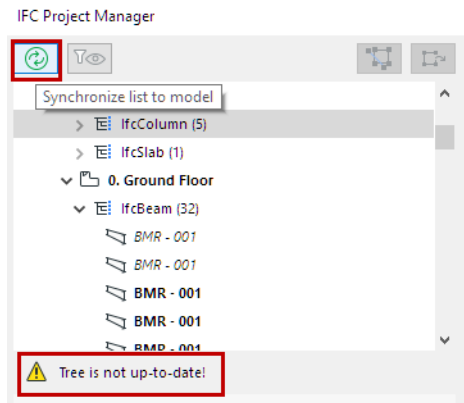
Filter Data Shown in Tree

Click **List visible model elements only** to narrow the tree to those elements that are visible in the current Archicad model view.

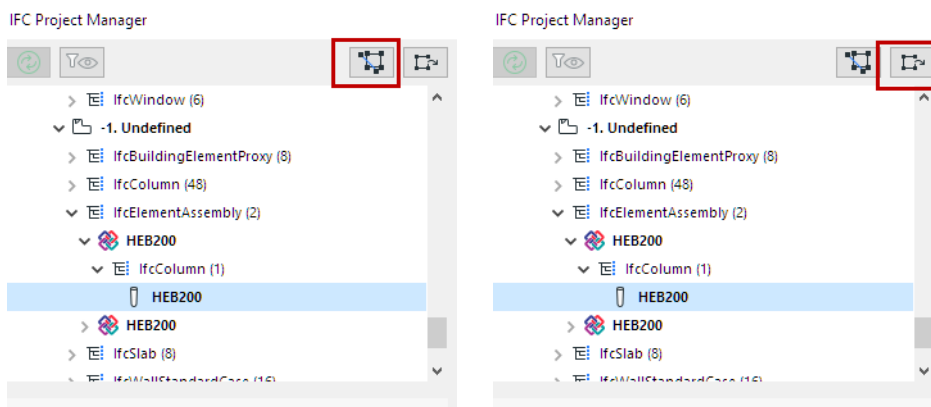


Synchronize Project Manager Data with Model

When elements are created or deleted in the Archicad model, the tree will not register these changes automatically. If a refresh is needed, a message appears below (“Tree is not up-to-date!”). Click the “Synchronize” button to refresh it.



Find Selected Model Elements in Tree; Find Selected Tree Elements in Model



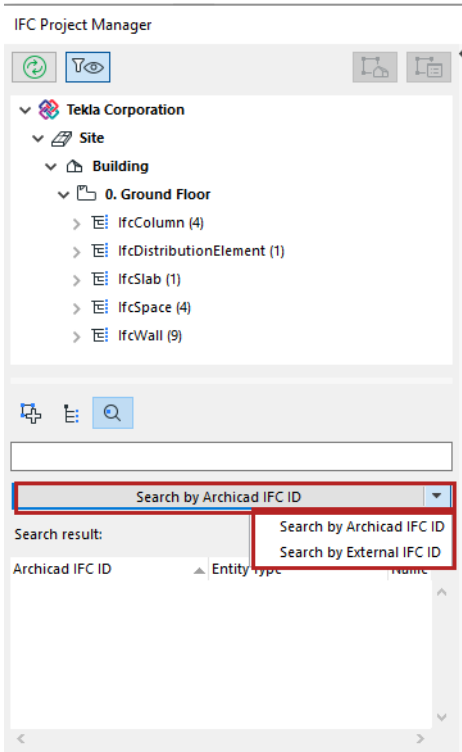
- To find and select IFC Entities of the tree in the Archicad model, select them in the tree and click **Show list selection in model**. The program displays and selects the elements in the active window (Floor Plan or 3D view only).
- To find and select Archicad model elements in the tree, select them in the Floor Plan or 3D window, and click **Show model selection in list**.

Find Tree Elements by IFC Identifiers

An IFC model checker might alert you to modeling errors. You can locate such elements in IFC Project Manager based on their IFC identifiers:

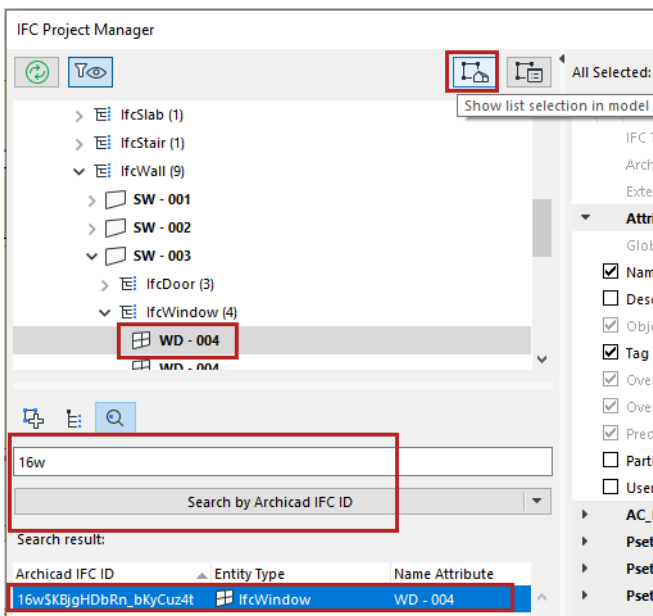
- Archicad IFC ID (the Globalid IFC attribute automatically assigned by Archicad to each element)
- External IFC ID (the Globalid IFC attribute assigned to each IFC element imported to Archicad)

Switch to the search mode using the Find by GlobalId tool (third icon underneath the tree).



Note: The External IFC ID is created when importing an IFC model to Archicad using either Merge or Hotlink IFC. This is done to prevent existing Archicad elements from being overwritten by identical elements being imported from IFC. (The External IFC ID stores the original ID of the model elements as defined in the external file).

Type the identifier (or any portion of it) that you are searching for into the empty field below the tool, and then click the “Search” button.



- Select any item in the “Search result” list to show that element in the tree.
- Click “Show list selection in model” to find and select the element in the Archicad model.

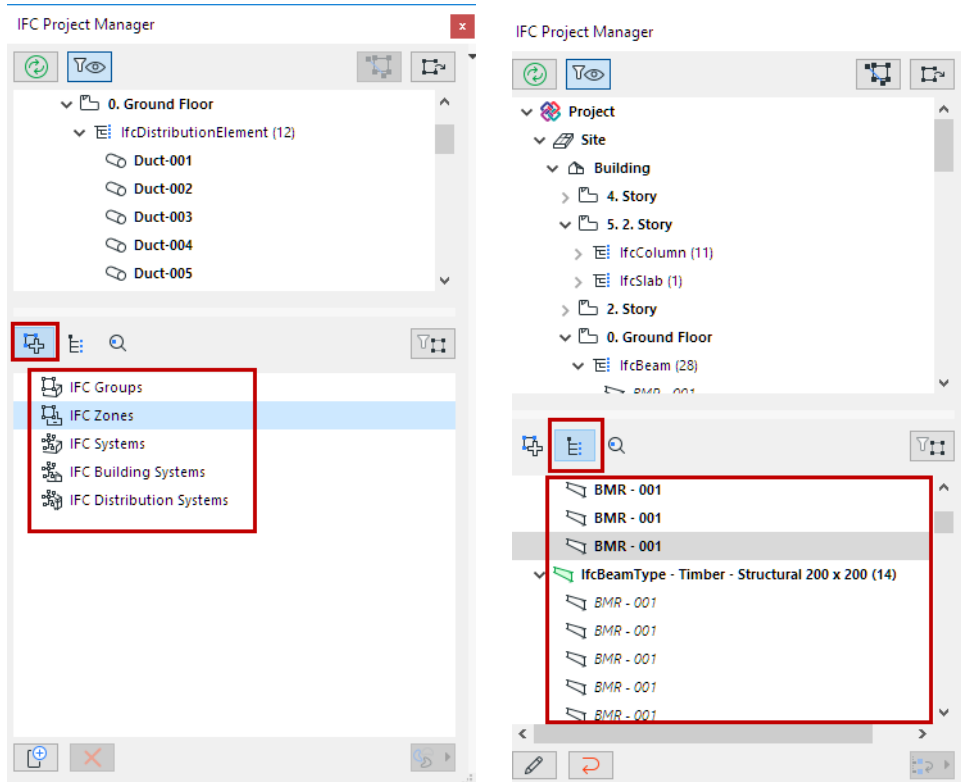
IFC Assignments View vs. IFC Type Products View

You can group IFC elements by Assignments and/or IFC Type Product.

[See the descriptions of IFC Assignments and IFC Type Product.](#)

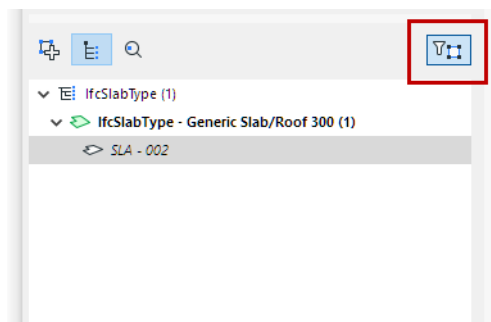
To view these groupings in IFC Project Manager:

Switch to “Assignments” or “Type Products” view/definition mode using the first or second icon, respectively, underneath the tree.



View Items by Assignment or Type Product

To narrow the displayed Assignment/Type Product data to that of the selected elements only, click the “Filter for selected element(s)” button.



Related Topics

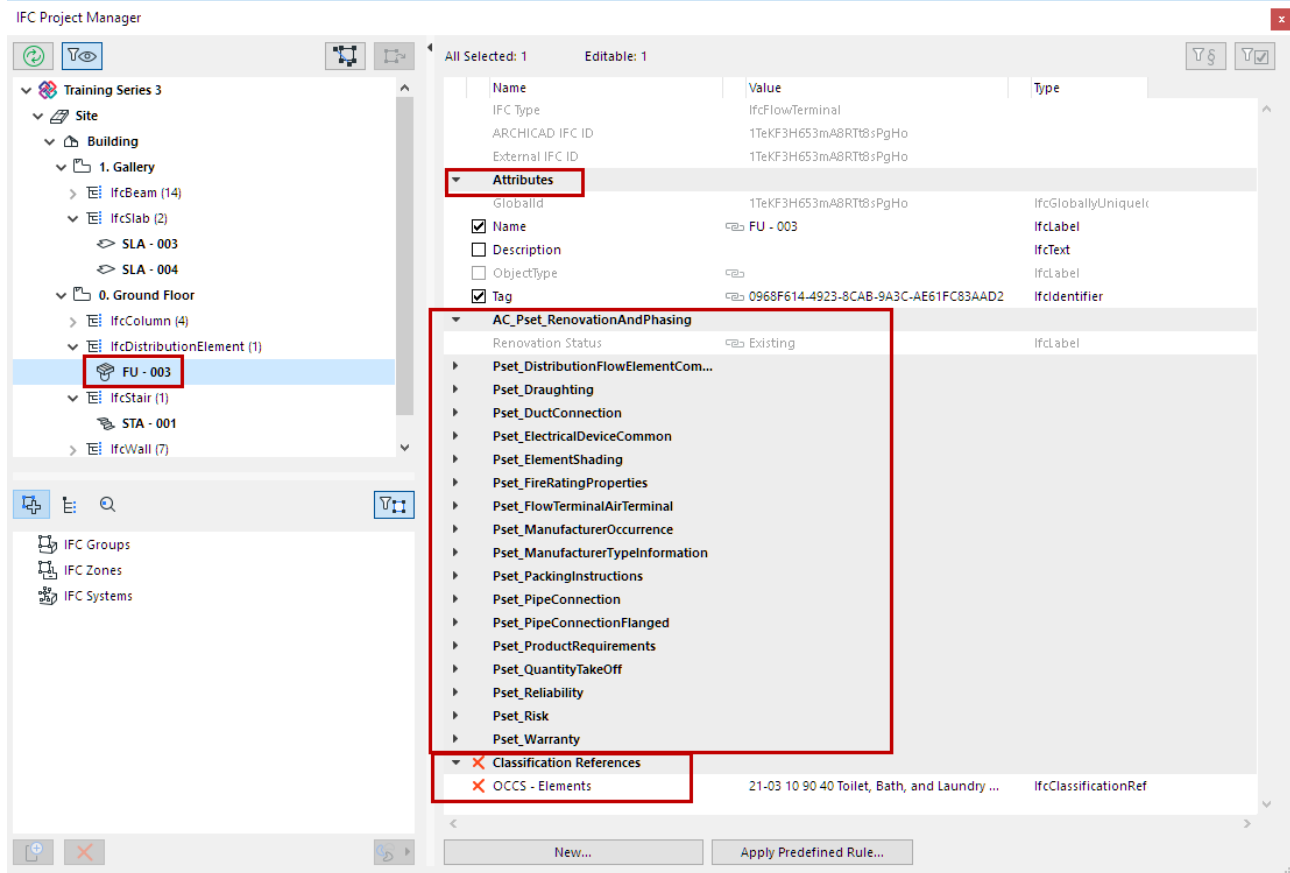
[Using Assignments in IFC Project Manager](#)

[Using Type Products in IFC Project Manager](#)

Navigating in IFC Project Manager: Attributes and Properties

Select a project element in the tree: for this selected item, the right side of IFC Project Manager displays:

- available IFC Attributes
- Properties grouped in Property Sets
- Classification References



In case of multi-selection, the IFC data of the last-selected element are shown.

At the top of the list, the numbers of selected/editable elements are shown.

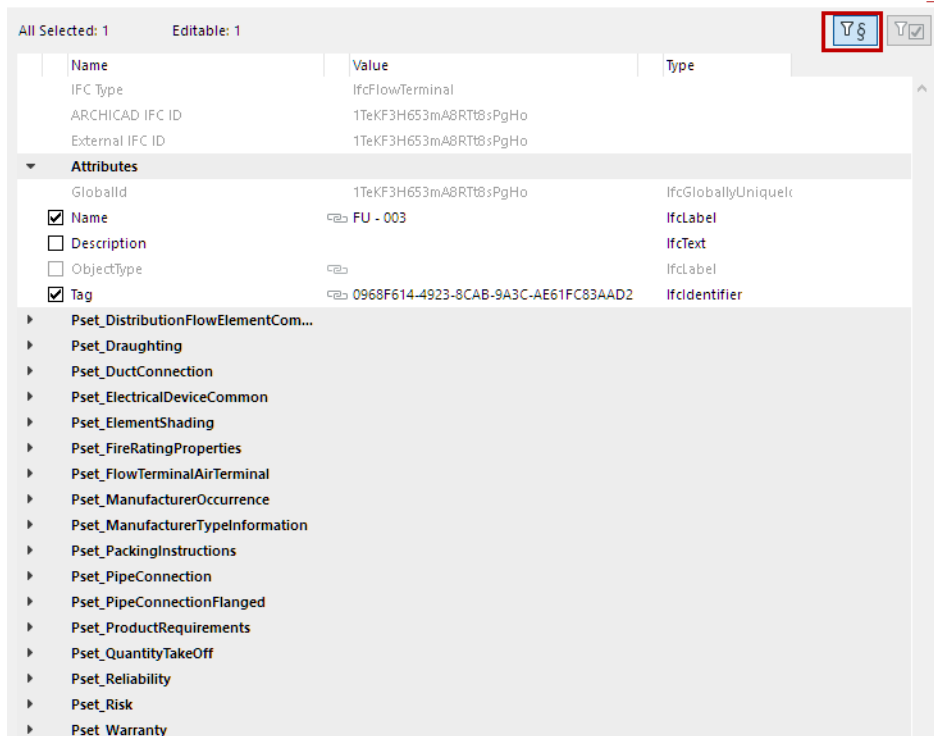
Many IFC Property and Attribute are defined *automatically*, based on the Property Mapping rules defined in the IFC Translator for Export.

Filter Data Shown in IFC Project Manager

The predefined IFC data types are very numerous, and you usually don't need to see all of them at once in IFC Project Manager. Use the two filter buttons at top right to cut down on the data shown here:

Show only Properties from Active Preview Translator

Activate this button to display only those data based on to the Property Mapping preset of the Preview Translator.



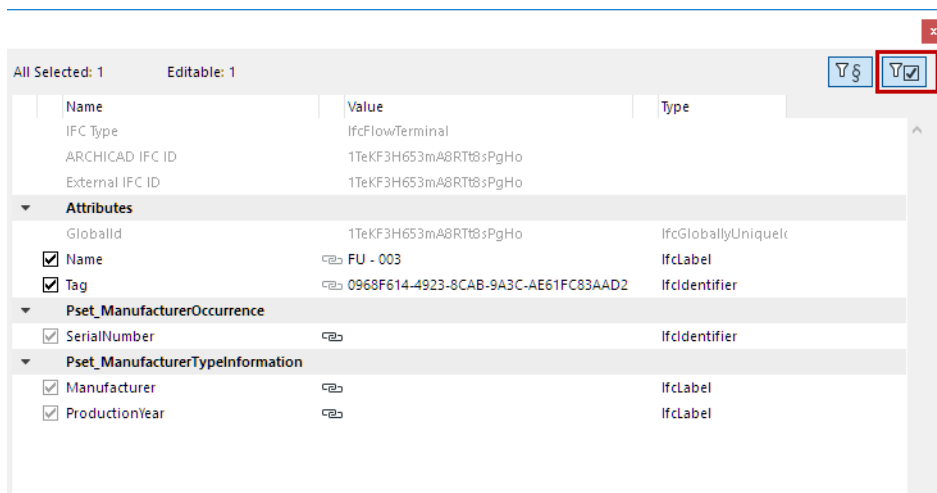
Show only Properties from Active Preview Translator (Property Mapping for IFC Export)

See also [Preview Translator and Property Mapping for IFC Export](#).

If you do not activate this filter, then IFC Project Manager also displays any new properties created here and in the Element Settings dialog boxes, as well as custom data added during an import process.

Show Only Items with Values

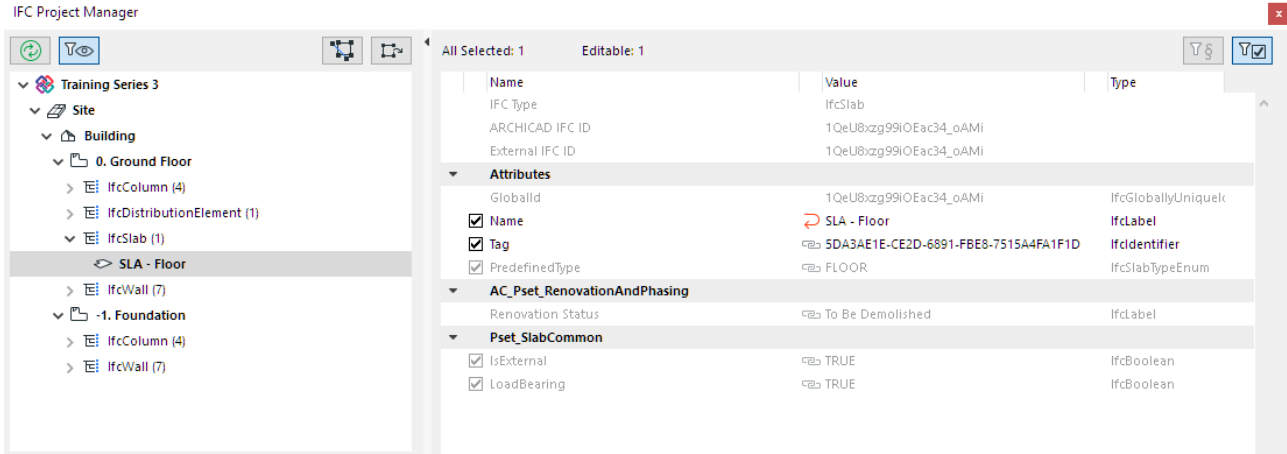
Use this toggle to display only those data which have assigned values: that is, its checkbox is enabled.



Note: A property whose field is empty, but whose checkbox is enabled, counts as a property that has a value assigned.

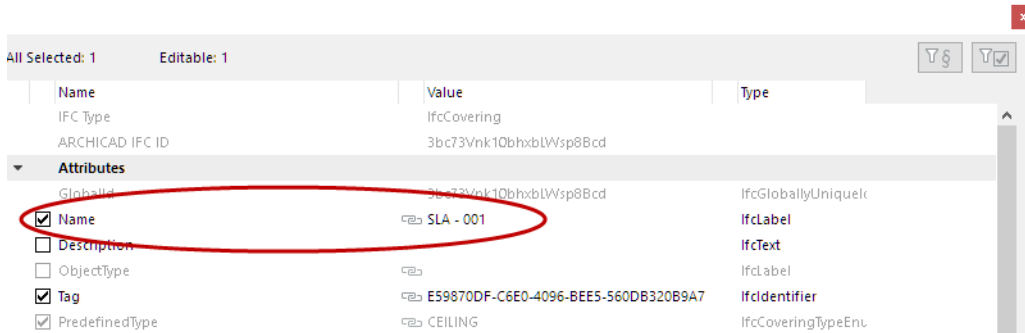
Assign or Edit IFC Property/Attribute Values

For any selected element in IFC Project Manager, overwrite any editable property or attribute value as needed.



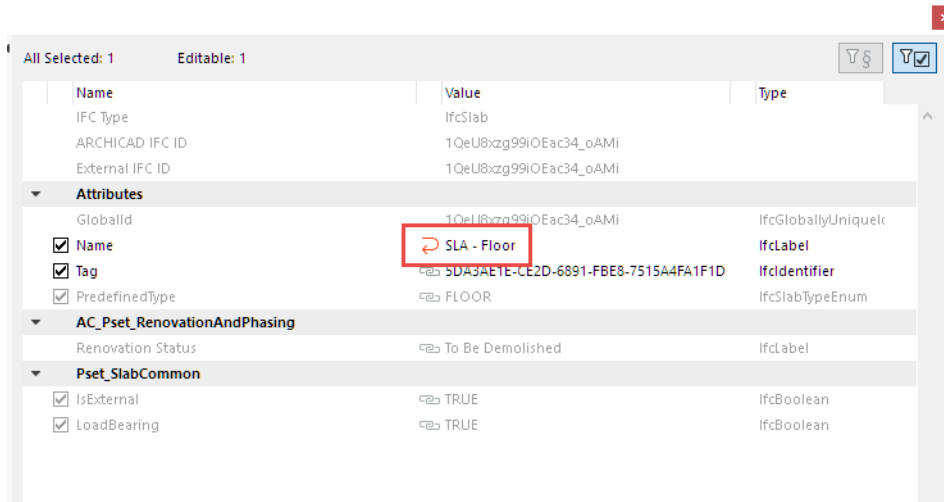
The display of Attributes and IFC Properties in IFC Project Manager indicate the data status:

- **BLACK:** Editable items can be overwritten with a custom value.
- **Checkbox:** An active checkbox indicates that this property or attribute has a value.

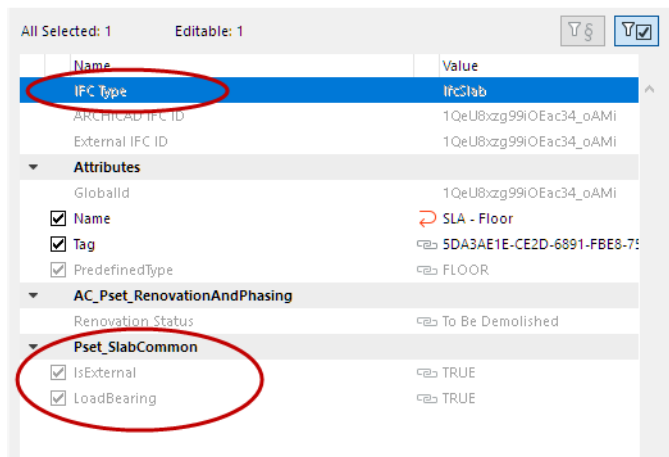


Note: Don't forget that if you enable "Show only items with values", then items without a value are not shown here at all. [See Filter Data Shown in IFC Project Manager.](#)

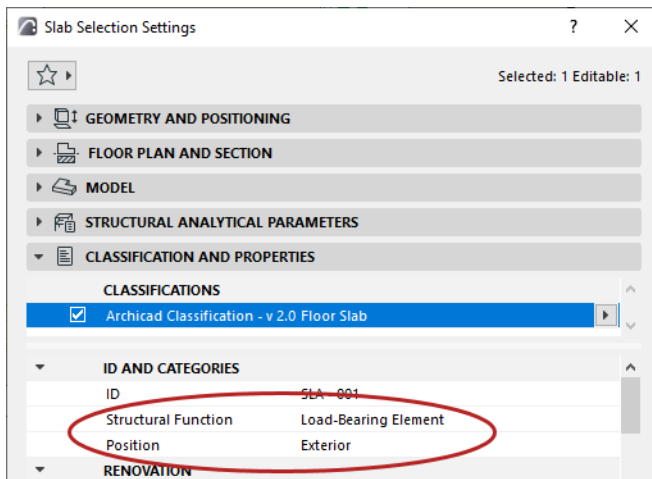
- **RED ARROW:** A customized value is shown with a red arrow: click it to revert to the original value.



- **GREY:** cannot be modified here in the IFC Project Manager.
 - The element’s **IFC Type** depends on the Type Mapping definition of the Preview IFC Translator. This item is greyed and not editable here.
- **Chained:** This value is derived from some other interface. For example:



- The **IsExternal** and **LoadBearing** IFC properties (see image below) are greyed and not editable here.
- These values depend on what is set in the Archicad element’s Settings dialog: the “LoadBearing” value comes from its Structural Function; and the “IsExternal” value comes from the Position.



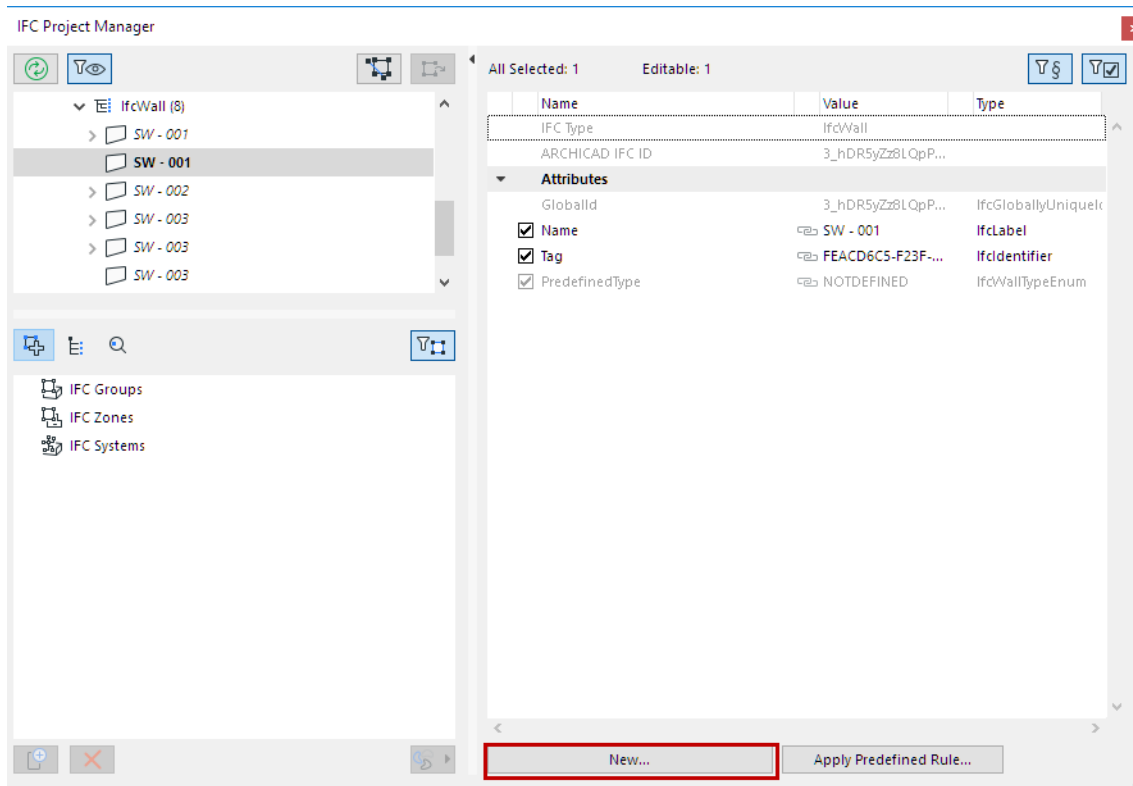
- If a Property Set no longer contains any Properties, it will be deleted from the list automatically.

Create New, Custom IFC Property

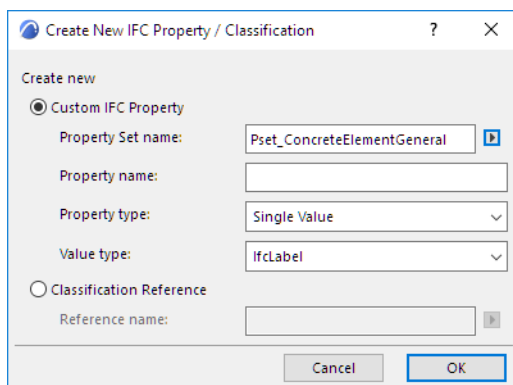
IFC Properties are additional (mostly IFC Type-specific) parameters assigned to an IFC Entity. Standard, so-called “IFC2x3 Schema” data are defined by buildingSMART and stored in property sets whose names begin with the prefix “Pset_”

You may wish to create a custom Property or Property Set (e.g. an office-standard Property Set).

1. In IFC Project Manager, click **New**.



2. In the appearing dialog box, enter a new custom Property Set name (Pset); or else choose from the list of existing Property Set names defined previously for the same element type (click the arrow icon to access this list). For example, if you are creating a new Property for an IfcWall, the list contains all the Pset names earlier assigned to other IfcWalls.



Note: To avoid errors (and to keep the standard property definition rules), do not use the standard “Pset” prefix when giving a name to your custom Property Sets.

3. Enter a new Property name.
4. Choose a Property type (single, enumerated, complex, etc.) as described in the following table.

Property type	Definition
Single Value	A property object which has a single (numeric or descriptive) value assigned.
Enumerated Value	A property object whose value is chosen from an enumeration.
Bounded Value	A property object which has a maximum of two (numeric or descriptive) values assigned, the first value specifying the upper limit and the second value specifying the lower limit
Table Value	A property object which has two lists of (numeric or descriptive) values assigned, corresponding to a table with two columns.
List Value	A property that has several (numeric or descriptive) values assigned, corresponding to an ordered list.
Reference Value (IfcCalendarDate only)	A property object which references to calendar date-type entity (day, month, and year).

5. Set the value type to label, text, integer, boolean, etc.

See also [Unit Conversion for IFC Export](#)

Value type	Definition
IfcAreaMeasure	REAL type value of the extent of a surface. Value must be set in square meters, but is recalculated at export based on Area Unit settings.
IfcBoolean	A defined data type of simple BOOLEAN type. Value: TRUE or FALSE.
IfcIdentifier	An identifier is an alphanumeric STRING (max. 255 characters) which allows an individual thing to be identified. It may not provide natural-language meaning.
IfcInteger	A defined data type of simple INTEGER type. The number of bits contained in the IfcInteger is unrestricted, but in practice it is implementation-specific.
IfcLabel	A label is the term by which something may be referred to. It is a STRING (max. 255 characters) which represents the human-interpretable name of something and shall have a natural-language meaning.
IfcLengthMeasure	REAL type value of a distance. Value must be set in meters, but is recalculated at export based on Length Unit settings.
IfcLogical	A defined data type of simple LOGICAL type. Value: TRUE, FALSE or UNKNOWN.
IfcMonetaryMeasure	REAL type value of an amount of money without regard to its currency. The currency unit can be set at Currency Unit settings.
IfcPlaneAngleMeasure	REAL type value of an angle in a plane. Value must be set in radians, but is recalculated at export based on Angle Unit settings.
IfcPositiveLengthMeasure	Length measure shall be greater than zero. Value must be set in meters, but is recalculated at export based on Length Unit settings.

Value type	Definition
IfcRatioMeasure	REAL type value of the relation between two physical quantities that are of the same kind. Input given in percent is expressed as a decimal value: e.g. 25% becomes 0.25.
IfcReal	A defined data type of simple REAL type. The domain of Real is all rational, irrational and scientific real numbers. Here the precision is unconstrained, but in practice it is implementation-specific.
IfcText	Alphanumeric STRING of characters which is intended to be read and understood by a human being. It is for information purposes only. No character number limitation.
IfcTimeMeasure	REAL type value of the duration of time periods. The time unit can be set at Time Unit settings.
IfcVolumeMeasure	REAL type value of the solid content of a body. Value must be set in cubic meters, but is recalculated at export based on Volume Unit settings.
More Types...	See the definitions at “alphabetical listing” > “Defined Types”: https://standards.buildingsmart.org/IFC/RELEASE/IFC2x3/TC1/HTML/

Note: All other “Measure”-type properties not listed in the above table are handled and exported without any unit settings.

- Newly created IFC data will appear in the defined Property Set or under “Classification References” in the IFC Project Manager, as well as in the element Settings dialog among the listed properties.
- Such newly created IFC data appear with a red X in front, meaning that you can delete them at any time.

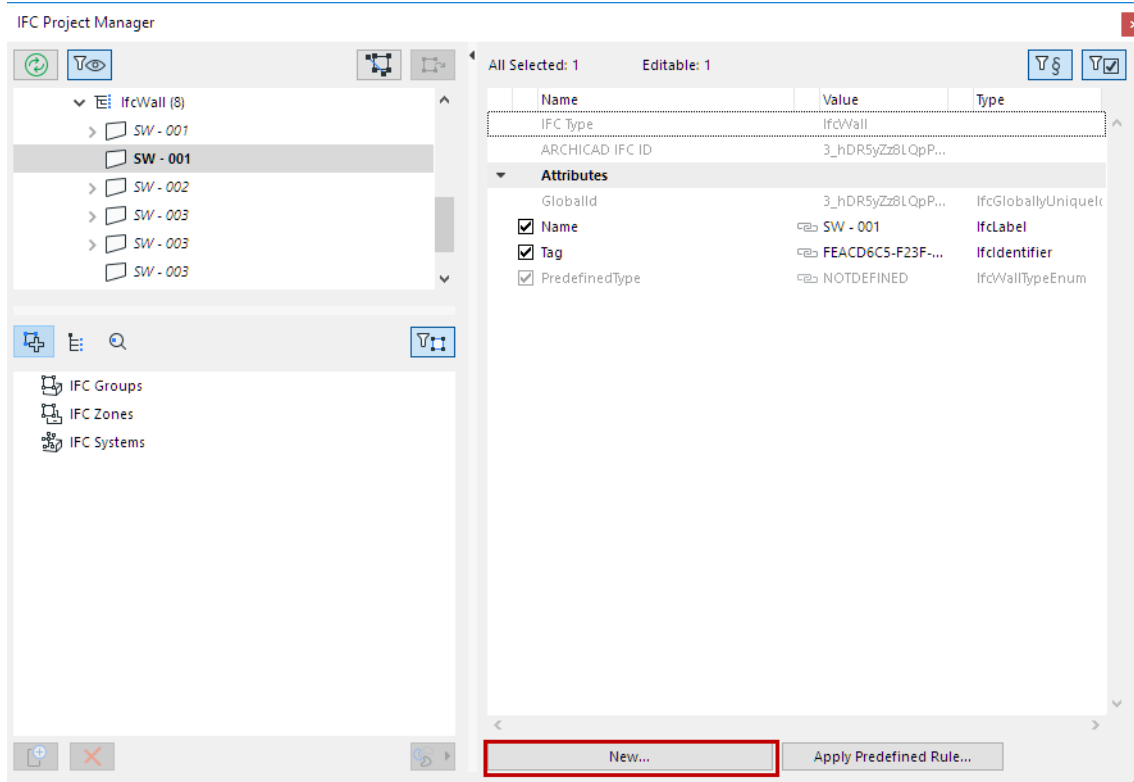
Create New Classification Reference

A Classification Reference (IfcClassificationReference) arranges IFC elements into a category. You can assign an Item Reference identifier, a classification Name attribute and some other optional parameters to all project elements (such as the building, the stories, the building elements, or the zones).

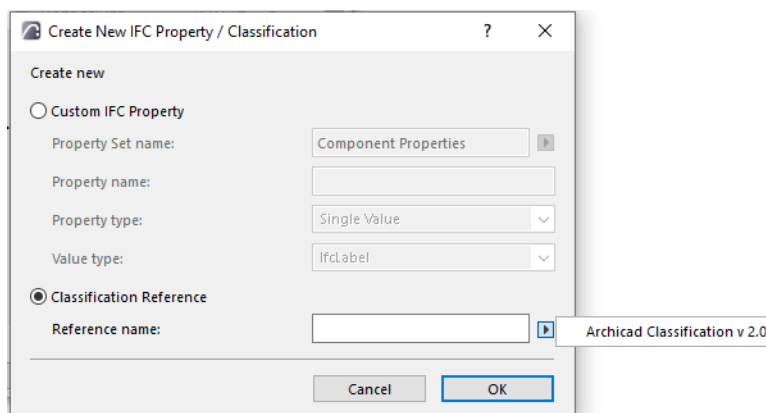
You can classify elements with this system both in IFC Project Manager and the element Settings dialog.

Example

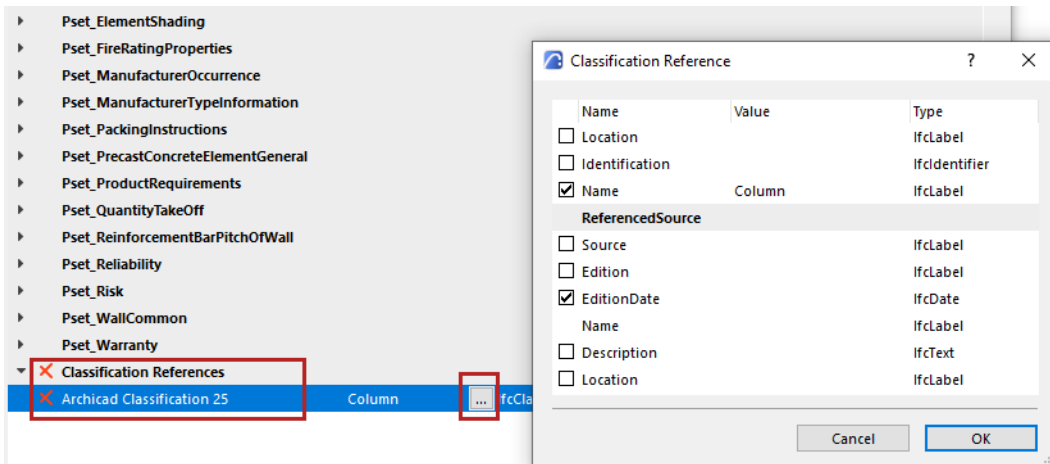
1. In IFC Project Manager, click **New**.



2. Click the **Classification Reference** button and assign it a new name. Here, too, you can choose from the list of existing Classification Reference names defined previously for the same element type (click the arrow icon to access this list).



- Assign values. The Classification Reference has 8 available items (the most important ones are the “Name” and the identifier (“Identification”) of the classification reference data) for defining/editing.



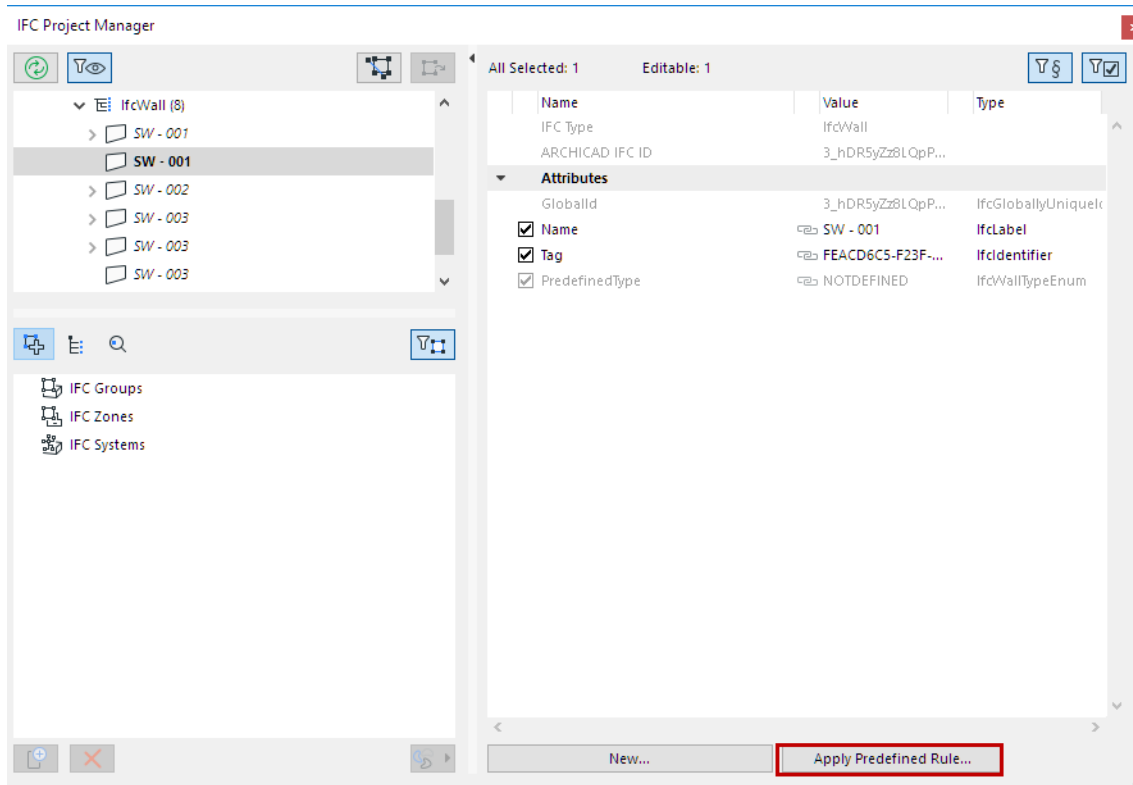
To define values of a custom-defined Classification Reference data, type them manually or use the Apply Predefined Rule command:

[See Apply Predefined Rules.](#)

Apply Predefined Rules

Use this function to create custom-defined Classification Reference data based on automatic (.xml-based) rules.

1. In IFC Project Manager, click **Apply Predefined Rule**.



There are three types of predefined rules, based on their differing structures:

- “Tree list” rule: select an element from a tree-structure database to create the new data
- “Table” rule: select a row from a table to create the new data
- “Command” rule: start the rule-command to create the new data

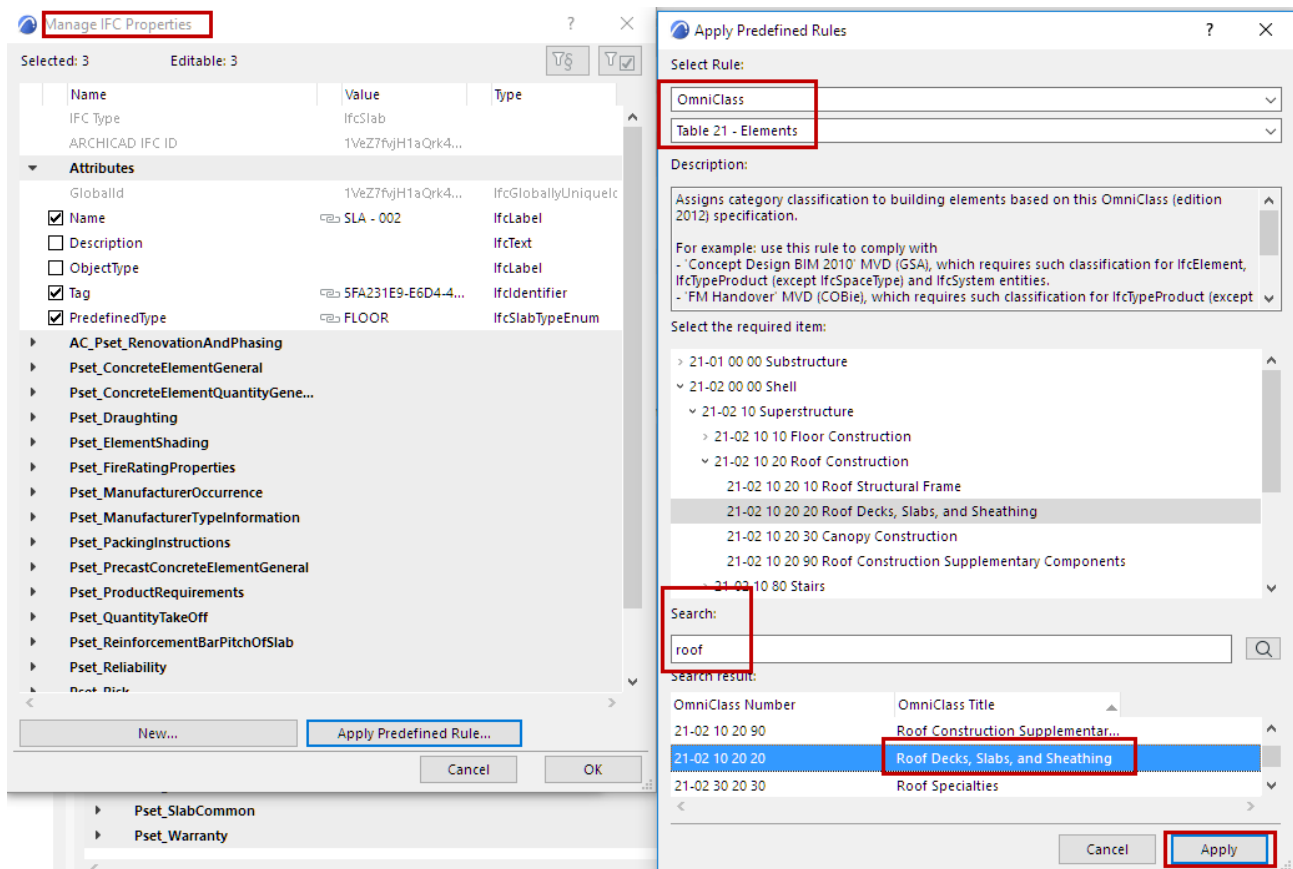
The program contains factory-defined built-in rules (these vary depending on your localized version of Archicad).

- The available rules are accessible from the following folder, and the newly created rules should also be saved to this location:
 - On Windows:
Graphisoft\Archicad\Defaults\IFC Rules
 - On Mac:
Applications\Archicad\Defaults\IFC Rules

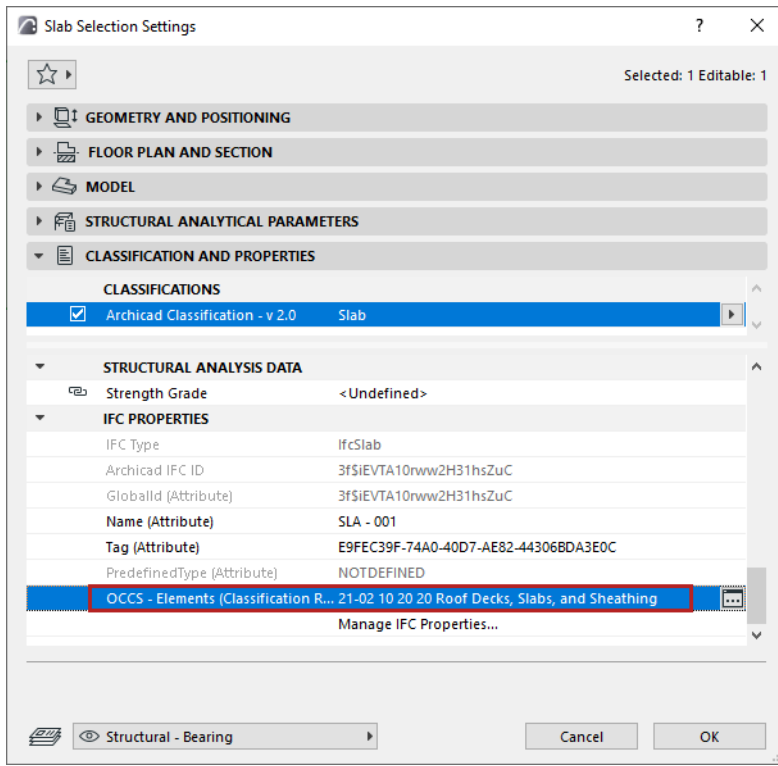
Apply Predefined Rule: Example

Assign classifications to a number of roof slabs (IfcSlabs) using the “OmniClass / Table 21 - Elements” rule:

1. In the model, select the Slab elements to be classified (you can use Find & Select). Open the Settings dialog box and click on “Manage IFC Properties” from the Classification and Properties panel. Alternatively, select the corresponding IfcSlab entities from the IFC Project Manager tree.
2. Click on **Apply Predefined Rule**. From the dialog box, choose the “OmniClass / Table 21 - Elements” rule.
3. In the database Search field, enter the text “roof”, and click the search icon to execute the search. (This searches among the rules in the databases (tree list and table) based on any partial text.)
4. From the Search result area, select the “Roof Decks, Slabs, and Sheathing” title.



- Click “Apply”. The Classification Reference is generated, with the name of “Roof Decks, Slabs, and Sheathing”, with an ItemReference value of “21-02 10 20 20”, and additional data corresponding to the OmniClass standard.



Notes

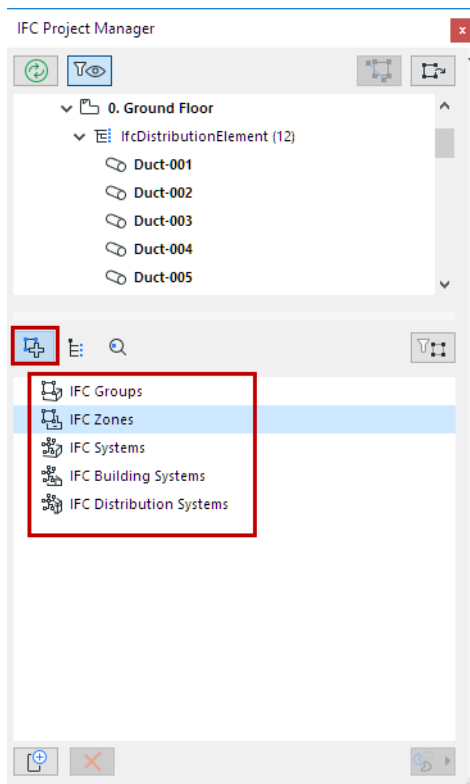
- Once you select an element, only the applicable rules are available in the “Select Rule” list (for example, if you have selected IfcProject, then the “Concept Design BIM 2010 (US GSA) / Project Client/Owner and Project Architect” rule is available). All the non-applicable rules are grayed.
- IfcElement includes building, distribution (MEP) and furnishing elements.
- Certain rules are not available in Settings dialogs if the rules are related to elements which do not have Settings dialogs (for example, IfcBuilding, IfcBuildingStorey, IfcOccupant, IfcProject and IfcGroup).
- You can create and edit your own rule in .xml format. It is recommended to peruse the structure of an existing rule file (which may contain multiple rules), and to create a similar new rule by duplicating it under a different name.

Using Assignments in IFC Project Manager

Use IFC Assignments (IfcRelAssigns) to create groups of project elements. The major Assignment Types of the IFC4 standard can be seen in IFC Project Manager.

- IFC Groups
- IFC Zones
- IFC Systems
- IFC Building Systems
- IFC Distribution Systems

For more information on each Assignment type, see [IFC Assignments](#).



Assignment Types in IFC Project Manager

To group IFC elements by IFC Assignment in IFC Project Manager:

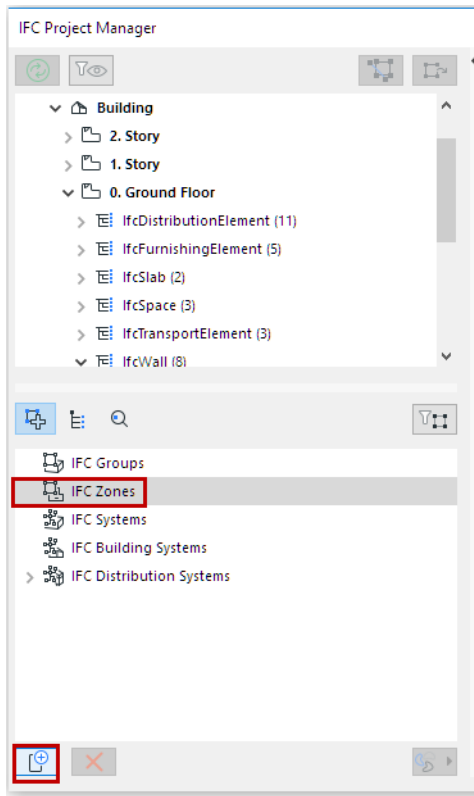
1. Use the New button to create a new Assignment under an Assignment Type
2. Drag relevant IFC entities from the top tree view into the newly created Assignment Relation item

This process is described below.

Create New Assignment Relation

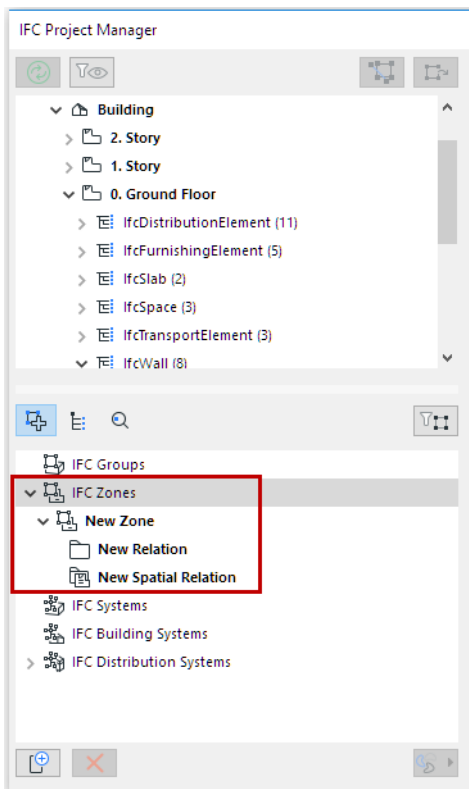
1. Switch to Assignment view mode by clicking the Assignments icon. (See illustration above).
2. Click on an Assignment type to select it (e.g. IFC Zones)

3. Click the “New” button at the bottom of the window.



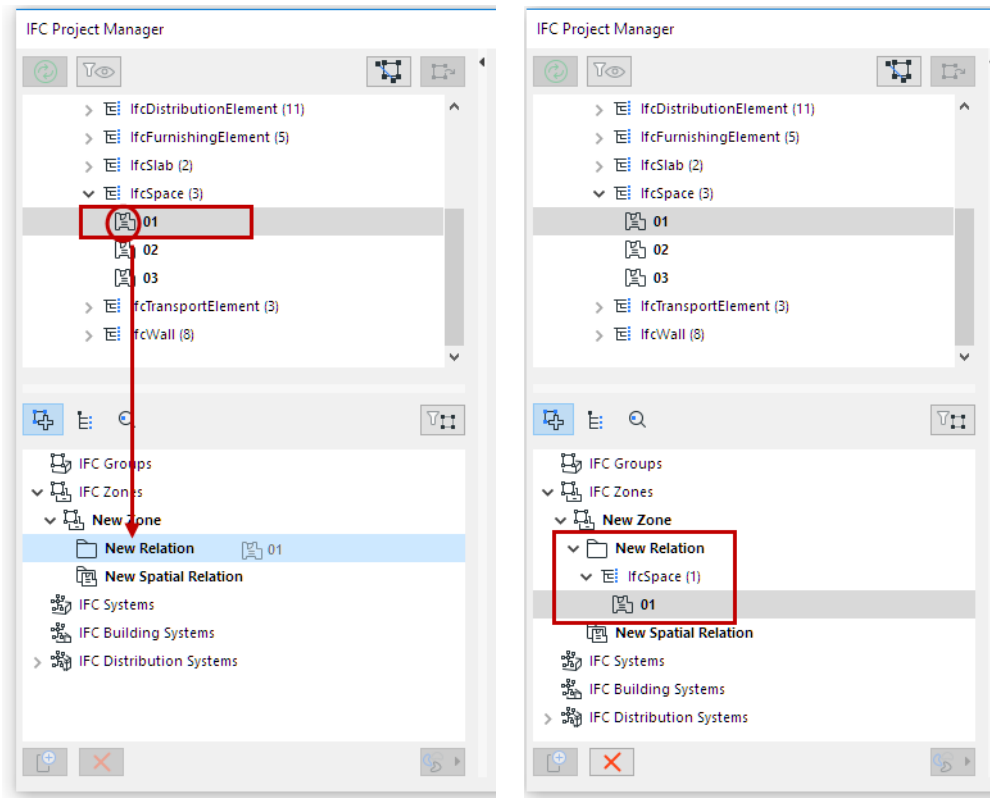
As a result:

- A **New Assignment** (e.g. New Zone) is created for the selected Assignment Type.
 - A **New Relation** folder is created within the new Assignment. You can now add related IFC entities to this folder.
 - A **New Spatial Relation** node is also created within the new Assignment. (Not available for IFC Groups). You can add related spatial entities to this node.



4. Drag related items from the IFC Entity tree above into the New Relation folder and/or the Spatial Relation node.

Hint: Shift+click to multi-select, and grab the entities by their icons to drag them.



Click and Drag IFC Entity into New Relation Folder

[See Examples of Assignments.](#)

Which Entities Can Be Added?

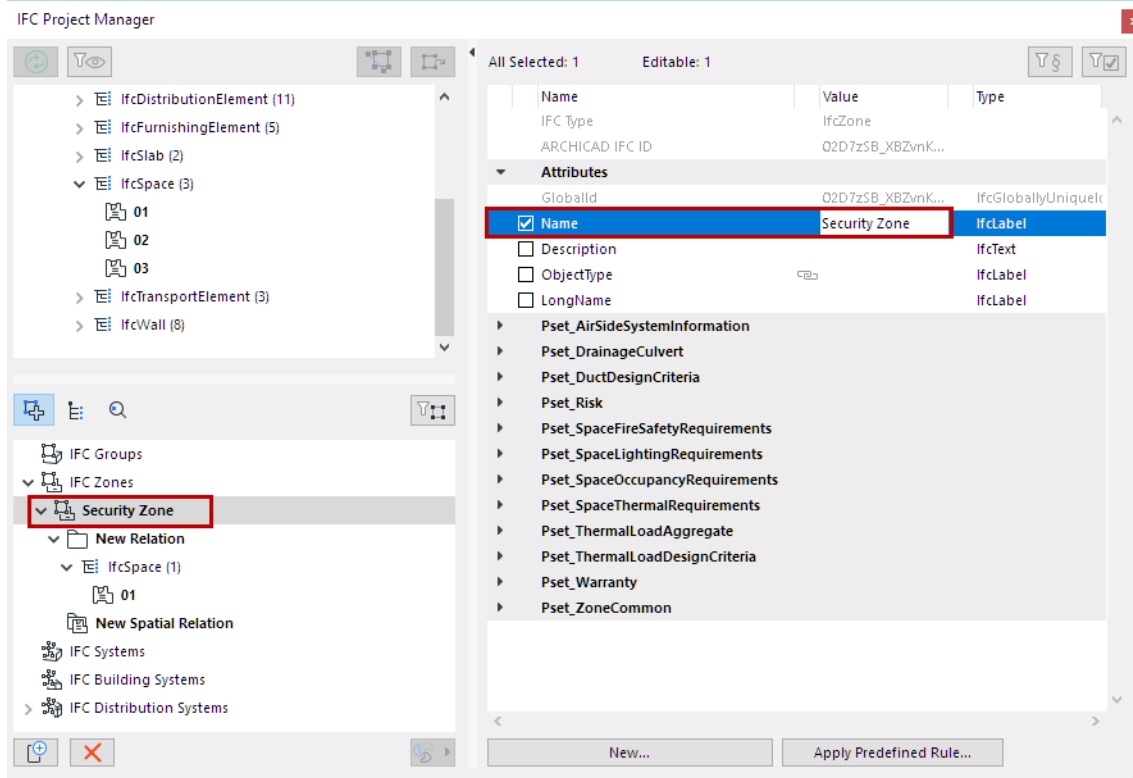
Relation Folder: Entities that can be added to a Relation Folder depend on the Assignment type:

- IFC Groups: Drag any project elements here to create logical group(s)
- IFC Zones: Drag IfcSpace elements here
- IFC Systems: Drag IFC spatial types (IfcSite, IfcBuilding, IfcBuildingStorey and IfcSpace) and any of their related elements
- IFC Building Systems: Drag related building elements here. Possible items: IfcBuildingElement, IfcFurnishingElement, IfcElementAssembly, IfcTransportElement
- IFC Distribution Systems: Drag MEP elements (IfcDistributionElement and/or sub-elements) here.

Spatial Relation: Spatial IFC elements (Site, Building, Story, Zone) can be added to the Spatial Relation node.

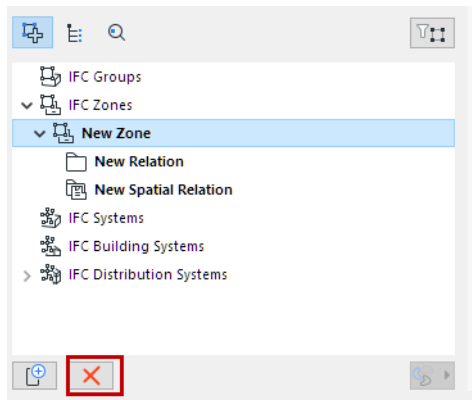
Rename Assignment Item

Rename any newly created item using the Name Attribute field on the right side of the dialog.



Delete Assignment or Entity

To delete a Relation entity or Assignment, select it and click the “Delete” (red X) button.

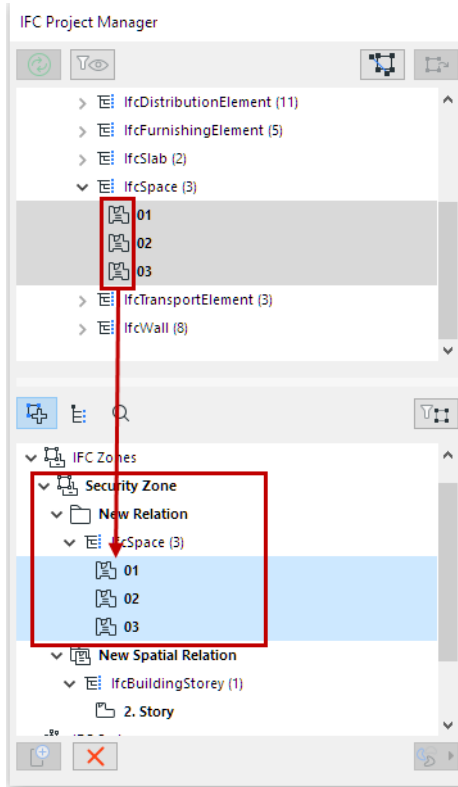


Manage Assignment Hierarchy

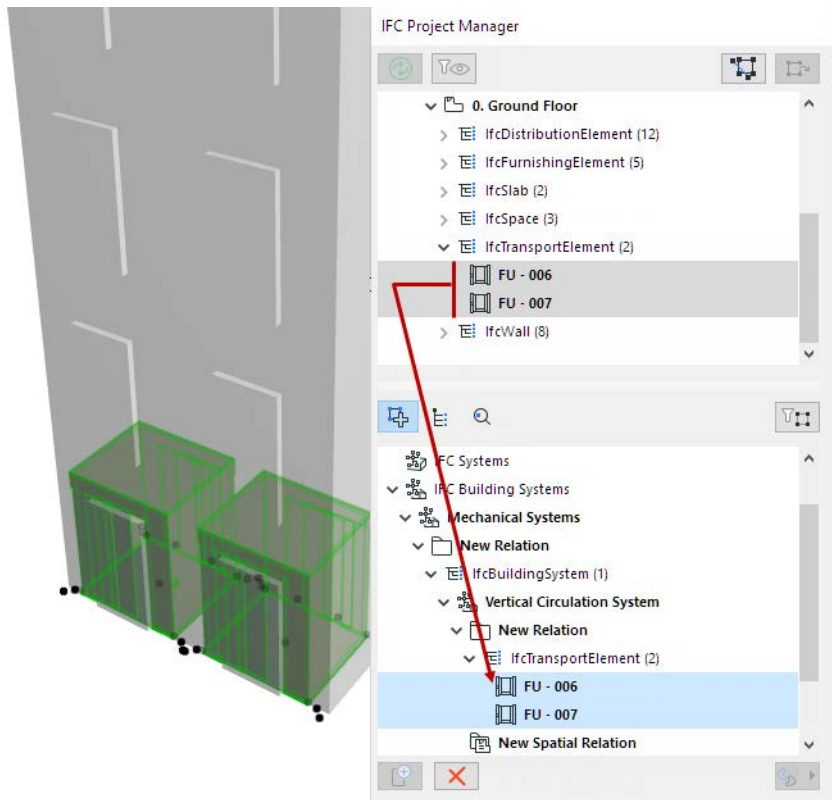
- Within a particular Assignment Type, drag and drop entities among different Relation folders
- Define new hierarchies: Select an existing Assignment or Assignment Type item and click “New” to define a sub-item. This way, you can create a multi-level hierarchy within an Assignment Type.
- To copy an item, hold down Ctrl/Alt while dragging it
- To make a child Assignment into a parent Assignment, select it and drag it outside of the dialog box; it will reappear in the hierarchy as a parent Assignment

Examples of Assignments

- Create an IfcZone named “Security Zone”, which will group together all of the project’s IfcSpaces that have a security function.



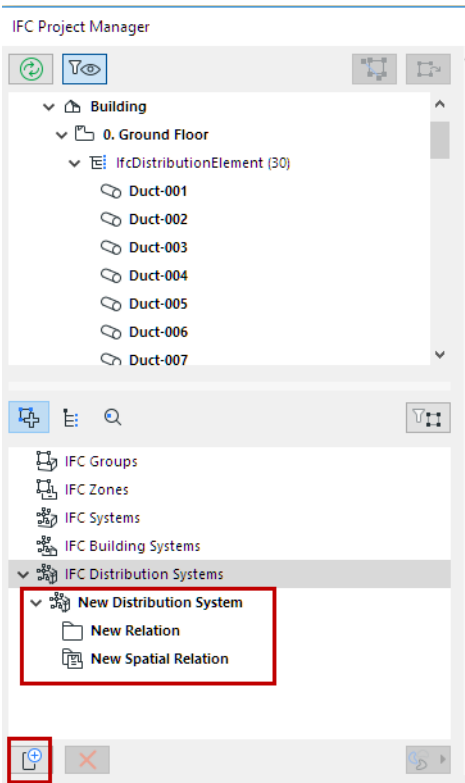
- Group elevators in a vertical circulation system, which is a child system of a mechanical system.



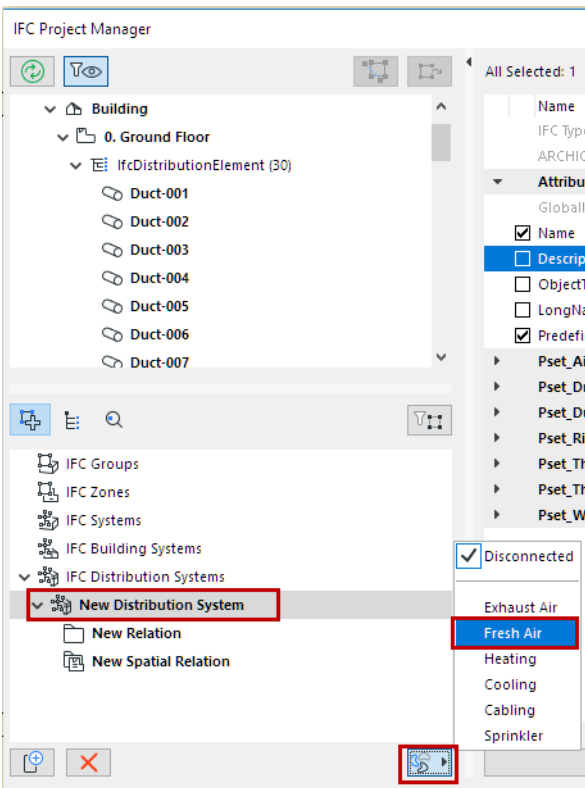
MEP Modeler Elements: Automatic Assignment to IFC Distribution System

MEP Modeler elements can be automatically added to the **IFC Distribution System** Assignment, corresponding to each element's MEP System. For example:

1. At the IFC Distribution Systems Assignment Type: Click the New button to create a new, empty IFC Distribution System.



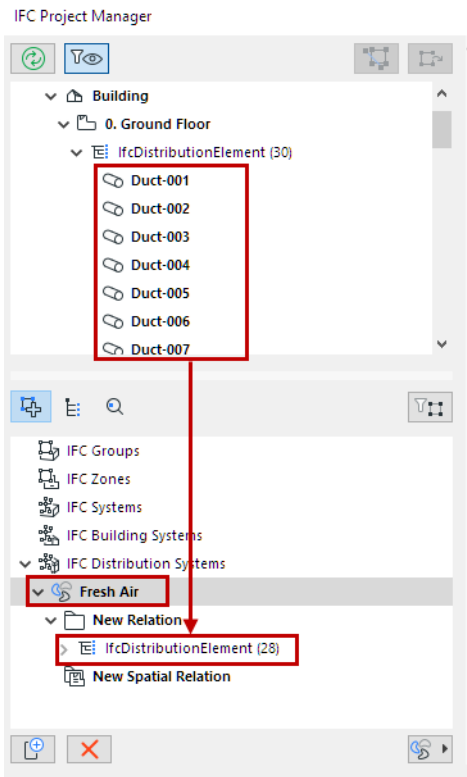
2. Select this new Distribution System, then click on the MEP icon at the right (underneath the Assignment list).



3. Choose the desired MEP System from the drop-down menu (e.g. Fresh Air). The Distribution System is renamed accordingly.

- All elements that belong to this MEP System (e.g. Fresh Air) are automatically added from the tree into the “New Relation” folder.

Note: You can also add any other Distribution element manually to this folder.



To delete MEP System elements from an IFC Distribution System, select the Distribution System item (e.g. Fresh Air), and choose “Disconnected” from the drop-down menu of the MEP System options.

Using Type Products in IFC Project Manager

The IFC Project Manager displays all the IFC Type Products that exist in the project, plus all the elements which refer to them. IFC Type Products are another way to group project elements.

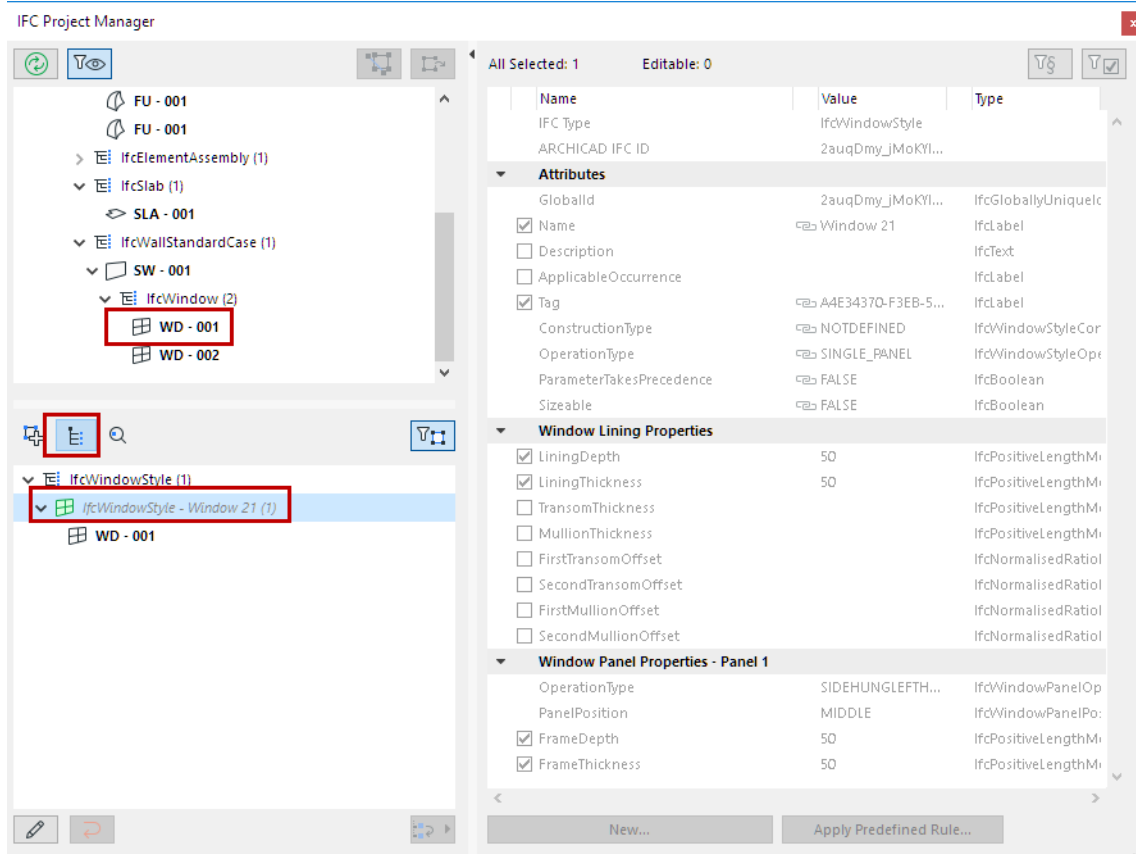
To see elements by Type Product, switch to the Type Products view/definition mode using the second icon underneath the tree.

Property Mapping rules can be defined for Type Products.

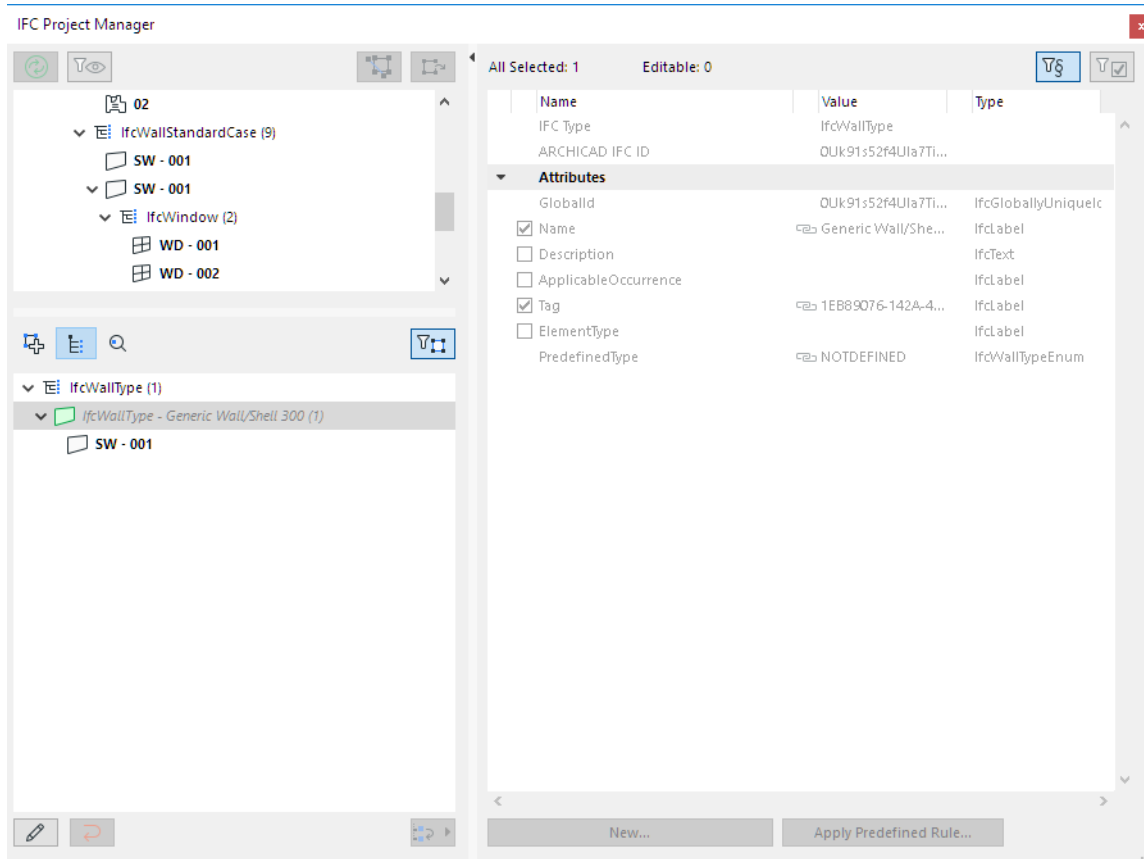
See an example here: [Example 4: Mapping Rule by IFC Type Product](#).

An IFC Type Product defines a particular style/type of other entities by relating to them with common IFC Attributes and Properties. For example, *IfcWindowStyle* is an IFC Type Product, to which many windows (*IfcWindow*) refer.

IFC Type Product entities are generated for Archicad elements based on the Type Mapping definition in the Translator for Export.



Select a Type (e.g. IfcWallType) to check its IFC data on the right side of the IFC Project Manager. The IFC data of the resulting IFC Type Product are not editable by default; they are displayed in italicized gray type, and all IFC data on the right side are grayed.



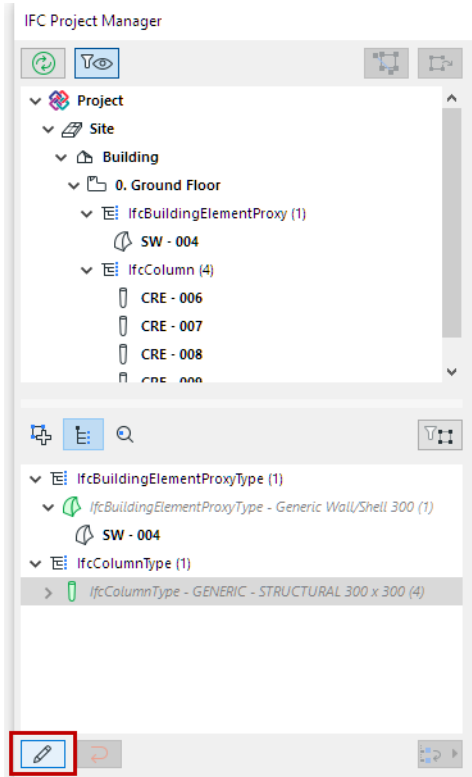
Edit IFC Type Product

You may wish to edit an IFC Type Product. For example:

- Change its Name Attribute
- Define a value for an Attribute or standard Property that has no value yet
- Modify existing Property data
- Create a new custom Property or Classification Reference

See [Create New, Custom IFC Property](#).

To edit the IFC Type Product, first enable it for editing: click on the “Edit/New Type” button.



Then make the desired modifications in the attributes/properties on the right.

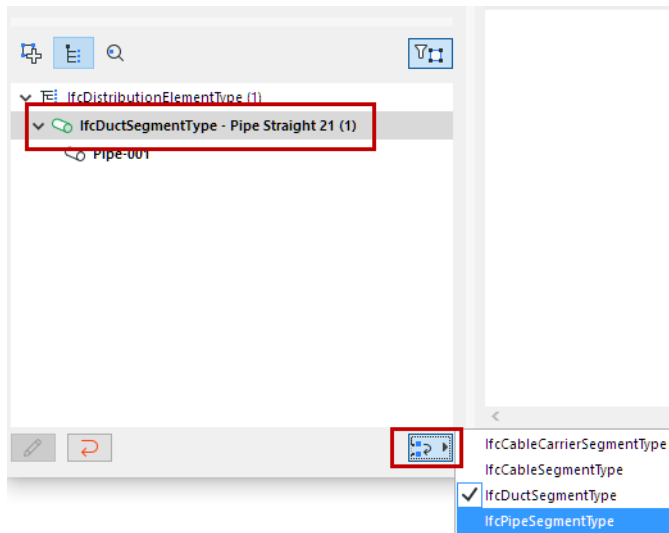
Change IFC Type Product Definition - Subtype

For some Archicad elements, definition as a particular IFC Type Product is not always clear-cut - typically the case for HVAC elements (IfcDistributionElements).

Suppose you have an MEP Modeler-defined Pipe element, classified as IfcFlowSegment. The program will automatically choose a subtype of the IfcFlowSegmentType (e.g. IfcPipeSegmentType).

To change this to a different subtype of the IfcFlowSegmentType (such as IfcDuctSegmentType):

1. Make the IFC Type Product editable, using the “Edit/New Type” button.
2. Use the “Change Type Product” drop-down menu.

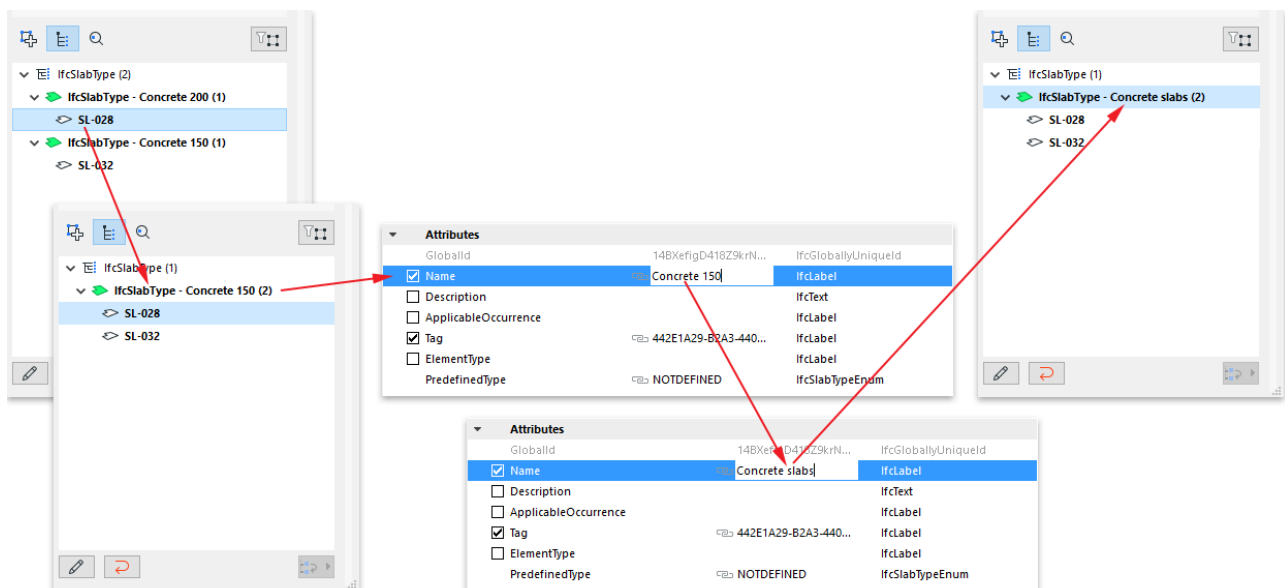


Combine IFC Type Products Into One

It is possible to combine several IFC Type Products entities into a single IFC Type Product. This works for all element types except Doors/Windows (see below).

For example, suppose you have two concrete Slabs of different thicknesses. By default, these will be classified as two separate IFC Type Products (IfcSlabType), because thickness is a significant property of slabs. But you want to combine them into a new, single type named “Concrete slabs.”

1. Select one of the Type folders and make it editable using the “Edit/New Type” button.
2. From the other Type folder, select the IfcSlab drag it onto first Type folder.
3. Change the Name Attribute of the Type folder (which now contains two elements) to “Concrete slabs” (see illustration below).



Combine IFC Type Products of Doors and Windows

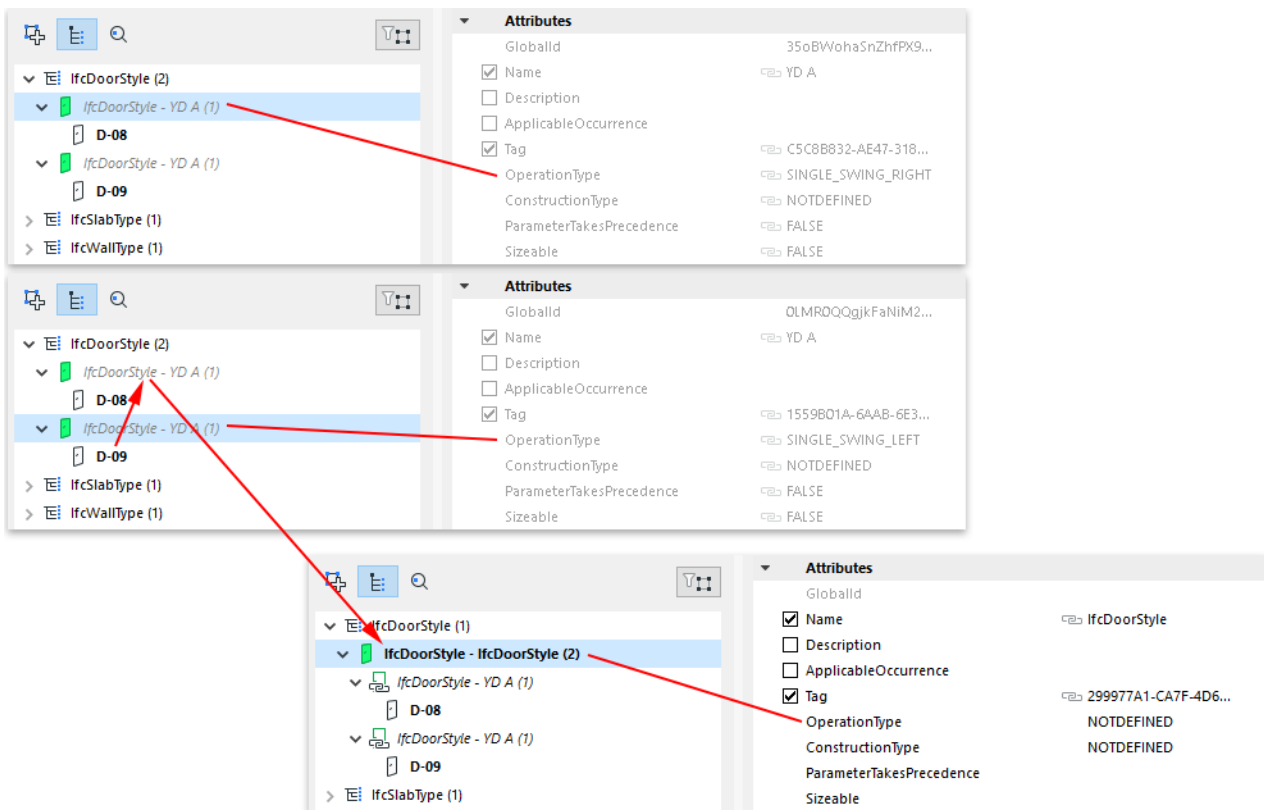
Combining doors or windows is more complicated, because some of their major parameters are generated from GDL data, and cannot be modified as a result of combining elements.

Such parameters are the OperationType Attribute and the panel and lining attributes. Suppose you have two doors that are identical in all their data (e.g. size, panel type), but their Operation Types are different: one opens to the right, the other to the left. These two doors will be classified under two different IFC Type Products: these two Types will have the same name, but their “OperationType” IFC Attributes will differ.

To have both doors belong to the same IFC Type Product, select one of the doors and place it in the other (second) type.

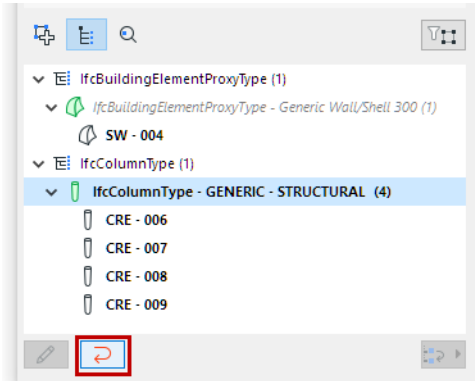
As a result, a new, common type is created that uses all of the data of the second type (such as Name Attribute, Properties, Classification Reference, etc.). This new common type is the one that will be exported, and which will be assigned to both of the doors. This common type can be renamed and its data modified.

At the same time, the two original types will remain as so-called linked types: they will link only data which is generated and which differ for each of the two doors (in this case, their OperationType Attributes). In other words, both doors will belong to a single type, which includes all of the data they have in common, but they will also maintain certain IFC data that were different for each and cannot be combined (see illustration below).

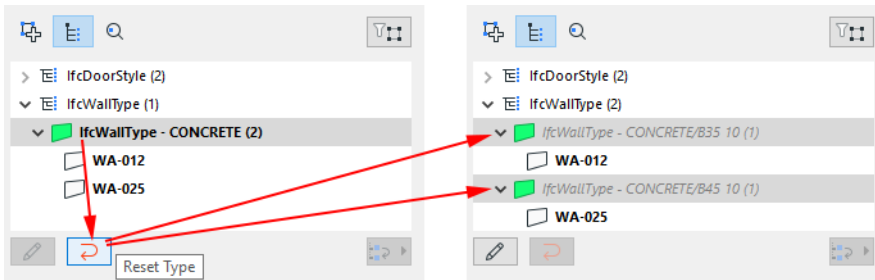


Reset Type

“Reset Type” means that all edited IFC data will revert to their original values, and the new data will be deleted.



- When Reset Type is applied to a selected IFC Type Product (e.g. IfcWallType) that has been edited, the Type's original data are restored (newly modified IFC data are lost), and the Type is no longer editable. If the Type now contains new elements as a result of combining Types, then those elements will return to their original locations.
- When Reset Type is applied to a single element (e.g. IfcWall) that has been moved into a different Type, that element will return to its original location.



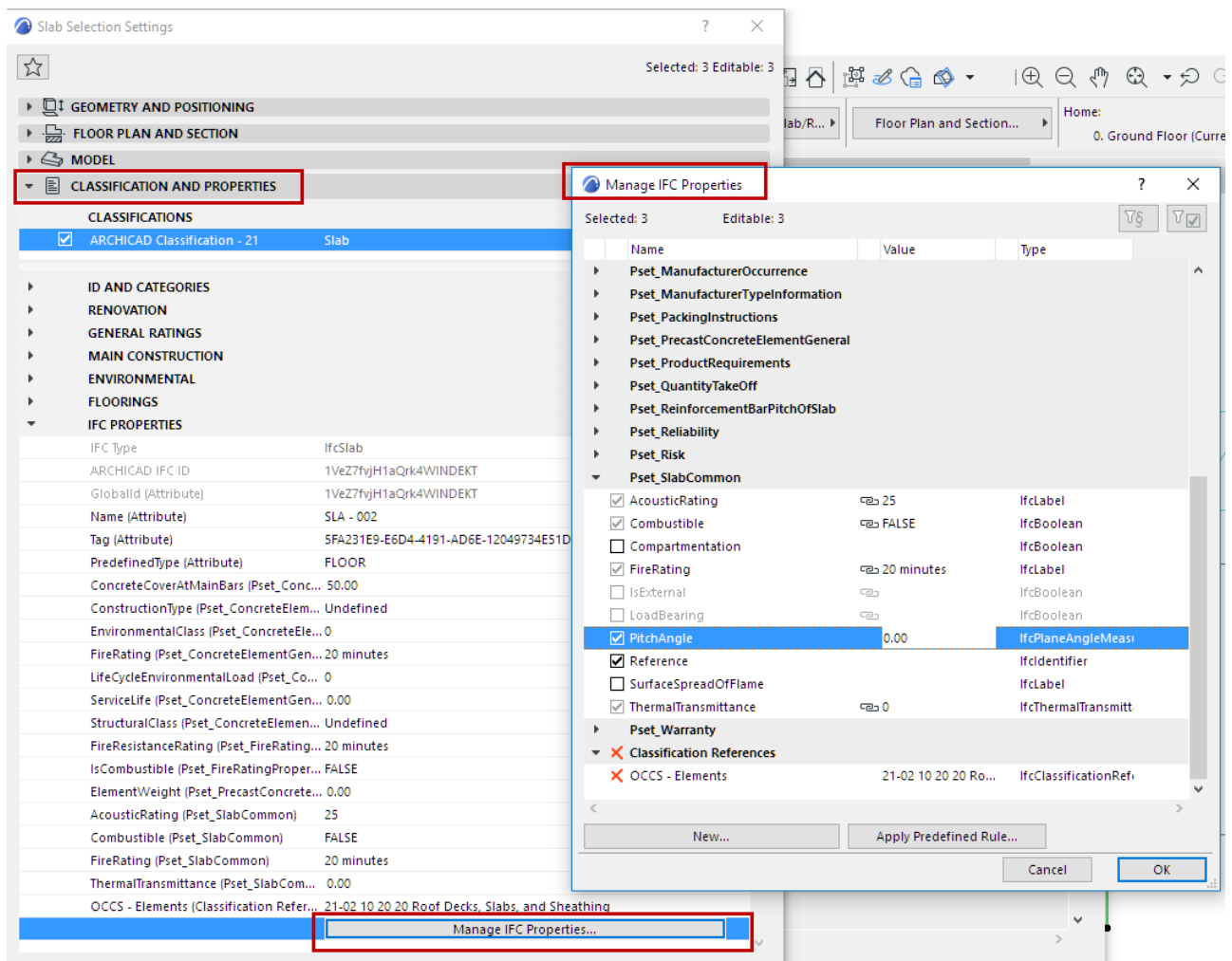
Manage IFC Data at Element Level

Archicad elements created from imported IFC model elements display their standard and custom IFC data, at the element level, in their respective Settings dialog boxes.

New IFC data can be added to the elements manually or automatically, based on predefined rules in the **Manage IFC Properties** dialog, which is opened from Element Settings (Classification and Properties panel). The interface of the Manage IFC Properties and all its functions are the same as the right side of the IFC Project Manager dialog box.

For detailed function descriptions, see [IFC Project Manager](#).

The new data appear in the Classification and Properties panel of the element Settings dialog, under **IFC Properties**. Here, you can assign values to the available data fields that are still empty.

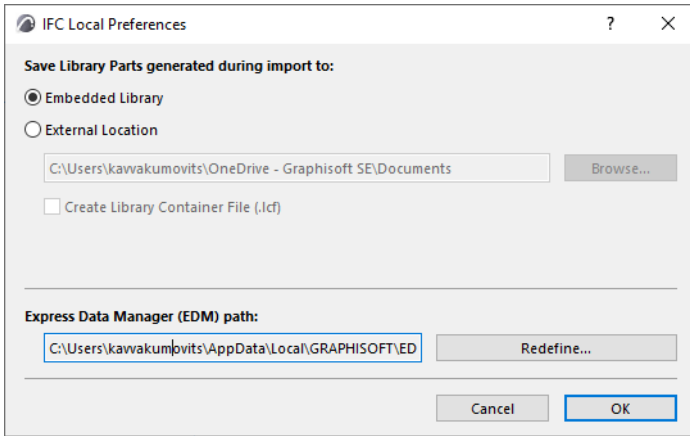


Working with IFC Data: Element Settings vs. IFC Project Manager

	Element Settings dialog	IFC Project Manager
management level	element-level only	project level
IFC data owner	<ul style="list-style-type: none"> • building elements • Zone (IfcSpace) 	<ul style="list-style-type: none"> • building elements • spatial elements: <ul style="list-style-type: none"> – IfcProject – IfcSite – IfcBuilding – Story (IfcBuildingStorey) – Zone (IfcSpace) • IFC Assignments • IFC Type Products • IFC Containers
Functions	<ul style="list-style-type: none"> • collects and lists IFC data • enables you to create, edit and delete IFC data • enables you to use predefined rules • filters Mapping properties • filters only IFC data having values 	<ul style="list-style-type: none"> • collects and lists IFC data • enables you to create, edit and delete IFC data • enables you to use predefined rules • filters Mapping properties • filters only IFC data having values • navigates between the Archicad model and the IFC model • displays Zone (IfcSpace) containment • enables you to assign elements directly to the IfcSite or IfcBuilding

IFC Local Preferences

The settings defined at IFC Local Preferences (**File > Interoperability > IFC**) affect the way that IFC data are created and managed.



Save Library Parts Generated During Import To

Set the default location of imported Library Parts when opening or merging an IFC project.

[See *Library Parts Location*.](#)

Express Data Manager (EDM) path

The EDM (Express Data Manager) toolkit is the engine which creates an Archicad model from the imported IFC model, and which creates an IFC file out of the exported Archicad model.

During the import-export process, the engine generates temporary files at the path defined here. File size varies depending on model size. Before redefining the path, make sure the selected drive is writable and has enough space.

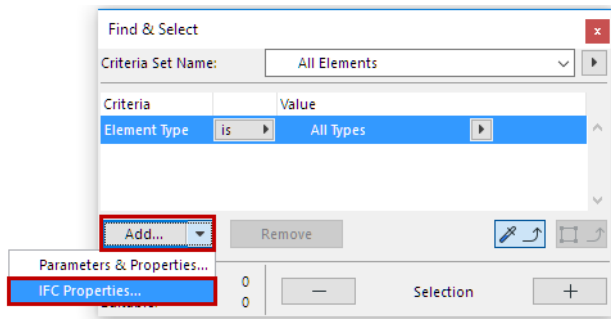
Define Element Criteria using IFC Data

Element criteria can include IFC data in the following functions:

- Find & Select
- Reserve Elements by Criteria (Teamwork)
- Interactive Schedule Schemes - list IFC data assigned to current project elements

In these dialog boxes:

1. When defining element criteria, choose “IFC Properties” from the “Add” pop-up list.



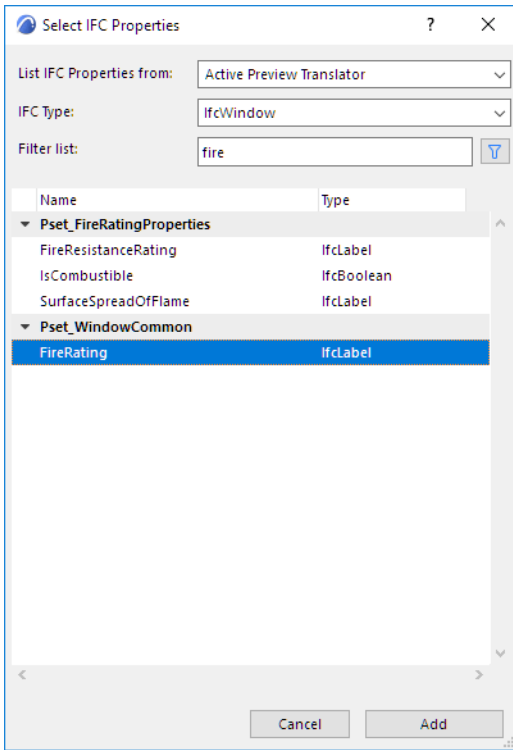
2. The Select IFC Properties dialog box appears.

Select IFC Properties

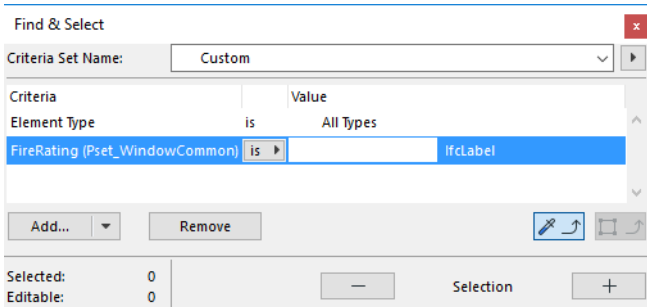
All IFC data added to project elements are listed here.

You can narrow the displayed data:

- Include IFC properties from the **Active Preview Translator** (as opposed to those currently used in the Project). For example, you may want to define a Find & Select criteria set that includes properties (e.g. Fire Rating) which may not yet exist in your project;
- Find only the IFC data of a chosen IFC Type; and/or
- Search for IFC data by typing any text string in the “Filter” list field.

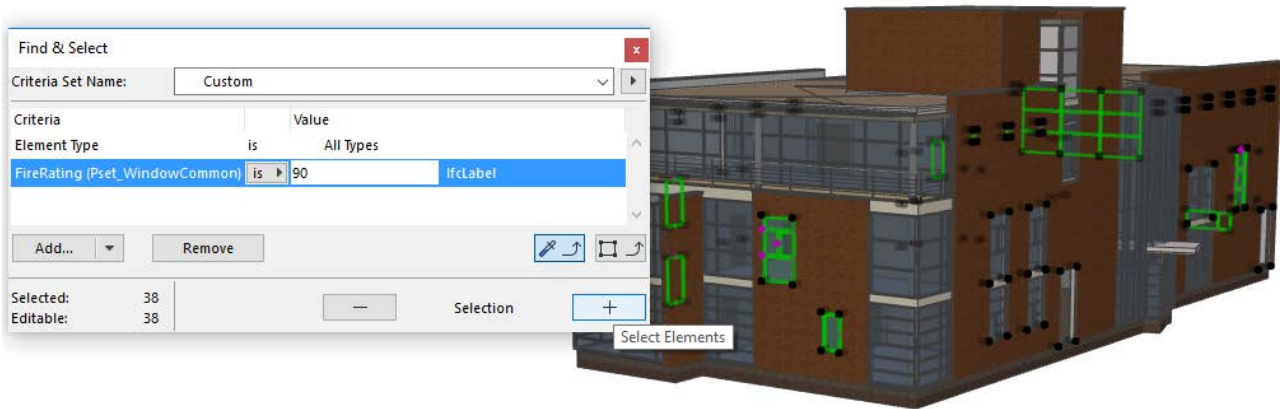


3. Select the IFC data item to use as element criteria. Click **Add**.
 - Multiselection is available
 - If you select a Property Set in the list, then all of the Properties it contains are added to the criteria.
 - If you select an Attributes folder or a Classification References folder, then all items contained in the folder are added to the criteria



4. Define operators and values for the criteria according to their type (label, integer, boolean etc.)

Note: In Find & Select: Get values for IFC data criteria from the model by applying the Pick Up Settings and Copy Settings tools to the owner element of the same IFC data type.)



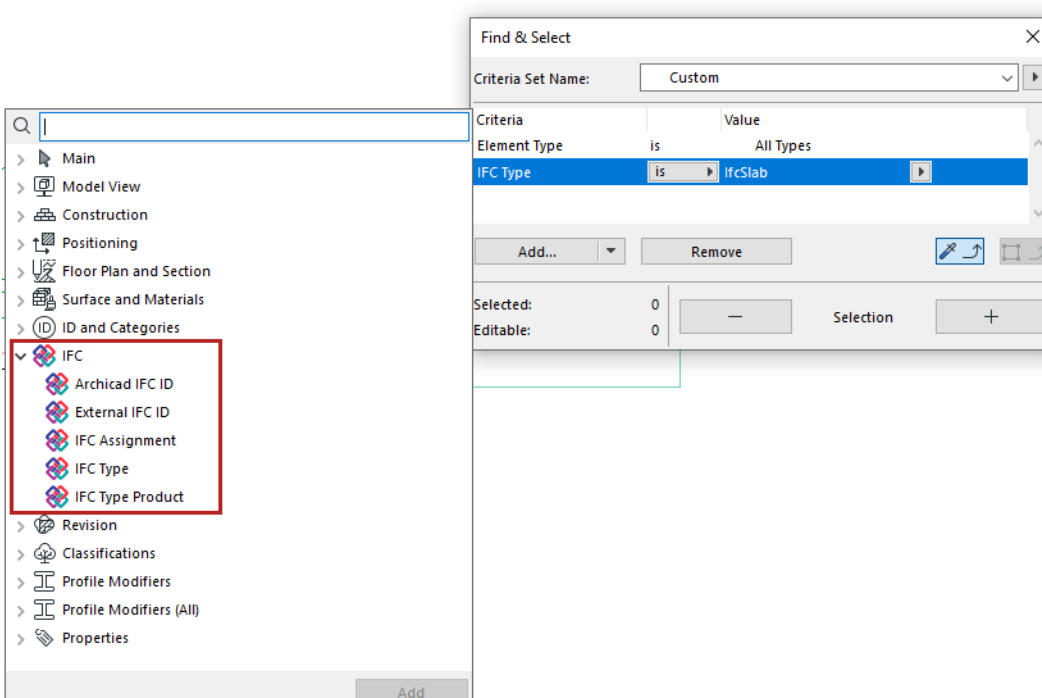
Using IFC Properties in IES Schemes

- If you select a Property Set from the list, then all of the Properties it contains are added as fields.
- If you choose “IFC Assignment” as the parameter for Schedule Field, then the table will list (in non-editable form) all those IFC Assignments (their named branch) to which the element belongs. If an element has multiple Assignments, all of them will be listed, separated by a “,”.
- If you list an element’s “IFC Type Product” data, then the Schedule table will generate two columns: The “IFC Type Product - Type” (e.g. 'IfcWallType') and “IFC Type Product - Name”) (the Name Attribute of the IFC Type Product).

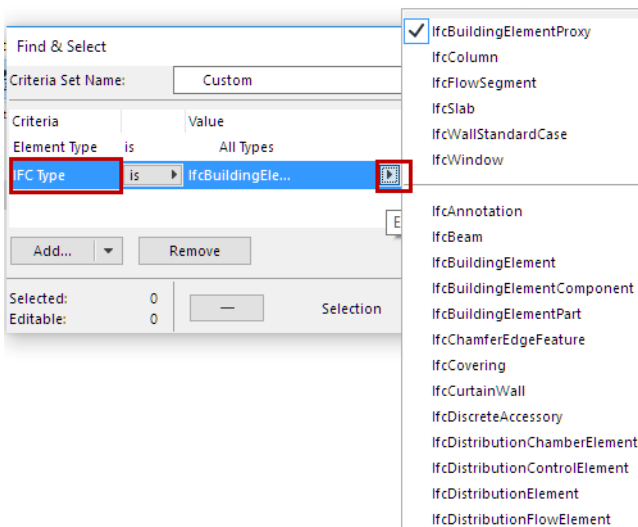
Use IFC Data Types as Element Criteria

From the criteria or field selection interface (e.g. the Find & Select dialog):

1. Click the **Add** button.
2. From the appearing Properties list, choose an IFC Data Type:



- Archicad IFC ID (the Globalid IFC attribute automatically assigned by Archicad to each element)
 - External IFC ID (the Globalid IFC attribute assigned to each element upon IFC import)
 - IFC Assignment, e.g. IFC Group.
 - IFC Type (see below)
 - IFC Type Product, e.g. IfcColumnType.
3. For IFC Type: the list is divided between all types available in the project (above the line) and all types that can be displayed and modeled in Archicad (below the line).

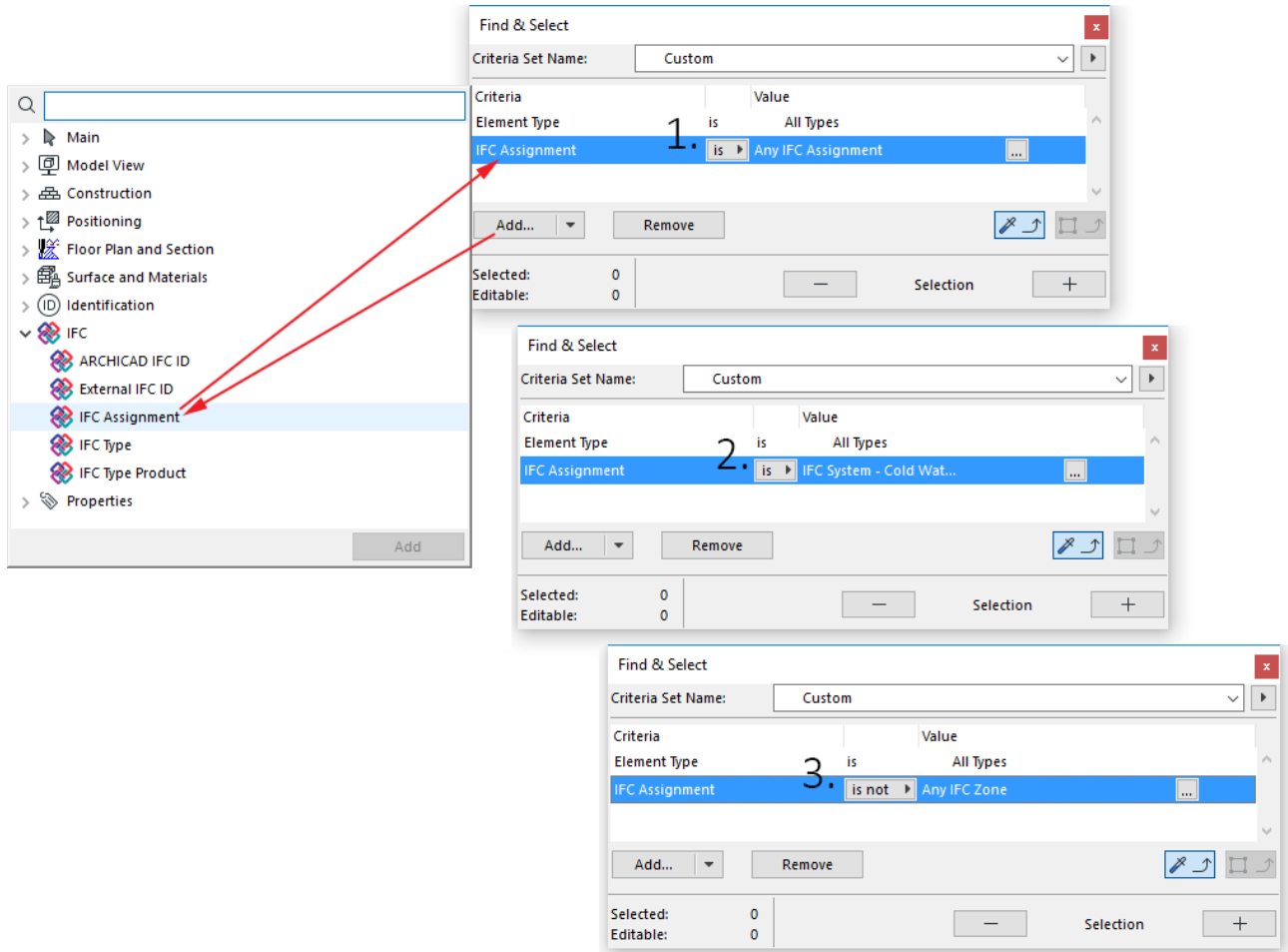


The IFC Type list includes all so-called “parent” element type (these are collector entities that are above the other elements in the IFC entity hierarchy) - for example, IfcBuildingElement.

The following are examples of Find & Select possibilities using IFC data as criteria:

1. Find all elements that are members of any Assignment.
2. Find all elements of a particular IFC System.

3. Find all Archicad Zones that have not yet been grouped into an IFC Zone.



Note: In Find and Select: the “Pick Up Settings” function does not work for IFC Assignment and IFC Type Product criteria.

IFC Translators: Overview

Importing and exporting model data using IFC takes place according to the settings of the IFC Translator you are using. An IFC Translator defines the rules for which elements should be converted and how they should be interpreted - whether in Archicad or in the external application reading the IFC file.

[Basic Workflow for Using IFC Translators](#)

[Important Note on Translators and Template Files](#)

[IFC Translators Dialog Box - Overview](#)

[Preview Translator](#)

[Managing Translators](#)

Basic Workflow for Using IFC Translators

Archicad provides predefined, factory-default IFC Translators whose settings are optimized for the particular import/export purpose.

Most Archicad users doing IFC export and import do not have to adjust any Translator settings. All you have to do is select a predefined translator from the Open dialog box (when importing a model in IFC format), or from the Save As/Merge to IFC dialog box (when exporting an Archicad model to IFC format).

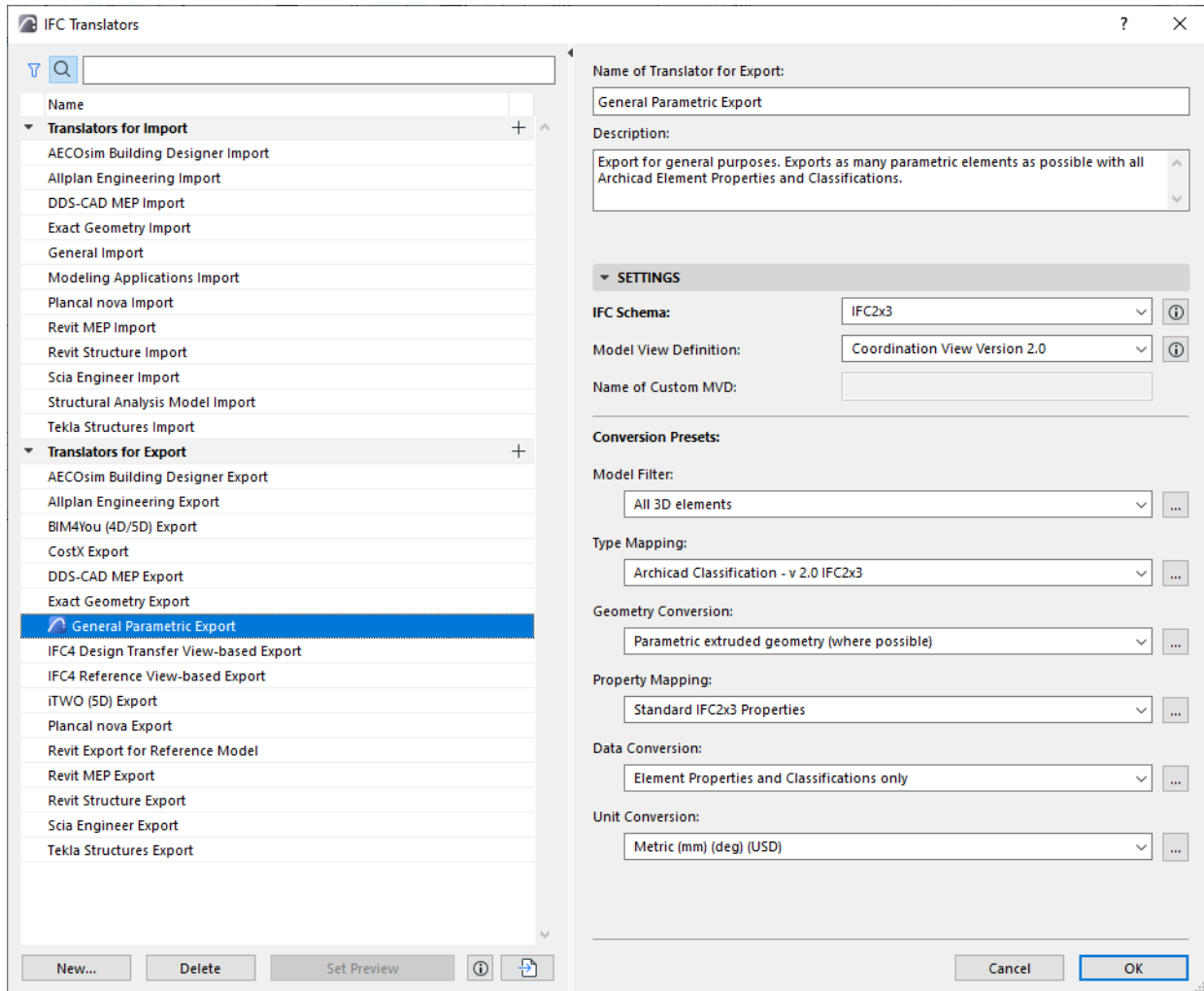
[See *Export IFC Model from Archicad and Import IFC Model to Archicad*.](#)

Important Note on Translators and Template Files

- When you import an IFC project using **Open** or **Hotlink IFC**: You will choose an IFC Translator for Import from an external Template file, selected by you.
- When you import an IFC project using the **Merge** command: You will choose an IFC Translator for Import from the current host project.

IFC Translators Dialog Box - Overview

The **IFC Translators** dialog box (**File > Interoperability > IFC > IFC Translators**) allows you to view or modify Translator settings, or to create new Translators.



Translators are listed by name on the left side. The Translator name tells you what it is for: e.g. “Import from Modeling Applications” or “Export to Tekla Structures.”

They are divided into **Translators for Import** and **Translators for Export**. The two groups of translators are not interchangeable, since they contain different settings.

- As needed, use the Search and Filter buttons at the top of the Translator list to locate a Translator.
 - The Filter button enables you to limit the list by IFC Schema: IFC2x3, or IFC4

[See IFC Schema \(Translators for Export only\) for more information.](#)

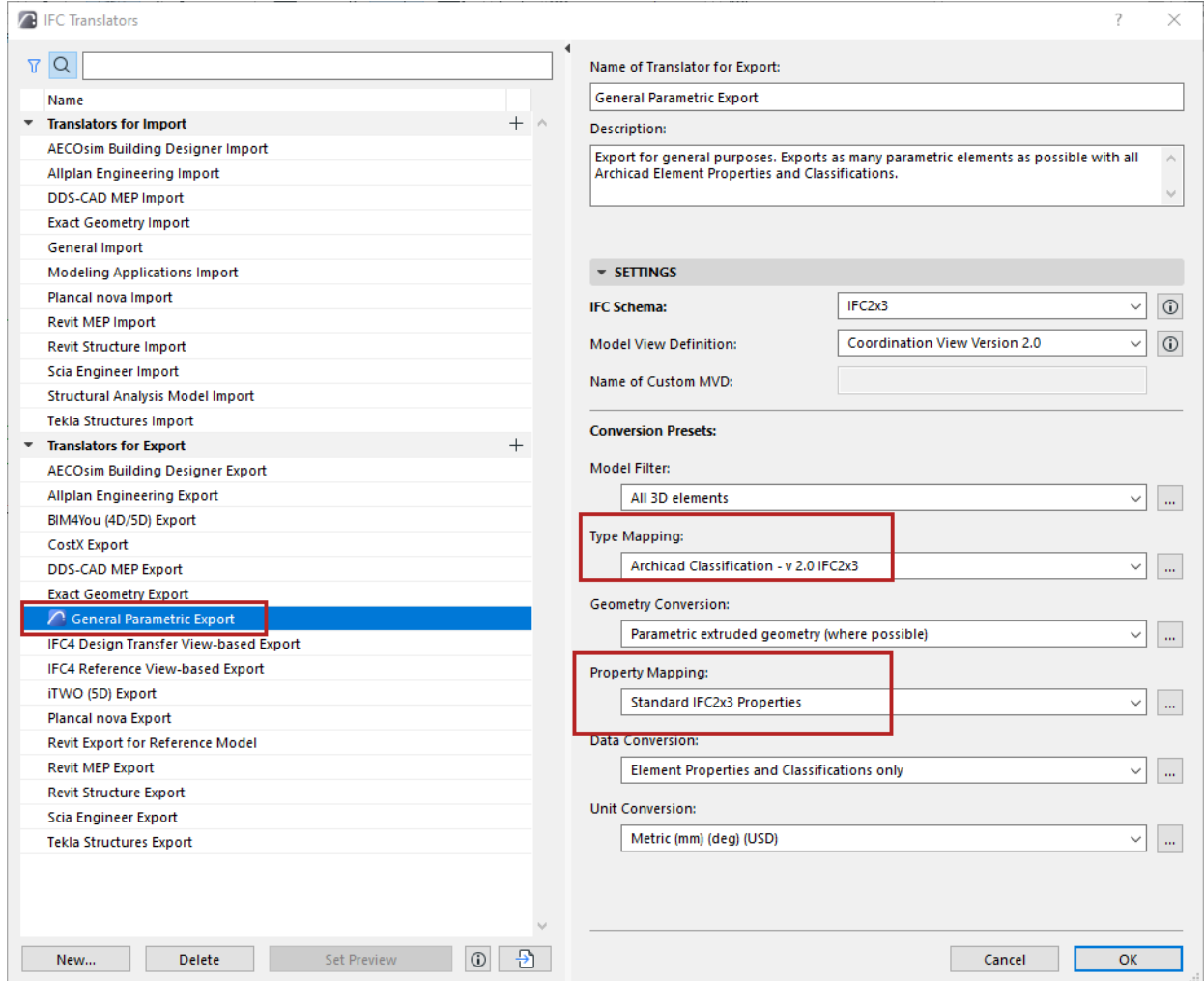
For complete details on all Translator settings, see the following sections:

[Translators for Import \(Detailed Settings\)](#)

[Translators for Export \(Detailed Settings\)](#)

Preview Translator

One of the Translators for Export is the Preview Translator, marked with the Archicad icon:



The Preview Translator also provides default mapping definitions needed to work with IFC data in Archicad. These essential mapping definitions are found in two presets of the Preview Translator:

- **Type Mapping for Export preset**

The *Type Mapping preset* of the Preview Translator assigns an IFC Type to each exported Archicad element. The IFC Type, in turn, affects which IFC properties are available to the element.

See also [Type Mapping for IFC Export](#).

- **Property Mapping for Export preset (formerly IFC Scheme Settings)**

Based on IFC Type, the *Property Mapping preset* of the Preview Translator provides the set of IFC Properties used for IFC data-related functions in Archicad. For example, you can find/select/list IFC Properties in the following parts of the Archicad interface:

- criteria in Find & Select
- criteria in Interactive Schedules
- Labels (e.g. Classification and Properties Label, ID and Properties Label)
- IFC Project Manager
- Element Settings (Classification and Properties panel)

All these interfaces use the Property Mapping that is set in the Preview Translator.

[See also Property Mapping for IFC Export.](#)

Use the Preview Translator:

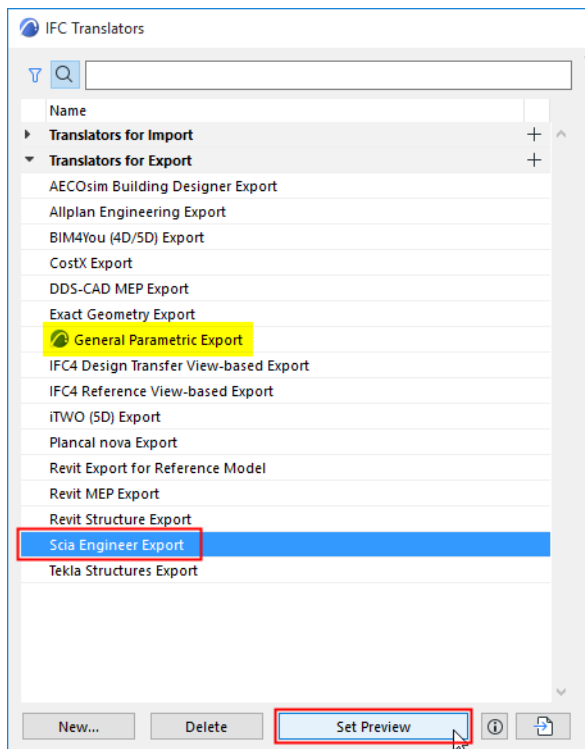
- to export the model using the Properties visible in the current Archicad model
- to see the IFC Properties in Archicad as they are used in a particular IFC standard. For example, to work with the IFC Properties as defined by the COBie 2 Export standard for the US, go to the IFC Translators dialog box and switch the Preview Translator to “COBie 2 Export (US)”.

Note: The Property Mapping preset contains Translator-specific settings. In Archicad versions prior to 21, these settings were project-specific, and managed in IFC Scheme Setup.

For information on migrating legacy IFC Schemes, see the Migration Guide.

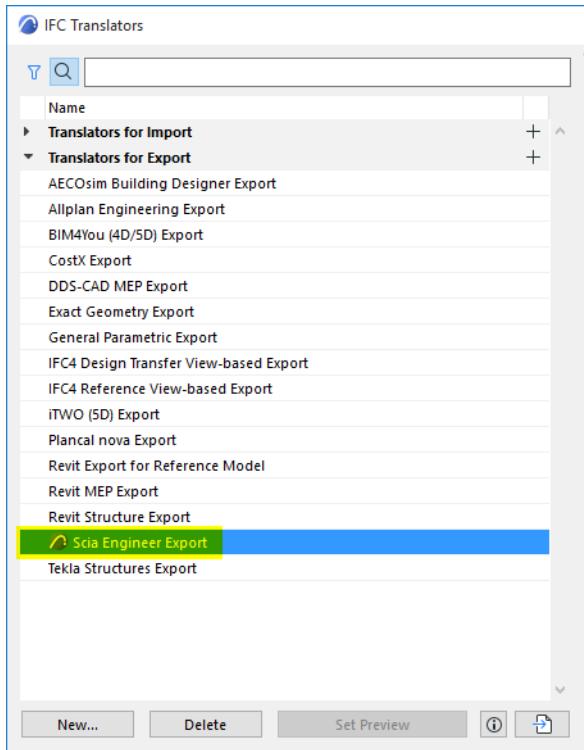
Set a Different Translator as the Preview Translator

1. Go to **IFC Translators**. (File > Interoperability > IFC)
2. Select a Translator for Export from the list.
3. Click **Set Preview** at the bottom of the dialog box.



Set Preview Translator

4. The selected Translator is now the Preview Translator, marked with the blue Archicad icon:



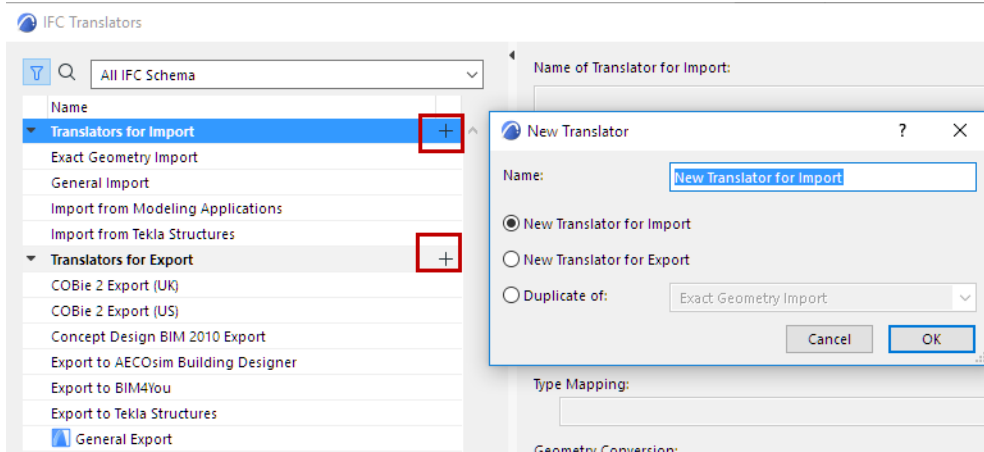
Newly Defined Preview Translator

Managing Translators

Create New/Duplicate Translator

Do one of the following:

- Click the **New** button at bottom left, or
- Click the **Plus** button at the right of the Translators group (Import or Export).



Delete Translator

Select a Translator, then click the **Delete** button at bottom left.

- Multi-selection is not available.

- At least one Translator for Import and one Translator for Export must remain - you cannot delete the last one of either group.

Note: If you delete the Preview Translator, you are prompted to select a different Translator for Export to serve as the Preview Translator. ([See also *Preview Translator*, below.](#))

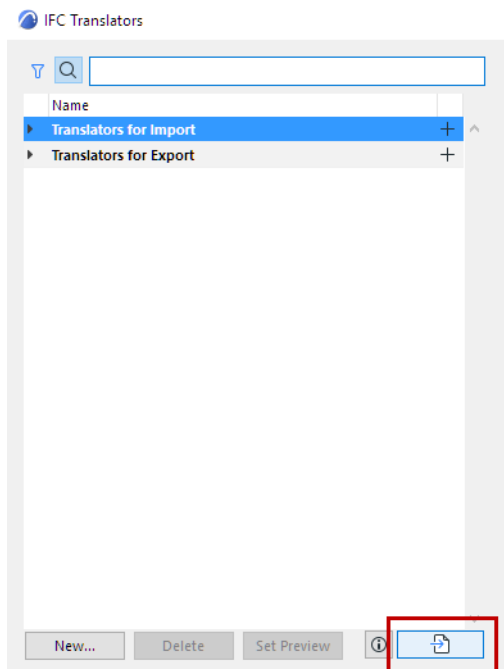
Cancel Changes in IFC Translators

In the IFC Translators dialog box, clicking Cancel will discard all changes you have made not only to the main dialog box, but also in the sub-dialogs (the specific Conversion Preset settings dialogs.)

Import Translators From Archicad Project

To import Translators from another Archicad project:

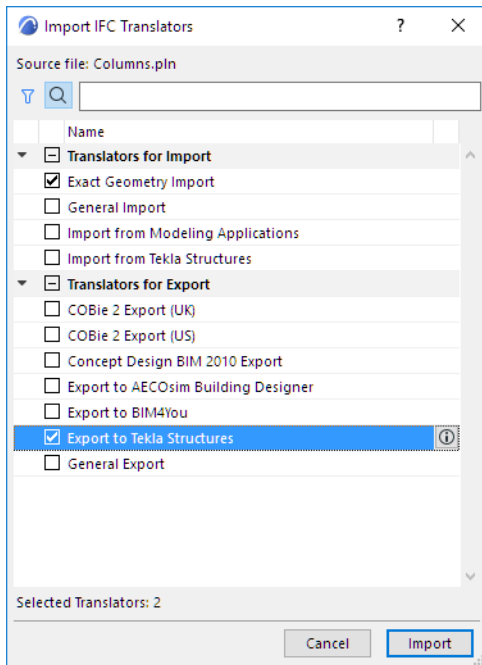
1. Click the import button at bottom left.



2. Browse for the desired file type: PLN, PLA, or TPL

Note: You can no longer import IFC Translator files in XML format. ([See *Update IFC Translators*](#).)

3. Click **Open**. In the appearing dialog, select one or more IFC Translators to import from the selected project.



4. Click **Import**. In case your project already contains Translators of the same name as the imported ones, you are prompted to choose how to manage these (Replace, Import with new file name, or skip.)

Importing Mapping Definitions

Certain IFC Translator Mapping definitions (Type Mapping for Import/Export, and Property Mapping) are also imported to your project if you use the **Import BIM Content** or **Download BIM Content** commands.

[See *Import BIM Content*.](#)

[Watch the Video](#)

[Watch the Video](#)

Translators for Import (Detailed Settings)

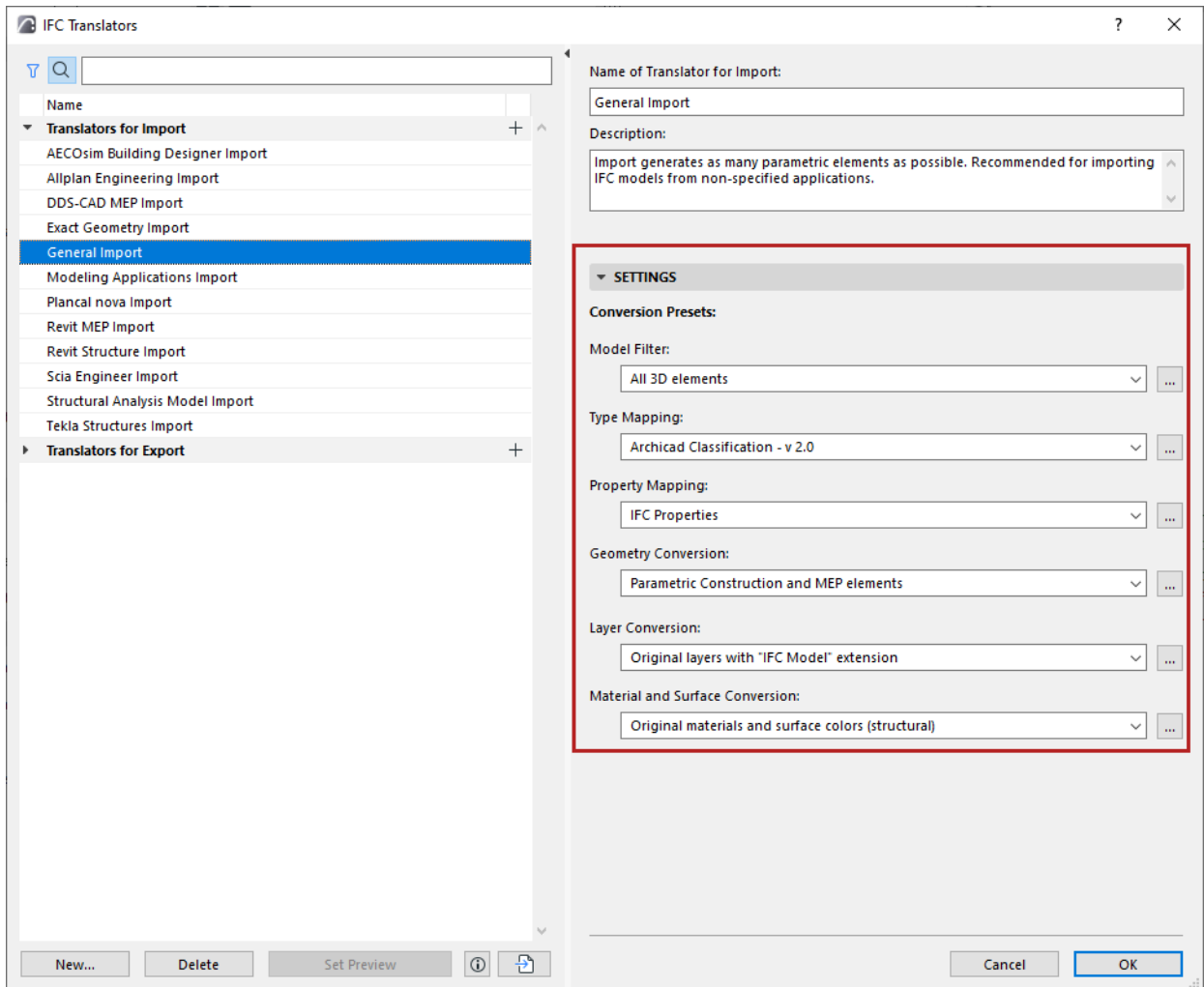
An IFC Translator for Import defines the rules for handling model elements when importing an IFC-based file to Archicad.

For an overview of this topic, see [Overview of IFC Import: Defining how IFC Data are Converted to Archicad](#).

The **IFC Translators** dialog box (**File > Interoperability > IFC > IFC Translators**) allows you to view or modify Translator settings, or to create new Translators.

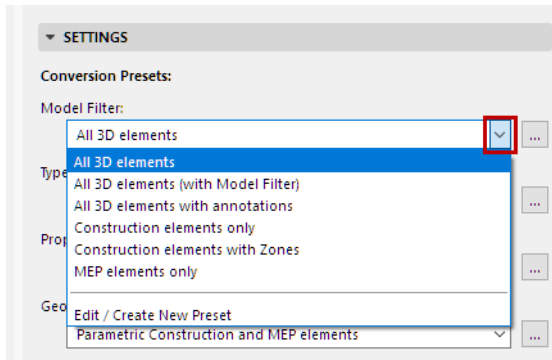
Select a **Translator for Import** from the list.

The selected Translator's settings, divided into six Conversion Presets, are shown on the right side.



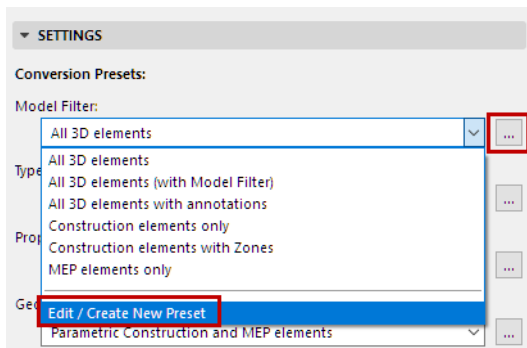
Working with Conversion Presets

For any category of Conversion Preset, click the dropdown list to view or select a different Preset. The Preset name gives you an idea of how it will affect the imported IFC model, if this Preset is used in a given IFC Translator for Import.

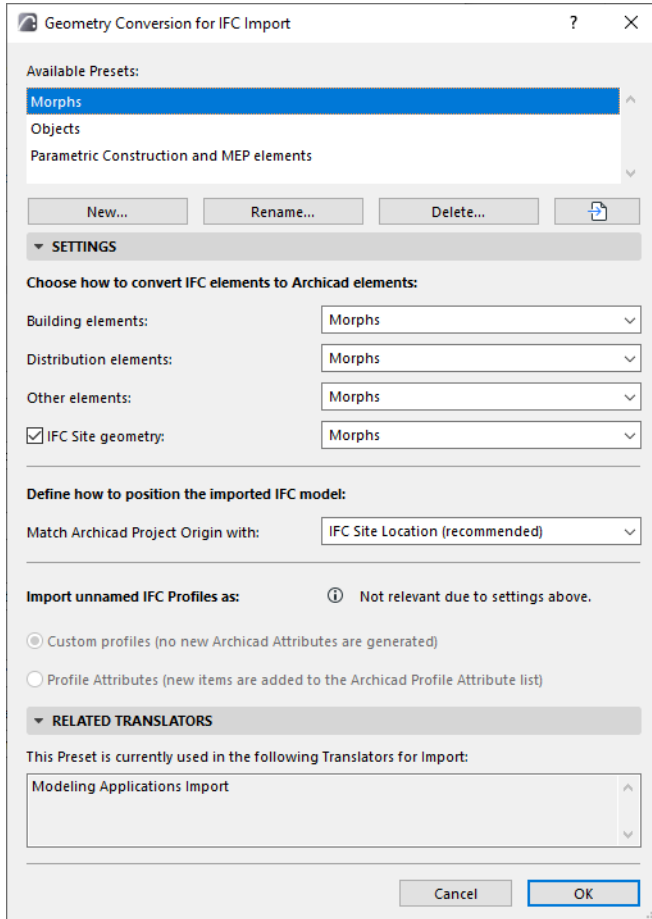


Choose a different preset from this list if needed.

To adjust settings or create a new preset, select **Edit/Create New Preset**, or click the three buttons at the right of the Preset.



The Preset dialog opens (for example, the Geometry Conversion for IFC Import).



For each of the IFC Import Preset dialogs, view the following:

- **Available Presets** for this Conversion process (e.g. Geometry Conversion)
- **Settings** (this is where the import is defined in the greatest detail, for various data types, as defined for the currently selected preset at the top)
- **Related Translators** (all the currently defined IFC Translators for Import which use this Preset)

All Settings of each type of Conversion Preset (used by IFC Translators for Import) are detailed in the following sections.

[Model Filter for IFC Import](#)

[Type Mapping for IFC Import](#)

[Property Mapping for IFC Import](#)

[Geometry Conversion for IFC Import](#)

[Layer Conversion for IFC Import](#)

[Material and Surface Conversion](#)

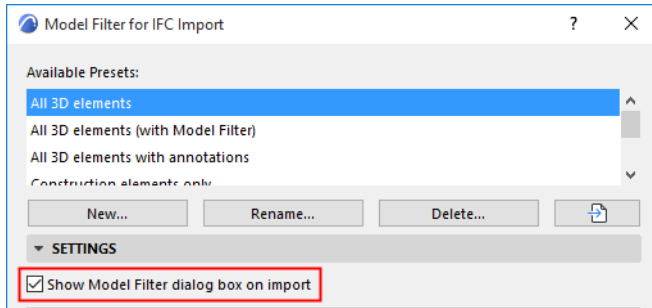
[Replace missing Renovation Status](#)

Model Filter for IFC Import

File > Interoperability > IFC > IFC Translators

Choose a Translator for Import and open the Model Filter preset.

Show Model Filter on import



If this option is active, then - after you issue an Open or Merge IFC command - the Model Filter window will open.

This gives you the chance to overwrite, for the purposes of the current import process, the elements to be imported.

See [Filter Model at Import \(Secondary Filter\)](#).

Select 3D Elements to Import

- By Structural Function

Use this as an additional (element-level) filter for import purposes, which takes into account the elements' "LoadBearing" IFC Property at import.

- "Load-Bearing elements only" means that only those elements whose IFC Property "LoadBearing Element" has a "True" value will be imported from the IFC file to Archicad.

Notes:

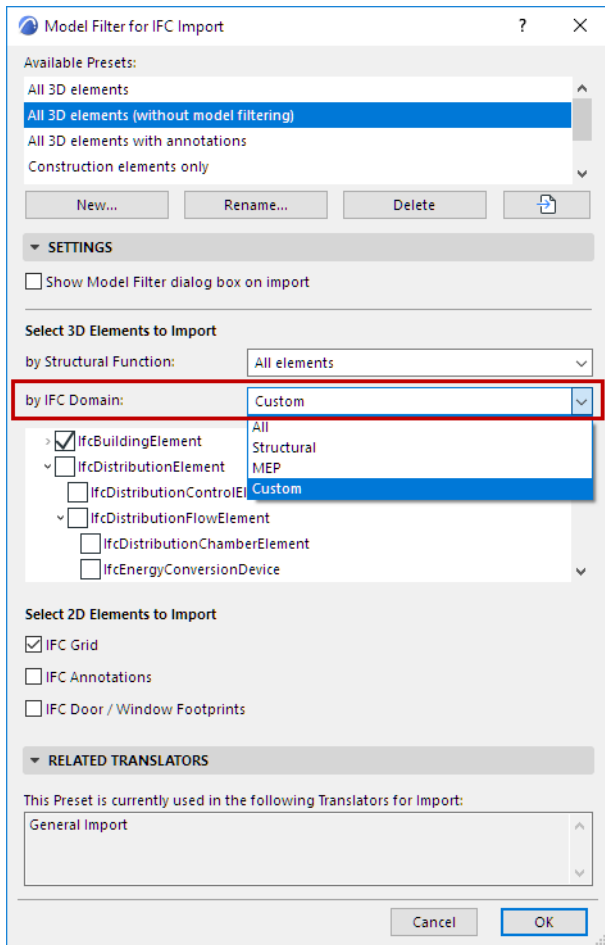
- If no Structural Function classification has taken place in the Archicad project, or if you have not finished the classifying process in Archicad, then you should choose "All Elements" here: either of the other two settings ("Load-Bearing elements only" or "Non-Load-Bearing elements only") can result an empty IFC file, or one that is missing elements you might need.
- IfcPile and IfcFooting entities are load-bearing structural elements by definition, although they have no "LoadBearing" property in the IFC2x3 Schema. They are always handled as load-bearing elements at import.

- By IFC Domain

Use the IFC Domain to filter according to the following criteria for IFC import:

- "All" includes all elements from the model
- "Structural" includes only the structural building elements (IfcBuildingElement)
- "MEP" includes only the mechanical elements (IfcDistributionElement)

See the exact composition of each Domain filter in the tree list below. Modifying this filter will create a “Custom” filter.



Select 2D Elements to Import

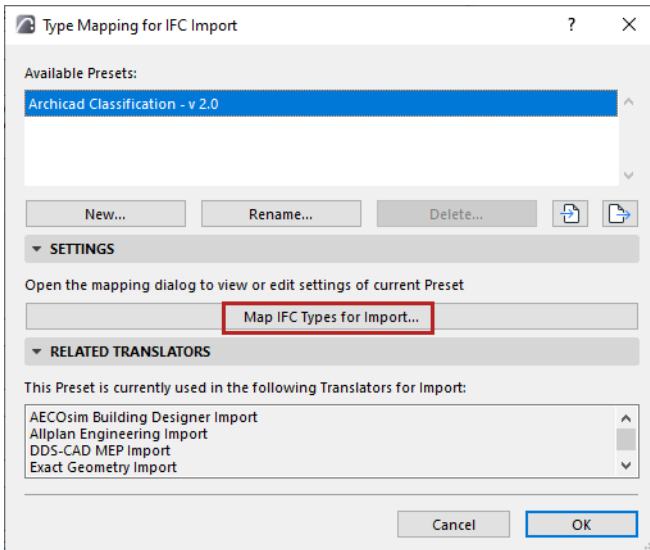
- **IFC Grid:** During import, all IfcGrid elements of the IFC file will be imported and converted into grouped Archicad Grid Elements.
- **IFC Annotations:** Check this to import all IfcAnnotation-type elements (including exploded dimension elements) of the IFC file into Archicad. They will be converted into 2D elements (texts and lines).
- **IFC Door/Window Footprints:** During import, all generated doors and windows will be displayed in floor plan views with correct opening direction and sizes, based on their imported 2D symbols

Type Mapping for IFC Import

File > Interoperability > IFC > IFC Translators

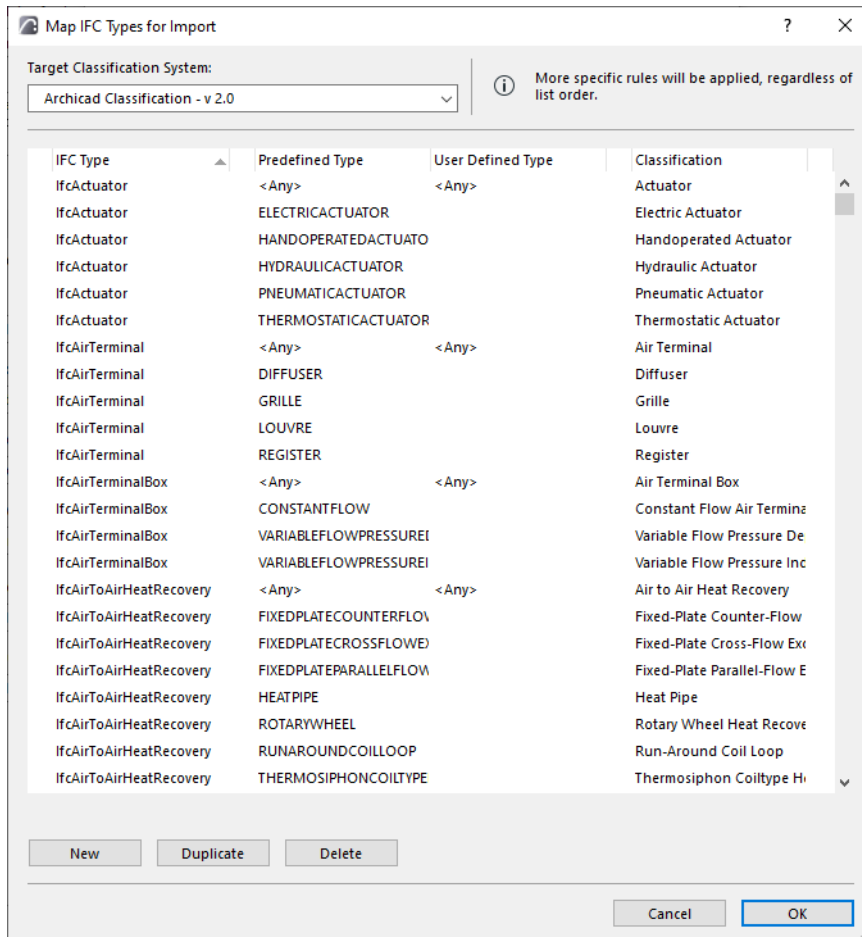
Choose a Translator for Import and open the Type Mapping preset.

From this preset dialog, click **Map IFC Types for Import** to access the mapping definitions.



Use this chart to define how to classify IFC elements imported to Archicad, based on their IFC Type.

At the top (“Target Classification System”), choose a Classification system in Archicad whose definitions you want to use.



The IFC Type (far left column) is mapped to the Classification (far right column) as defined here. You can further narrow the mapping according to Predefined Types (if any) and User Defined Types.

“Any” means that the mapping does not take the Type definition into consideration when assigning the classification in Archicad, even if the imported element has such a definition.

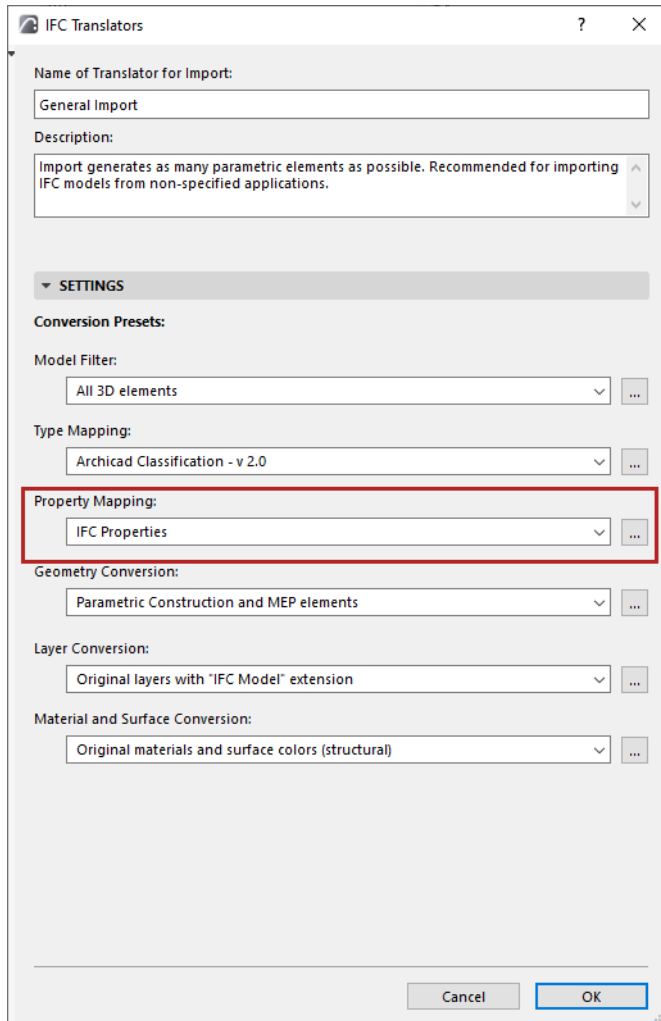
[Watch the Video](#)

Property Mapping for IFC Import

File > Interoperability > IFC > IFC Translators

Choose a Translator for Import and open the Property Mapping preset.

Use this Conversion Preset to define how to handle the properties of IFC elements imported to Archicad.



Related topics:

[Choose Import Method](#)

[Using the Mapping Table](#)

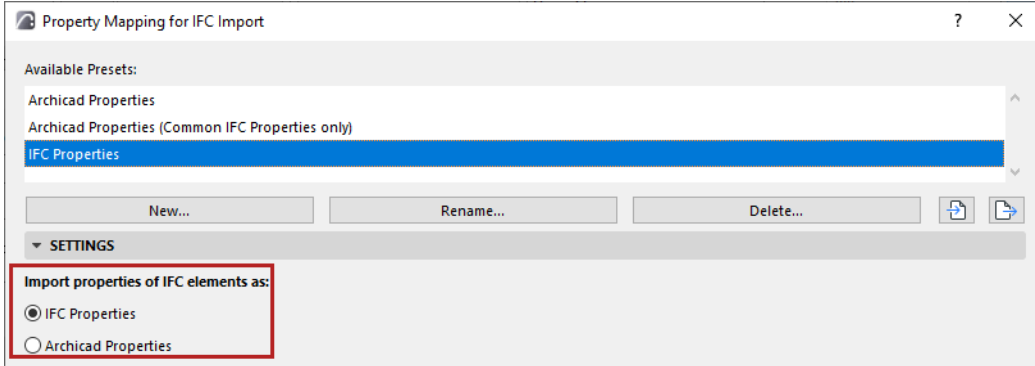
[Assigning Property Values in Archicad \(Based on IFC Property Mapping\)](#)

[Importing IFC Data to Archicad Categories](#)

[Replace missing Renovation Status](#)

Choose Import Method

Choose either **IFC Properties** or **Archicad Properties**:



Import as IFC Properties

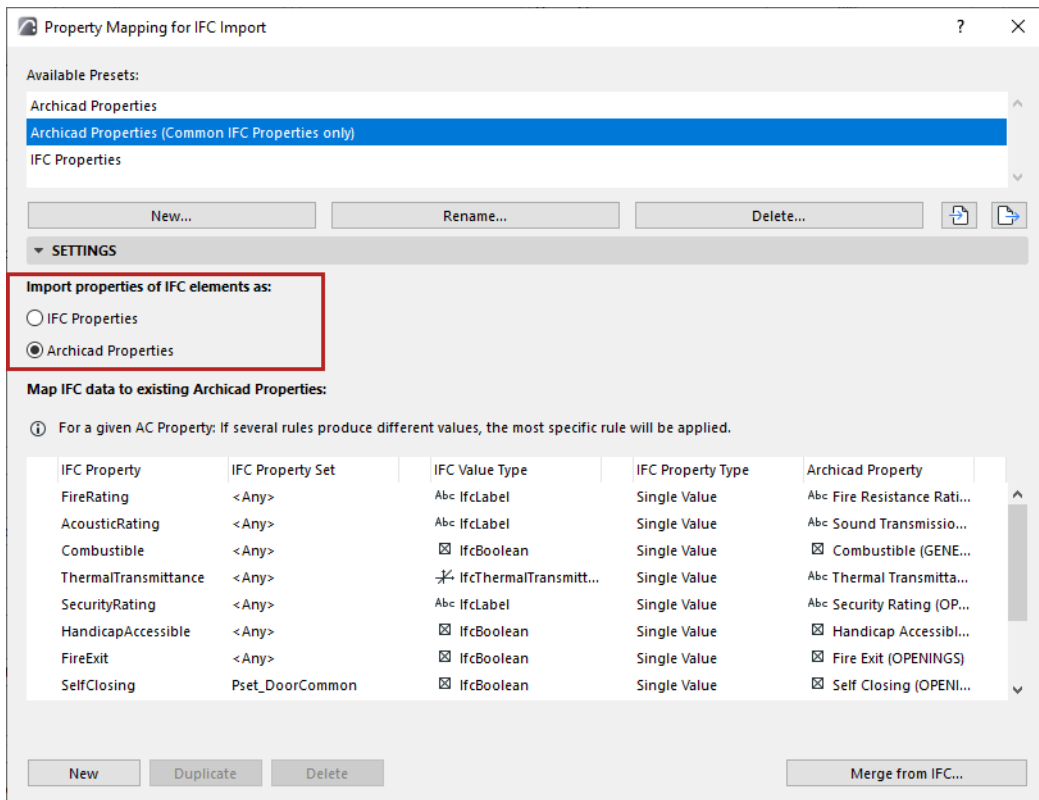
- This is the default option: Archicad imports all properties of the imported IFC elements.
- These data will now be available as IFC Properties in various parts of the Archicad interface, such as Find & Select and Scheme Settings of element lists. (This is how IFC properties were imported in earlier AC versions.)
- If you choose this, you do not have to use Property Mapping.

Import as Archicad Properties

This option can be useful if you wish to make certain IFC property data available for Archicad functions which use Archicad Properties (but not IFC Properties). These functions include:

- Graphic Overrides
- Expression-defined Properties
- Collision Detection
- Autotext Labels

This option activates the mapping table. Use the table to define which IFC property values to import to which existing Archicad Property.



- Each IFC Property in the list (defined using the first four columns from the left) is mapped to an Archicad Property of the current project (far right column).
- No new Archicad Properties are generated in the current project.
- IFC properties not listed in this mapping table are ignored during import.

Remember:

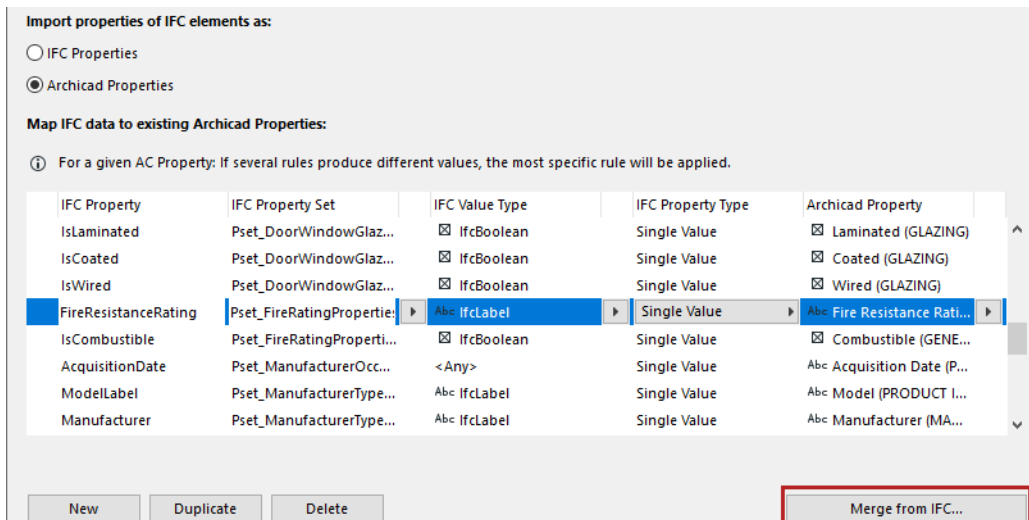
- The set of Archicad Properties available to any element in Archicad depends on the element’s Classification.
- Each Archicad Property has a default value (assigned in Property Manager).
- For elements imported from IFC, this mapping table lets you change their Archicad Property value from default to a custom value, based on the applicable IFC data if available.

Using the Mapping Table

- Each row represents a mapping rule. You can sort the rows by clicking the column headers.
- Click on any rule, then use the list item controls to set data types or to enter custom text.
 - In each rule, use the four IFC data columns - Property, Property Set, Value Type, and Property Type - to narrow the set of IFC data to be considered. (These are all IFC standard definitions.)

To add IFC Properties to the mapping list:

- Click **New**, or
- Click **Merge from IFC** and browse for an IFC file. The file's IFC properties will be appended to the list.



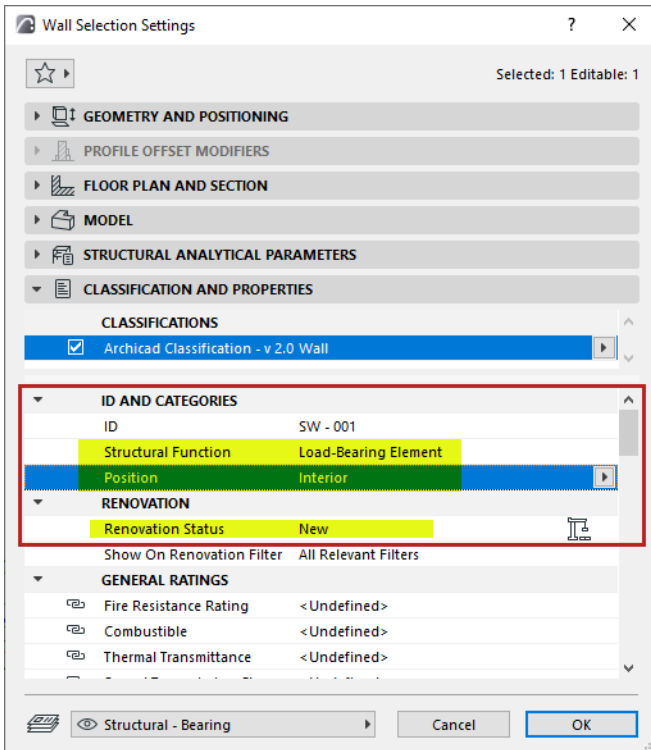
- Use the **New**, **Duplicate** and **Delete** buttons to create and delete mapping rules.

For details and examples, see [Assigning Property Values in Archicad \(Based on IFC Property Mapping\)](#)

Importing IFC Data to Archicad Categories

The following Archicad Categories use the corresponding data from imported elements. (These are Categories, rather than Archicad Properties.) This data is filled in automatically, without reference to IFC Translator settings.

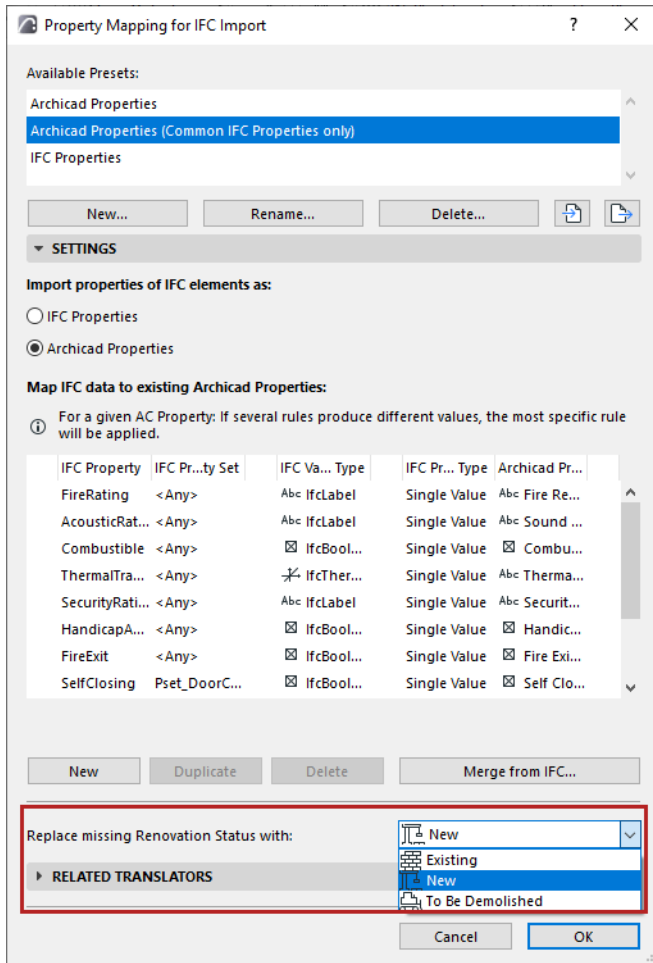
- Renovation Status
- Structural Function
- Position



You can also map these IFC values manually, in the mapping table, to any Archicad Property.

Replace missing Renovation Status

Choose a Renovation Status in Archicad (New, Existing or Demolished) for imported elements that do not have an assigned renovation status in IFC.



Notes:

- Nearly every model exchange scenario will lack automatic mapping of renovation status. The reason is that the IFC2x3 standard has no standard format for renovation status, and so the various applications will export this data only as custom IFC properties.
- In model exchange with Revit, the Archicad “Renovation Status” item is mapped to Revit’s “Phasing” item, provided that the IFC export and import on the Revit side is carried out using the [IFC Model Exchange with Archicad Add-In](#).

Assigning Property Values in Archicad (Based on IFC Property Mapping)

For general information on setting up the Property Mapping preset: see [Property Mapping for IFC Import](#).

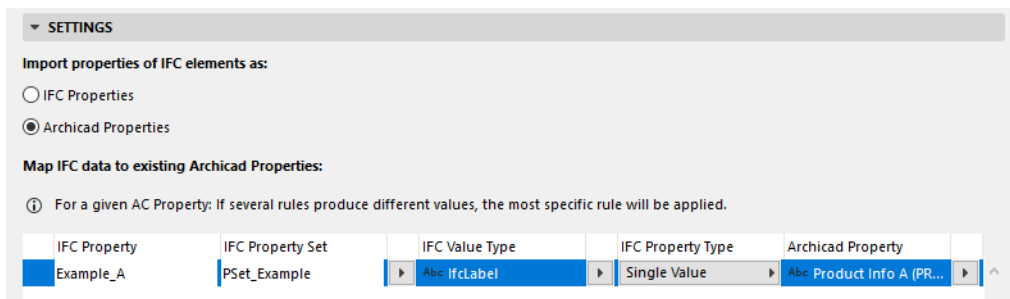
For each imported element, Archicad attempts to execute the applicable mapping rules.

A mapping rule is applicable if the Archicad Property is available to the element (based on its Classification). The value of the mapped IFC property is assigned to the element's corresponding Archicad Property. (It is a custom value.)

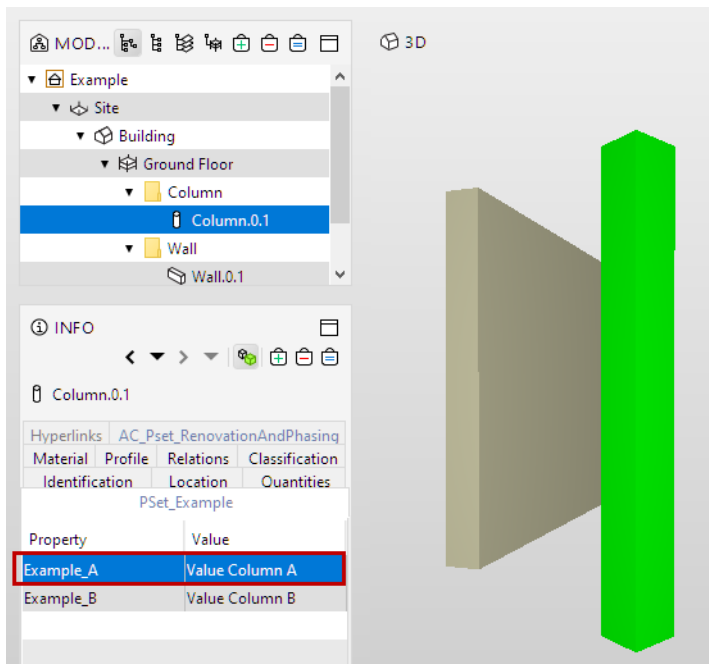
Example: Mapping Rule Applied

We have set up a mapping rule in the Property Mapping for Import preset of the current IFC Translator.

According to this rule, the IFC property called Example_A is mapped to the Archicad Property called Product Info A.

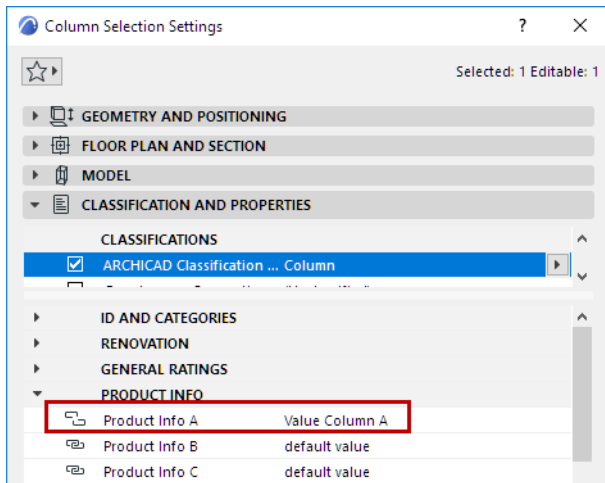


A Column, with an IFC data called "Example_A", is imported from an IFC file.



Element's Data in Source IFC File (as seen in Model Viewer)

The mapping rule is successfully applied: the Column element generated during import has a property called Product Info A, which is assigned a custom value (in Archicad Element Settings, the Property value is displayed with the unlinked icon - it is no longer linked to its default value, set in Property Manager.)



Target Archicad Property with Custom Value, after Element Import from IFC

If a mapping rule refers to IFC property data that do not exist for the incoming element, then the element's Archicad Property value becomes Undefined.

Incompatible Data Types

- If the mapped IFC element has a value, but it cannot be interpreted by the corresponding Archicad Property (e.g. incompatible data types): no custom Property value is assigned. The element uses the Archicad Property's default value.

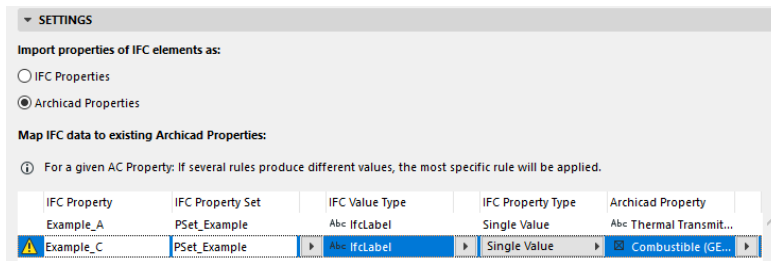
Note on Data Types:

- IFC and Archicad use different data type definitions: [See Data Types for Properties in IFC and Archicad](#). When you map an IFC Property to an Archicad property, their data types do not necessarily have to coincide. Archicad attempts to convert the data.
- If data conversion is not possible, the Session Report provides details on which elements' IFC data could not be converted.

Example

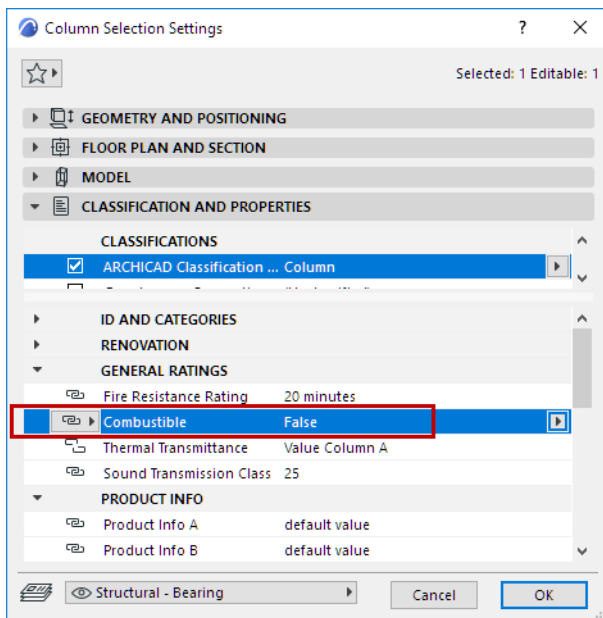
Two mapping rules are shown. One of these maps the "Example_A" IFC data (a string-type IfcLabel) to the Thermal Transmittance Archicad Property, which is a String-type value. The mapping rule is successfully applied: the element generated during import shows a String-type custom value for the Archicad Property of Thermal Transmittance.

The second mapping rule maps the “Example_C” IFC data to the Combustible Archicad Property, which is a True/False type value. The data types do not match, so the mapping rule gets a Warning icon.



Mapping Rule to “Combustible” Property Referencing Incompatible Data Type

The targeted Archicad Property does not get any custom value: it remains linked to its default value.



Property Mapping with Incompatible Data Types: No Change to Archicad Property Value

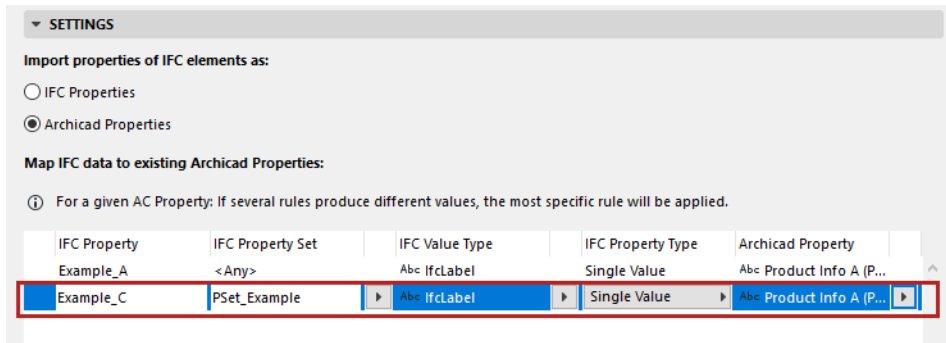
- If no mapping rule applies to an element’s given Property, then no custom Property value is assigned. The element uses the Archicad Property’s default value.

Conflicting Mapping Rules

The mapping table may include several IFC Properties mapped to the same single Archicad Property. In this case, Archicad will use the most specific rule. (The order in which the rules appear in the mapping table does not matter.)

A rule becomes more specific as its IFC data properties are more narrowly defined. Within a mapping rule, each field to the right further narrows its definition.

Here, two IFC data (Example_A and Example_C) are mapped to the same Archicad Property. In this case, the more specific rule is automatically applied. Example_C is more specific, since it is narrowed down at the level of IFC PropertySet.

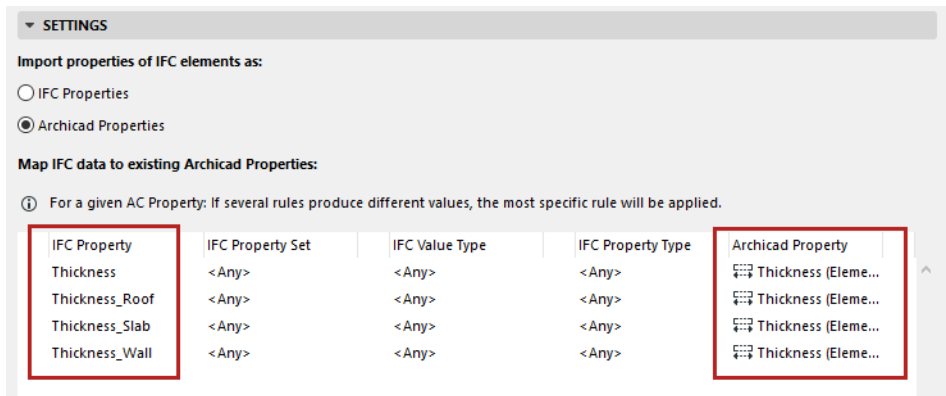


Multiple IFC Properties mapped to single Archicad Property

In case of two or more equally specific mapping rules: the rules are ignored, and no mapping takes place.

Several Thickness-related IFC Properties are mapped to a single Thickness Property in Archicad. If an imported IFC element comes with multiple Thickness properties, then Archicad has no way of determining which Property value to use in Archicad.

The mapping rules are *ignored* for this imported element, and its Archicad Property value remains the default one.

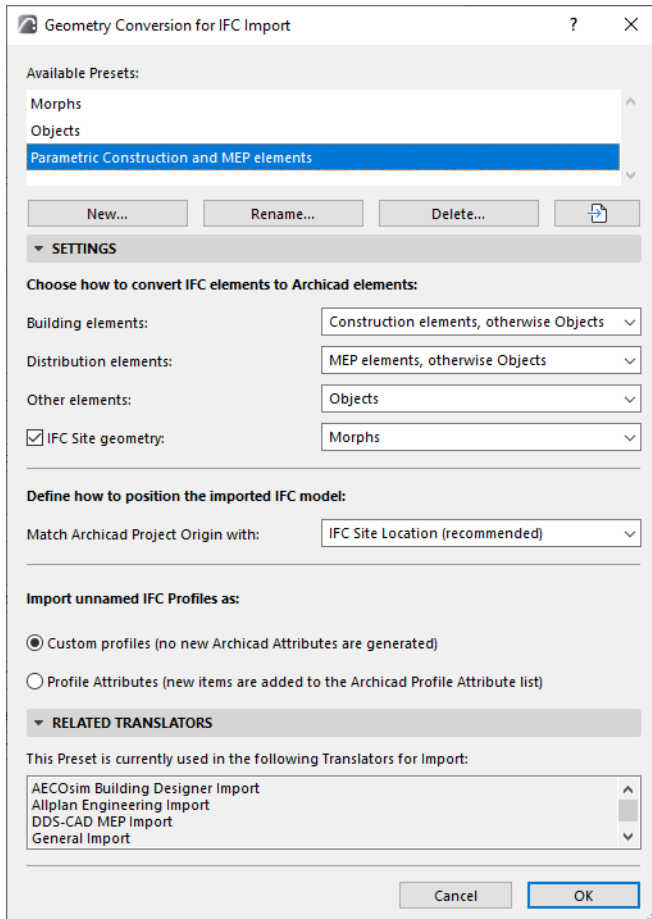


Geometry Conversion for IFC Import

File > Interoperability > IFC > IFC Translators

Choose a Translator for Import and open the Geometry Conversion preset.

Use this dialog box to define how to convert the geometry of IFC elements imported to Archicad.



Building Elements

Convert to:

- Construction elements, otherwise Objects
- Construction elements, otherwise Morphs

With either of these options, IFC elements whose supertype is IfcBuilding (IfcWall, IfcColumn, IfcBeam, etc.) are imported as editable Archicad elements (“Construction elements”: Wall, Column, Beam, etc.)

The advantage of converting the imported elements to AC tool-based construction elements is that they remain parametric, and Archicad functions (such as automatic intersections) will apply to them.

Even if you choose the “Construction element” option, some elements cannot be imported in this form: such as non-extruded (BREP geometry) elements, or elements having no counterpart in Archicad.

For such elements, you can choose to convert them as either an editable Morph or an Object (“otherwise Objects/otherwise Morphs”).

- Morphs

- Objects
 - Conversion of elements to Morphs or Objects guarantees perfect geometry import.
 - The resulting Morphs will be editable.
 - The resulting Objects will be Library Parts, whose subtype will match the IFC entity type: for example, an IfcColumn converted to an Archicad Object will have the subtype “Column”.

Distribution elements

This option applies to imported elements whose domain is MEP (IfcDistributionElement).

These elements can be converted to one of three results:

- Morphs
- Objects
 - Conversion of elements to Morphs or Objects guarantees perfect geometry import.
 - The resulting Morphs will be editable.
 - The resulting Objects will be Library Parts, whose subtype will match the IFC entity type: for example, an IfcColumn converted to an Archicad Object will have the subtype “Column”.
- MEP Elements: this option generates parametric MEP elements from the project’s MEP elements. Even if you choose this option, the MEP IFC model might not contain sufficient data to generate the parameters, in which case non-parametric MEP Equipment elements will be generated, or - in case of limited data - Objects will be generated.

Other elements

This option defines how to convert elements that are neither building elements nor distribution elements (such as furnishing and transportation elements).

- Morphs
- Objects
 - Conversion of elements to Morphs or Objects guarantees perfect geometry import.
 - The resulting Morphs will be editable.
 - The resulting Objects will be Library Parts, whose subtype will match the IFC entity type: for example, an IfcColumn converted to an Archicad Object will have the subtype “Column”.

IFC Site geometry

Check this box if you want to include the geometry of the imported IfcSite.

Define whether the converted element should be an editable Morph or an Object. (IfcSite geometry cannot be converted to an Archicad Mesh.)

Define how to position the imported IFC model

Match Archicad Project Origin with:

- IFC Site Location (Recommended)
- IFC Global Origin

Import Unnamed IFC Profiles

Choose how to handle unnamed IFC profiles imported to Archicad:

- Custom Profiles (this option does not create any new attributes in the Archicad project)
- Profile Attributes (new Profile attributes are added to the Archicad project)

Notes on Geometry Conversion

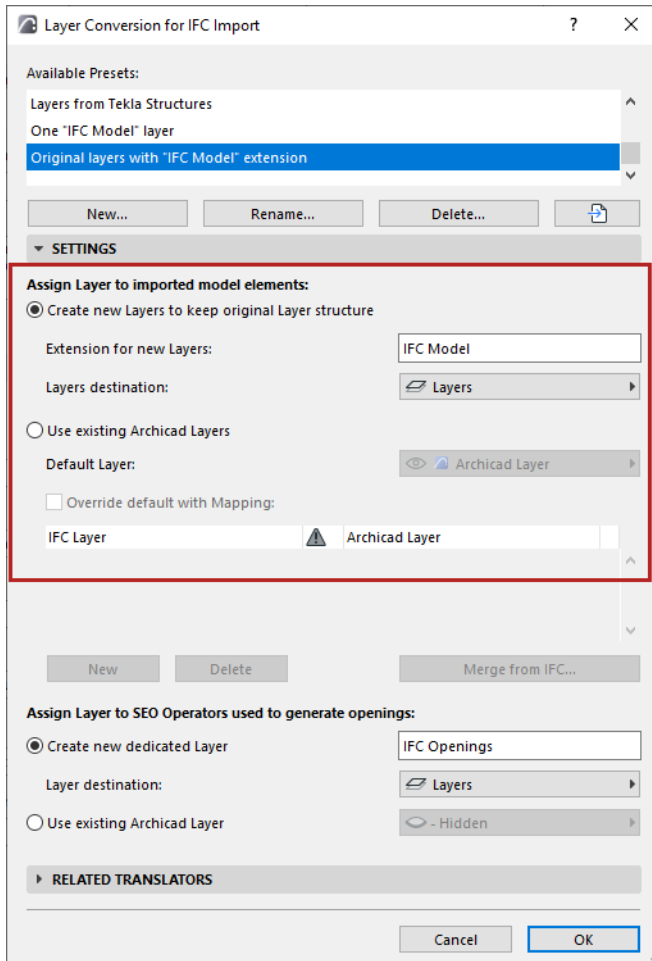
- To optimize the management of shared geometry, Archicad creates as few geometric elements (when generating Morphs) or as few Library Parts (when generating Objects) as possible.
- Objects can be placed in the project's Embedded Library, or else saved to a given linked library ([see IFC Local Preferences](#)).
- Some IFC models may contain elements with special geometry which cannot be converted to Morphs, even if you choose the Morph conversion option. In this case, Archicad will generate Objects, to achieve perfect geometry import.
- Imported IfcSpace elements will always be generated as Archicad Zones, which will always retain the imported elements' original geometry.

Layer Conversion for IFC Import

File > Interoperability > IFC > IFC Translators

Choose a Translator for Import and open the Layer Conversion preset.

Use this dialog to define the method for placing the imported elements onto Archicad layers.



Create new Layers to keep original Layer structure

Archicad will create layers using the original layer names (e.g. IfcPresentationLayerAssignment) of the IFC model, and will place the imported elements onto these layers.

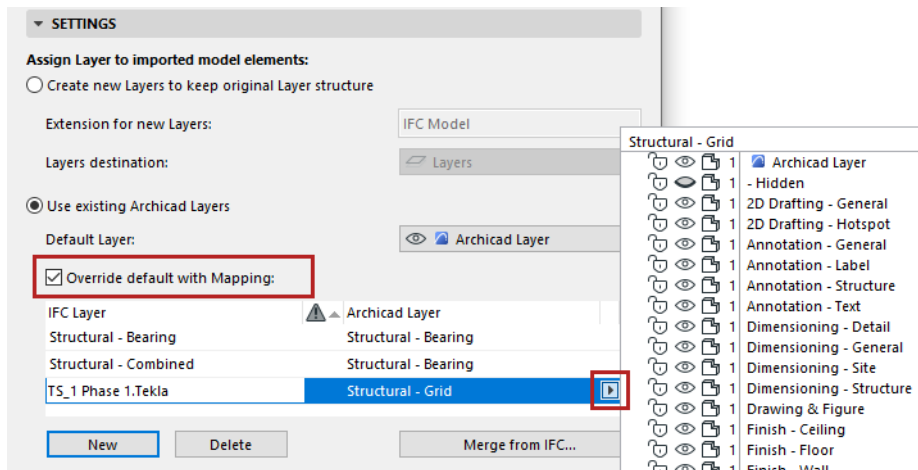
- Extension for new Layers: Add a custom extension (such as “IFC Model”) to each Layer name. This allows you to sort Layers by extension in Archicad’s Layer Settings.
- Layers destination: Choose a folder for IFC layers. Default destination is the root “Layers” folder.

Use existing Archicad Layers

From the Default Layer pop-up, choose an existing layer from the Archicad project on which to place imported elements.

Override default with mapping

Check this to map layers manually, instead of using the default Layer assigned from the pop-up.



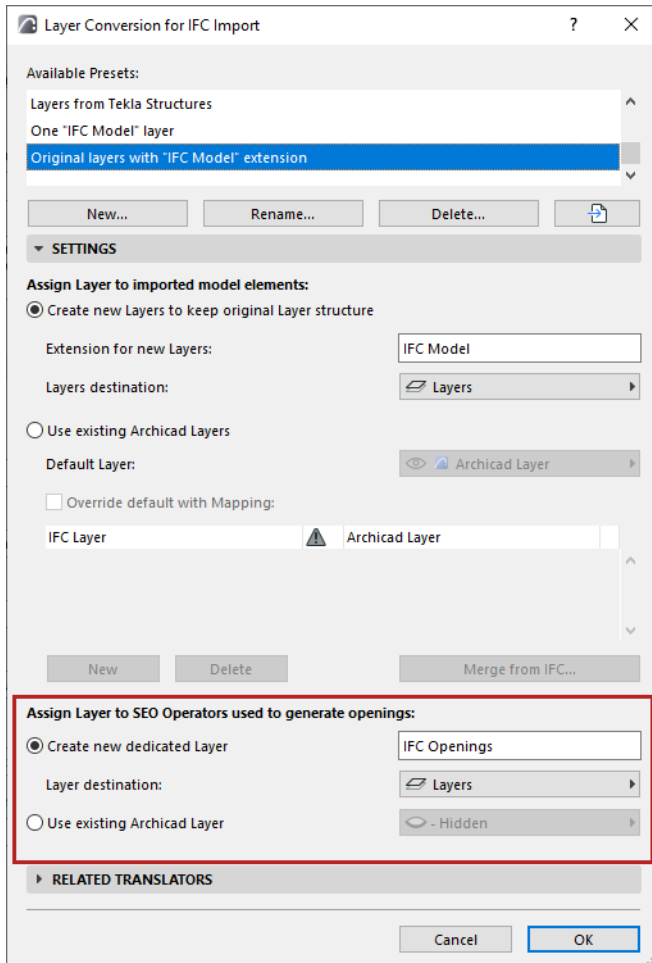
To add IFC Layer names to the mapping list:

- Click **new**, or
- Click **Merge from IFC** and browse for an IFC file. The file's layer names will be appended to the list.

Assign Layer to SEO Operators Used to Generate Openings

Some imported openings can be created in Archicad only by using Solid Element Operations. In such cases, Archicad uses operator elements to generate the openings automatically via SEO.

At IFC import, you can define the layer settings of these SEO operator elements:



- **Create new dedicated Layer:** Enter name for the new Layer.
- Layers destination: Choose a folder for the new Layer.
- **Use existing Archicad Layer:** Choose a layer from the Archicad project for SEO operator elements.
Note: If you move any of the target elements, make sure that the operator elements' layer is shown, and move the target and operator elements together: do this to avoid modifying the original geometry of the imported elements.

Notes on Layer Conversion:

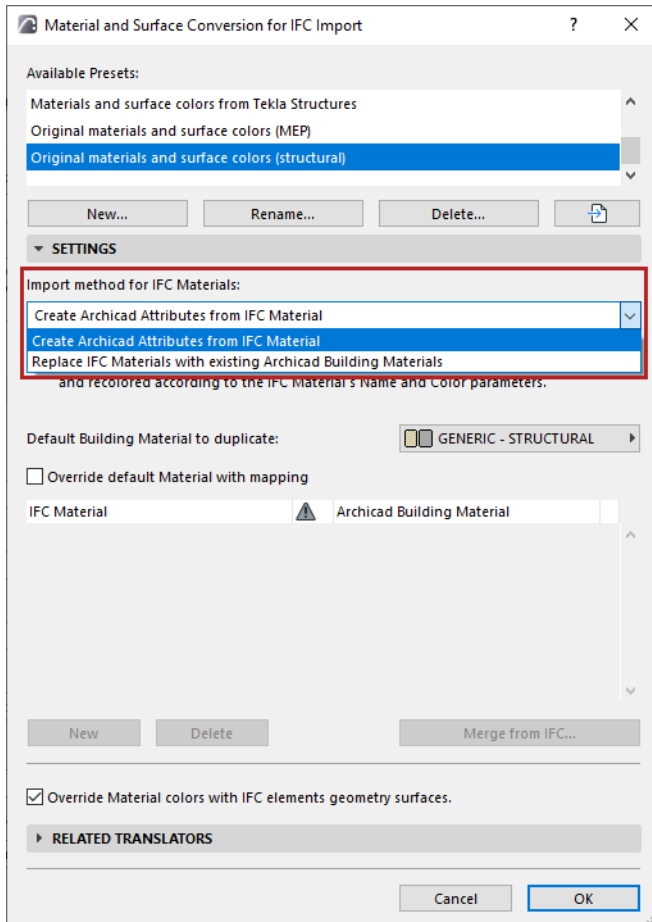
- The import process automatically sets an intersection group number for all imported layers, and this number will be different from the current project layers' intersection group number. This difference ensures that the elements on the imported layers will not intersect with the existing project elements.
- If the elements have been converted to their counterpart Archicad tool-based construction elements (that is, they have not been converted to Objects or Morphs; [see *Geometry Conversion for IFC Import* above](#)), then elements on identical layers might have their geometries modified due to automatic intersections. If you want to avoid this, set the intersection group number to 0.
- If layers of the same name already exist in Archicad, no new layers are created; the imported elements are placed onto the corresponding, existing Archicad layers.

Material and Surface Conversion

File > Interoperability > IFC > IFC Translators

Choose a Translator for Import and open the Material and Surface Conversion preset.

Use this dialog to choose how to assign Archicad Building Materials to imported IFC model elements.



There are two import methods:

Method 1: Replace IFC Materials with existing Archicad Building Materials

With this method, a single Archicad Building Material is assigned to all imported IFC elements. Choose this default Building Material from the pop-up below. No new Building Materials are generated.

If needed, you can override this default definition by using Material Mapping: [see *Override default Material with mapping*](#), below.

Method 2: Create Archicad Attributes from IFC Materials

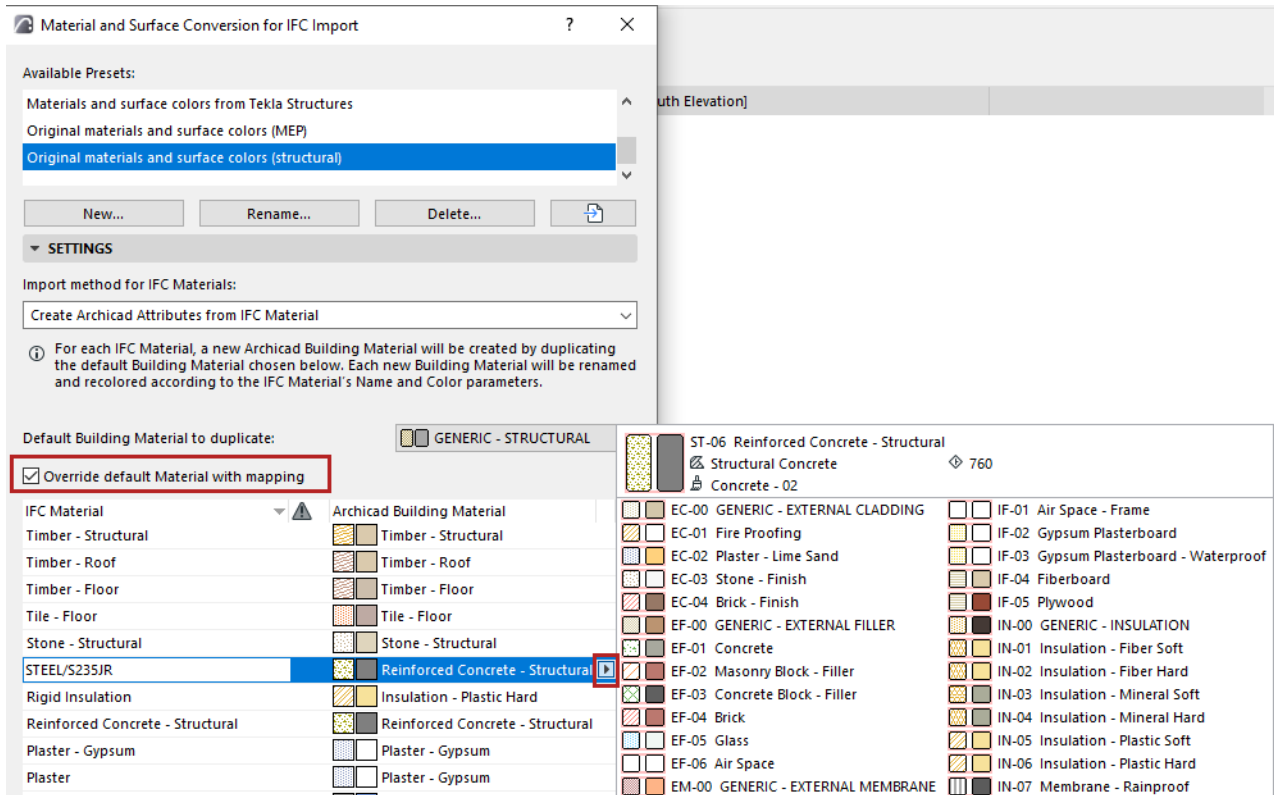
For each IFC Material, a new Archicad Building Material will be created: the default Building material chosen below will be duplicated.

The new Building material:

- is named using the IFC Material's Name parameter
- is assigned a color corresponding to the IFC Material Color parameter

Override default Material with mapping

Check this box to map individual materials manually, instead of using the default Building Material assigned from the pop-up.



To add IFC Materials to the mapping list:

- Click **New**, or
- Click **Merge from IFC** and browse for an IFC file. The file's IFC Material names will be appended to the list.

Note:

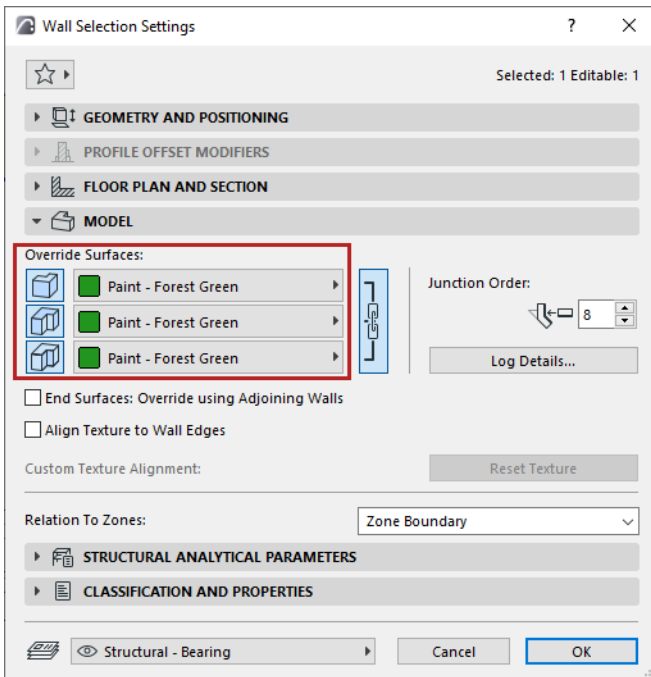
- Some of the default IFC Translators for Import include predefined mapping rules, whose settings you are free to expand or edit. It is recommended to duplicate the default translator before modifying it.
- You cannot import or export these mapping tables as separate files (e.g. xml); you must import/export them together with entire translator file itself.

Override Material colors with IFC element geometry surfaces

IFC elements in IFC models can have two colors: a material color and an additional color for the surface geometry color.

If you check this **Override Material colors** box, Archicad will create Surface attributes using the color assigned to the IFC element's surface geometry.

To check and modify the resulting surface overrides in Archicad, view the Model panel of the respective Element Settings dialog box.



Translators for Export (Detailed Settings)

An IFC Translator for Export defines the rules for handling model elements exported from Archicad to an IFC format.

For an overview of this topic, see [IFC Translators: Overview](#).

The **IFC Translators** dialog box (**File > Interoperability > IFC > IFC Translators**) allows you to view or modify Translator settings, or to create new Translators.

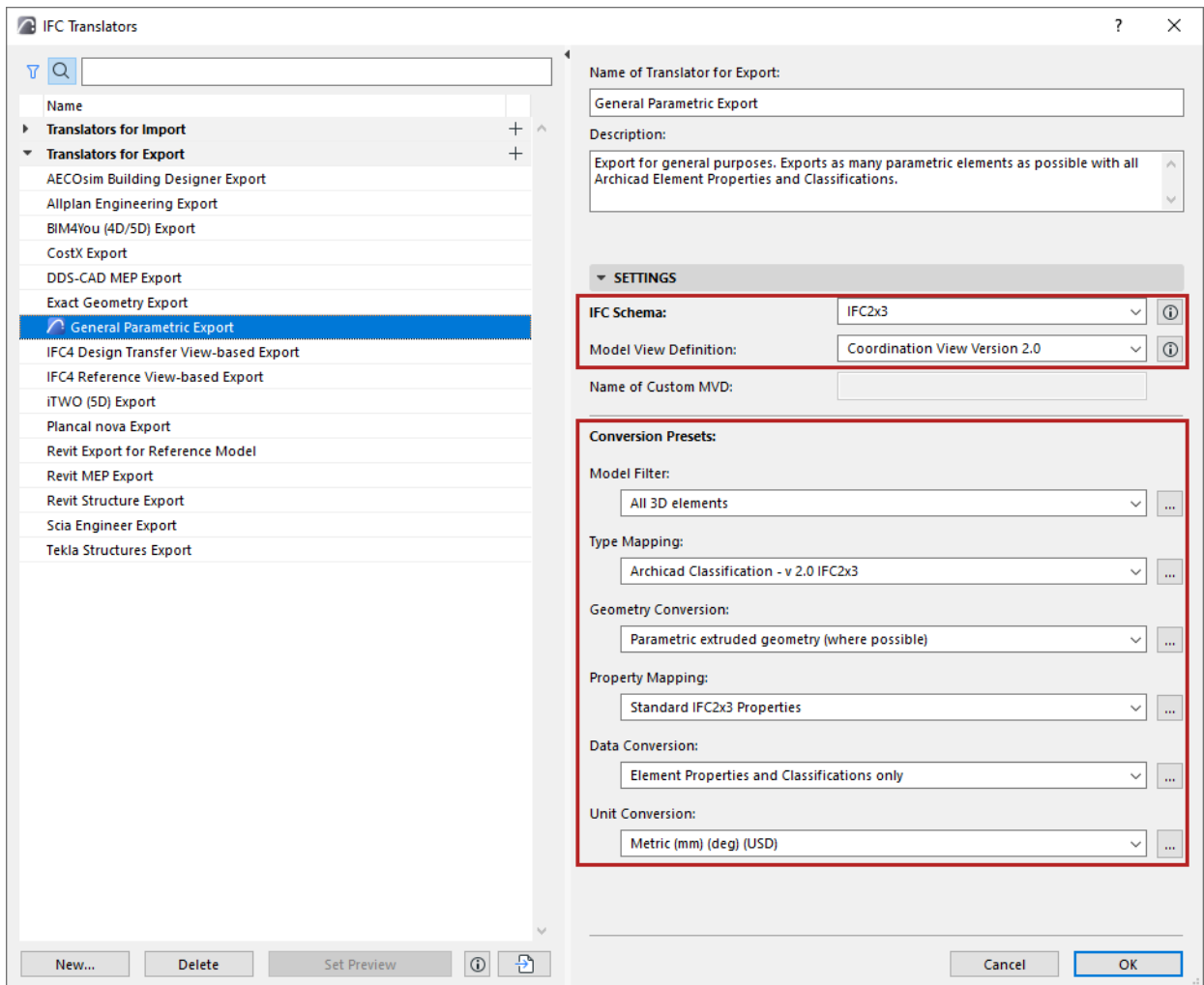
Select a **Translator for Export** from the list.

The selected Translator's settings, divided into six **Conversion Presets**, are shown on the right side.

One of the Translators for Export is the **Preview Translator**. The Preview Translator exists to provide default mapping definitions needed to work with IFC data in Archicad.

For details, see [Preview Translator](#).

The **IFC Schema** and **Model View Definition** categories help in setting up presets that are compatible with particular standards (see details below).



IFC Schema (Translators for Export only)

An IFC Schema is a particular version of the IFC standard. Archicad Translators can support either Schema IFC2x3 (recommended) or IFC4.

Note: IFC2x3 is recommended - this is the most widely used version. Choose IFC4 only if you are certain that it is supported by the application that will receive IFC data using this Translator.

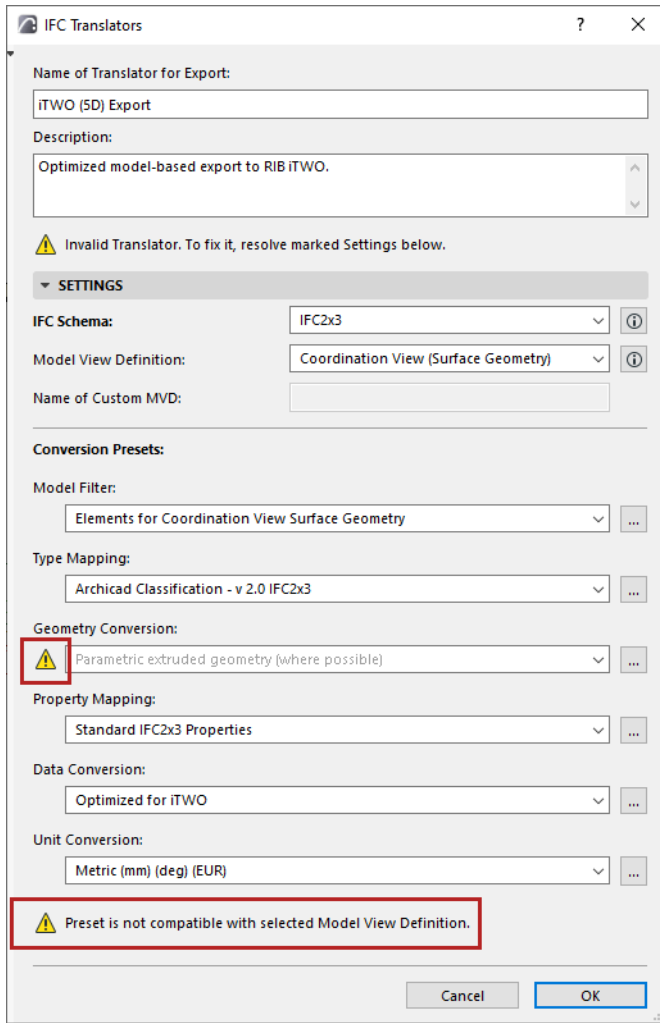
Model View Definition (Translators for Export only)

A *Model View Definition* (MVD) is a recommendation for which data and elements the IFC model should include, depending on the purpose of the model exchange. If your exported IFC model must conform to a predefined standard MVD, choose that MVD from the list.

Note: Available MVD's depend upon the IFC Schema chosen above: IFC2x3 or IFC4.

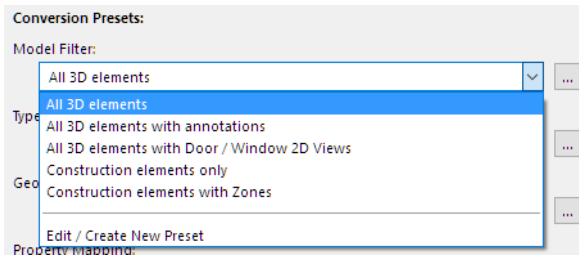
For a more detailed explanation of MVD's and their characteristics, see [Model View Definitions](#).

Once you choose an MVD, then certain Translator for Export settings must conform to the requirements of that MVD. If you change a Translator setting so that it no longer conforms to the selected MVD's requirements, then the Translator becomes invalid, and you get a warning in the dialog box. The problematic setting is also marked:

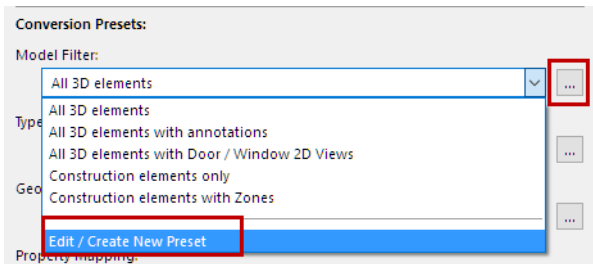


Working with Conversion Presets

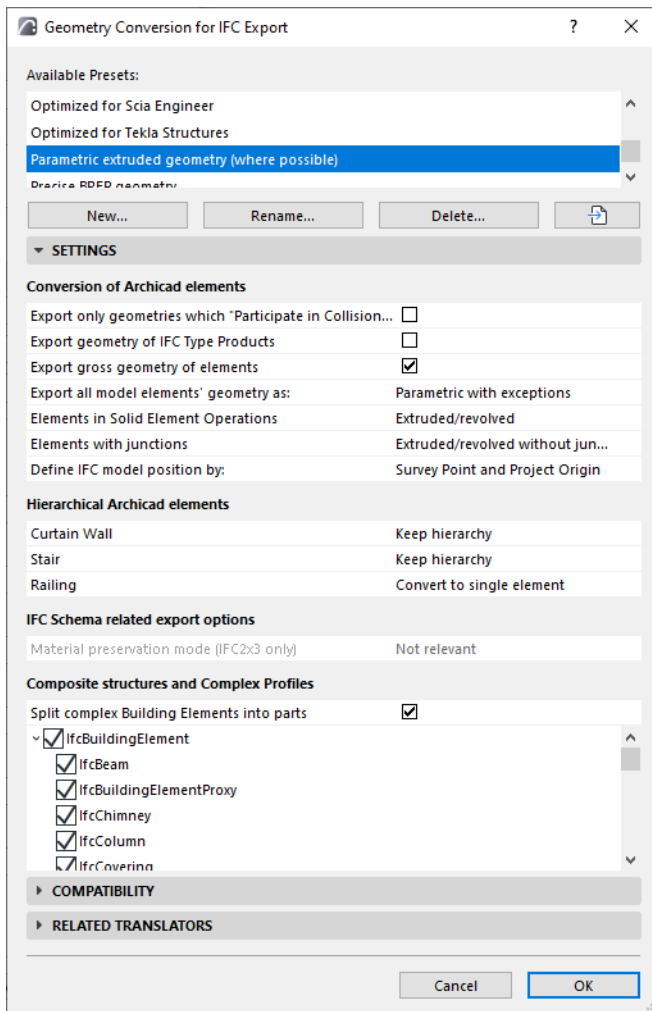
For any category of Conversion Preset, click the dropdown list to view or select a different Preset. The Preset name gives you an idea of how it will affect the exported IFC model, if this Preset is used in a given IFC Translator for Export.



To adjust settings or create a new preset, select **Edit/Create New Preset**, or click the three buttons at the right of the Preset.



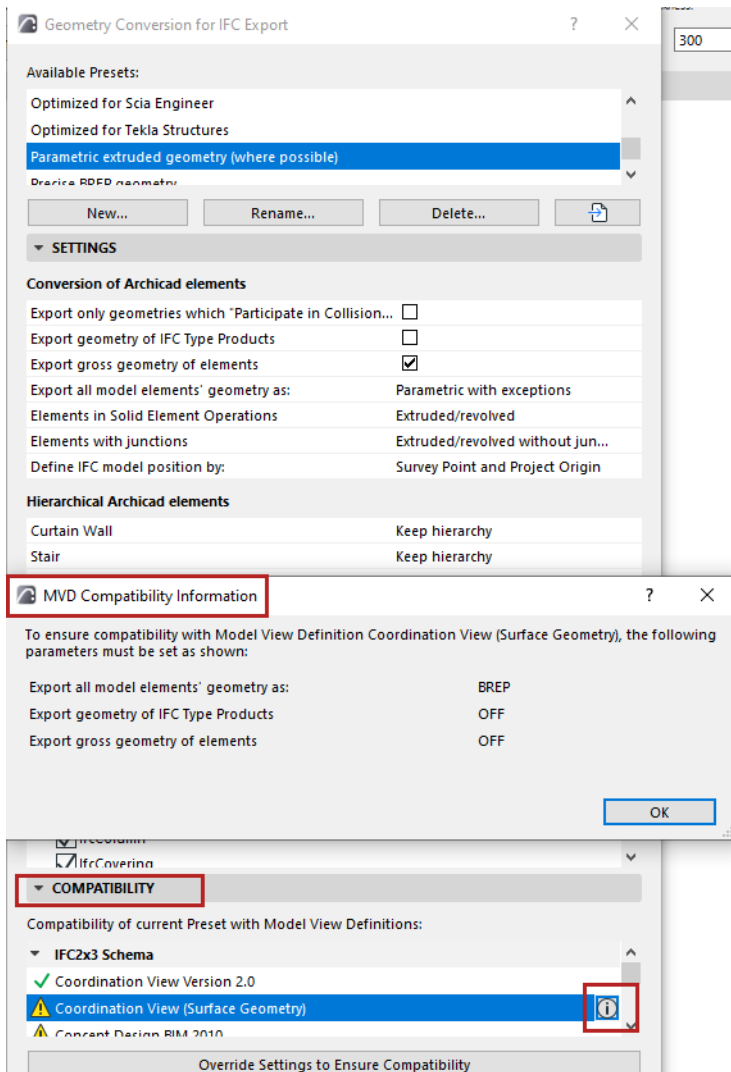
The Preset dialog opens (for example, the Geometry Conversion for IFC Export).



For each of the IFC Export Preset dialogs, view the following:

- **Available Presets** for this Conversion process (e.g. Geometry Conversion)
- **Settings** (this is where the export is defined in the greatest detail, for various data types, as defined for the currently selected preset at the top)
- **Compatibility:** Feedback on whether the current settings of this preset conform to particular Model View Definitions.

Click the **Info** button to see the settings conflicts, if any. (If the current Translator for Export does not use that particular MVD, then this conflict need not be a problem.) To ensure compatibility with the MVD, you can adjust the settings as shown in the Info dialog, or just click the **Override Settings** button to solve the issue.



- **Related Translators** (all the currently defined IFC Translators for Export which use this Preset)
- All Settings of each type of Conversion Preset (used by IFC Translators for Export) are detailed in the following sections.

[Model Filter for IFC Export](#)

[Type Mapping for IFC Export](#)

[Geometry Conversion for IFC Export](#)

[Property Mapping for IFC Export](#)

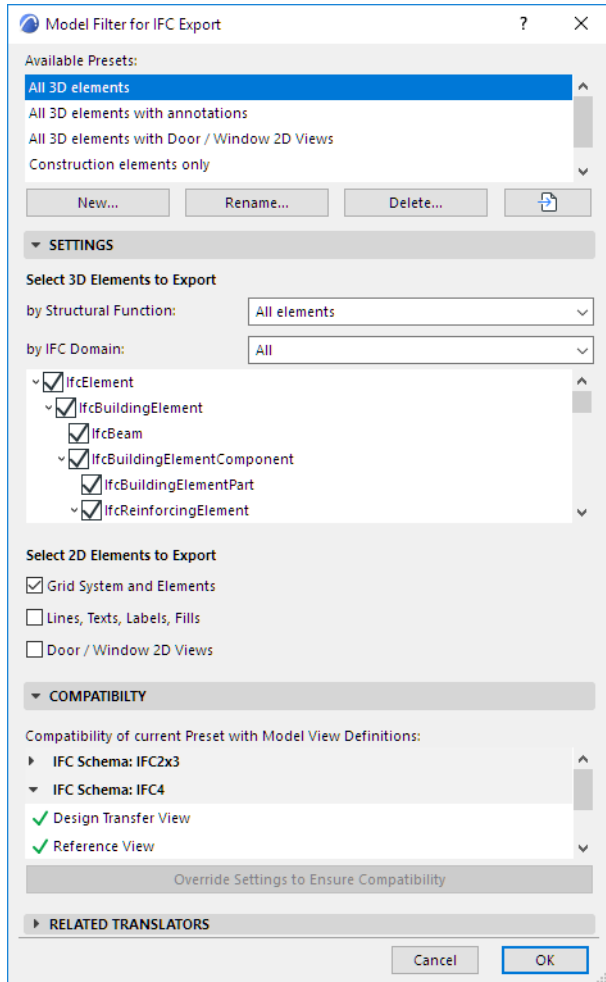
[Data Conversion for IFC Export](#)

[Unit Conversion for IFC Export](#)

Model Filter for IFC Export

File > Interoperability > IFC > IFC Translators

Choose a Translator for Export and open the Model Filter preset.



Select 3D Elements to Export

This filter defines which Archicad 3D elements should be exported to IFC. You can filter by the elements' Structural Function and/or by their IFC Domain.

- By Structural Function

This is an additional filter to the IFC Domain filter set below.

- This filter considers the Archicad element's "Structural Function" category: Either Load-Bearing or Non-Load Bearing. This is set for each element in its Classification and Properties panel of Element Settings.
- "Load-Bearing elements only" means that only those elements classified as "Load-Bearing Element" (plus Archicad Zones) will be exported from Archicad to IFC.

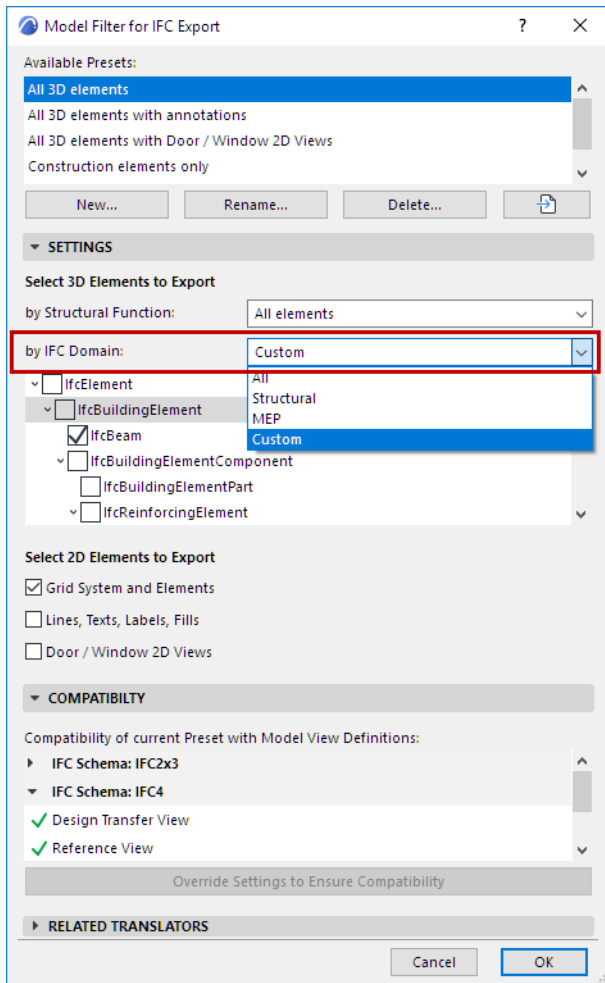
- By IFC Domain

Note: An element's IFC Domain depends on its Type Mapping preset.

- "All" includes all elements from the model

- “Structural” includes only the structural building elements
- “MEP” includes only the mechanical elements (IfcDistributionElement).

See the exact composition of each Domain filter in the tree list below. Modifying this filter will create a “Custom” filter.



Select 2D Elements to Export

- **Grid System and Elements:** Check this to include the Grid Elements and the grid members of Grid Systems in the exported file.
Grid elements will show up in the IFC structure as IfcGrid.
- **Lines, Texts, Labels, Fills:** Check this to export these 2D elements, plus all dimension types.
 - 2D elements will show up in the IFC structure as IfcAnnotation.
 - Dimensions will be exploded into lines and texts, since the IFC2x3 standard documentation does not include a dimension element.
 - The inclusion of 2D elements in the exported file is also affected by the “Elements to Export” filter at “IFC Save Options”: if a 3D view is currently active, the 2D elements can be included only if the “Entire project” option is selected in that filter. ([See Filter Model at Export.](#))
 - IFC4 Reference View does not allow annotations. To ensure compatibility for IFC4 Reference View, leave this box unchecked.

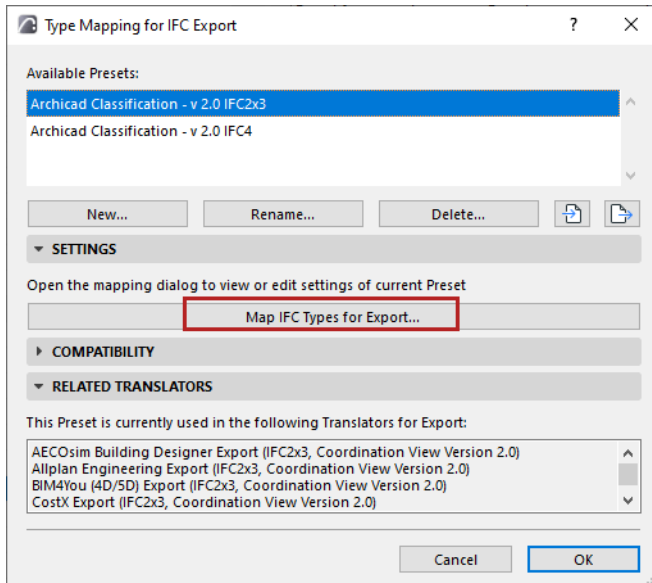
- **Door/Window 2D Views:** Check this box to include the 2D symbols of doors and windows in the export process, in addition to 3D model geometry. This is handy if the recipient program recognizes these data and can correctly display, for example, the door opening directions.

Type Mapping for IFC Export

File > Interoperability > IFC > IFC Translators

Choose a Translator for Export and open the Type Mapping preset.

From this preset dialog, click the **Map IFC Types for Export** to access the mapping definitions.



Topics in this Section

[About Type Mapping for Export](#)

[Mapping by Element Type](#)

[Mapping by Classification](#)

[Watch the Video](#)

About Type Mapping for Export

When a model is exported to IFC format, every one of its elements is assigned an IFC Type.

This dialog defines how to assign an IFC Type to each exported Archicad element.

There are two ways to take care of Type Mapping for IFC Export:

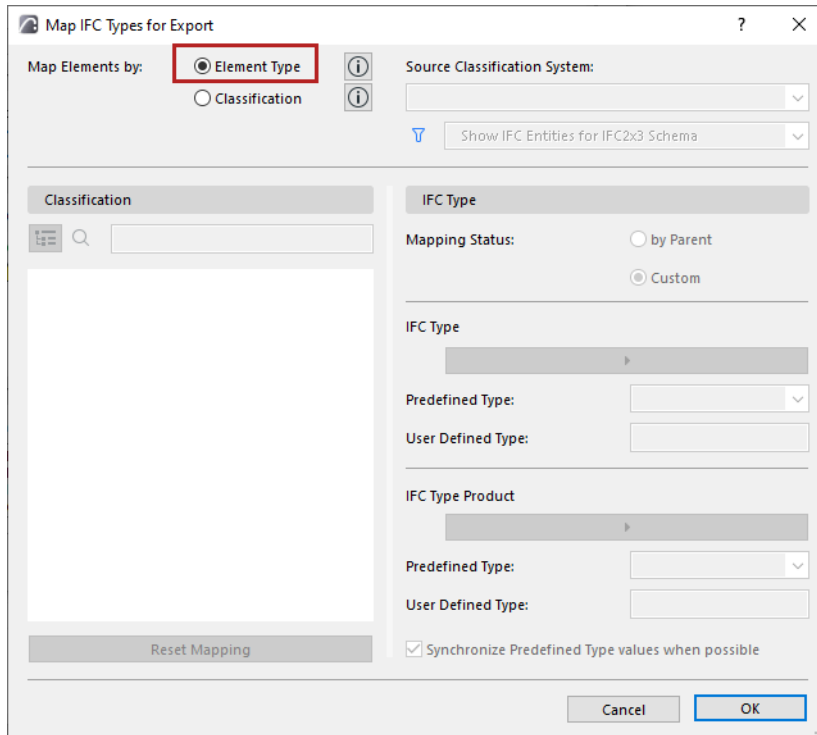
- **By Element Type**
- **By Classification**

Each is described below.

Mapping by Element Type

Assigns each Archicad Element Type automatically to a default, basic IFC Type, corresponding to its Archicad tool or (for GDL objects) to its Object Subtype.

If you choose this method, no further manual mapping is needed. The rest of the dialog controls are inactive.



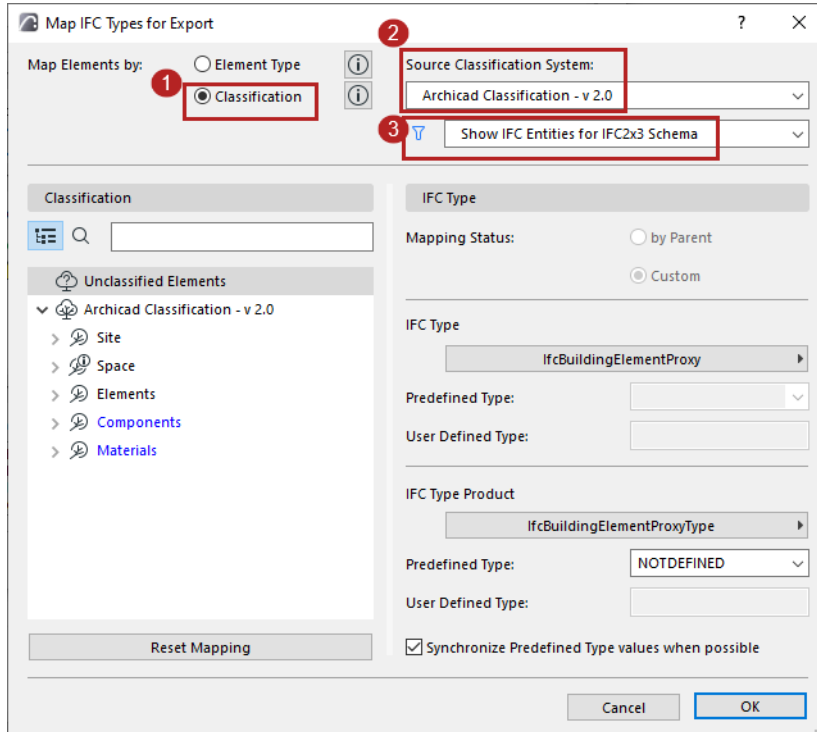
For more details, see [Built-in IFC Type Mapping for Archicad](#).

Mapping by Classification

Maps each Archicad Classification to an IFC Type, Type Product, and/or Predefined Type. This method allows for flexible and detailed IFC Type classification, according to specific IFC standards.

To use this method:

1. Select Mapping by **Classification**.



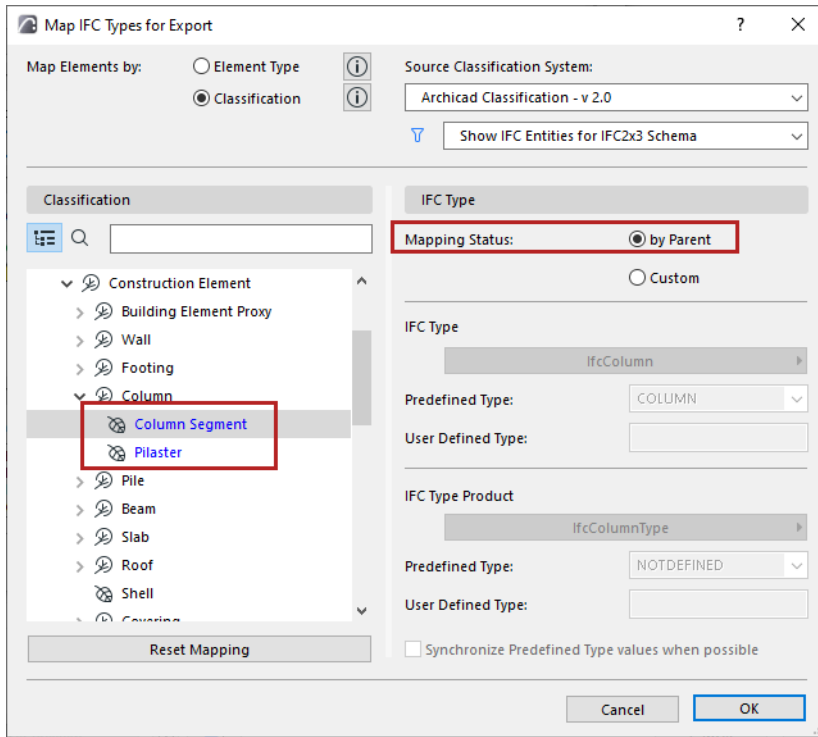
2. Choose a **Classification System** (among those available in the Archicad project). The Classifications of the selected System are now listed on the left side of the dialog.
3. Set the **Filter** to display IFC Type definitions by the IFC4 Schema, the IFC2x3 Schema, or both. This is relevant for fixing erroneous mapping definitions (see [How to Fix Error Warnings in the Type Mapping Dialog](#)). By default, the filter automatically corresponding to the IFC Schema of the current preset.

Now you can map the Classifications on the left to the IFC Type definitions on the right, as detailed below (By Parent and Custom).

Classification Mapping by Parent

For a selected Classification (e.g. Pilaster): “By Parent” maps it to the same IFC Type as its parent item (e.g. Pilaster gets the same IFC Type as Column, that is: “IfcColumn”).

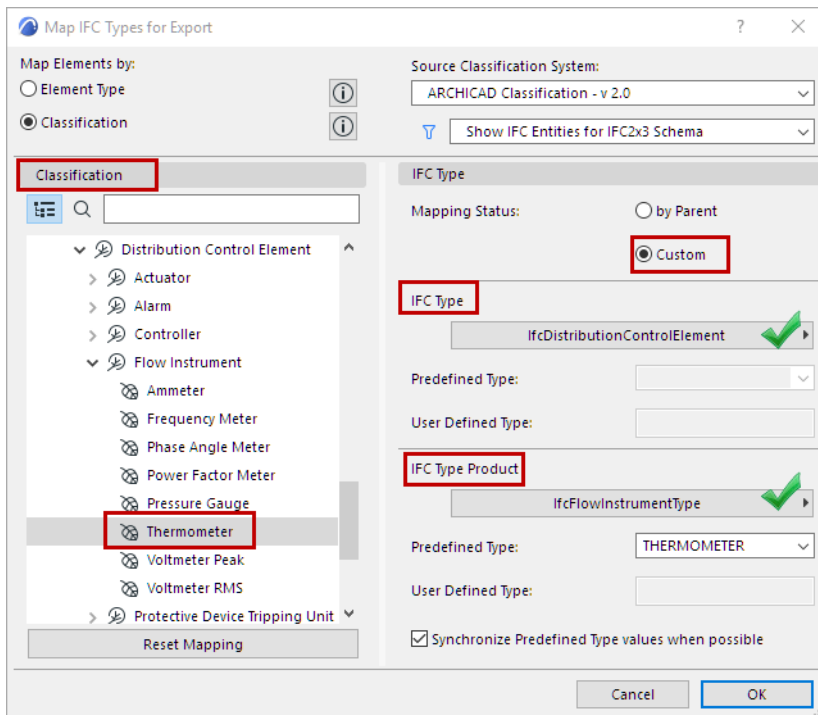
Classifications mapped “by Parent” are shown in blue: you don’t have to define its IFC Type, because it will automatically get the same one as its parent.



Custom Classification Mapping

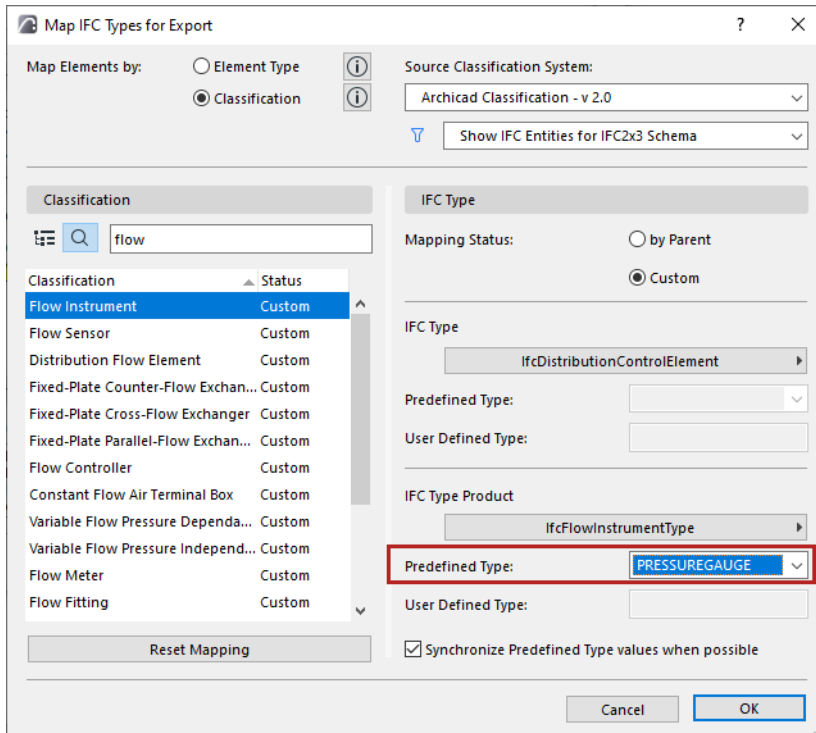
Maps the Classification to an **IFC Type** and **IFC Type Product** selected by you. This assigns more specific IFC Type definitions to the exported elements.

For example, here an element is classified in Archicad as Thermometer, and it is mapped to the IFC Type and IFC Type Product assignments as shown: just click the pop-ups to access a list of valid definitions.



Map to IFC Type and IFC Type Product

If the chosen **IFC Type** or **IFC Type Product** have **Predefined Types**, you can map those too. Here, the Flow element's Type Product is assigned the Predefined Type of PRESSUREGAUGE:



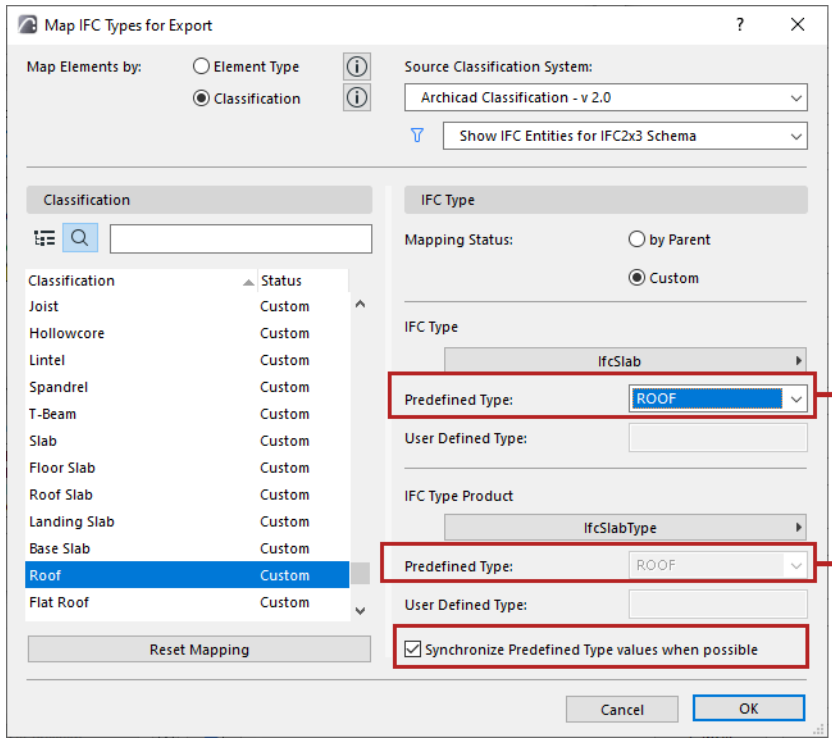
Choose a Predefined Type

User Defined Type

If the list of Predefined Types does not contain what you need, choose USERDEFINED and then enter the desired value in the text field below.

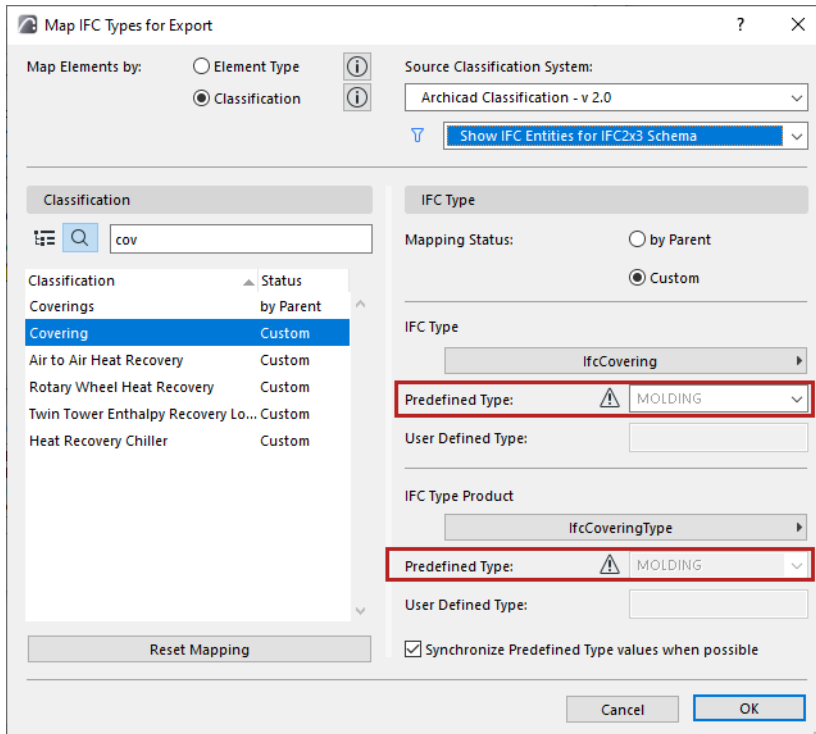
Synchronize Predefined Type Values

In most cases, the Predefined Types for IFC Type and IFC Type Product should be identical. To speed things up, the **Synchronize** checkbox is on by default:



Invalid Predefined Types

If a Classification element is mapped with a Predefined Type that is not available in the current Schema (e.g. the MOLDING value is available in IFC4 but not IFC2x3), then this value will be grayed out, and will not be exported. (The Translator preset will still be valid.)



Reset Mapping

Click this button to undo the mapping definitions made since opening the dialog.

Note: The Type Mapping preset of the *Preview Translator* has an additional function: to define the default IFC Type classification of elements in the current Archicad project.

See also [Preview Translator](#).

Related Topics

IFC Type

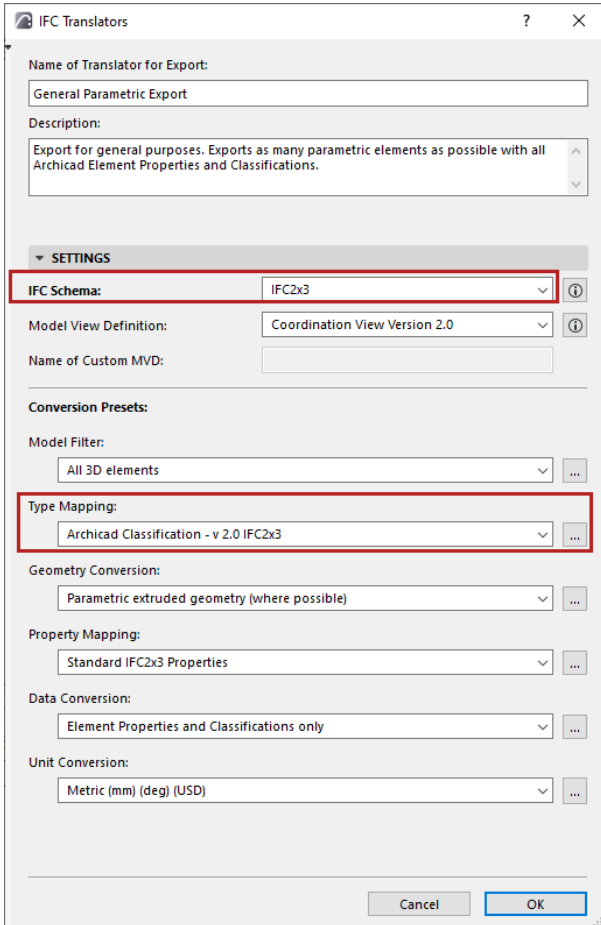
How to Fix Error Warnings in the Type Mapping Dialog

[Watch the Video](#)

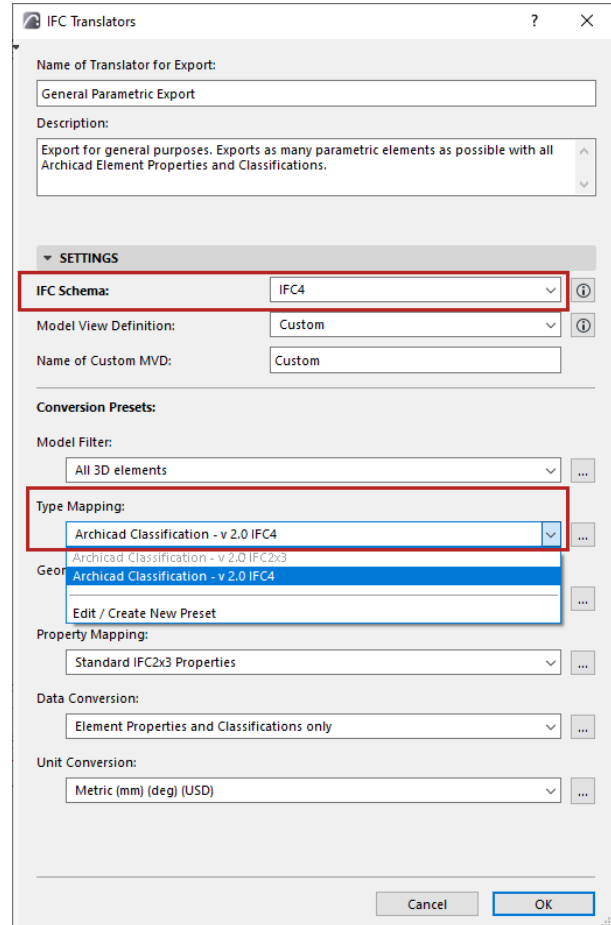
How to Fix Error Warnings in the Type Mapping Dialog

Archicad includes two Presets for Type Mapping for IFC Export. Each Preset is optimized for a supported IFC Schema: IFC2x3 or IFC4.

If you use the predefined Presets, as shown in the images below, the mapping will be correct.



Type Mapping Preset for IFC2x3 Schema

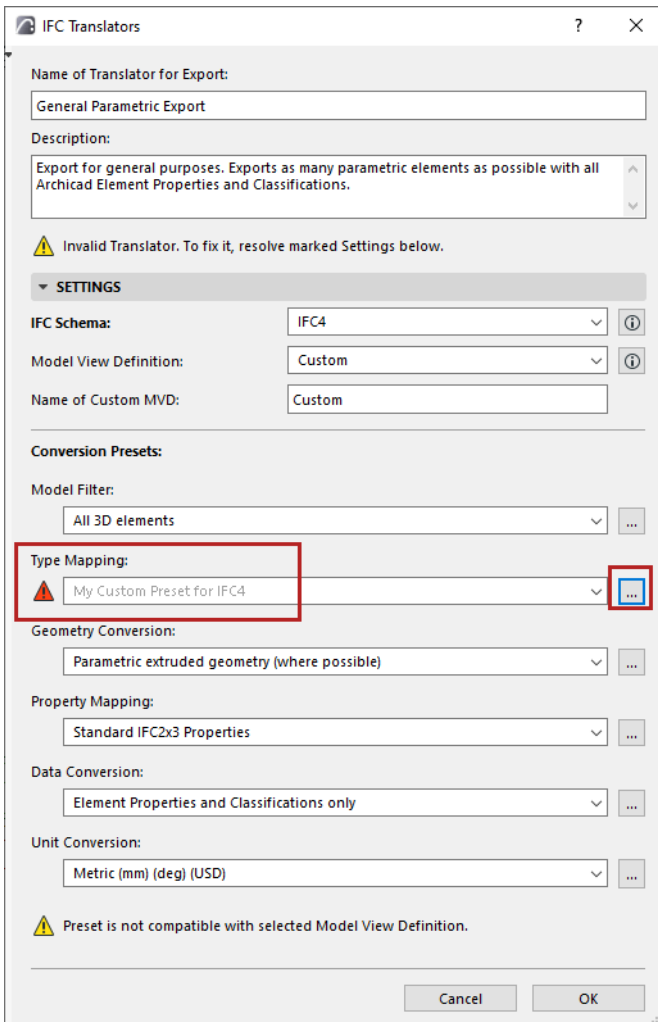


Type Mapping Preset for IFC4 Schema

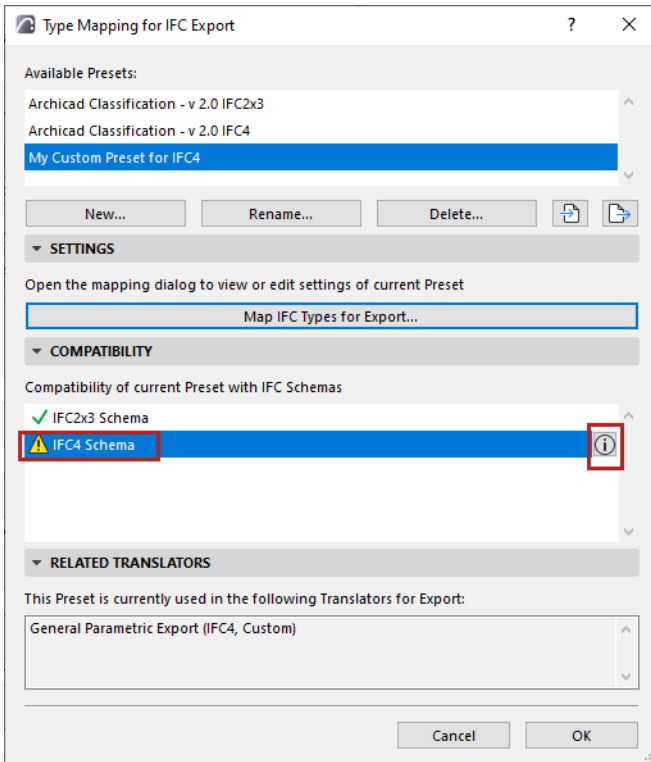
If you *customize* the Type Mapping preset or change the Schema definition, this may result in mapping definitions that are obsolete or incorrect, because the IFC type definitions differ somewhat for IFC2x3 and IFC4.

Archicad's IFC Translators dialog and the Type Mapping subdialog will alert you about any incompatible Type Mapping definitions, and help you fix them.

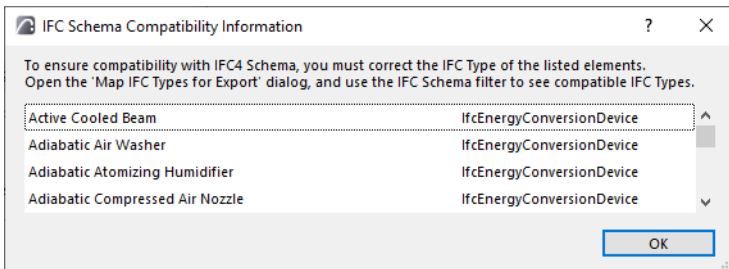
Here, you have a custom Type Mapping Preset that is supposed to be compatible with the IFC4 Schema. But the warning indicates a problem:



Click the three-dot button to open the Type Mapping Preset dialog. Here, too, the problem is indicated.



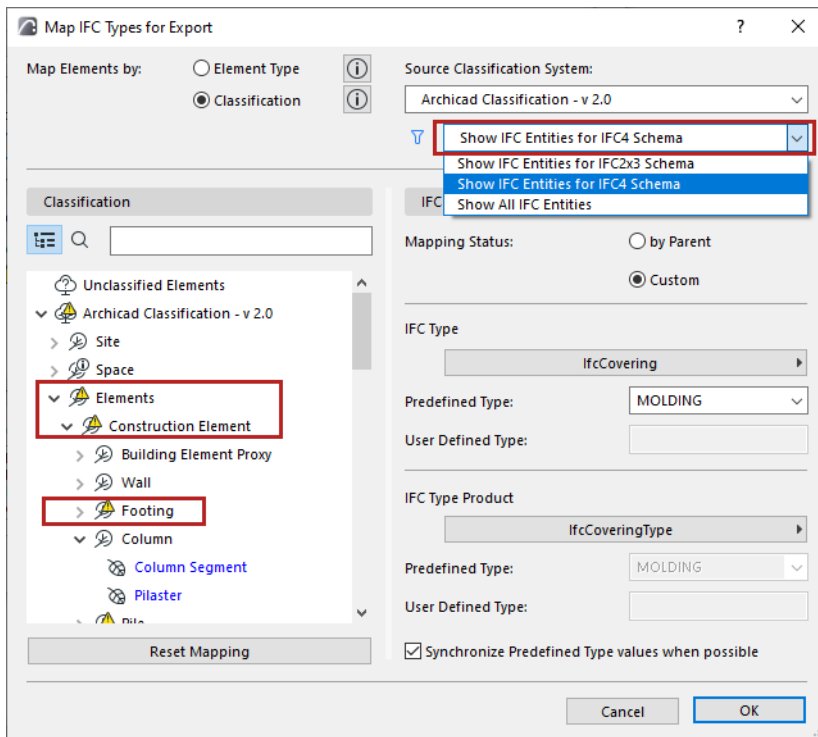
Click the **Info** button to find out more.



Click **OK** to return to the main dialog, then click to open **Map IFC Types for Export** dialog...

Set the filter to “Show IFC Entities for IFC4 Schema,” as shown in the image below.

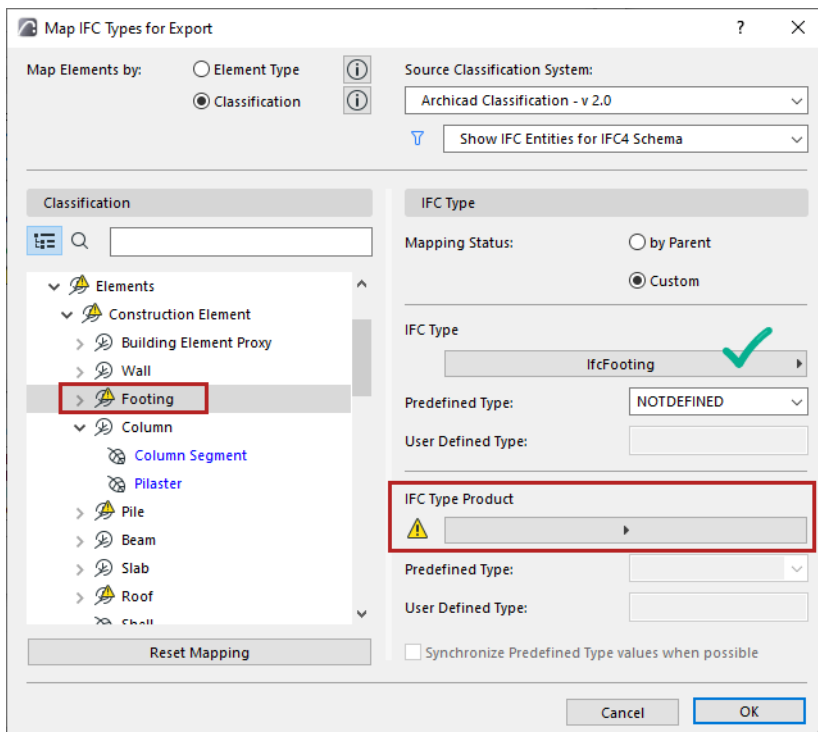
This will set up the dialog box so that you are forced to choose mapping definitions compatible with IFC4.



In the left-hand tree view, you can see the Classification items that have errors: they are marked with warning triangles. These items were probably mapped correctly for IFC2x3, but since naming standards have changed or been expanded, you have to re-map these items to use the corresponding new terms for IFC4. This will be easy.

1. Select the problem Classification item (e.g. Footing).

Its mapping definitions are on the right. The IFC Type is okay, but the IFC Type Product is missing.

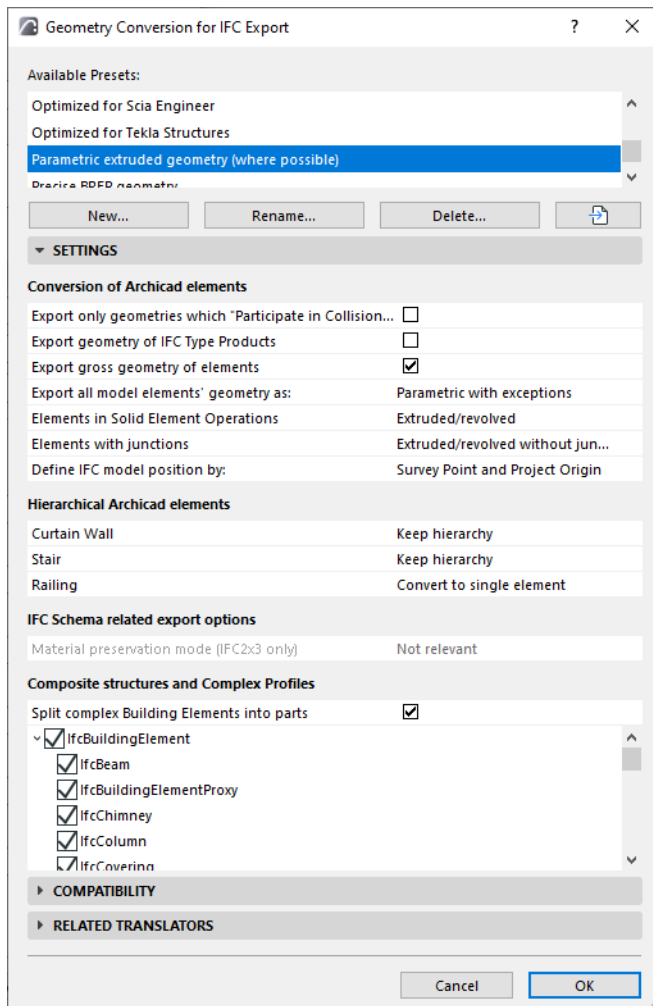


2. Click the **IFC Type Product** pop-up to see compatible mapping choices. Select one and click **Choose**. Repeat this re-mapping for each of the problem elements to ensure that your Type Mapping preset is compatible with IFC4.

Geometry Conversion for IFC Export

File > Interoperability > IFC > IFC Translators

Choose a Translator for Export and open the Geometry Conversion preset.



These settings define how to convert the geometry of Archicad elements exported to IFC.

The conversion settings are divided into categories:

Conversion of Archicad Elements

Hierarchical Archicad Elements

IFC Schema-Related Options

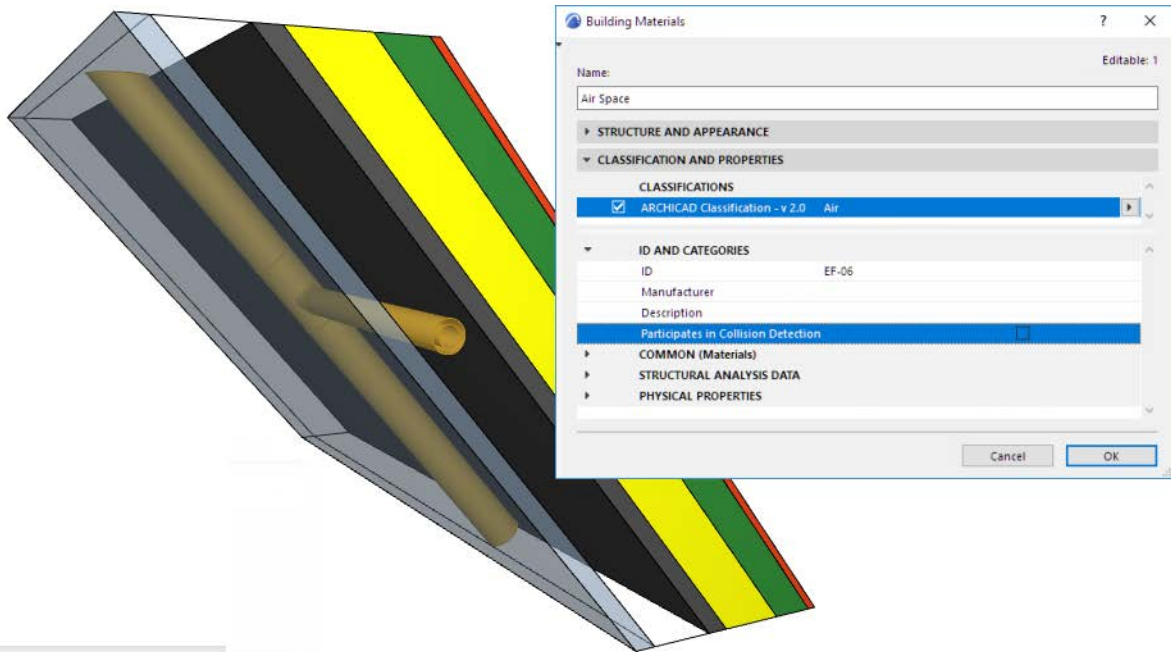
Composite Structures and Complex Profiles

Conversion of Archicad Elements

Export only geometries which “Participate in Collision Detection”

Exports only elements whose Archicad Building Material includes the “Participates in Collision Detection” option ON. (This is set at **Options > Element Attributes > Building Materials > Properties**).

Use this option to export composite elements and complex profiles, which don’t have solid geometries, as real air gaps. This way, MEP engineers (who receive the Archicad IFC file) can place pipes in the gaps without collision detection; only element parts which have real geometry will participate in collision detection.



[See also Collision Detection.](#)

Export Bounding Box

Exports the dimensions of the building elements' bounding box.

Export Geometry of IFC Type Products

Assigns a representative geometry to each Type Product. (This option is required for Concept Design BIM 2010.)

Each type (e.g. IfcFurnishingElementType) will use a representative geometry of the elements that belong to it (e.g. IfcFurnishingElement).

Export gross geometry of elements (Export holes separately)

Elements containing Doors, Windows and/or Openings are exported with their gross geometry. For example, a host Wall is exported with its gross geometry, without considering the hole created by a window.

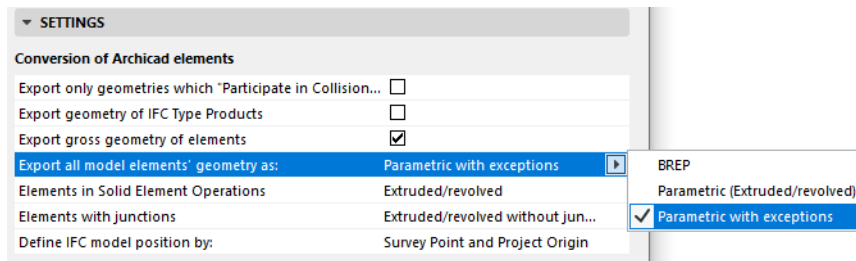
Use this option to ensure accurate cost calculation using the IFC model, where the opening geometry will be considered separately.

Note: Openings are exported by checking IfcOpeningElement in the Model Filter for IFC Export. This is generally checked by default.

If the **Export gross geometry** option is *unchecked*, then elements cannot be created using extrusion. In this case, extruded elements with holes are exported as BREP!

Export all model elements' geometry as:

This describes how the exported element geometry will be interpreted. Choose one of the three options from the pop-up: BREP; Parametric (Extruded/revolved); or Parametric with exceptions. See below.



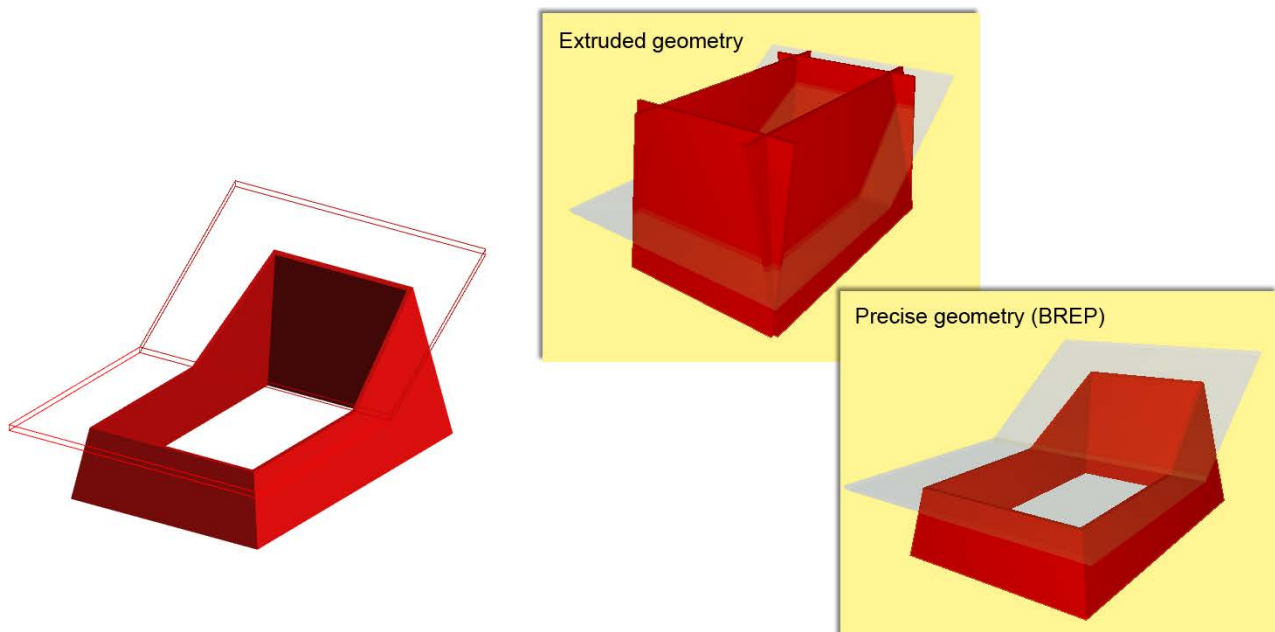
BREP

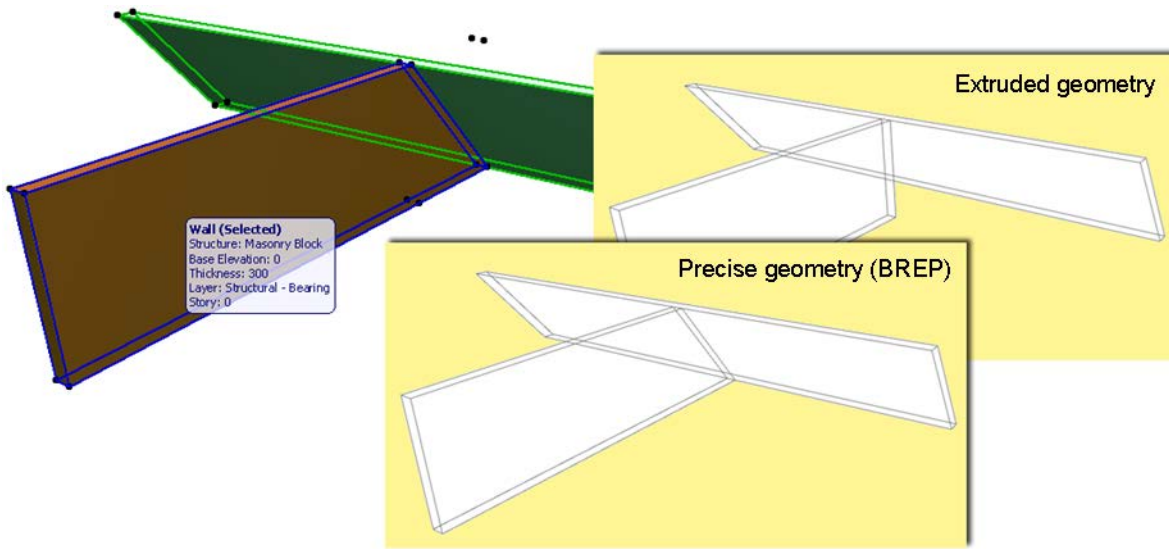
- All model elements are exported using BREP (Boundary Representation) geometry, together with their junctions.
- This method is useful in the “reference model” workflow.
- Provides the most exact reproduction of element geometry, together with its specialized sections and junctions.
- Exported elements are transformed into non-editable elements, and their parameters are lost.
- The elements will be exported in the color displayed in the current Archicad view, including colors from Graphic Overrides, Renovation Status, and Issue highlights.

Parametric (Extruded/revolved)

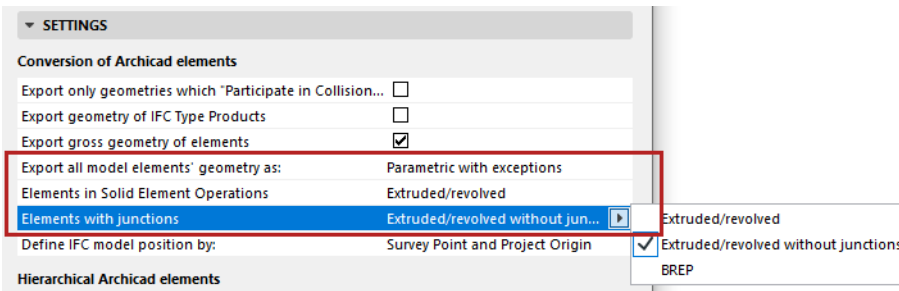
- All model elements are exported using Parametric (Extruded/revolved geometry.)
- This is the format usually supported by static analysis programs, in which element parameters are important, but their special cut angles (such as the slanted edge of a slab) are not important.
- Retains the elements' parameter values (such as thickness, height, location of reference line or edge).
- The skin structure of composite materials cannot be fully preserved because of the IFC standard's limitations.
- Certain specialized sections are not retained.

Note: Morphs, Objects, Shells and certain Walls or Beams with unusual profiles cannot be exported as parametric: these are always exported using BREP.





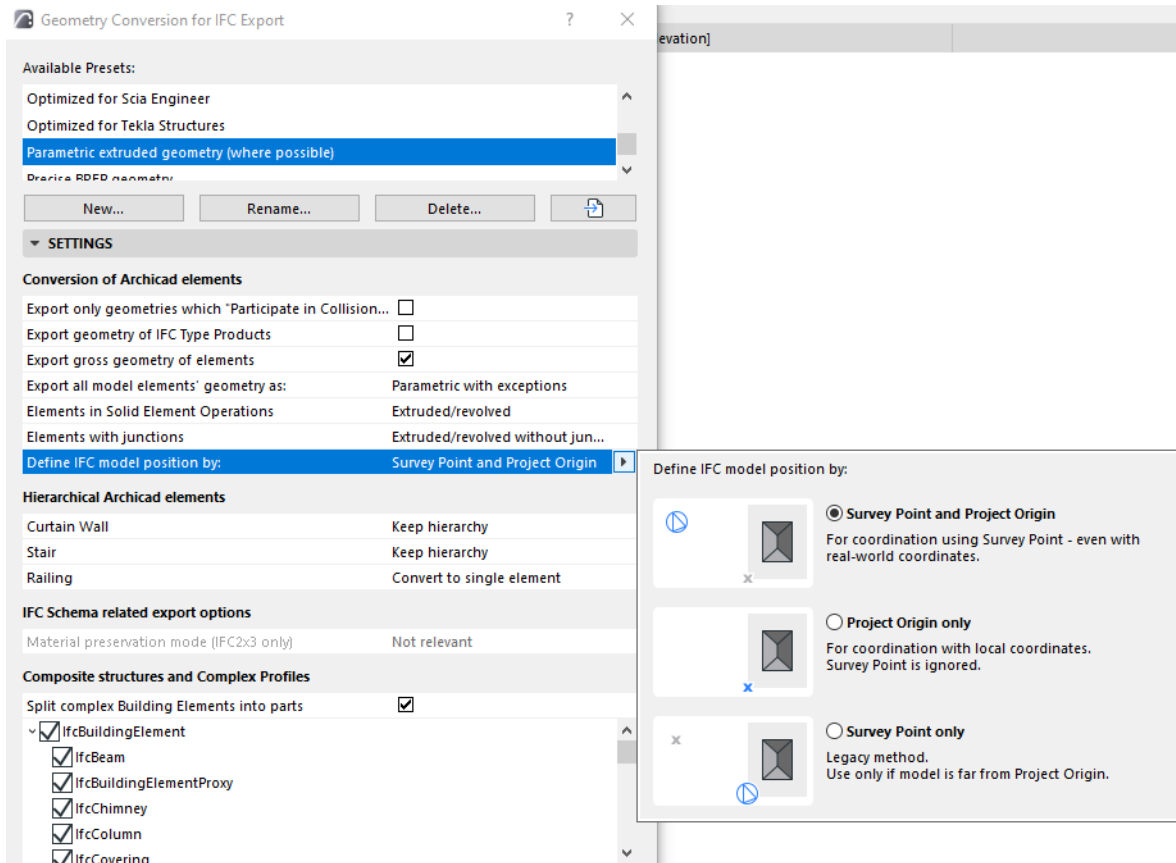
Parametric with exceptions



- This exports elements as parametric (as described above). However, you can set separate export preferences for **Elements in Solid Element** Operations and **Elements with junctions** (including slabs with slanted edges):
 - BREP (as described above)
 - Extruded/Revolved (as described above)
 - **Extruded/Revolved without junctions**: Elements are exported without the priority-based intersections, resulting in a faster export process.
 - Recommended for structural analysis programs, for which the elements' reference lines/surface information is sufficient, and detailed intersections are not required.

Define IFC model position by:

Choose to locate the IFC Site entity using the **Survey Point** and/or **Project Origin**.



- **Survey Point and Project Origin** (default): Used in most coordination workflows.
- **Project Origin only**: Use this if model position is unimportant (e.g. to use the model to create budget calculations). This workflow is most typical in German-speaking markets.
- **Survey Point only**: Use this only if the model is far from the Project Origin.

Notes:

- The program considers the Survey Point even if it is not visible in the exported view
- To assign geographic property data, use the Survey Point's Georeferencing data. These properties can be shared with any project participants via IFC.

See also [Survey Point](#).

Note on Export of IFC Site Geometry:

As of Archicad 23, all elements that are IFC Site Geometry are exported to IFC as solid bodies.

Archicad elements are IFC Sites if they are:

- Classified using "IfcSite" as their IFC Type, or
- Saved as Objects having the Subtype "IFC Site"

Mapping IFC Location Data From Archicad and Revit

IFC	Archicad 20 and up	Revit
IFC Global Origin	Survey Point	Survey Point
Position of IfcSite entity	Project Origin	Project Base Point

Hierarchical Archicad Elements

Choose a conversion option for each type of hierarchical element: Curtain Walls, Stairs, Railings.

- **Convert to single element:** The hierarchical element is exported as a **single IFC entity**, which contains within itself the geometry of all its sub-elements. The hierarchical nature of the original element (e.g. Stair, with all its separate sub-components) is thus lost.
- **Keep hierarchy:** Retain the original element's hierarchical structure, including sub-elements.

See also [Exporting Hierarchical Elements from Archicad as IFC Containers](#).

IFC Schema-Related Options

Material preservation mode (IFC2x3 only)

Use this control to define whether and how to explode complex elements into Building Element Parts. This choice depends on whether you need to retain the material definitions of the components.

Choose one of the following Material preservation modes:

- **Never explode elements.** Material preservation is not guaranteed.
This is the default option: complex elements (composites and Profiles) are not exploded, because doing so multiplies the number of Building Element parts in IFC.
- **Explode only when necessary to preserve materials.** The program will explode an element only if that is the only way to preserve its material definition.
- **Explode all elements into parts, preserve materials.** All complex elements are exploded into parts, and material preservation is guaranteed.

Notes:

- These Material preservation options are relevant for IFC2x3 only.
- For IFC4, use the control below: see [Split Complex Building Elements into parts](#).

Composite Structures and Complex Profiles

Split Complex Building Elements into parts

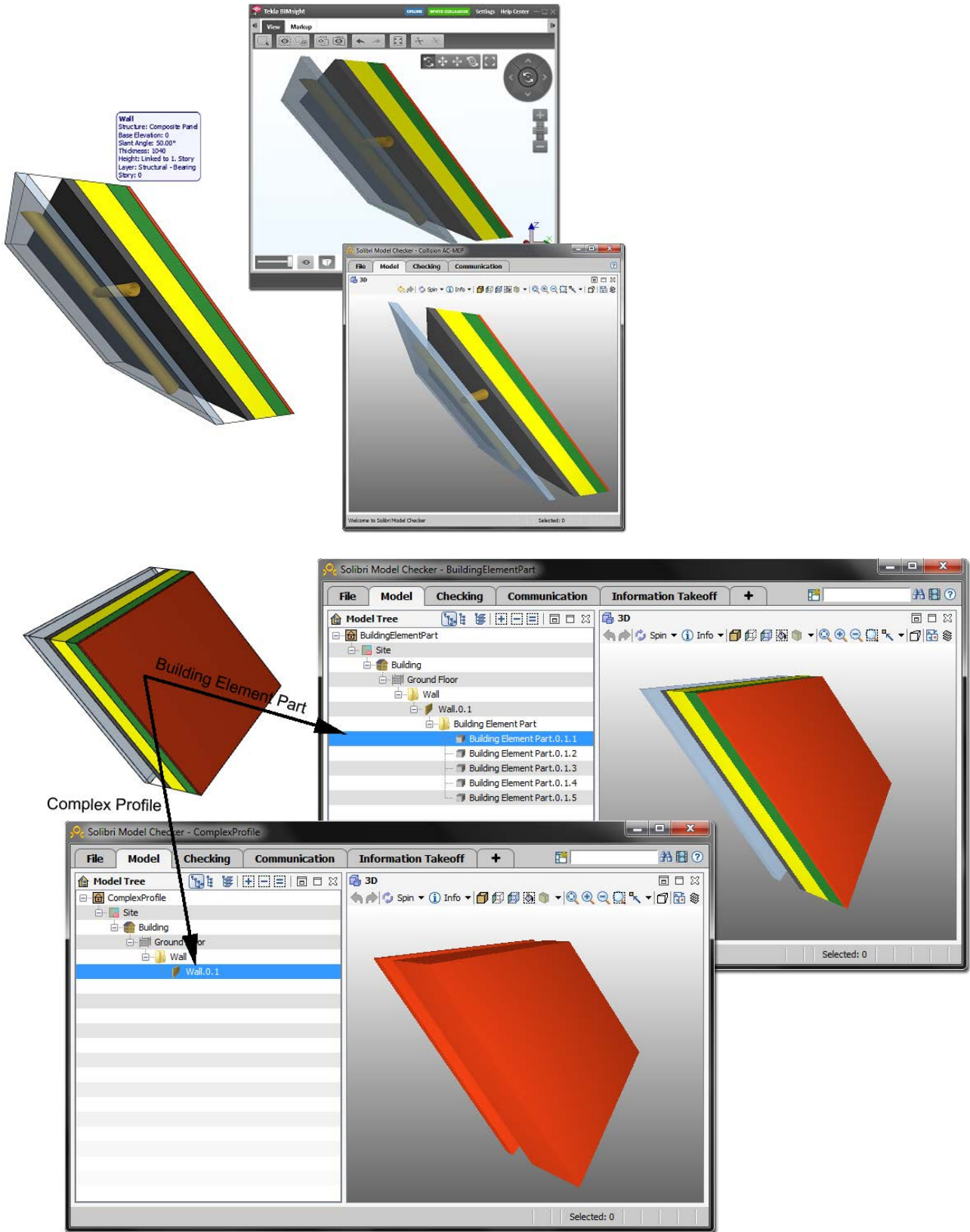
Use this control to split complex elements into Building Element Parts. This option enables you to comply with IFC4 Reference View. Material definitions are not affected by this option (IFC4 retains material definitions irrespective of whether complex elements are split.)

This option enables composite elements and Complex Profiles to be split into so-called "Building Element Parts". With this option, the receiving Structural Engineer will know that a composite Wall includes an air gap component.

If you check the box:

- the tree structure below is editable: you can choose specific element types to be split into parts. (The unchecked element types will not be split up.)

If the box is not checked: no complex elements will be split.

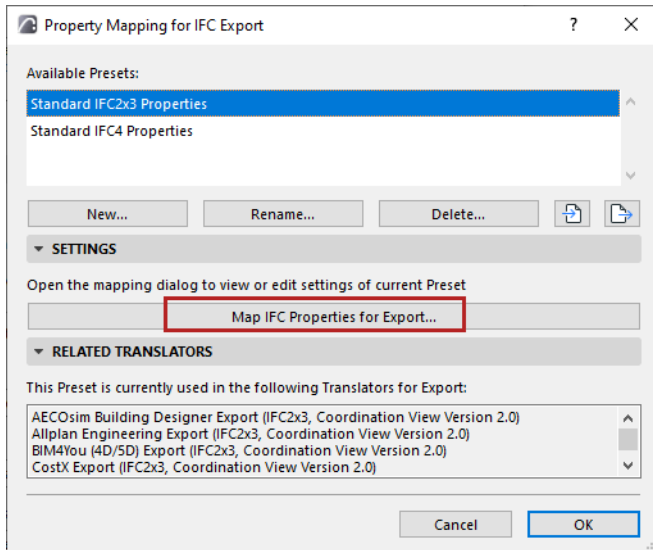


Property Mapping for IFC Export

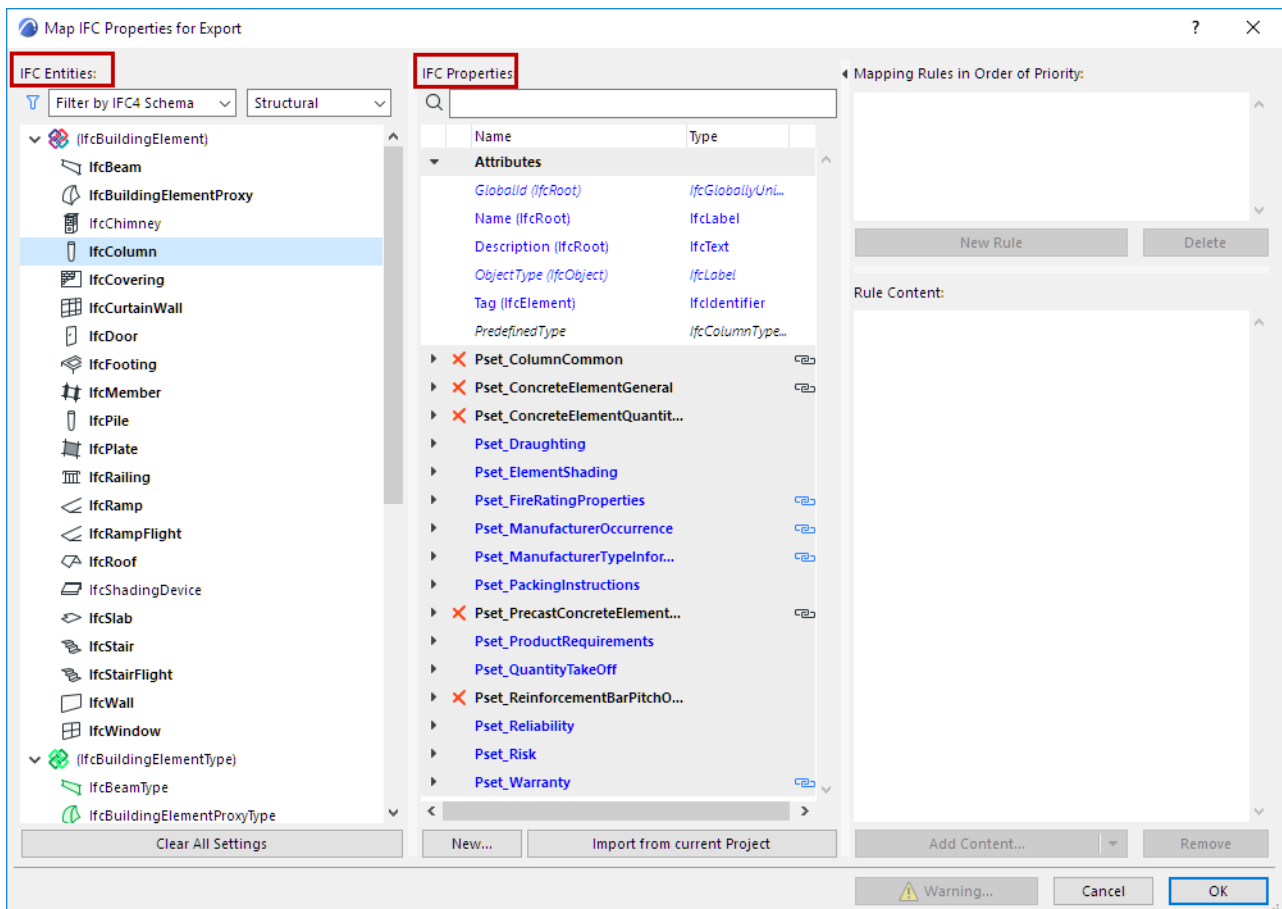
File > Interoperability > IFC > IFC Translators

Choose a Translator for Export and open the Property Mapping preset.

From this preset dialog, click the **Map IFC Properties for Export** to access the mapping definitions.



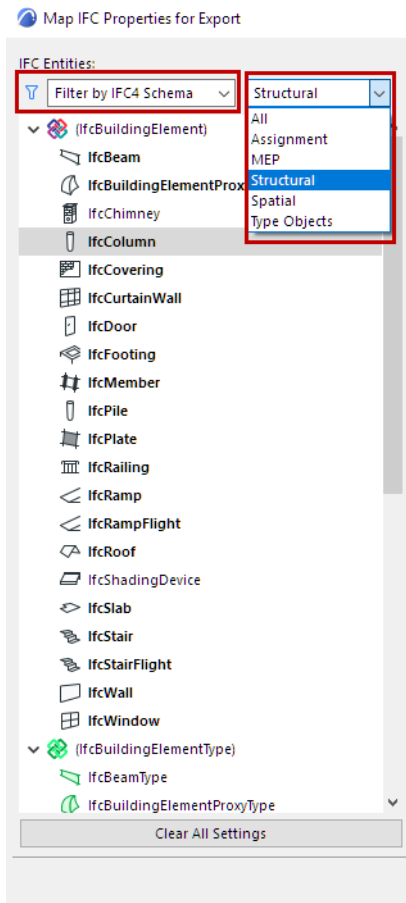
These mappings define the IFC properties assigned to elements when exported from Archicad.



Tree list of IFC Entities

IFC Entities are listed in the tree at left.

- Use the pop-ups if needed to filter the tree list:
 - filter by Schema (IFC2x3 or IFC4)
 - **Note:** A parent entity, if it can be used for property mapping, will be displayed even if it does not belong to the Schema in the current filter
 - filter by domain (Structural or MEP). For example, the Structural domain contains the building element (IfcBuildingElement) types (IfcBeam, IfcColumn, etc.), while the MEP domain contains the IfcDistribution types.
 - filter by IFC Assignment types, IFC Type Objects, or IFC Spatial element types (IfcSpatialStructureElements: IfcSite, IfcBuilding, IfcBuildingStorey and IfcSpace).

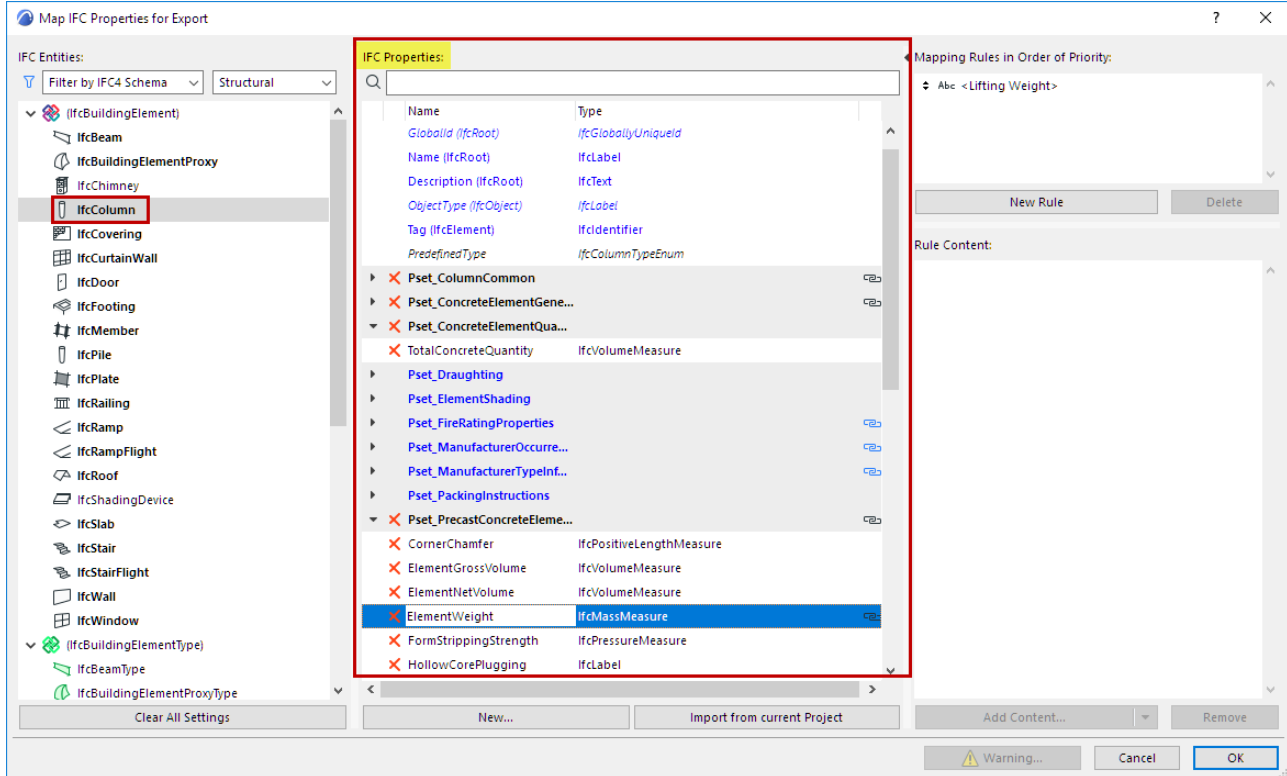


Filter IFC Entities in Tree List

Note: IFC Entities in parentheses are necessary to the IFC Schema, but have no Archicad elements assigned to them directly.

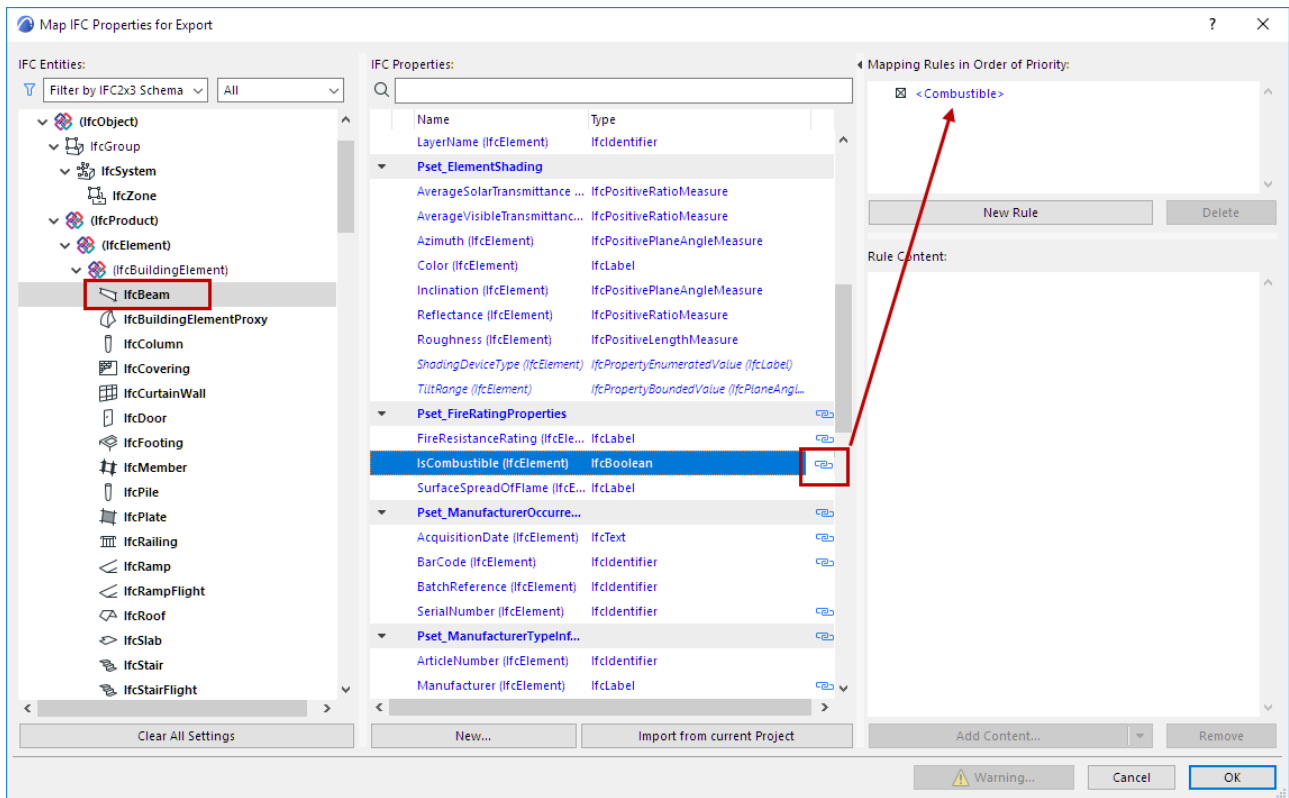
List of IFC Properties

The IFC Properties of each selected entity are listed in the **IFC Properties** column.

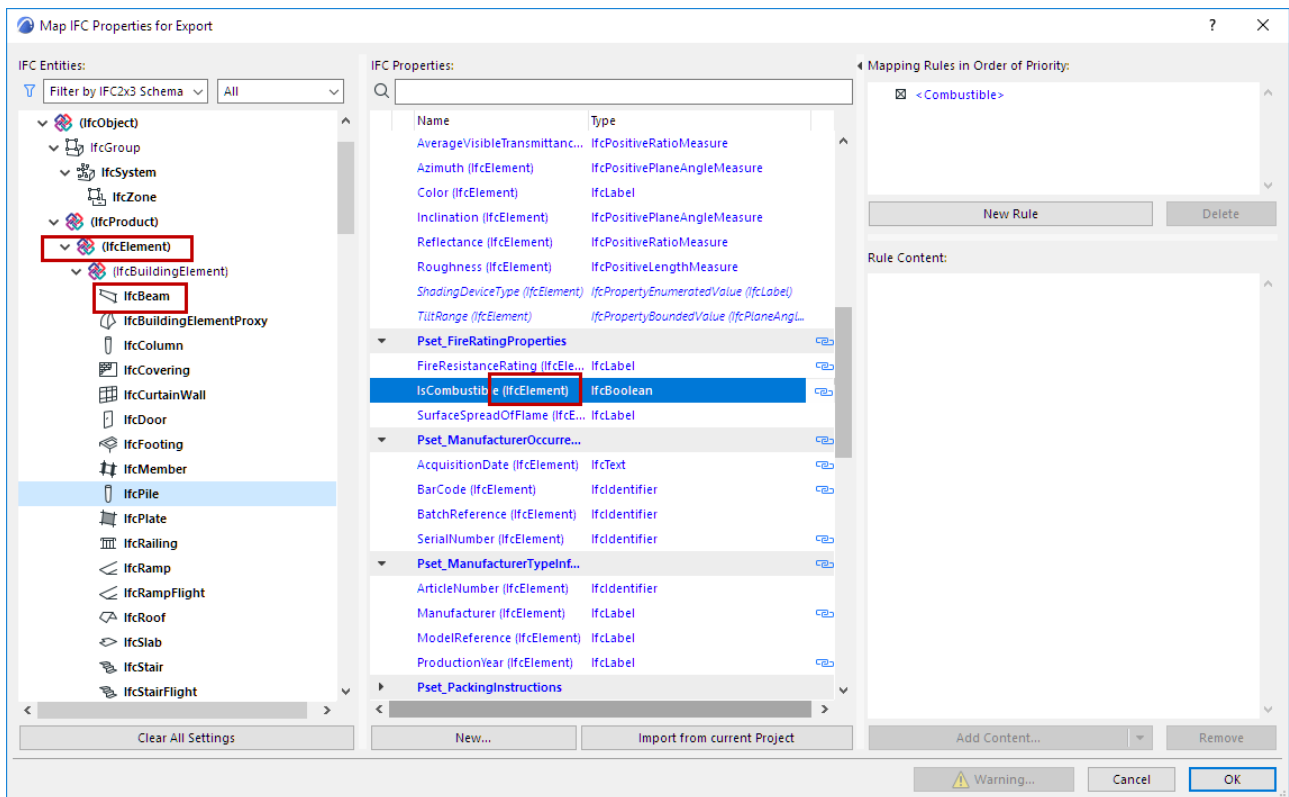


- Property data assigned to a parent element (such as IfcElement) are automatically assigned to all its child elements (e.g. IfcBeam, IfcColumn, IfcSlab, IfcWall, etc.)
- **X**: Delete any data item by clicking the red X. Attributes cannot be deleted.
- **BOLD TYPE**: IFC Entities which have property set data assigned to them are shown in **BOLD**.
- **BLUE TYPE**: Properties of a child element are shown in blue type. You can see its parent type name in parentheses.
- Chain icon: Indicates that the property has a mapping rule. View its mapping rule at the right side of the dialog box.

See [How to create a Mapping Rule](#).



Property *assignment* can be modified only at the parent level. Thus, to create a common property for all building elements, you don't have to create it for every single building element type (IfcWall, IfcColumn, IfcSlab, etc.): instead, create it once at a higher level (e.g. at IfcElement).



Add New Property/Classification Reference

1. Select the item.
2. Click **New**.

The same function is available in the IFC Project Manager and Element Settings dialog boxes.

See [Create New, Custom IFC Property](#).

Import from Current Project

Click this to add all the custom IFC Property Sets and Properties of the current project (defined in either IFC Project Manager or Element Settings) to the Property Mapping preset.

Note: If a custom property assigned to an IFC element type (for example IfcWall) is not yet available in the current project, it is not displayed in the IFC management dialogs until one instance of the type has been defined in the model.

Clear All Settings

Delete the entire contents (mapping rules, Properties and Classification References) of the current Property Mapping Preset.

Related Topics

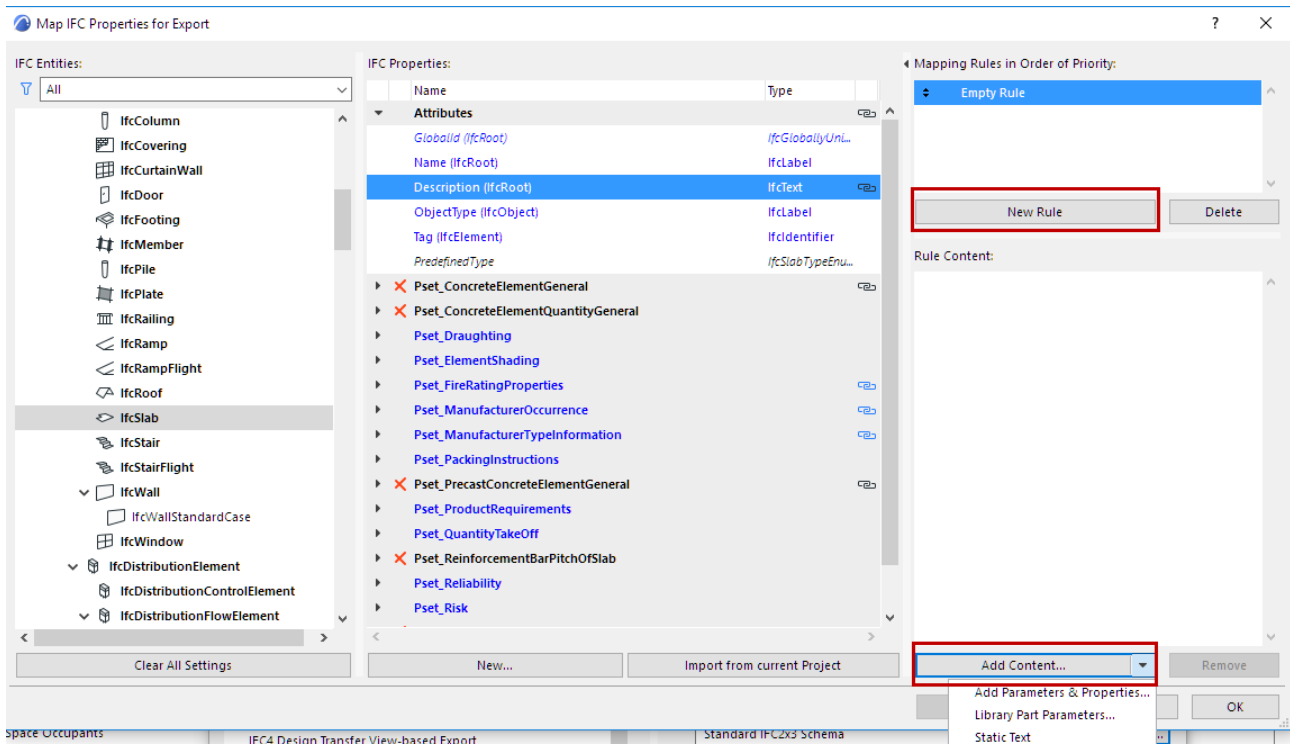
[Predefined Property Mapping \(Export Archicad to IFC\)](#)

[Preview Translator](#)

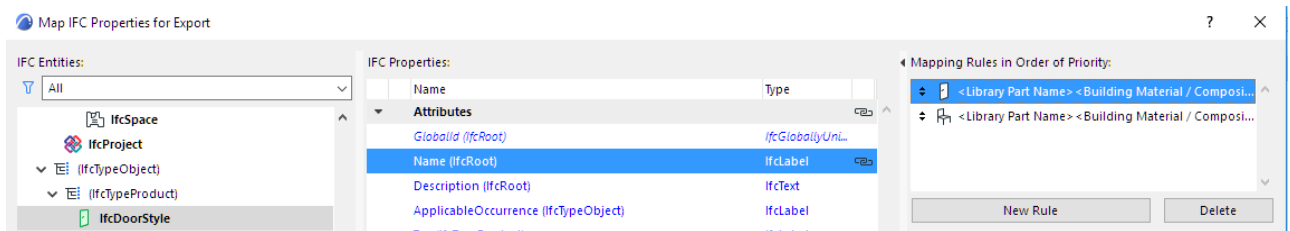
How to create a Mapping Rule

1. Select the IFC Property which you would like to map. Data in italic type cannot be mapped.
2. Click on the “New Rule” button. The “Empty Rule” field appears.
3. from the “Add Content” drop-down button, select fields from among the content types:
 - Parameters & Properties
 - Library Part Parameters
 - Static Text

Note: If a rule contains more than one field, the fields are always a concatenation of fields converted to a string.



You can assign multiple rules to a single property. This is useful, for example, if several tools will generate the same IFC element type. Rules will be applied in the order in which they appear on the user interface: the first valid rule will be applied.

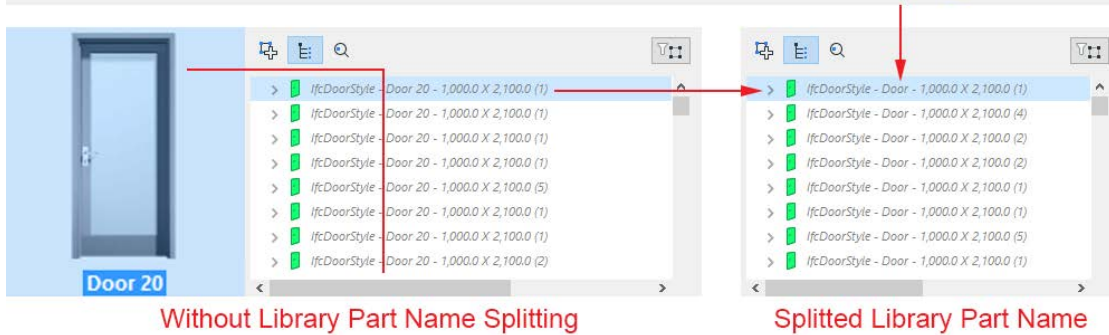
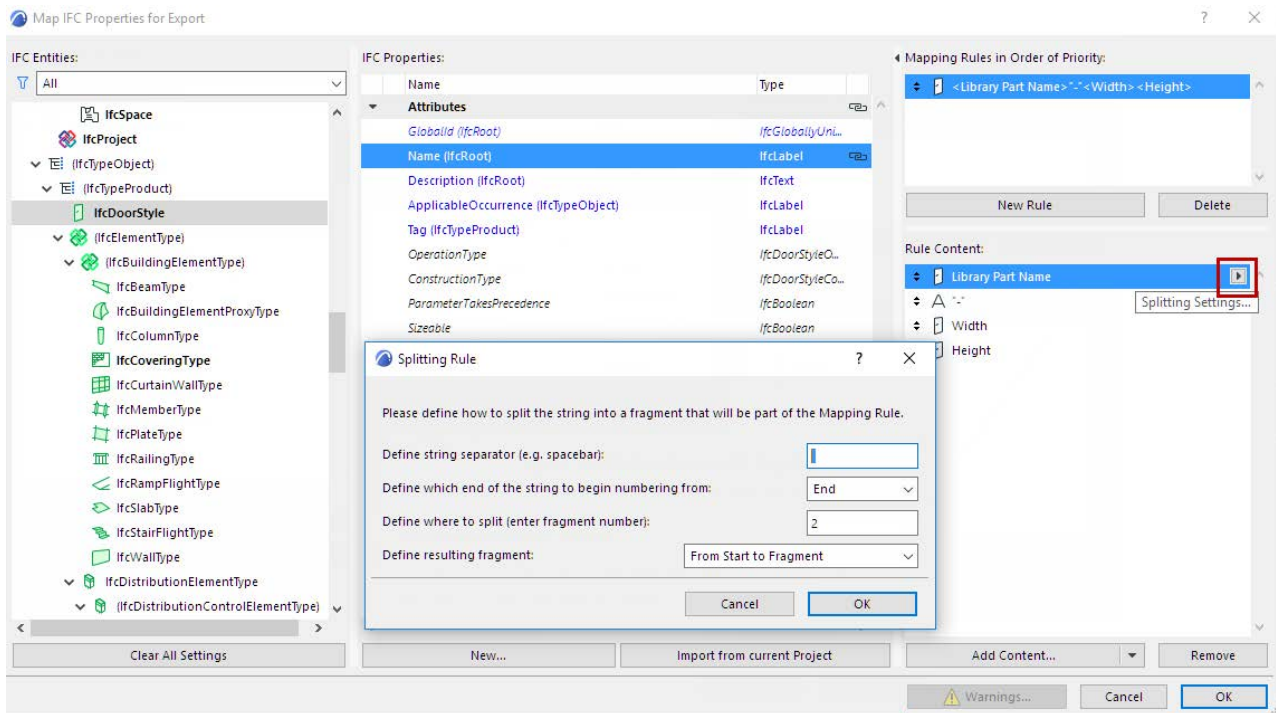


Split a string

A string-type field can be split.

In the Rule Content, for a selected string-type field: Click on the pop-up to open Splitting settings:

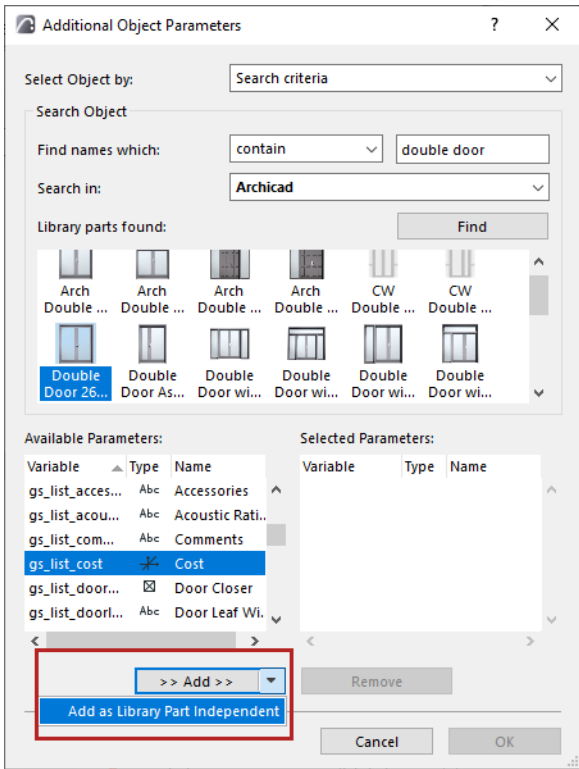
For example, use a split to cut out a particular fragment of a Library Part Name.



Library Part Parameters

Library Part Parameters can be set in two ways:

- Library Part dependent:** The selected Library Part parameter will be in effect only for elements that contain this parameter. If the Library Part parameter was defined at the Object subtype level, then the parameter mapping will take place for the child elements, too. For example, if we are mapping the “gs_list_cost” variable parameter of the GS Door Object subtype, then the mapping will be in effect for all Doors (and ifcDoors) which are child Library Part elements of the GS Door (such as Door, Double Door, Metal Door, etc.) See examples below.
- Library Part independent:** the parameter variable (e.g. “gs_list_cost”) will be in effect for the mapping, regardless of which Library Part it was chosen from. For example, if you select the “gs_list_cost” parameter of the Double Door for mapping as a Library Part Independent parameter, then the mapping will take place for every Door that accesses the “gs_list_cost” parameter. If this door parameter is used for mapping at the IfcElement level, then it will be mapping for all IFC entities which are child entities of the IfcElement and which include the “gs_list_cost” parameter (such as Furnishing elements, e.g. Armchair and Chair). See examples below.



Note: If a mapping rule involves a Library Part parameter which is not loaded in the project, then the rule, the mapped IFC data, and the mapped IFC Entity data are all shown in red.

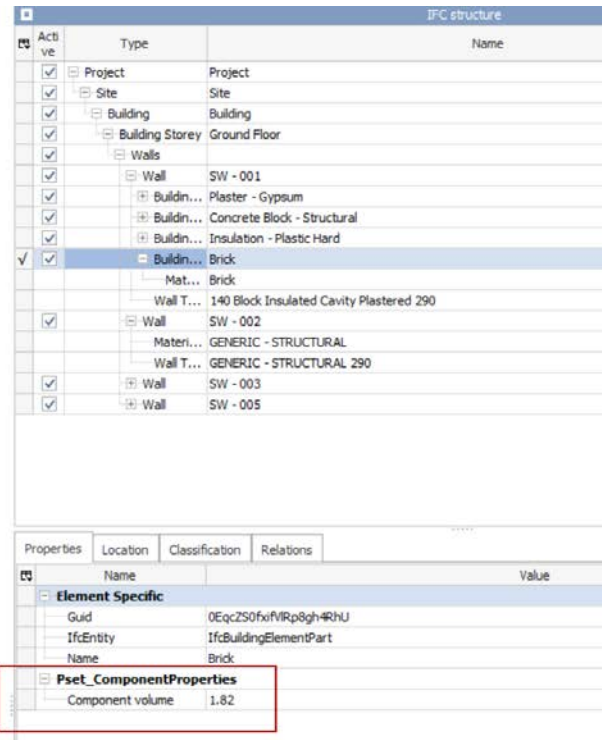
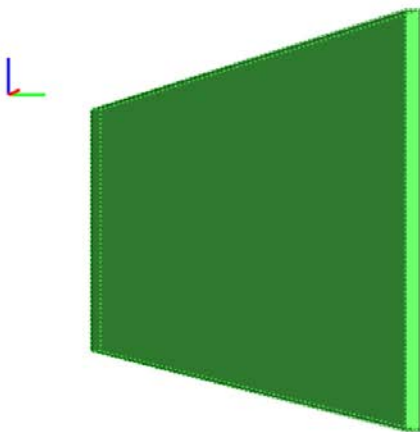
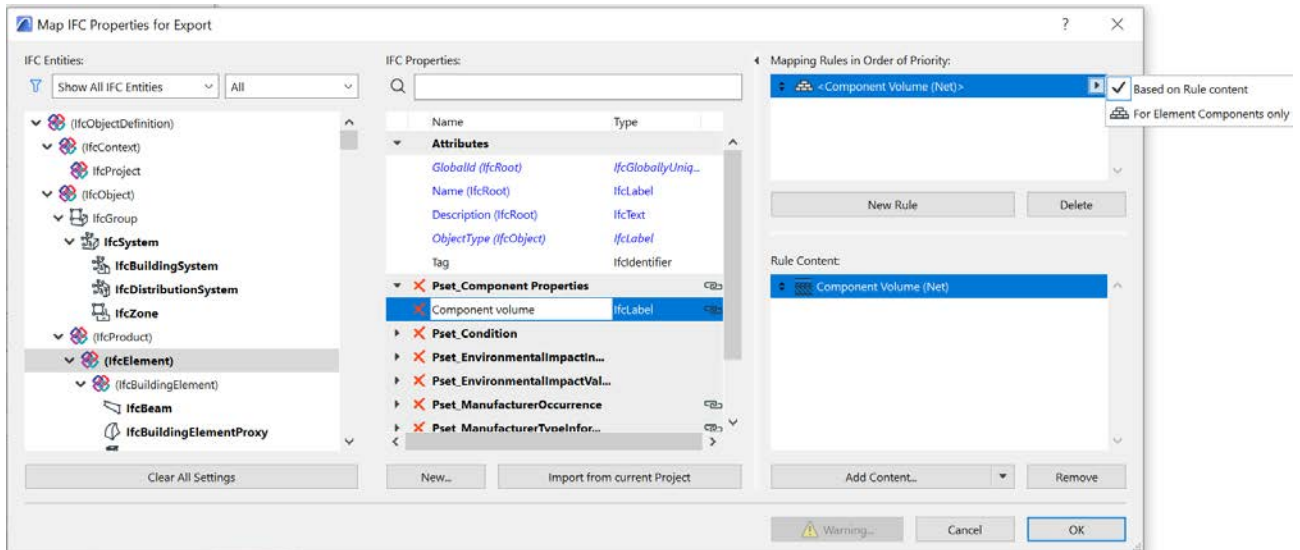
Related Topic

- [Examples of Property Mapping Rules](#)
- [Map Component-Level Properties](#)

Map Component-Level Properties

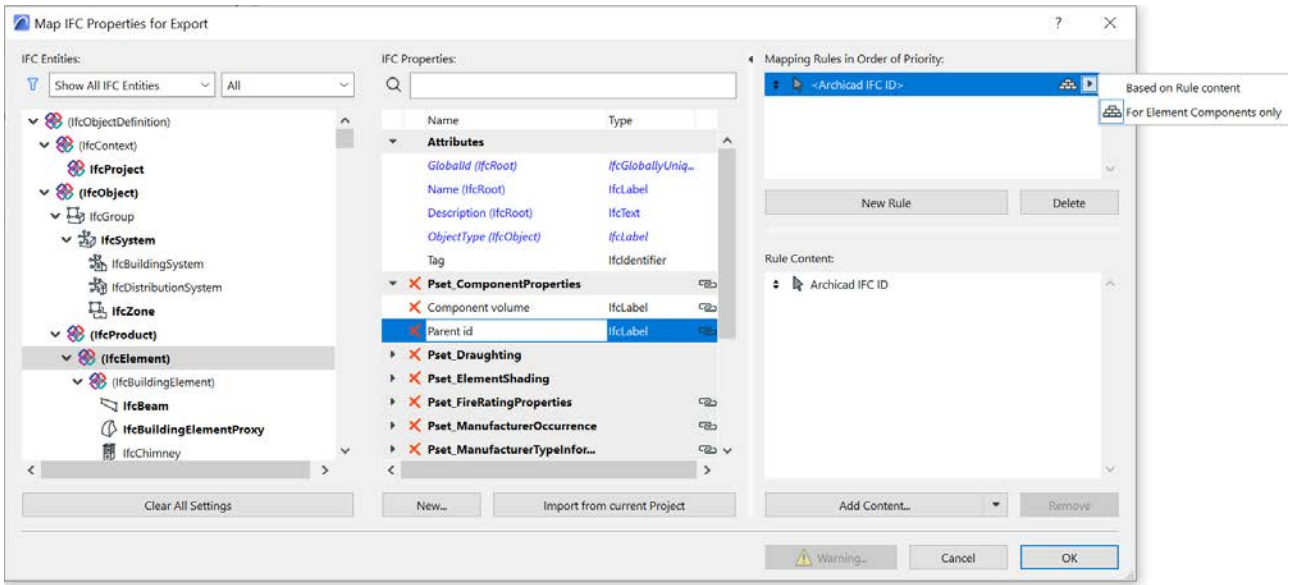
By default, IFC export automatically interprets component-level properties wherever applicable.

The default setting is “Based on rule content” which extracts property values of each element by skin. (This works for each part of simple, composite, and profile elements alike.)



You can also specify a property-mapping rule to apply ONLY to components (but not the parent element as such).

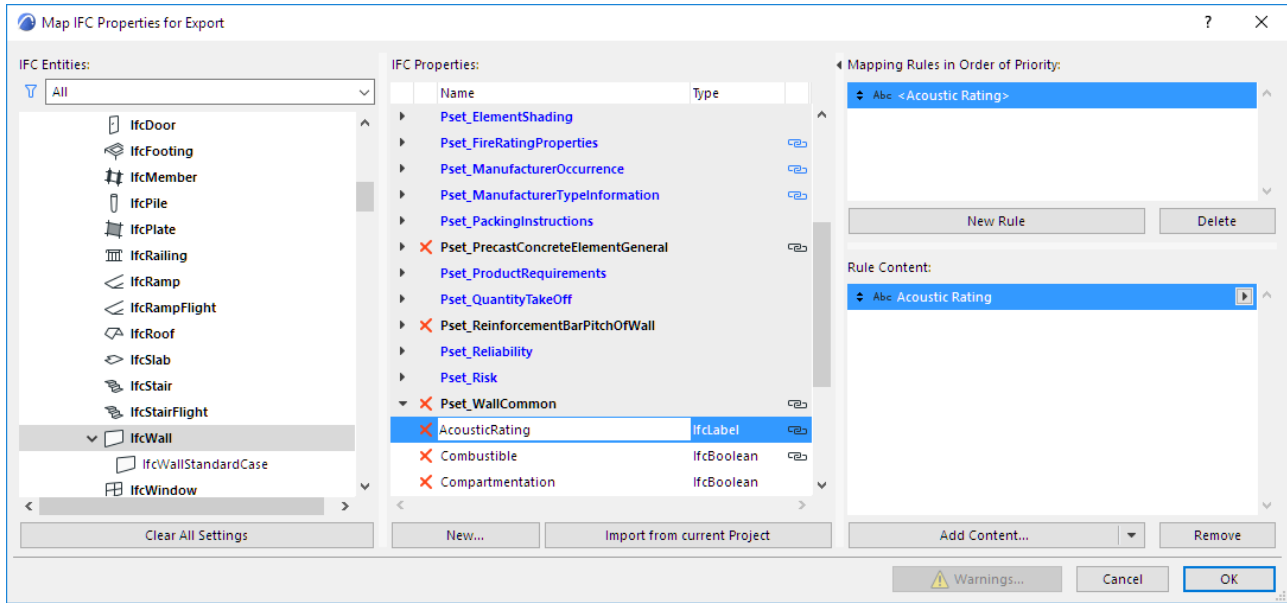
For example, create a rule to show the Archicad ID of each component’s parent element. Choose “Element components only” from the Mapping Rule pop-up:



Examples of Property Mapping Rules

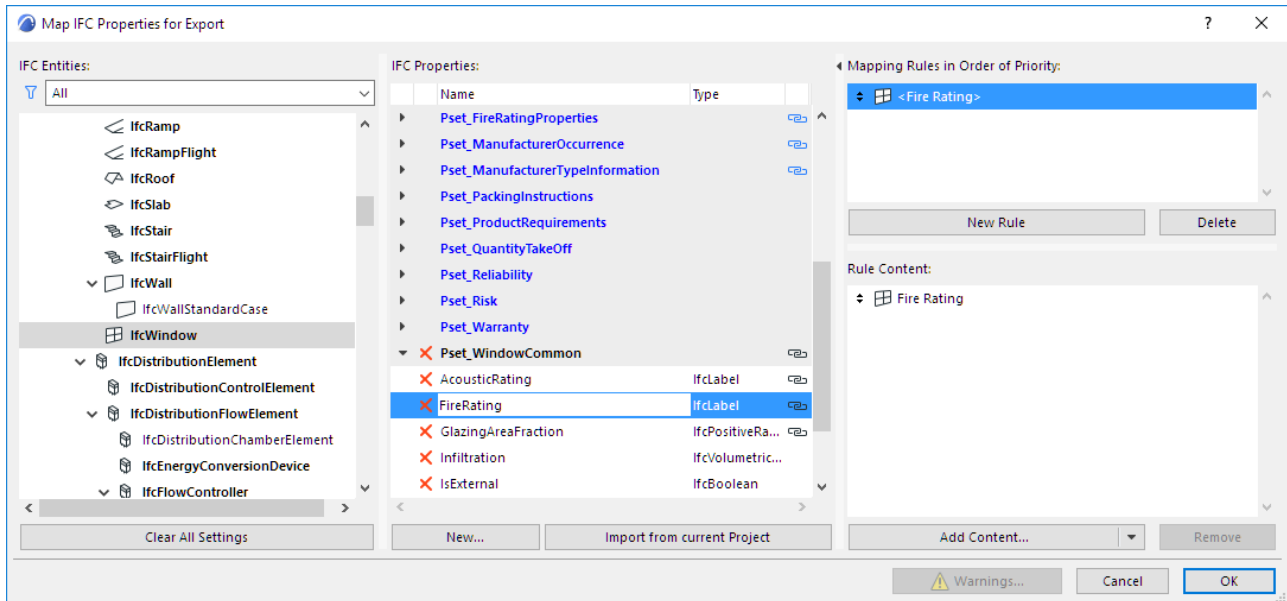
Example 1

Map the user-defined “Acoustic Rating” property (Options > Property Manager) to the “AcousticRating” property defined for the IfcWall.



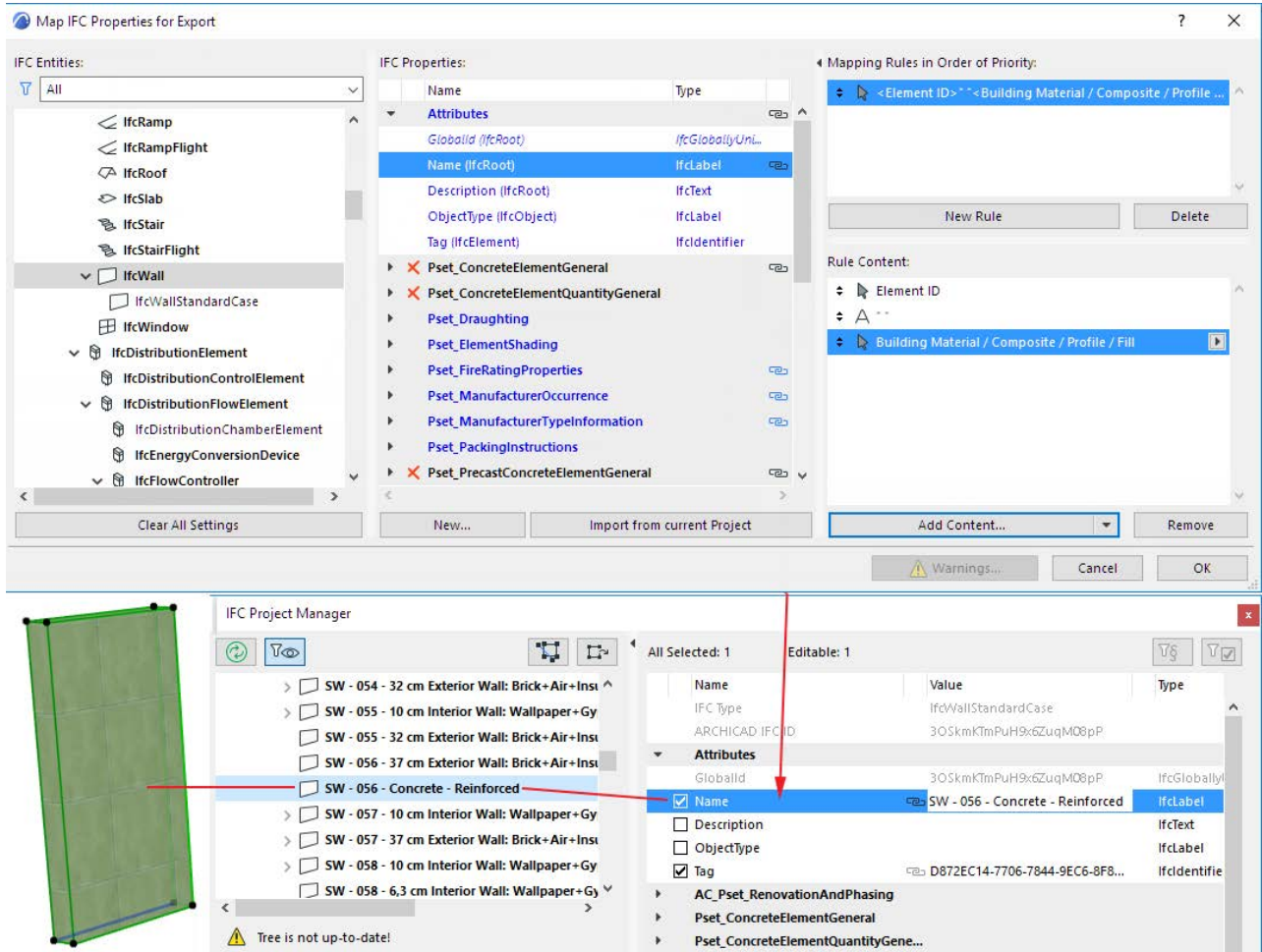
Example 2

Map the “Fire Resistance Rating” parameter of windows to the “FireRating” property defined for the IfcWindow.



Example 3

Define the “Name” Attribute of IfcWall elements using a combination of Wall “ID” and “Building Material/ Composite” Name.

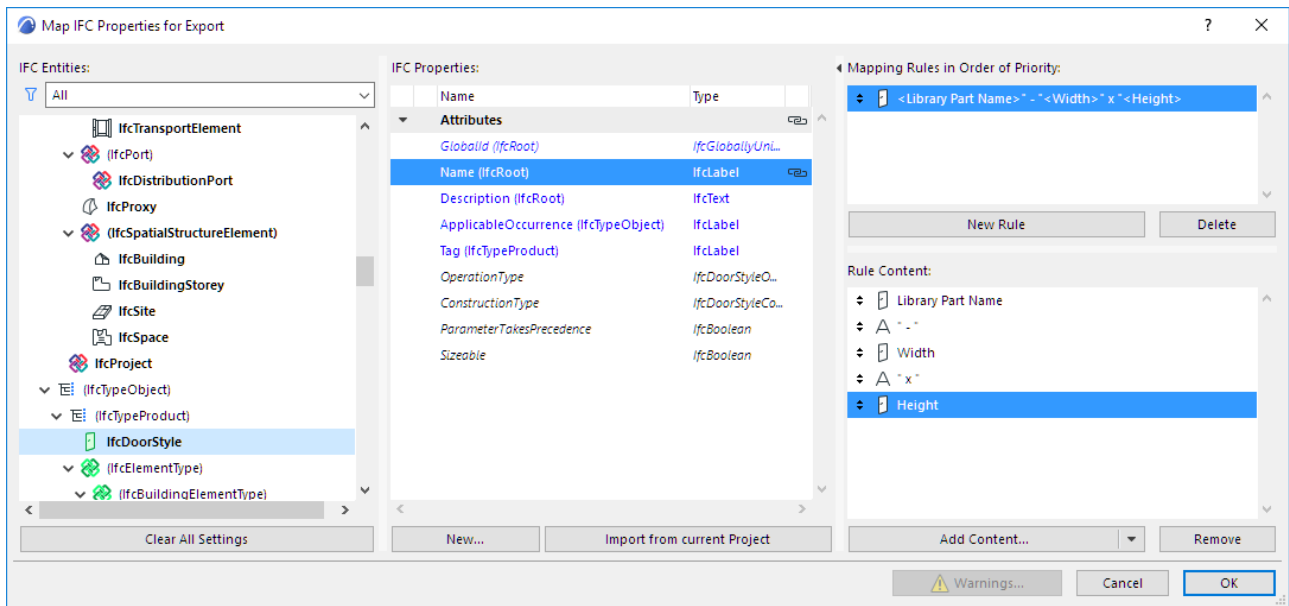


Example 4: Mapping Rule by IFC Type Product

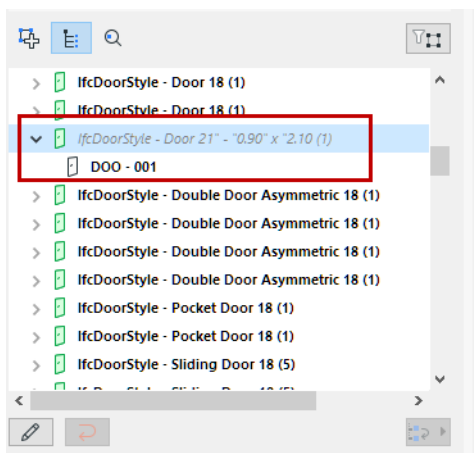
You can apply data mapping rules to the IFC data of IFC Type Product elements.

For example, generate the names of Door types (Name Attribute of IfcDoorStyle) from the combination of the Library Part Name, the Width and Height parameters of a Door.

Define the “Name” Attribute for IfcDoorStyle by using a fragment of the Door “Library Part name” + static text “-” + “Width” + static text “X” + “Height”.



This rule overwrites the default IfcDoorStyle creation rule ([see IFC Type Product](#)) with new IfcDoorStyle names (you can double-check the result in the IFC Project Manager).

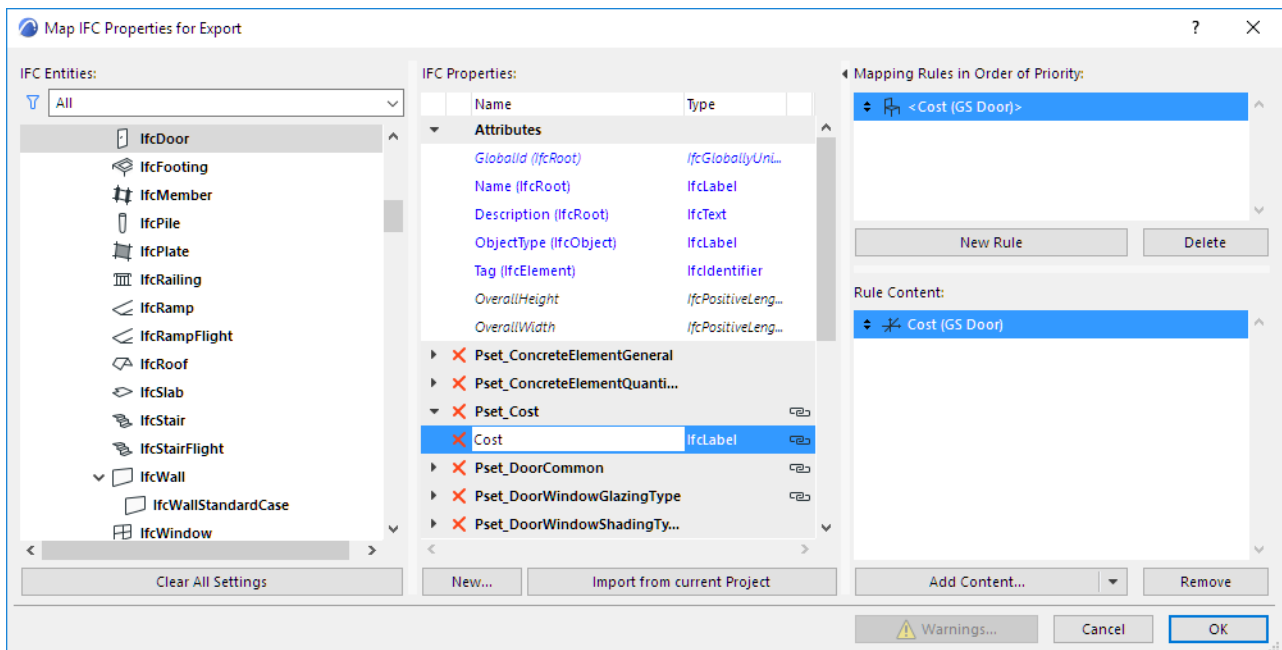
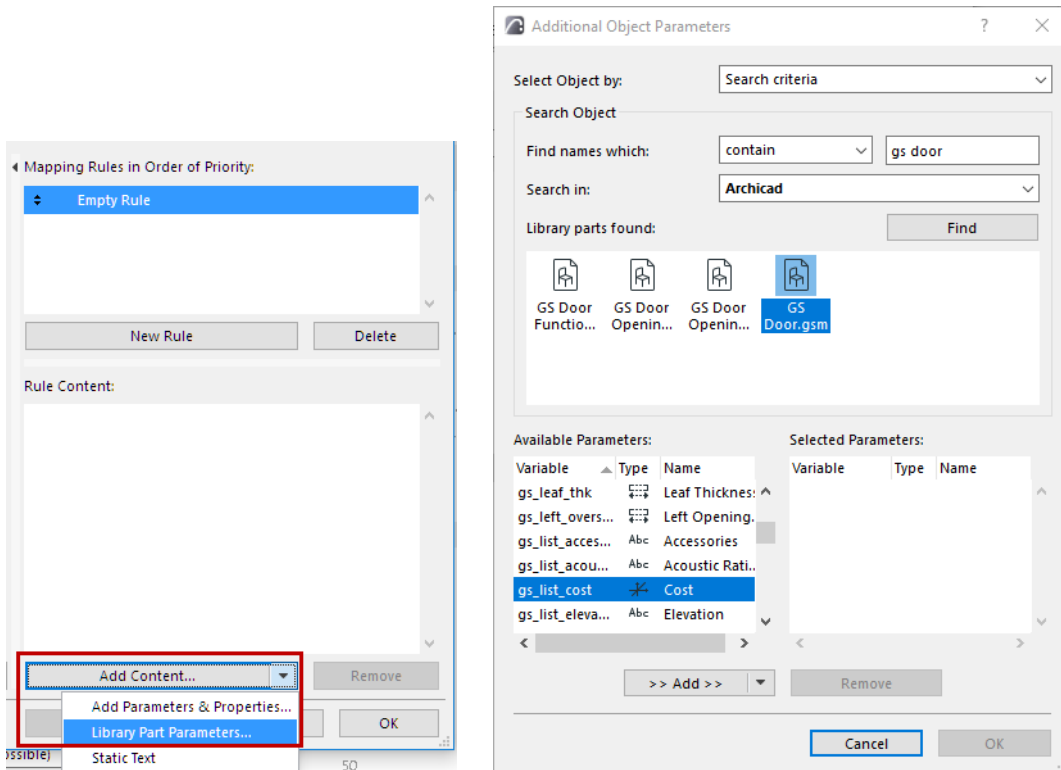


Example 5

Define the “Cost” Property for IfcDoor as Library Part dependent parameter.

For mapping, choose the “gs_list_cost” parameter of the “GS Door” Library Part.

To find this parameter, go to the **Add Content** drop-down button and choose **Library Part parameters**. From the appearing dialog box, search for the `gs_list_cost` parameter as shown:

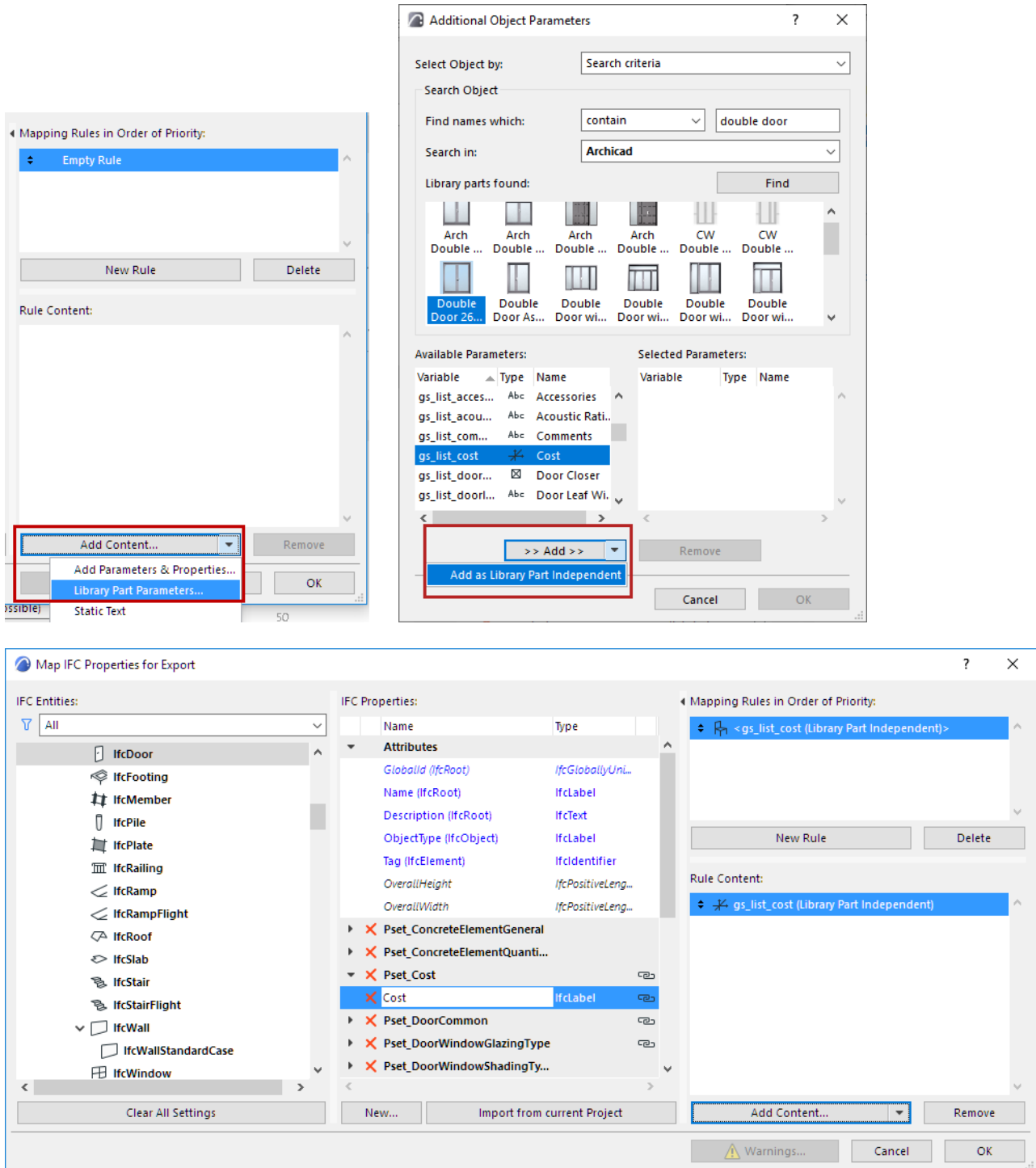


The mapping rule is valid for the Library Parts which are child elements of the “GS Door” subtype (e.g “Door”, “Double Door”, “Metal Door”, etc.).

Example 6

Define the “Cost” Property for IfcDoor as Library Part independent parameter. For mapping, choose the “gs_list_cost” parameter of the “Double Door”.

To find this parameter, go to the **Add Content** drop-down button and choose **Library Part parameters**. From the appearing dialog box, first select the Double Door library part, then search for the gs_list_cost parameter as shown. However, make sure to add it as a **Library Part Independent** parameter:



Example 7

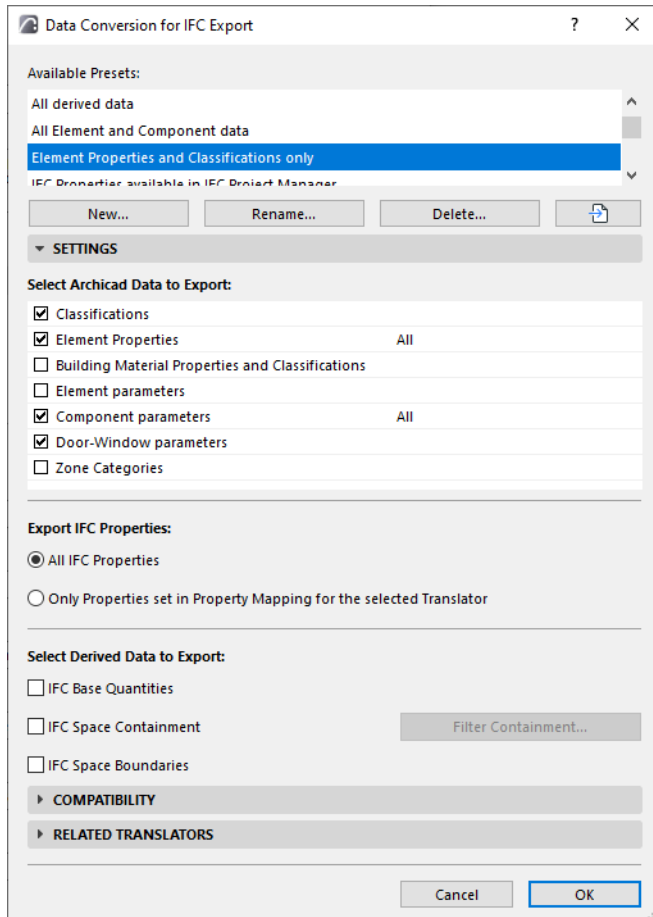
Define the “Cost” Property for **IfcElement** as Library Part independent parameter. For mapping, choose the “gs_list_cost” parameter of the “Double Door”. The mapping rule is valid for not just IfcDoor elements but **all other Library Parts** which will be converted to IFC Entities that are child element of IfcElement (e.g. IfcWindow, IfcFurnishingElement, IfcDistributionElement, IfcTransportElement etc.) and contain the “gs_list_cost” parameter (e.g. “Armchair”, “Desk”, “Basin”, “Sink” etc.).

Data Conversion for IFC Export

File > Interoperability > IFC > IFC Translators

Choose a Translator for Export and open the Data Conversion preset.

Use this dialog to define which data, in addition to element geometry, should be exported to IFC. Such data can be useful in data exchange with facility management, energy analysis or cost estimation programs.



Select Archicad Data to Export

Check the boxes of the Archicad Data types you wish to export.

The required data varies depending the purpose of the resulting IFC file (and the software that will be used to open and/or edit it.)

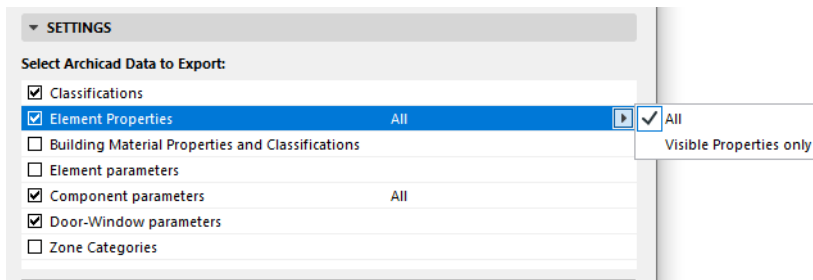
- **Classifications**

Archicad Classifications are exported as IFC Classification Reference data.

- **Element Properties**

Check to export Archicad Properties as IFC Properties. (Archicad Properties are defined in **Options > Property Manager**, and assigned to individual elements in their Settings dialog boxes or through the Interactive Schedule.)

- Select this list item and use the pop-up to choose to export **All** properties, or **Visible properties only**.



Note: The Visible Properties option is only relevant if you have loaded an Add-on which controls property visibility in Archicad. If no such add-on is loaded, then all properties are exported in every case.

- **Building Material Properties and Classifications**

Check to export Archicad Building Material Properties (defined at Options > Element Attributes > Building Materials) as IFC Properties, and Building Material Classifications as IFC Classification References.

In IFC 2x3: Exported Building Material data is associated to the exported elements.

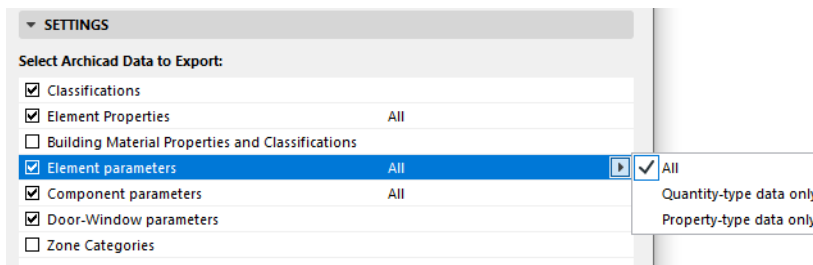
In IFC4: Exported Building Material data is associated to IFC Materials.

- **Element Parameters**

Check to export Archicad element parameters that convert to IFC Quantities or IFC Properties, depending on their type.

Note: Choosing this option will significantly increase IFC file size. Export parameters only if you know that the target application can read these data.

To filter the exported parameters by type: Select this list item and use the pop-up to choose to export **All** parameters, or **Quantity-type/Property-type data only**.



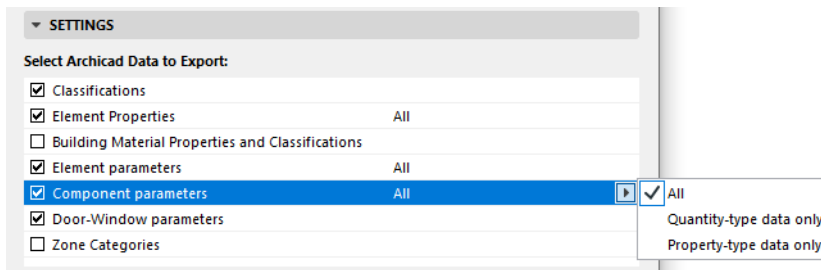
- Quantity-type data include e.g. Weight, Length, Area, Volume, Time. Useful for data exchange with programs that can read quantities.

- Property-type data include e.g. Library Part parameters.

- **Component Parameters**

Export parameters of Composite Skins and Complex Profile components as IFC Properties or IFC Quantities, depending on their type.

To filter the exported parameters by type: Select this list item and use the pop-up to choose to export **All** parameters, or **Quantity-type/Property-type data only**.



- Quantity-type data include e.g. Weight, Length, Area, Volume, Time. Useful for data exchange with programs that can read quantities.
- Property-type data include e.g. Library Part parameters.

As a result, the exported Component data can be interpreted in IFC receiving applications, such as Solibri.

- **Door-Window Parameters**

Export detailed information about Archicad openings as IFC Door-Window Lining or Panel properties.

- **Zone Categories**

Export the Zone Categories data (Code and Name) of Archicad Zones as IFC Space Classification Reference data (ItemReference and Name).

Export IFC Properties

Choose an option for which IFC Properties and Classification References to export:

- All IFC Properties (all which are visible in the IFC Project Manager, no filter on)
- Only Properties set in the Property Mapping Preset for the selected Translator. (These properties are shown in the IFC Project Manager if you turn on the **Show Only Properties from Preview Translator** filter)

Select Derived Data to Export

Check one or more types of data to export to the IFC model.

- **IFC Base Quantities**

Check this to export Quantity Takeoff parameters (size, area and volume) to IFC. Useful in cost estimation applications.

The following table summarizes the base quantities by entity types automatically calculated and exported when using this option. The values of IfcSite's base quantities can be set manually at **Info > Project Info** (Site Gross Perimeter and Site Gross Area).

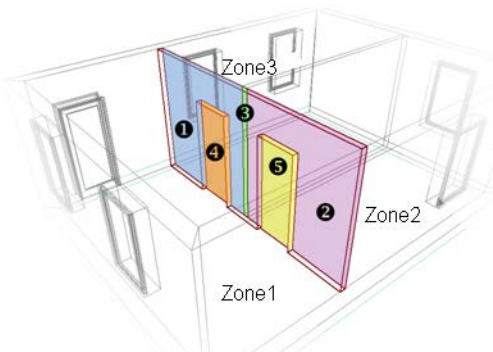
IfcSite	Zone (IfcSpace)	IfcWall	IfcCurtainWall	IfcBeam
GrossArea	Height	Length	Width	Length
GrossPerimeter	FinishCeilingHeight	Height	Length	Width (rectangle profile only)
	FinishFloorHeight	Width	Height	Depth (rectangle profile only)
IfcBuilding	GrossFloorArea	GrossFootprintArea	GrossSideArea	CrossSectionArea
GrossFloorArea	NetFloorArea	NetFootprintArea	NetSideArea	OuterSurfaceArea
	GrossCeilingArea	GrossSideArea		GrossSurfaceArea
Story (IfcBuildingStorey)	NetCeilingArea	NetSideArea	IfcDoor/ IfcWindow/ IfcOpening	TotalSurfaceArea
Height	Gross WallArea	Gross Volume	Height	Gross Volume
NetHeight	NetWallArea	NetVolume	Width	NetVolume
GrossHeight	GrossPerimeter		Area	NetSurfaceArea ExtrudedSide
GrossFloorArea	NetPerimeter	IfcSlab	Depth	
	Gross Volume	Width	Volume	IfcColumn/ IfcMember
	NetVolume	Perimeter	Perimeter	Length
	SpaceNetFloorArea BOMA	GrossArea		Width (rectangle profile only)
	SpaceUsableFloor AreaBOMA	NetArea		Depth (rectangle profile only)
		Gross Volume		CrossSectionArea
		NetVolume		OuterSurfaceArea
				TotalSurfaceArea
				GrossVolume
				NetVolume

Export IFC Base Quantities

- **IFC Space Containment:** Export data defining the relationship between Archicad Zones and objects they contain. This feature is useful in facility management applications.
 - Space Containment is relevant only for elements, within an Archicad Zone, which were created using the Object, Lamp, Morph or Slab tools.
 - For Object, Lamp or Morph elements: These elements are in space containment only if their home story is the same as that of the Zone, and if the center of the elements' bounding box falls within the Zone's 2D polygon.
 - A Slab is in space containment if its reference line falls within the Zone top/bottom, and if the Slab polygon falls within the Zone polygon.
 - Click **Filter Containment** to limit the contents of the Space Containment by their IFC Element Type.

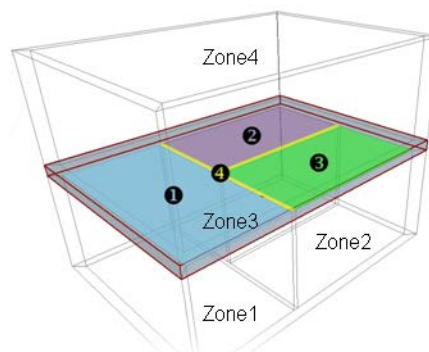
- **IFC Space Boundaries:** Export geometry data of Archicad Zones. (Useful in thermal calculation applications.)

Archicad Zones include precise geometry data that are useful to thermal calculation software applications. Zones are 3D solids in geometrical aspect and in normal cases are bordered by two slabs and four walls. Space boundaries define the logical connection between Archicad Zones (IfcSpace) and the building elements that enclose them. In practice, Walls, Slabs, Roofs, Windows, Doors etc. all have different thermal conductivity properties. If you activate the “IFC Space boundaries” option, Archicad will export Space boundaries and their relations (IfcRelSpaceBoundary) together with the Zones (IfcSpace) to the IFC file. In other words, Archicad will calculate the position, size and adjacency of the elements that border each Zone. Archicad divides the Zone boundaries according to the areas defined and cut by connected elements and openings.



Space Boundaries (defined by Walls):

- 1 - Wall part defined by Zone3
- 2 - Wall part defined by Zone2
- 3 - Wall part cut by connected wall
- 4 and 5 - Area defined by door opening



Space Boundaries (defined by Slab):

- 1 - Slab part defined by Zone1
- 2 - Slab part defined by Zone3
- 3 - Slab part defined by Zone2
- 4 - Slab part without thermal convection

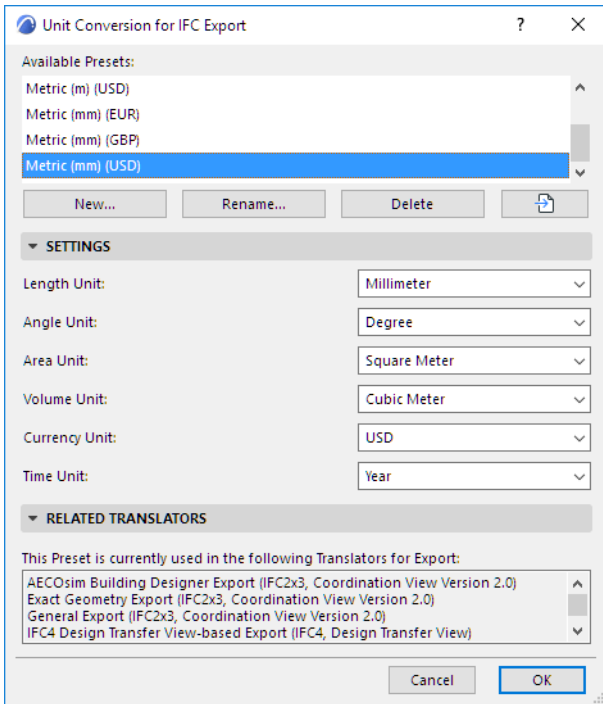
See also [Energy Evaluation: Direct BIM to BEM](#).

Unit Conversion for IFC Export

File > Interoperability > IFC > IFC Translators

Choose a Translator for Export and open the Unit Conversion preset.

Set unit types globally for the export of all coordinates, geometric parameters and “Measure”-type IFC Properties.



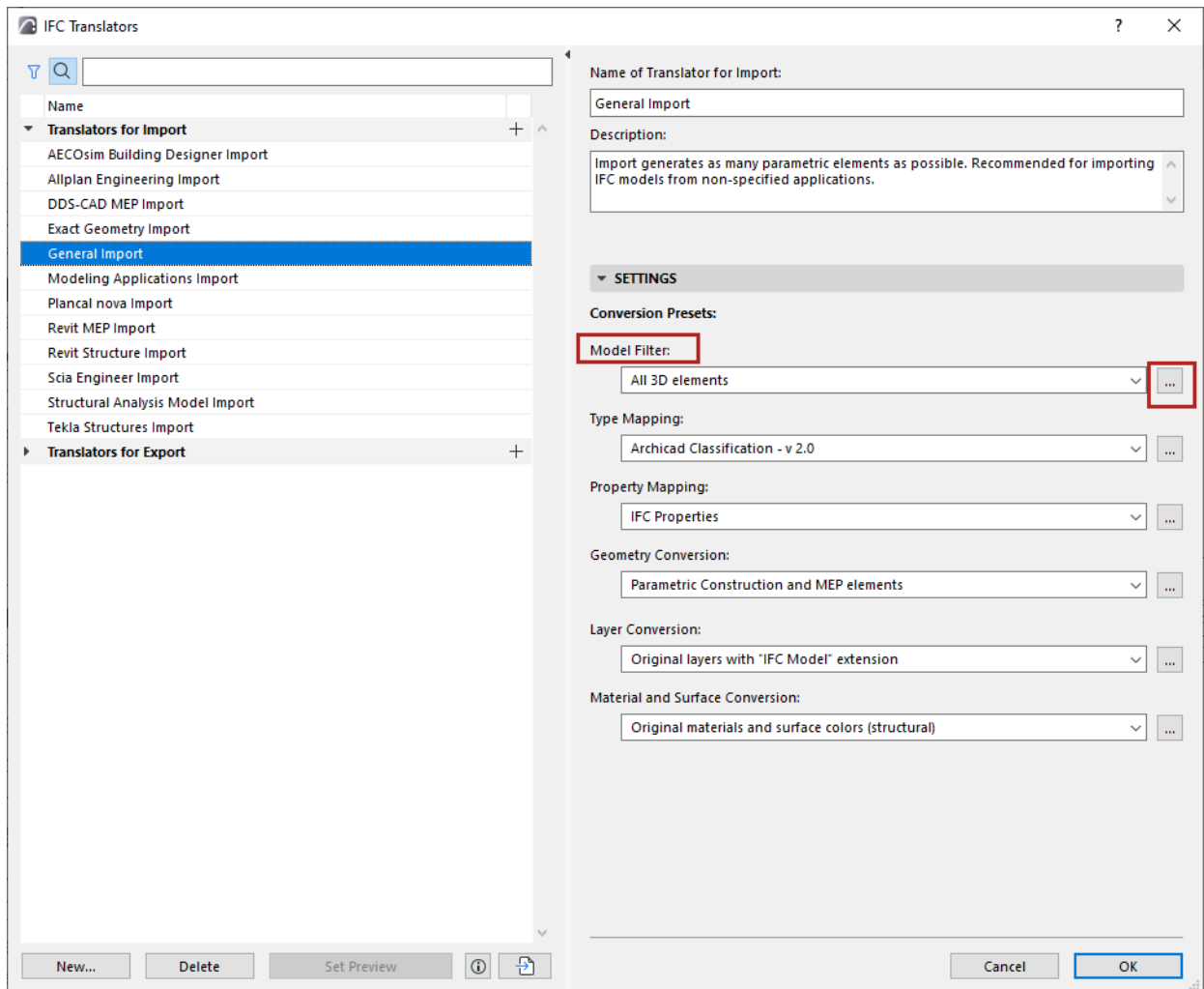
Filter Model at Import (Secondary Filter)

When Hotlinking, Merging or Opening an IFC file, you must select an IFC Translator for Import, which controls the conversion process. Every IFC Translator for Import has a filtering component (see *Model Filter for IFC Import*), which defines which elements of the IFC model should be placed in Archicad (and which should be skipped).

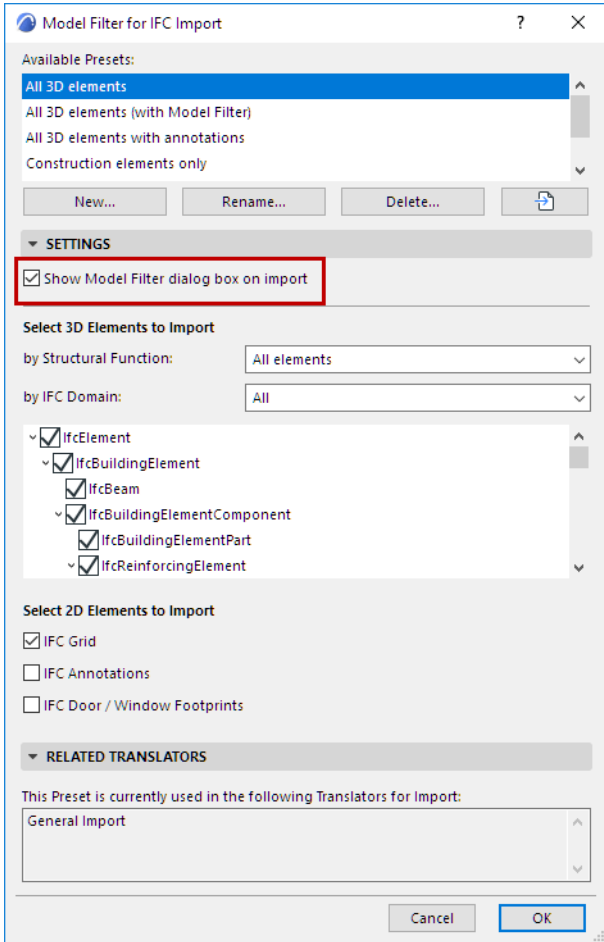
Each Translator for Import also contains another, secondary Model Filter function. This dialog (Model Filter for Import) allows you to further fine-tune the set of imported IFC elements, based on more specific element categories - e.g. type, story, owner.

However, this secondary Model Filter for Import dialog does not appear by default. To use this feature, turn it on in IFC Translator Settings:

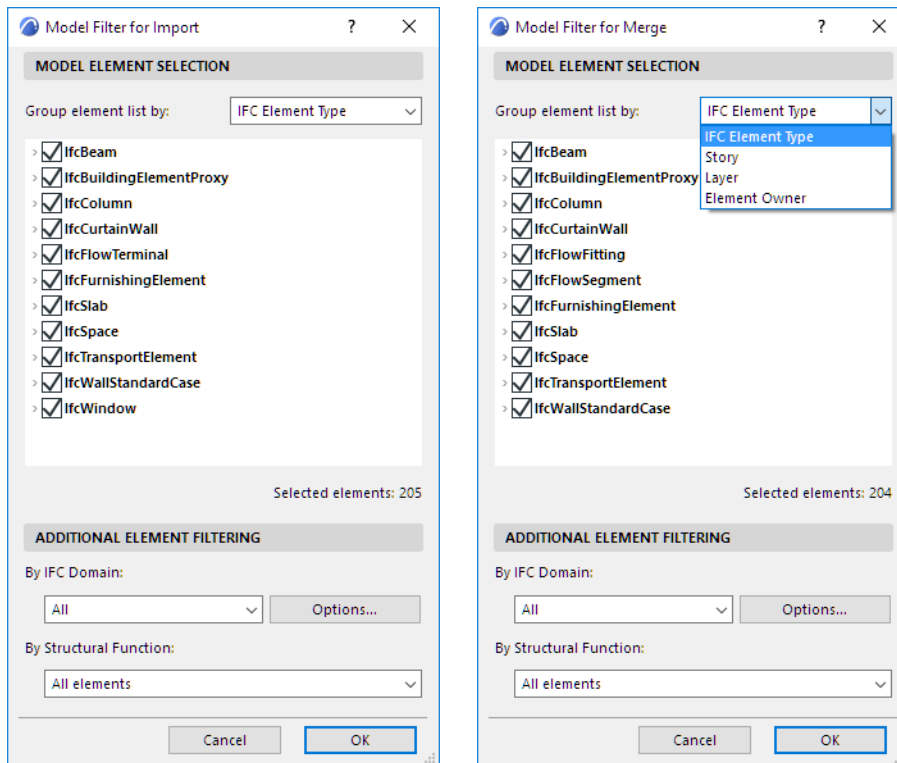
1. Open the template file whose IFC translators are being used.
2. Go to **File > Interoperability > IFC > IFC Translators**.
3. Select a Translator for Import.
4. Under Settings, open **Model Filter Settings**.



5. Check the **Show Model Filter dialog box on import** on import box.



Now the **Model Filter for Import** dialog will appear when you open, hotlink or merge an IFC file to Archicad. Use it to apply custom element filters to the current import process.



Group element list by

To make filtering easier, sort the list items or grouping them as follows:

- **IFC Element Type**
- **Story:** List elements by their story location
- **Layer:** List IFC element types and elements by layer
- **Element Owner:** List element types and elements according to who created them in IFC

Note on Element Owner

- Elements in IFC models always have an owner attached. If the IFC file is exchanged among many users and programs, an element may have multiple owners.
- The IFCProject entity, like all IFC entities, also has an owner. When an IFC project is *imported* to Archicad, the IFCProject entity's owner are added to Archicad's Project Info. At *export* from Archicad, the model elements are all assigned to this owner.

Additional Element Filtering

If needed, filter elements further by IFC Domain or by Structural Function.

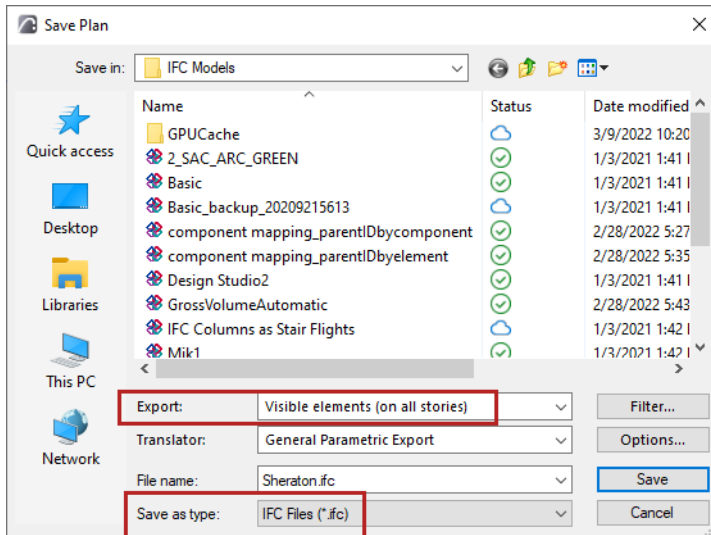
Filter Model at Export

Each IFC Translator contains settings for filtering the elements to be exported.

However, you can further fine-tune the set of exported elements during the export process.

IFC Save Options: Elements to Export

1. Go to **File > Save as**.
2. Choose an IFC file format.



3. In the **Export** field, choose an option:

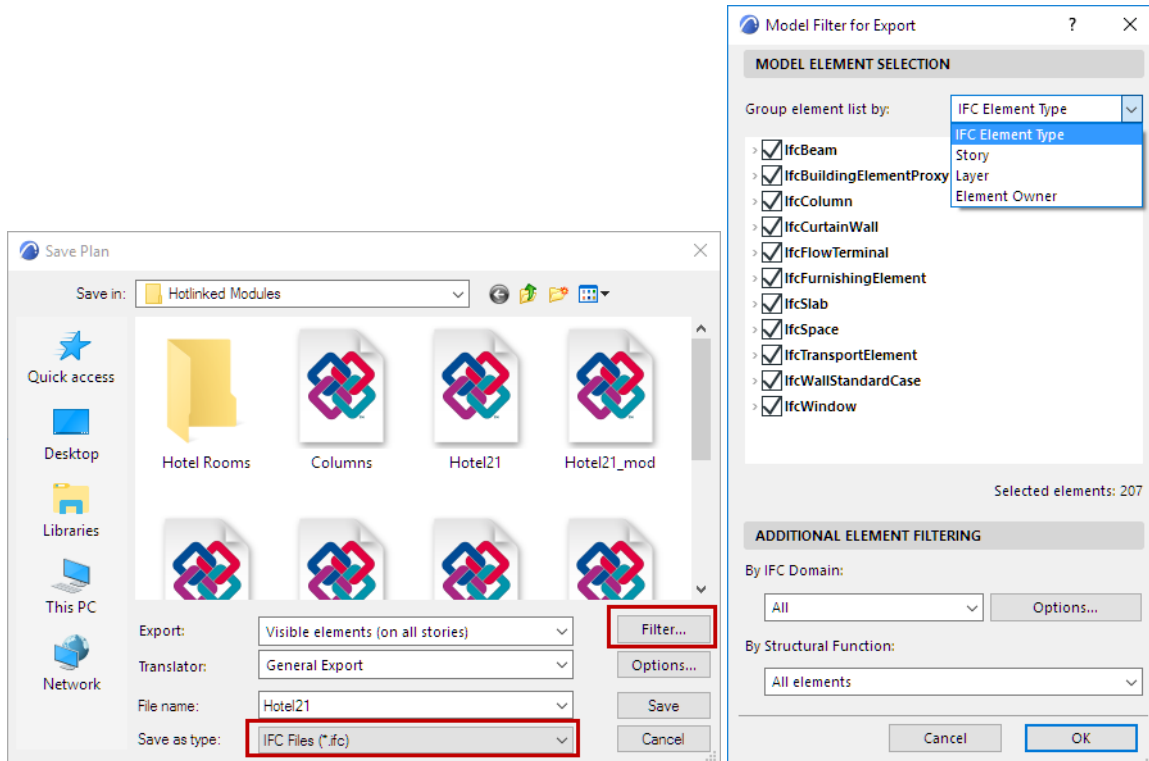
- **Selected elements only** in the current view (this option is only available if the current view includes some selected elements)
- **Visible elements on all stories**. When saving from Floor Plan view, you can choose to export visible elements on just the **current story**. Visibility depends on display settings (Layers, Partial Structure display, etc.).
- **Entire project**, regardless of any selection or layer settings.

Notes:

- In order to export 2D drawing-type elements, annotation element or Grid elements, it is not enough to meet the criteria of the filters listed above (e.g. export based on selected or visible elements); you must also specifically enable their export, using the translator settings. ([See Select 2D Elements to Export.](#))
- Publisher always uses the 'visible' element export mode, independently of the Model Filter settings of the applied IFC Translator for Export.

IFC Save Options: Model Filter

From the Save dialog, click the **Filter** button to open this dialog box.



Use this list to further filter the elements to be exported.

Group element list by

To make filtering easier, sort the list items or grouping them as follows:

- **IFC Element Type**
- **Story:** List elements by their story location
- **Layer:** List IFC element types and elements are listed by layer.
- **Element Owner:** element types and elements are listed according to who created them. (At export, all elements are considered as belonging to the same owner.)

[See also Note on Element Owner.](#)

Additional Element Filtering

If needed, filter elements further by IFC Domain or by Structural Function.

Appendix: IFC Terms and Concepts

The following sections provide additional detail about terms and concepts used in IFC.

[Model View Definitions](#)

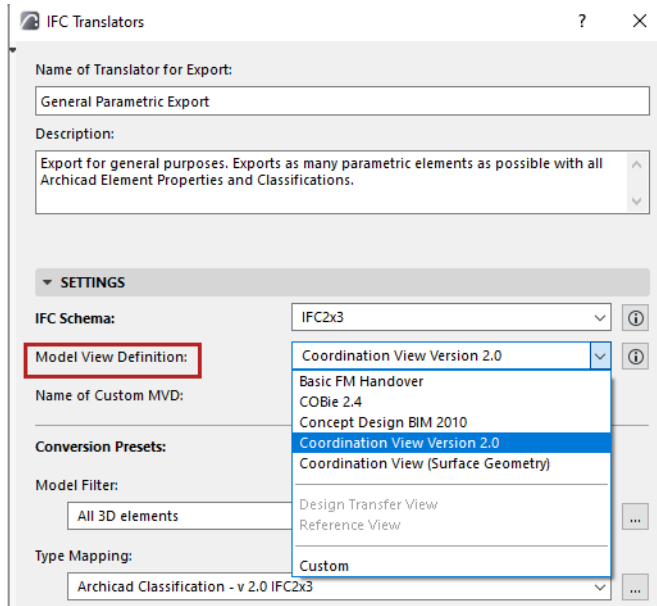
[IFC Data Types](#)

Model View Definitions

A *Model View Definition*, MVD (or IFC View Definition) is a recommendation for which data and elements the IFC model should include, depending on the purpose of the model exchange.

Model View Definitions are either defined within buildingSMART International, or by other organizations and interest groups.

Archicad IFC Export and Import supports the following Model View Definitions (choose an MVD in the Translator for Export Settings):



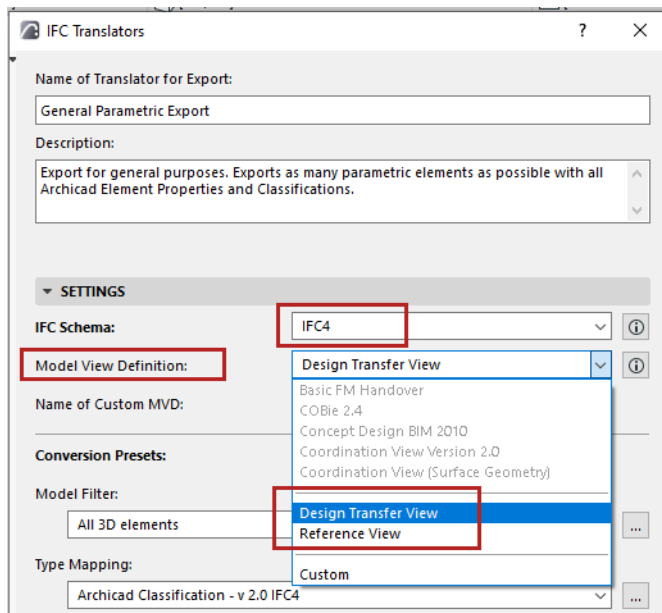
IFC2x3 Schema

- The **Coordination View** is currently the most widely implemented view of the IFC Schema. Its main purpose is to allow sharing of building information models among the disciplines of architecture, structural engineering, and building service. It contains definitions of spatial structure, building, and building service elements. Archicad is certified for both import and export of IFC2x3 Coordination View 2.0.
- The IFC2x3 **Coordination View (Surface Geometry)** is a simplified publishing format (i.e. a subset) of the Coordination View. This format is suitable for viewing (since all IFC viewers support this format), design coordination, clash prevention during design, and clash checking, similarly to IFC4 Reference View. “Coordination View (Surface Geometry)” means that every element will be exported with its BREP (boundary representation) geometry. This method comes closest to reproducing the real shape of the element, together with its specialized sections, connections and solid operations. (However, the element’s parameters are lost, and BREP elements from an imported IFC file are transformed into non-editable elements.)
- The IFC2x3 **Basic FM Handover View** is an extended version of the Coordination View (developed by buildingSMART) that defines the general requirements for design applications to enable the handover of facility management information. The basic scope can be summarized as the space and equipment list for the spatial and technical systems of a facility. Technically, the Basic FM Handover View fulfills the following requirements above and beyond those of the abovementioned Coordination View: ability to assign furnishing and equipment components to spaces (Space Containment, IFC System), to assign

spaces to zones (IFC Zone), to assign a classification to spaces and components (Classification Reference), to assign manufacturer base properties to components (standard and custom IFC Properties), to assign doors and windows to spaces (Space Boundaries), to assign type information for components (IFC Type Product) and to export base quantities for all project components and spatial structures. This MVD is required in various projects defined by buildingSMART and other organizations. An example is the COBie (Construction-Operations Building information exchange), a specification used in the handover of Facility Management information. It is a spreadsheet data format for the delivery of a subset of building model information, rather than geometric model information. Archicad's BIM-quality models and IFC data exchange capabilities produces data output that is easily converted into COBie documentation, with the help of free or commercial conversion programs.

- There are several other Model View Definitions (generally the extended versions of the Coordination view) specified by organizations or development teams outside of buildingSMART International.

IFC4 Schema



The **IFC4 Schema** further divides the Coordination View into two Model View Definitions:

Note: The IFC4 Schema is not supported by all applications. Use an MVD of the IFC4 Schema only if you are certain that it is supported by the application that will receive IFC data.

- **IFC4 Reference View:** Suitable for all BIM workflows based on reference models, where the exchange is mainly one-directional. Requested modifications of the BIM data, mainly of the shape representation, are handled by a change request to the original author.
 - When exporting to IFC4:
 - the relation connections of chained Walls and Beams are omitted
 - Surfaces of Building Materials are omitted
- **IFC4 Design Transfer View:** Provides building information with support for editing of interconnected elements: inserting, deleting, moving, and modifying physical building elements and spaces, within the limited scope of parametric exchange. An example of a target scenario is an architect providing building design information to an engineer for a particular discipline, where geometric modifications may be needed. Note that the Design Transfer View is not meant for round-trip model exchange scenarios.

All Model View Definitions can be extended by definitions which support additional exchange requirements:

- the Quantity Take-off add-on view adds the ability to transmit base quantities for all spatial and building elements
- the Space Boundary add-on view adds building elements to space relationships to support models in thermal and energy analysis
- the 2D Annotation add-on view supports the exchange of additional 2D element representations and annotations of building models

IFC Data Types

This chapter summarizes the main IFC data types available in Archicad.

[IFC Entity](#)

[IFC Model Hierarchy](#)

[IFC Type](#)

[IFC Type Product](#)

[IFC Container](#)

[IFC Attributes](#)

[IFC Properties](#)

[IFC Classification Reference](#)

[IFC Assignments](#)

IFC Entity

In an IFC model, the project information (generated, for example, from an Archicad project) is represented as a set of IFC Entities - such as IFC Type, Attribute, Assignment, Type Product. Each IFC Entity (for example, an IfcWall) includes a fixed number of IFC Attributes, plus any number of additional IFC Properties.

Some IFC Entities express characteristics of other entities. Some of these correspond to Archicad attributes. These correspondences are mapped automatically when the IFC model is generated or exported, or when the IFC model is imported.

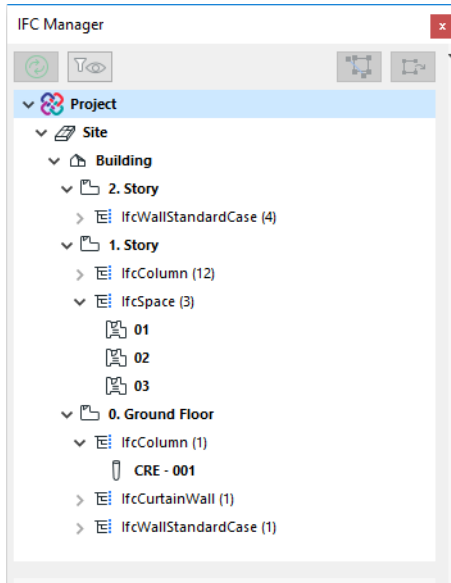
Archicad Attribute	IFC Entity
Layer	IfcPresentationLayerAssignment
Building Material	IfcMaterial
Surface	IfcSurfaceStyleRendering
Composite	IfcMaterialLayerSet (extruded geometry) or IfcMaterialList (BREP geometry)
Profile	IfcProfileDef

You can set different preferences for converting Layers, Building Materials and Surfaces of imported IFC models: use the conversion settings of Import translators.

See [Layer Conversion for IFC Import](#) and [Material and Surface Conversion](#).

IFC Model Hierarchy

An IFC model is composed of IFC Entities built up in a hierarchical order. As shown in the following illustration, each story level (IfcBuildingStorey) has its corresponding story in Archicad.



In Archicad, all elements and objects are linked to a home story. Thus, by default, the IFC model hierarchy lists them under the IfcBuildingStorey of the same name as the elements' home story. However, in IFC Project Manager, elements can be assigned directly to the IfcSite or IfcBuilding. So, for example, neighboring buildings (e.g. defined by the Morph tool) and site context elements (trees, fences, roads, etc.) can be assigned to the site instead of the project building with its story system.

By default, the IFC elements (for example the IfcBuildingElements) and the IfcSpace entities are on the same hierarchy level in an IFC model. However, Archicad Objects and Morphs - regardless of their Classification - can be displayed on and, for IFC export purposes, linked to the Archicad Zone (IfcSpace) that contains them, instead of to their home story.

IFC Spatial Elements

IFC Spatial Elements are **IfcProject**, **IfcSite**, **IfcBuilding**, **IfcBuildingStorey**, as well as **IfcSpace**. They are at the top of the IFC Project Manager Hierarchy.

These elements can only be managed through the IFC Project Manager, where you can also edit their IFC-related data.

The IFC Attribute values of these spatial elements, however, are derived from the data of the current Archicad project.

IfcSite is the geographic position of the IfcProject. It can have a geometry, but a geometry is not required. In Archicad, site geometry is represented by Mesh elements, Site-type Objects, or elements with "Site Geometry" Classification.

Archicad can handle and import one building (**IfcBuilding**) only. But it can import more than one IfcSite, even though the IfcSite is above the IfcBuilding in the hierarchy. Thus, when importing an IFC file that contains multiple buildings, you can choose only one to import.

[See *Building and Site Selection*.](#)

However, other applications can export a building's topography into multiple IFC Sites. In this case, importing the IFC Building will import all of the IFC Sites, merged into the one IFC Site that actually contains the imported IFC Building. As a result, you will end up with a single IFC Site and a single IFC Building, but the IFC Site includes all of the IFC Sites.

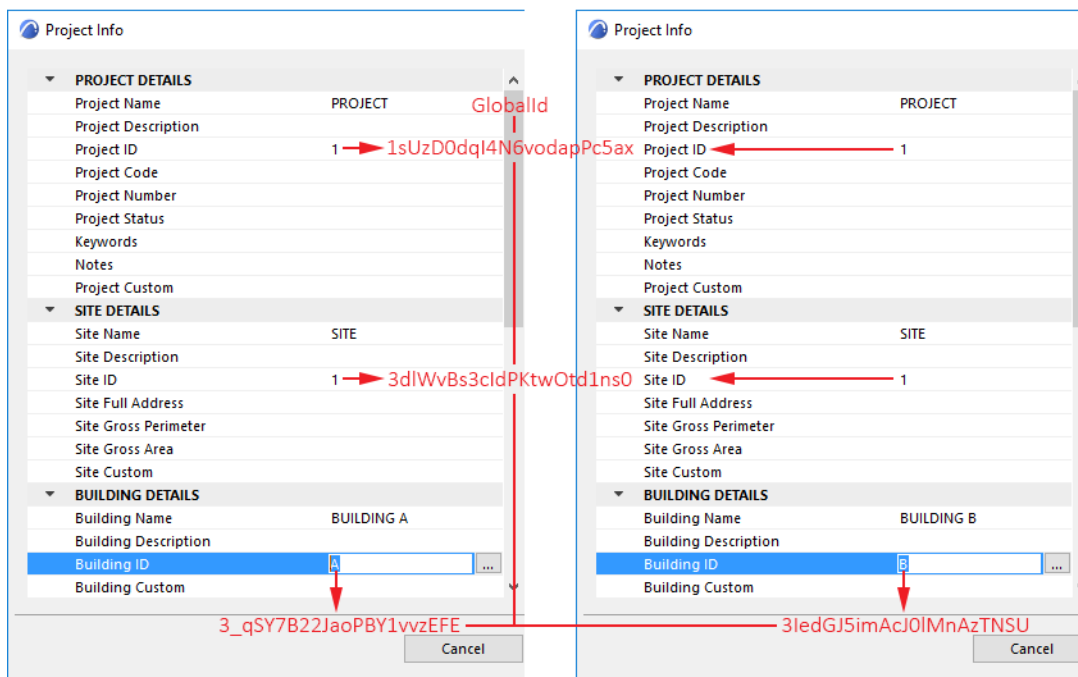
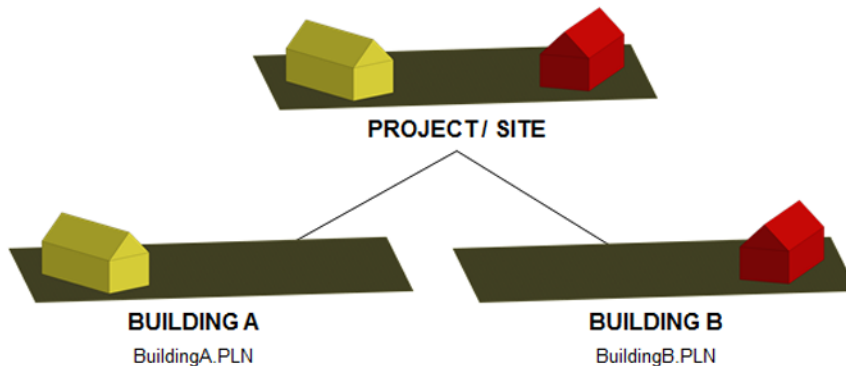
How to Control Global ID (IFC Attribute) Based on Archicad Project Info

By default, Archicad assigns identical GlobalID attributes to the following entities: IFC Project, IFC Site, IFC Building, and IFC Story. More precisely: the Global IDs will be identical, provided that the respective ID fields, defined at Archicad's Project Info dialog box (File > Info) contain identical keywords (or no value at all).

However, you can alter these keywords in the Project Info dialog box to control whether the spatial entities have identical or different GlobalIDs.

Suppose you have two separate buildings (stored in two different Archicad projects). You want both projects and both sites to share the same GlobalID; yet you want the various buildings on each site to have different GlobalIDs.

To achieve this, enter the same keyword in the "Project ID" and "Site ID" fields of both projects. Enter different keywords for the "Building ID" fields.



IFC Story entities have no corresponding ID field in the Project Info dialog box. The GlobalIDs of IFC Story entities are derived from their Building ID. If two buildings in two different Archicad projects have the same Building ID, then all stories of those buildings which have the same Story Number will have the same Global ID.

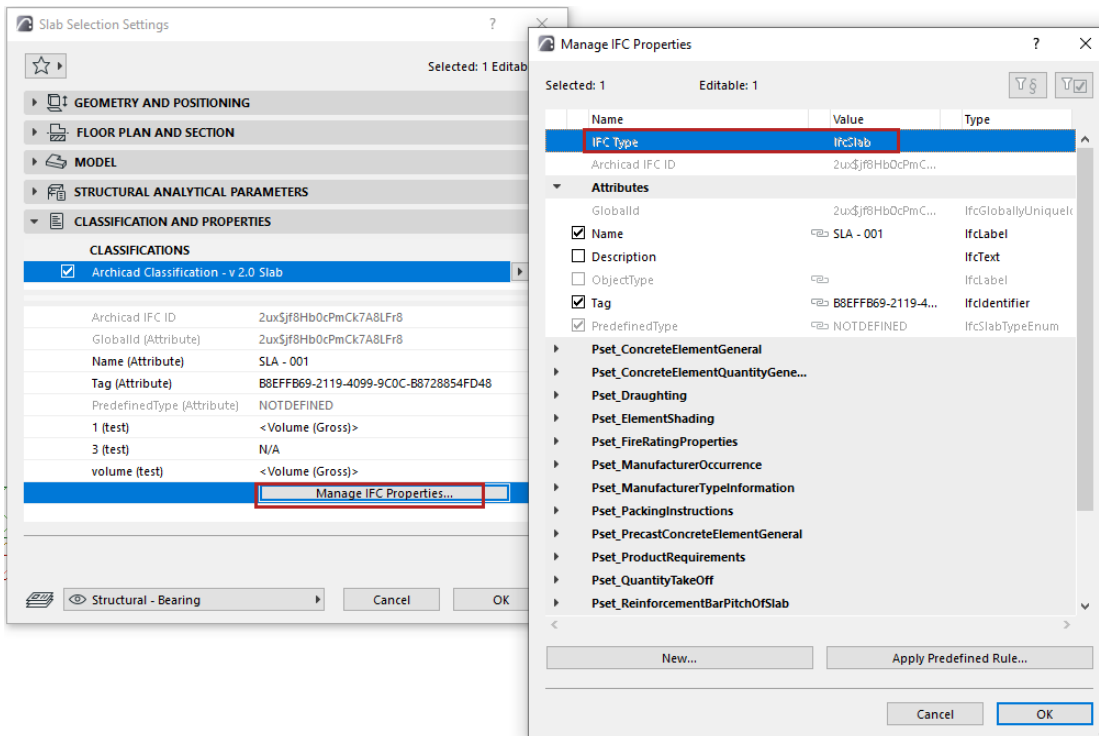
IFC Type

IFC Type is one kind of IFC entity - it groups IFC model elements into a hierarchical structure, represented by the tree structure in the IFC Project Manager.

When a model is exported to IFC format, every one of its elements is assigned an IFC Type. In Archicad, IFC Types are generated based on the Type Mapping definition (either by Tool or by Classification) in the Translator for Export.

See [Type Mapping for IFC Export](#).

in Archicad, view any element's IFC Type in the Manage IFC Properties window of Element Settings (or for any selected element in IFC Project Manager).



You can also use Find & Select to filter the project by "IFC Type" criteria, or use Interactive Element Schedule to list this data for Archicad elements. For example, you can select or list just the model elements having IfcSlab IFC Type.

See [Define Element Criteria using IFC Data](#).

IFC Type Product

An IFC Type Product defines a particular style/type of other entities by relating to them with common IFC Attributes and Properties. For example, IfcWindowStyle is an IFC Type Product, to which many windows (IfcWindow) refer.

IFC Type Product entities are generated for Archicad elements based on the Type Mapping definition in the Translator for Export.

[See *Type Mapping for IFC Export*.](#)

IFC Container

IFC Container (relating object of IfcRelAggregates) is an IFC entity that does not have its own body geometry, but its components (related objects of IfcRelAggregates) contain all geometry and structure-related data.

Exporting Hierarchical Elements from Archicad as IFC Containers

In an Archicad project, the IFC Translator for Export gives you the option of exporting hierarchical elements (Curtain Wall, Stair, Railing) to IFC as either a single element, or as a hierarchical one.

- If exported with the hierarchical method (“Keep Hierarchy”), it will be an **IFC Container** entity, and its sub-elements will be individual IFC elements. This reflects the multi-part hierarchy of the original Archicad element (e.g. Stair).
- If exported as a single element, it will be a **single IFC entity**, which contains within itself the geometry of all its sub-elements. The hierarchical nature of the original element (e.g. Stair, with all its separate sub-components) is thus lost.
- IFC export that uses the “Explode Composite and Complex Profile elements into parts” option generates **IFC Container** elements from building elements in the output IFC file: for example a composite wall element is saved as an “IfcWall”-type IFC Container element without geometry, and its skin parts will provide the geometry as “IfcBuildingElementPart” elements.

[These export options are found at *Geometry Conversion for IFC Export*.](#)

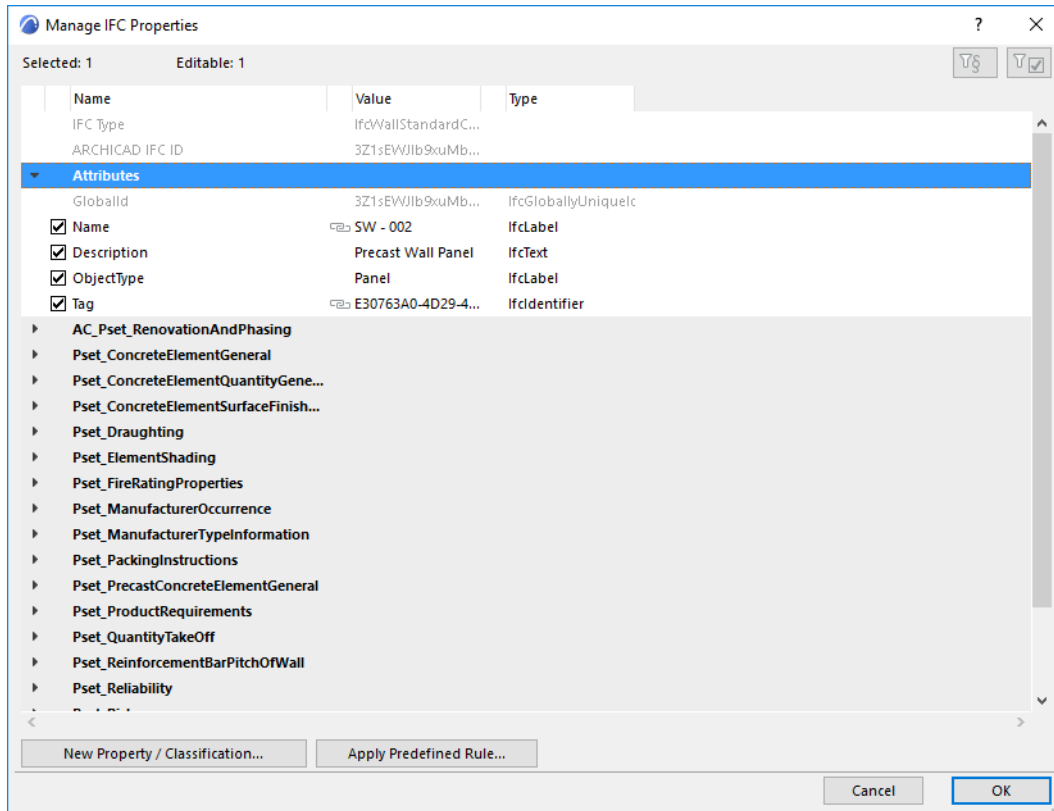
IFC Attributes

IFC Attributes are the main identifiers of IFC Entities. The names of IFC Attributes are fixed, having been defined by buildingSMART as part of the IFC standard code.

Most of these IFC Attribute names are derived from the project. For example, the IFC Attributes of an IfcWall building element:

- GlobalId: the Globally Unique Identifier of the IfcWall in the IFC model (not editable).
- OwnerHistory: the assignment of the information about the current ownership of the IfcWall.
- Name: its factory-default derived value is IfcWall’s Archicad ID (Wall Settings dialog box > Classification and Properties panel).
- Description: an optional textual description.
- ObjectType: an optional text to define a subtype of an element or add type information to it.

- Tag: its factory-default derived value is the IfcWall’s Archicad “Unique ID” (not the same as IFC GlobalId).



- OwnerHistory is directly attached to all independent objects, relationships and properties. Its components and values are derived from project data and hidden in the Archicad user interface. It includes, for example, the owning actor (OwningUser) (certain owner data is added to Archicad’s Project Info (File > Info)).

IFC Properties

IFC Properties are additional (mostly IFC Type-specific) parameters assigned to an IFC Entity. IFC Properties can be standard or non-standard.

- Standard, so-called “IFC2x3 Schema” data are defined by buildingSMART and stored in property sets whose names begin with the prefix “Pset_”
- Non-standard data are created by the exporting application using any property name and stored in any property set (the names of these property sets usually contain the name of the exporting application, or the IFC Model View Definition which requires the property).

[See *Property Mapping for IFC Export*.](#)

IFC Classification Reference

A Classification Reference (IfcClassificationReference) arranges IFC elements into a category. You can assign an Item Reference identifier, a classification Name attribute and some other optional parameters to all project elements (such as the building, the stories, the building elements, or the zones).

You can classify elements with this system both in IFC Project Manager and the element Settings dialog.

[See *Create New, Custom IFC Property*.](#)

IFC Assignments

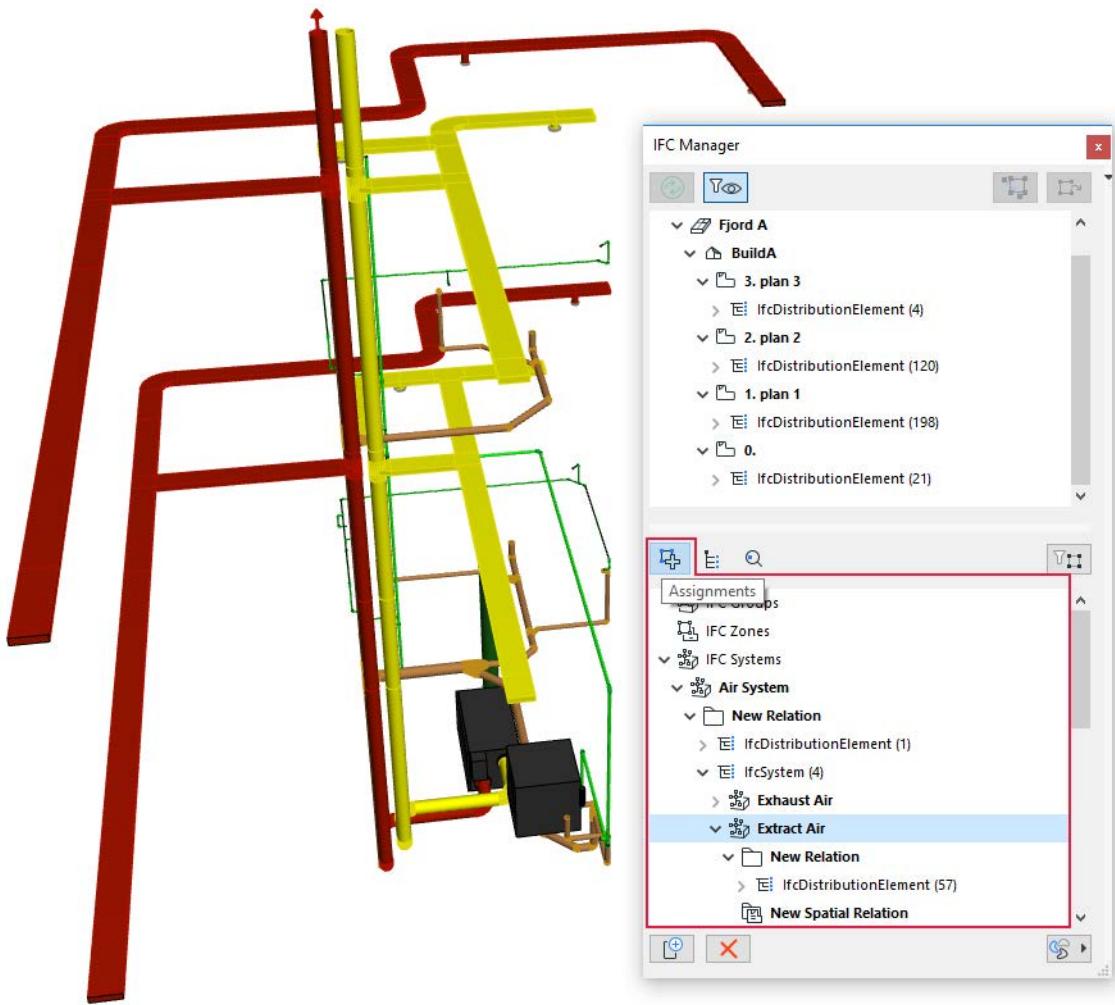
IFC Assignments (IfcRelAssigns) define the relationship among project elements. Each assignment type can have its own IFC Attribute (the most significant is the Object Type), and standard and custom IFC Properties. The major assignment types of the IFC standard are the following:

- **IFC Group** (IfcGroup): Use this to group any project elements (building element, story etc.) together. For example: the column and beam components of a frame standing are grouped together. Grouping by multilevel-hierarchy is also available: for example, you can group the frame standing groups in a 'frame-system' group.
- **IFC Zone** (IfcZone): Use this to group IfcSpace elements (the equivalent of Archicad Zones) together. For example: Archicad Zones having the identical function are grouped into an IfcZone named e.g. "Security Zone." Grouping by multilevel-hierarchy is also available for IfcZone assignments. For example, you can group some Archicad Zones (IfcSpaces) in an IfcZone (Security Zones) that is a part of a higher level IfcZone group (Governmental Zones).
- **IFC System** (IfcSystem): Use this to group any project elements (especially MEP elements) in a system hierarchy manually, by using MEP Modeler defined systems, or by importing systems exported from e.g. MEP applications. Definition of sub-system hierarchy (parent and child system) is also available. For example: group elevators in a vertical circulation system which can be a child system of a mechanical system. Or, assign plumbing elements to the cold water sub-system of a plumbing system. You can also tie systems to a spacial structure relation (IfcSite, IfcBuilding, IfcStorey or IfcSpace). For example, add to a duct system all the Archicad Zones (IfcSpace) that it runs through.
- **IFC Building Systems**
- **IFC Distribution Systems**

Archicad supports all of the above mentioned IFC Assignment types:

- the IFC Project Manager interface lets you define any assignments in your project and manage all their IFC data;
- all of the above mentioned assignment data are created at IFC model import (for example IfcSystems stored in an MEP-type IFC model, IfcZones defined in an FM application);
- all available IFC Assignments (defined manually or previously imported) are exported with a new IFC model mapped from the current Archicad project.

See [Using Assignments in IFC Project Manager](#).



Appendix: IFC Data Mapping in Archicad

This section contains two charts with detailed mapping information that is built in to Archicad.

[Predefined Property Mapping \(Export Archicad to IFC\)](#)

[Built-in IFC Type Mapping for Archicad](#)

Related Topic:

[Data Types for Properties in IFC and Archicad](#)

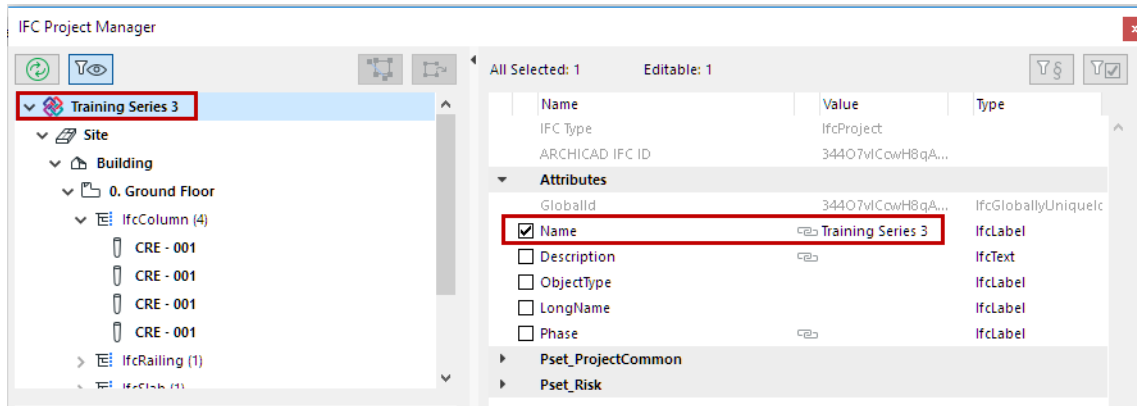
Predefined Property Mapping (Export Archicad to IFC)

The Property Mapping preset of the IFC Translator for Export defines the assignment of IFC Properties and Attributes when exporting Archicad model to IFC.

[See Property Mapping for IFC Export.](#)

Some Attribute and Property data come with a predefined mapping rule, which you do not have to create yourself in Property Mapping.

Although these rules do not appear in the Map IFC Properties for Export dialog box, they can be seen (if they have values) with a gray chain icon in the **IFC Project Manager** and Element Settings.



These rules are listed in the following table.

Archicad Command	Archicad Parameter	IFC Entity Types	IFC Data	IFC Project Manager
Project Info	Project Name	IfcProject	Attribute>Name	visible/editable
Project Info	Project Description	IfcProject	Attribute>Description	visible/editable
Project Info	Project ID	IfcProject	Attribute>GlobalId (see How to Control Global ID (IFC Attribute) Based on Archicad Project Info)	-
Project Info	Project Status	IfcProject	Attribute>Phase	visible/editable
Location Settings	Project North	IfcProject	Attribute>RepresentationContext>TrueNorth	-
Location Settings	Latitude	IfcSite	Attribute>RefLatitude	visible
Location Settings	Longitude	IfcSite	Attribute>RefLongitude	visible
Location Settings	Altitude	IfcSite	Attribute>RefAltitude	visible
Project Info	Site Name	IfcSite	Attribute>Name	visible/editable
Project Info	Site ID	IfcSite	Attribute>Global Id (see How to Control Global ID (IFC Attribute) Based on Archicad Project Info)	-

Archicad Command	Archicad Parameter	IFC Entity Types	IFC Data	IFC Project Manager
Project Info	Site Description	IfcSite	Attribute>Description	visible/editable
Project Info	Site Full Address>Site Address1	IfcSite	Attribute>SiteAddress>AddressLines	-
Project Info	Site Full Address>Site Address2			
Project Info	Site Full Address>Site Address3			
Project Info	Site Full Address>Site Postal Box	IfcSite	Attribute>SiteAddress>PostalBox	-
Project Info	Site Full Address>Site City	IfcSite	Attribute>SiteAddress>Town	-
Project Info	Site Full Address>Site State	IfcSite	Attribute>SiteAddress>Region	-
Project Info	Site Full Address>Site Postcode	IfcSite	Attribute>SiteAddress>PostalCode	-
Project Info	Site Full Address>Site Country	IfcSite	Attribute>SiteAddress>Country	-
Project Info	Site Gross Perimeter	IfcSite	BaseQuantity>GrossPerimeter	-
Project Info	Site Gross Area	IfcSite	BaseQuantity>GrossLandArea	-
Project Info	Building Name	IfcBuilding	Attribute>Name	visible/editable
Project Info	Building Description	IfcBuilding	Attribute>Description	visible/editable
Project Info	Building ID	IfcBuilding & IfcBuildingStorey	Attribute>GlobalId (see How to Control Global ID (IFC Attribute) Based on Archicad Project Info)	
Project Info	Contact Full Name>Contact Prefix Title	IfcPerson	Attribute>PrefixTitles	-
Project Info	Contact Full Name>Contact Given Name	IfcPerson	Attribute>GivenName	-
Project Info	Contact Full Name>Contact Middle Name	IfcPerson	Attribute>MiddleNames	-
Project Info	Contact Full Name>Contact Family Name	IfcPerson	Attribute>FamilyName	-
Project Info	Contact Full Name>Contact Suffix Title	IfcPerson	Attribute>SuffixTitles	-
Project Info	Contact ID	IfcPerson	Attribute>Id	-
Project Info	Contact Role	IfcPerson	Attribute>Roles	-
Project Info	Contact Department	IfcPerson	Addresses>Postal>InternalLocation	-

Archicad Command	Archicad Parameter	IFC Entity Types	IFC Data	IFC Project Manager
Project Info	Contact Company	IfcOrganization	Attribute>Name	-
Project Info	Contact Company Code	IfcOrganization	Attribute>Id	-
Project Info	Contact Full Address>Contact Address1	IfcPerson & IfcOrganization	Attribute>Addresses>Postal>AddressLines	-
Project Info	Contact Full Address>Contact Address2			
Project Info	Contact Full Address>Contact Address3			
Project Info	Contact Full Address>Contact Postal Box	IfcPerson & IfcOrganization	Attribute>Addresses>Postal>PostalBox	-
Project Info	Contact Full Address>Contact City	IfcPerson & IfcOrganization	Attribute>Addresses>Postal>Town	-
Project Info	Contact Full Address>Contact State	IfcPerson & IfcOrganization	Attribute>Addresses>Postal>Region	-
Project Info	Contact Full Address>Contact Post-code	IfcPerson & IfcOrganization	Attribute>Addresses>Postal>PostalCode	-
Project Info	Contact Full Address>Contact Country	IfcPerson & IfcOrganization	Attribute>Addresses>Postal>Country	-
Project Info	Contact E-mail	IfcPerson & IfcOrganization	Attribute>Addresses>Telecom>ElectronicMailAddresses	-
Project Info	Contact Phone Number	IfcPerson & IfcOrganization	Attribute>Addresses>Telecom>TelephoneNumbers	-
Project Info	Contact Fax	IfcPerson & IfcOrganization	Attribute>Addresses>Telecom>FacsimileNumbers	-
Project Info	Contact Web	IfcPerson & IfcOrganization	Attribute>Addresses>Telecom>WWWHomePageURL	-
Story Settings	Elevation	IfcBuildingStorey	Attribute>Elevation	visible/editable
Zone Settings	Name	IfcSpace	Attribute>LongName	visible/editable
Zone Settings	Number (No)	IfcSpace	Attribute>Name	visible/editable

Archicad Command	Archicad Parameter	IFC Entity Types	IFC Data	IFC Project Manager
(element)>Settings	Classification and Properties>ID	(IFC element type)	Attribute>Name	visible/editable
(element)>Settings	Classification and Properties>Classification	(IFC element type)	Attribute>ObjectType	visible/(editable in Export Translator, Type Mapping)
(element)>Settings	Classification and Properties>Classification	(IFC element type)	Attribute>PredefinedType	visible/(editable in Export Translator, Type Mapping)
(element)>Settings	Classification and Properties>Structural Function	(IFC element type)	Pset_...Common>LoadBearing	visible
(element)>Settings	Classification and Properties>Position	(IFC element type)	Pset_...Common>IsExternal	visible
(element)>Settings	Classification and Properties>Renovation Status	(IFC element type)	AC_Pset_RenovationAndPhasing>Renovation Status	visible
(element)>Settings	Classification and Properties>Classification	(IFC type product)	Attribute>ElementType	visible/(editable in Export Translator, Type Mapping)
(element)>Settings	Classification and Properties>Classification	(IFC type product)	Attribute>PredefinedType	visible/(editable in Export Translator, Type Mapping)
Schedule>(element)	General>Unique ID	(IFC element type)	Attribute>Tag	visible/editable
Window/Door Settings	Preview and Positioning>Width	IfcWindow / IfcDoor	Attribute>OverallWidth	visible
Window/Door Settings	Preview and Positioning>Height	IfcWindow / IfcDoor	Attribute>OverallHeight	visible
(automatic type naming rule)	Bldg Mat/Profile name, element dimensions. For multi-segment: Number of segments, maximum segment dimensions, segment lengths.	IfcColumn	Pset_ColumnCommon>Reference	visible/editable

Archicad Command	Archicad Parameter	IFC Entity Types	IFC Data	IFC Project Manager
(automatic type naming rule)	Bldg Mat/Profile name, element dimensions. For multi-segment: Number of segments, maximum segment dimensions, segment lengths.	IfcBeam	Pset_BeamCommon>Reference	visible/editable

Built-in IFC Type Mapping for Archicad

When an Archicad model is exported to IFC, each element is assigned an IFC Element Type.

Type Mapping is defined in the IFC Translator for Export, in the Type Mapping preset: either by Element Type or by Classification.

[See *Type Mapping for IFC Export*.](#)

The following 3 charts show the Type Mapping that results using the automatic, “by Element Type” method.

Archicad Tool

LibPart (By UnId)

MEP (By Tool)

Archicad Tool

Archicad Tool	IFCObjectType	IFCObject	IFCObject PredefinedType	IFCTypeObject Predefined-Type
Beam	IFCB eam		IFCB eamType	
Column	IFCC olumn		IFCC olumnType	
Wall	IFCW all		IFCW allType	
Shell	IFCS lab	ROOF	IFCS labType	ROOF
Roof	IFCS lab	ROOF	IFCS labType	ROOF
Slab	IFCS lab	FLOOR	IFCS labType	FLOOR
Mesh	IFCS ite			
Morph	IFCBuildingElementProxy		IFCBuildingElement-ProxyType	
Railing	IFCR ailing		IFCR ailingType	
Baluster	IFCM ember		IFCM emberType	
Handrail	IFCM ember		IFCM emberType	
Inner Post	IFCM ember		IFCM emberType	
Railing Panel	IFCM ember		IFCM emberType	
Railing Post	IFCM ember		IFCM emberType	
Rail	IFCM ember		IFCM emberType	
Toprail	IFCM ember		IFCM emberType	
Handrail Connection	IFCM ember		IFCM emberType	
Rail Connection	IFCM ember		IFCM emberType	
Toprail Connection	IFCM ember		IFCM emberType	
Handrail End	IFCM ember		IFCM emberType	
Rail End	IFCM ember		IFCM emberType	
Toprail End	IFCM ember		IFCM emberType	
Railing Node	IFCM ember		IFCM emberType	
Railing Segment	IFCM ember		IFCM emberType	
Railing Pattern	IFCM ember		IFCM emberType	
Stair	IFCS tair		IFCS tairType	
Riser	IFCM ember		IFCM emberType	
Stair Structure	IFCM ember		IFCM emberType	
Tread	IFCM ember		IFCM emberType	
Curtain Wall	IFCC urtainWall		IFCC urtainWallType	
Curtain Wall Frame	IFCM ember		IFCM emberType	
Curtain Wall Panel	IFCP late		IFCP lateType	
Curtain Wall Accessory	IFCDiscreteAccessory		IFCDiscreteAccessory-Type	

Archicad Tool	IFCObjectType	IFCObject	IFCObject PredefinedType	IFCTypeObject Predefined-Type
Curtain Wall Seg- ment	IFCDiscreteAccessory		IFCDiscreteAccessory- Type	
Curtain Wall Junction	IFCDiscreteAccessory		IFCDiscreteAccessory- Type	
Zone	IFCSpace		IFCSpaceType	
Door	IFCDoor		IFCDoorType	
Skylight	IFCWindow		IFCWindowType	
Window	IFCWindow		IFCWindowType	
Opening	IFCOpeningElement			
Object	IFCFurniture		IFCFurnitureType	
Lamp	IFCLightFixture		IFCLightFixtureType	
Fill	IFCAnnotation		Undefined	
Line	IFCAnnotation		Undefined	
Spline	IFCAnnotation		Undefined	
Polyline	IFCAnnotation		Undefined	
Circle	IFCAnnotation		Undefined	
Arc	IFCAnnotation		Undefined	
Dimension	IFCAnnotation		Undefined	
Angle Dimension	IFCAnnotation		Undefined	
Level Dimension	IFCAnnotation		Undefined	
Radial Dimension	IFCAnnotation		Undefined	
Text	IFCAnnotation		Undefined	
Label	IFCAnnotation		Undefined	

LibPart (By UnId)

LibPart (By UnId)		
Archicad	IFCObjectType	IFCTypeObjectType
curtainWallFrame	IFCBuildingElementProxy	IFCBuildingElementProxyType
curtainWallPanel	IFCBuildingElementProxy	IFCBuildingElementProxyType
stair	IFCStair	IFCSpatialStructureElementType
opening	IFCOpeningElement	IFCMemberType
wallOpening	IFCOpeningElement	IFCMemberType
wallDoor	IFCDoor	IFCDoorStyle
wallWindow	IFCWindow	IFCRampFlightType
roofOpening	IFCWindow	IFCRampFlightType
beam	IFCBeam	IFCBeamType
column	IFCColumn	IFCColumnType
covering	IFCCovering	IFCCoveringType
light	IFCFlowTerminal	IFCFlowTerminalType
curtainWall	IFCCurtainWall	IFCCurtainWallType
discreteAccessory	IFCDiscreteAccessory	IFCDiscreteAccessoryType
footing	IFCFooting	IFCFlowTreatmentDeviceType
furnishing	IFCFurnishingElement	IFCFurnishingElementType
railing	IFCRailing	IFCRailingType
ramp	IFCRamp	IFCRailingType
roof	IFCRoof	IFCRampFlightType
slab	IFCSlab	IFCSlabType
wall	IFCWall	IFCRampFlightType
wallEnd	IFCCovering	IFCCoveringType
gridElementMarker	IFCGridAxis	IFCFurnishingElementType
slabOpening	Undefined	IFCWindowStyle
transport	IFCTransportElement	IFCRampFlightType
buildingElementProxy	IFCBuildingElementProxy	IFCBuildingElementProxyType
ifcMechanicalFastener	IFCMechanicalFastener	IFCMechanicalFastenerType
ifcFastener	IFCFastener	IFCFastenerType
ifcExtruded	IFCBuildingElementProxy	IFCBuildingElementProxyType
ifcBeamExtruded	IFCBeam	IFCBeamType
ifcColumnExtruded	IFCColumn	IFCColumnType
ifcDiscreteAccessoryExtruded	IFCDiscreteAccessory	IFCDiscreteAccessoryType
ifcFootingExtruded	IFCFooting	IFCFlowTreatmentDeviceType
ifcOpeningExtruded	IFCOpeningElement	IFCMemberType

MEP (By Tool)

MEP (By Tool)			
Archicad	IFCObjectType	IFCTypeObjectType	IFCTypeObject PredefinedType
CableCarrierBend	IFCCableCarrierFitting	IFCCableCarrierFittingType	BEND
CableCarrierRigidSegment	IFCCableCarrierSegment	IFCCableCarrierSegmentType	NOTDEFINED
CableCarrierTee	IFCCableCarrierFitting	IFCCableCarrierFittingType	TEE
CableCarrierTransition	IFCCableCarrierFitting	IFCCableCarrierFittingType	REDUCER
CableCarrierWye	IFCCableCarrierFitting	IFCCableCarrierFittingType	TEE
DuctBend	IFCDuctFitting	IFCDuctFittingType	BEND
DuctEndCap	IFCDuctFitting	IFCDuctFittingType	EXIT or ENTRY
DuctFlexibleSegment	IFCDuctSegment	IFCDuctSegmentType	FLEXIBLESEGMENT
DuctObstruction	IFCDuctFitting	IFCDuctFittingType	OBSTRUCTION
DuctOffset	IFCDuctFitting	IFCDuctFittingType	TRANSITION
DuctRigidSegment	IFCDuctSegment	IFCDuctSegmentType	RIGIDSEGMENT
DuctTakeOff	IFCDuctFitting	IFCDuctFittingType	JUNCTION
DuctTee	IFCDuctFitting	IFCDuctFittingType	JUNCTION
DuctTransition	IFCDuctFitting	IFCDuctFittingType	TRANSITION
DuctVerticalObstruction	IFCDuctFitting	IFCDuctFittingType	OBSTRUCTION
DuctWye	IFCDuctFitting	IFCDuctFittingType	JUNCTION
PipeBend	IFCPipeFitting	IFCPipeFittingType	BEND
PipeCombinationWye	IFCPipeFitting	IFCPipeFittingType	JUNCTION
PipeInlineFlowDevice	IFCPipeFitting	<ifc_subtype parameter>	<ifc_predefinedtype parameter>
PipeInlineFlowDevice	IFCPipeFitting	<ifc_subtype parameter>	<ifc_predefinedtype parameter>
PipeRigidSegment	IFCPipeSegment	IFCPipeSegmentType	RIGIDSEGMENT
PipeSanitaryTee	IFCPipeFitting	IFCPipeFittingType	JUNCTION
PipeTakeOff	IFCPipeFitting	IFCPipeFittingType	JUNCTION
PipeTee	IFCPipeFitting	IFCPipeFittingType	JUNCTION
PipeTransition	IFCPipeFitting	IFCPipeFittingType	TRANSITION
PipeWye	IFCPipeFitting	IFCPipeFittingType	JUNCTION

Data Types for Properties in IFC and Archicad

IFC Data Type	Archicad Data Type	Description
Single String	String	Any text or number
Single Real	Number	Any number expressed in decimals
Single Integer	Integer	A whole number
Single Boolean	True/False	A logical true or false value
List String	Tags List	A tag or series of tags.
Enumerated String	Option Set	Provides a fixed set of options. The user can choose a value from this set.

User Interface Reference

In this section you will find the description of Archicad's user interface listed in 3 categories: **Controls; Dialog Boxes; and Tool Settings Dialog Boxes.**

The best way to use the User Interface Reference is to navigate to specific topics by using Archicad's context sensitive help.

Read more about this feature in: [Context-Sensitive Help](#).

Topics in this section:

[Controls](#)

[Dialog Boxes](#)

[Tool Settings Dialog Boxes](#)

Controls

This section of the User Interface Reference describes the following interface items.

[Action Center](#)

[Project Chooser](#)

[The 2D Navigator Preview Palette Controls](#)

[The 3D Navigator Preview Palette Controls](#)

[Control Box](#)

[Coordinates Palette](#)

[Search and Replace Text Dialog Box](#)

[Formatting Palette and Text Editor Controls](#)

[Renovation Palette](#)

[Teamwork Palette](#)

[Project Info](#)

[Project Notes](#)

[Project Preview](#)

[Architect Credits in BIMx](#)

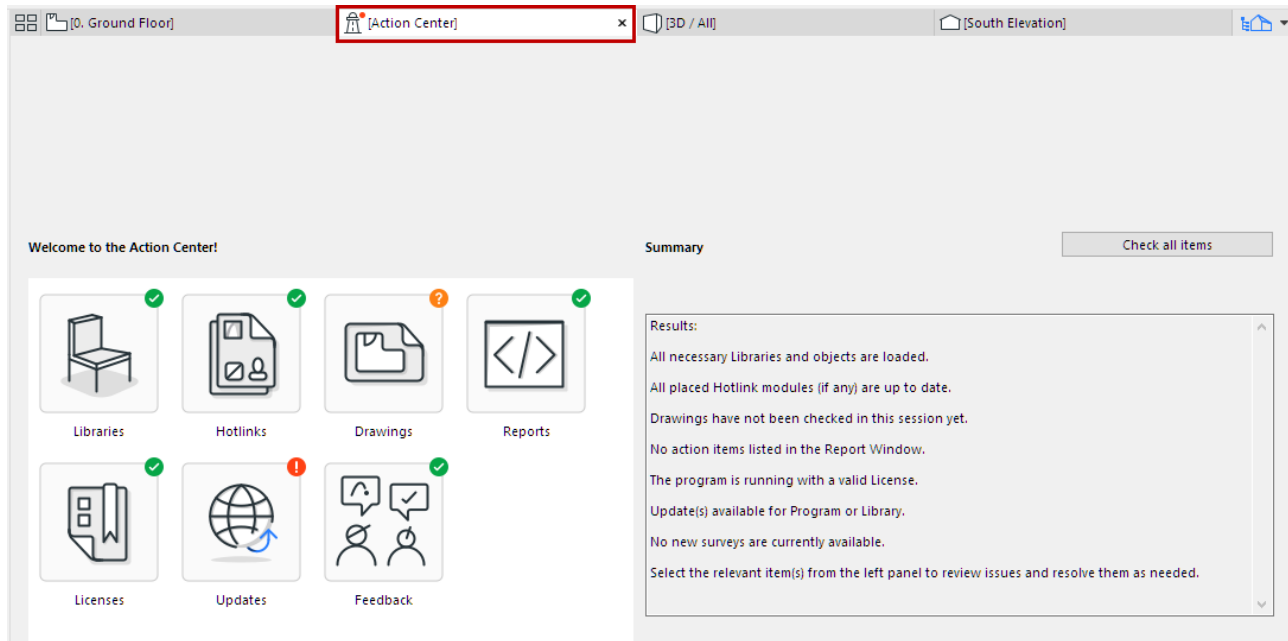
[Session Report Window](#)

[Solid Element Operations Palette](#)

[Classic 3D Navigation Toolbar](#)

Action Center

The Action Center is an Archicad Tab/window:



It collects all the information you need about the update status of the following project components:

- Libraries
- Hotlinks
- Drawings
- Reports (Session Reports)
- Licenses
- Updates (Archicad and Library)
- Feedback (User Surveys)

Use the Action Center to get information on their current status, issues that need to be resolved, and to run a check where needed.

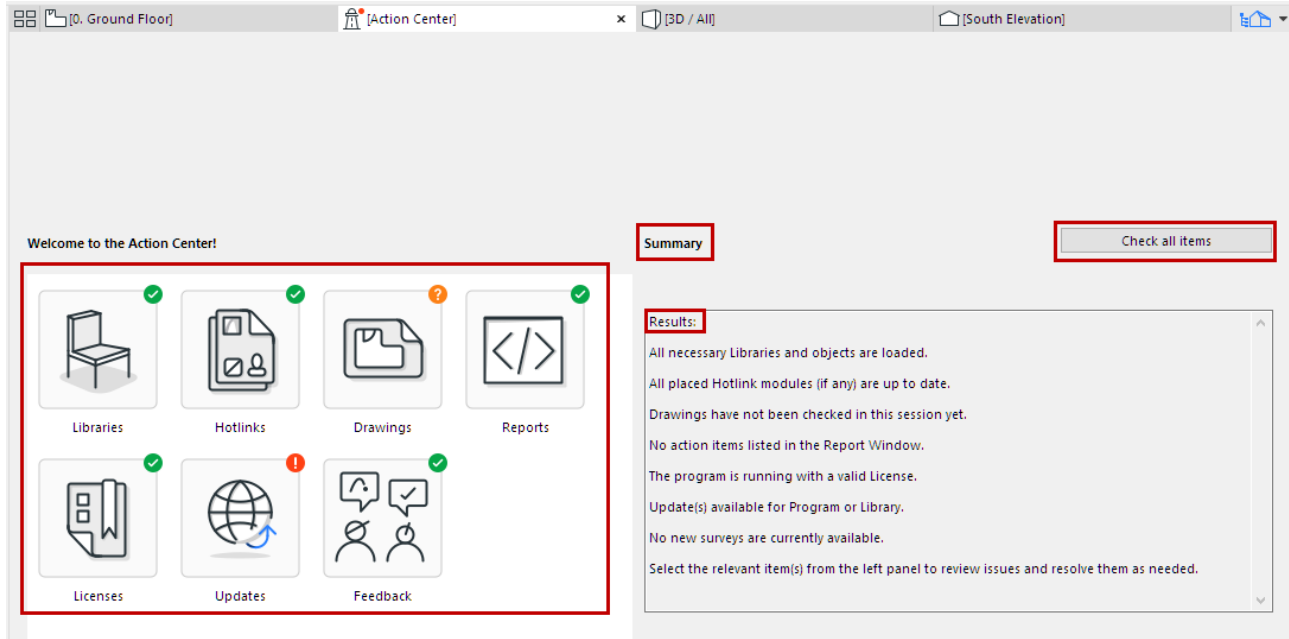
- Some of these items (e.g. Libraries) are checked **automatically** whenever you open a project.
- Other items (e.g. Hotlinks and Drawings) may require you to run a **manual** check.

Where Do I See the Action Center?

- The Action Center window appears **automatically** in the Tab Bar (or the current window, if you don't use the Tab Bar) whenever any item(s) need checking or action.
- At any time: use **File > Info > Action Center**

Action Center Summary

By default, you see the Summary page:



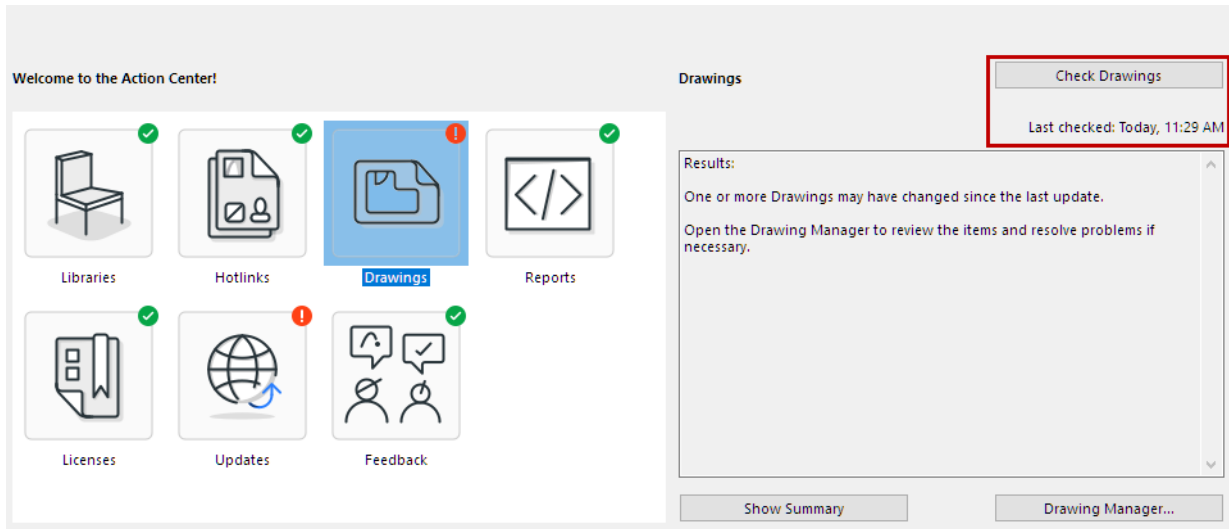
The icon of each project item indicates its current status.

Status of Project Items

Action Center Icon	What it Means
	No action needed Item was up to date as of the last check.
	Action needed Issues were found during the last check. Click the item and follow the instructions.
	Needs checking. This item needs a check to determine if anything is missing or out of date. Click the item and follow the instructions.

Note: The green checkmark means this item was up to date as of the last check. It is possible that some items shown with a green checkmark have been modified since the last check. Run a manual check to make sure.

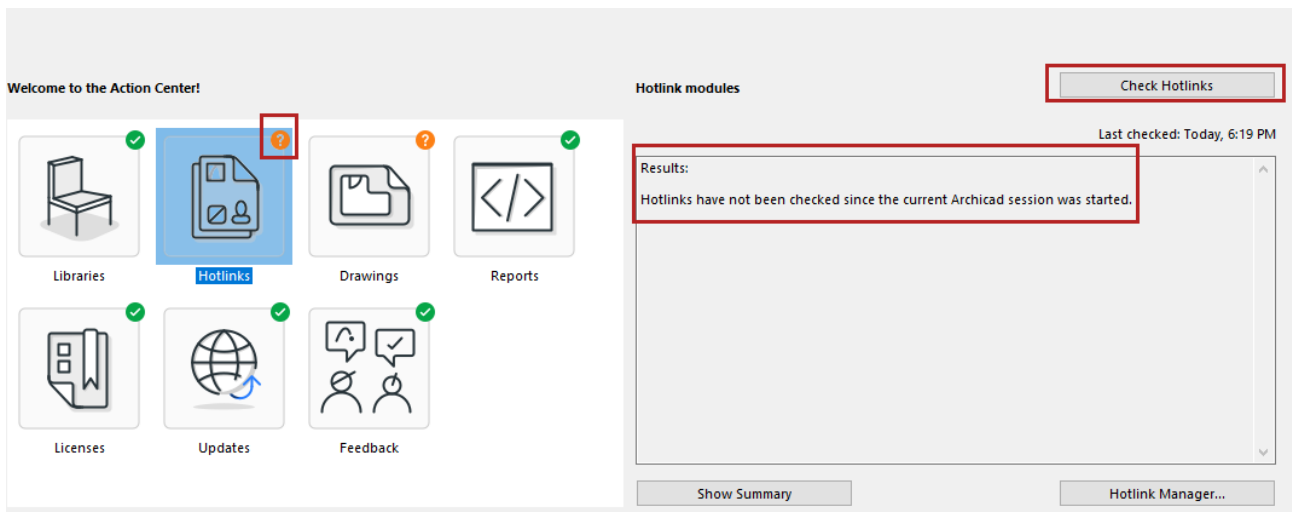
Click on any item for more info:



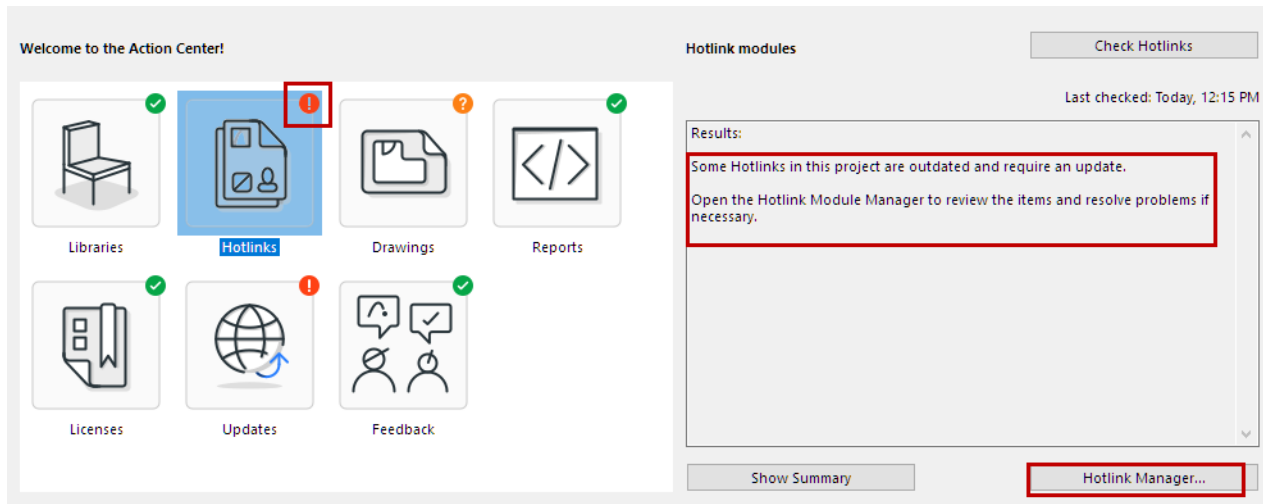
- In Action Center, for each item, the time of the last check is indicated.
- To run a manual check on any item, or all items:
 - For any selected item: click the context-sensitive **Check** button at top left
 - On the Summary page, click **Check all items**

Example

The Hotlinks item shows a question mark: manual checking is needed. Click the item, read the results, and click **Check Hotlinks** to run a check for missing sources or outdated hotlinks.



As a result of the manual check, the Action Center Hotlinks item now shows a red warning icon: an update is needed. Click the **Hotlink Manager** button to open it and take care of the update.



Management Dialogs Accessible from Action Center

[Library Manager](#)

[Hotlink Module Manager](#)

[Drawing Manager](#)

[Session Report Window](#)

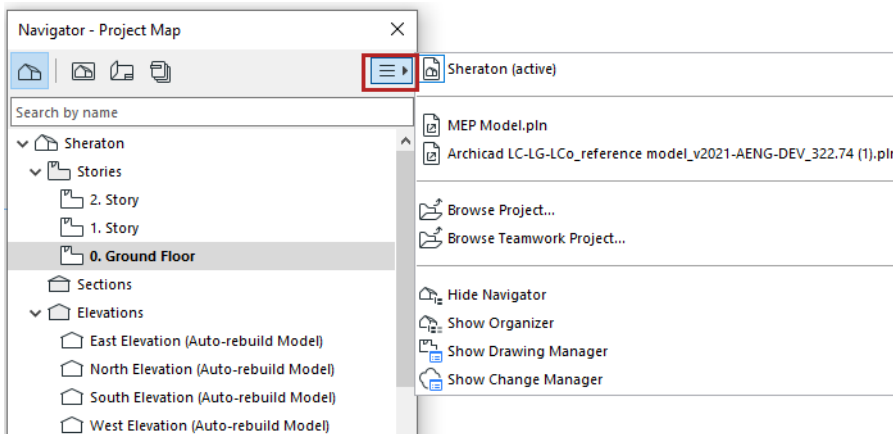
[License Information dialog](#)

See also:

[Updates for Archicad](#)

[Archicad User Feedback - Net Promoter Score Survey](#)

Project Chooser



Use this button to manage projects from the Navigator or Organizer.

The top section of the Project Chooser list displays the active Archicad project, plus any external project files you have accessed.

Browse: Browse a file directory or BIMcloud for an Archicad project whose content you want to use in the active project.

The rest of the items let you navigate to key palettes.

The 2D Navigator Preview Palette Controls

To show the 2D Navigator Preview Palette, use **Window > Palettes > Navigator Preview**.

The Navigator Preview palette has three uses:

- It shows a small preview of the Project Map or View Map item selected in the Navigator palette.
- It can be updated to show the a preview of the contents of the currently active model window.
- With zoom and pan operations inside the palette, you can update the view displayed in the active window.

Go: Click Go to jump to the view you select from the Navigator or in the Overview.

Note: Clicking Go is the equivalent of double-clicking one item on the Navigator tree view or double-clicking on the Overview inside the frame of the Overview.

Zoom Out: Clicking this results in reduction on the visible plan in the overview area.

Zoom: Sets the zoom on the plan visible in the overview area.

Zoom In: Clicking this results in magnification on the visible plan in the overview area.

Zoom Pop-up Menu: You can define the connection between the Navigator Preview and the frontmost window. You can also redraw preview from here.

- **Real-time Zoom:** If you choose this option, the zooming and panning operations made in the preview are continuously shown in the window.
- **Auto zoom:** With this option, the zooming and panning operations made in the preview do not affect the screen display until you release the mouse button.
- **Double-click to zoom:** With this option, the view does not reflect the display changes made in the preview. To update it, you need to either double-click inside the preview or click the **Go** button in the top right corner of the palette.
- **Redraw Preview:** This command updates the preview window if you have edited the contents of the active window.

For more information, see [Navigator Preview \(2D\)](#).

The 3D Navigator Preview Palette Controls

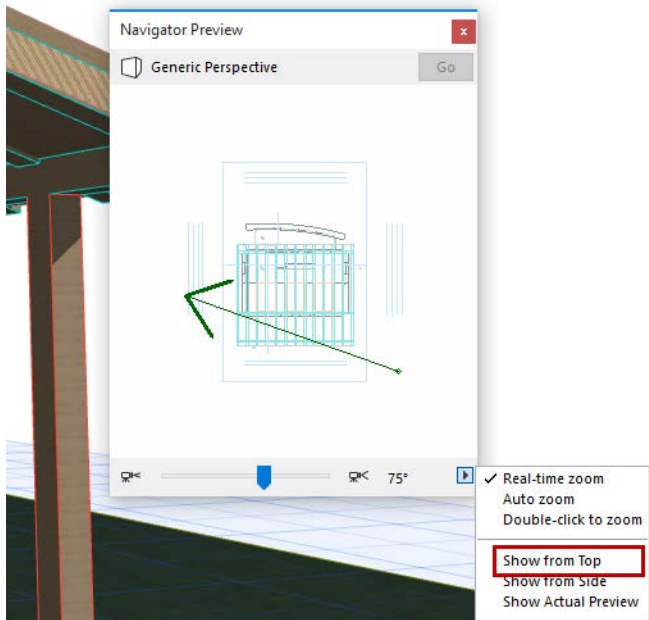
The 3D Navigator Preview Palette is only available when working in the 3D window.

[See also Navigator Preview \(3D\).](#)

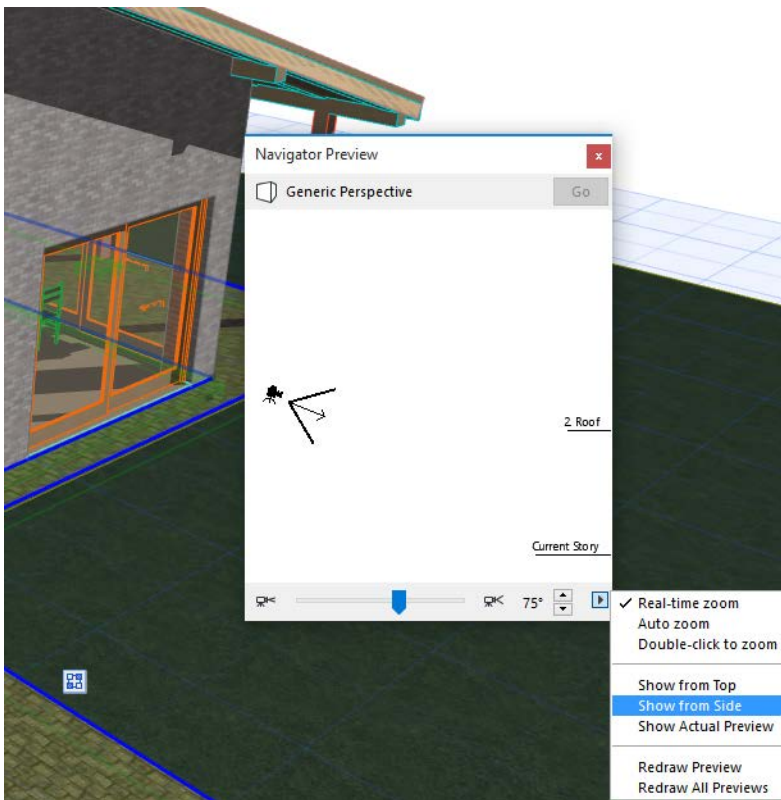
The options are different for Perspectives and Parallel Projections (Axonometrics).

If you are in a perspective view:

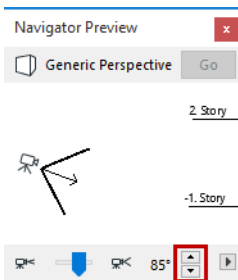
Choosing **Show from Top** from the pop-up at the lower right corner of the palette allows you to manipulate a **perspective** camera directly: move it around, fix the target point or the viewpoints.



Choosing **Show from Side** at the pop-up at the lower right corner displays only the **perspective** camera and story elevation marks. Click on the camera and hold the mouse button on the up or down arrow to adjust the camera's vertical position. (The story signs will start to move in the opposite direction, simulating the effect of camera motion on the model.) A caret will appear in the upper or lower right corner of the preview field to indicate when the camera position has reached the top or bottom of the model.

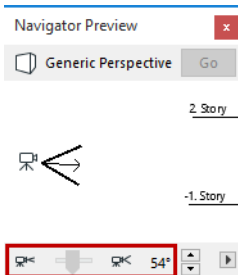


The double-arrow at the bottom of the window enables you to adjust the camera position from story to story (up or down) with just one click.



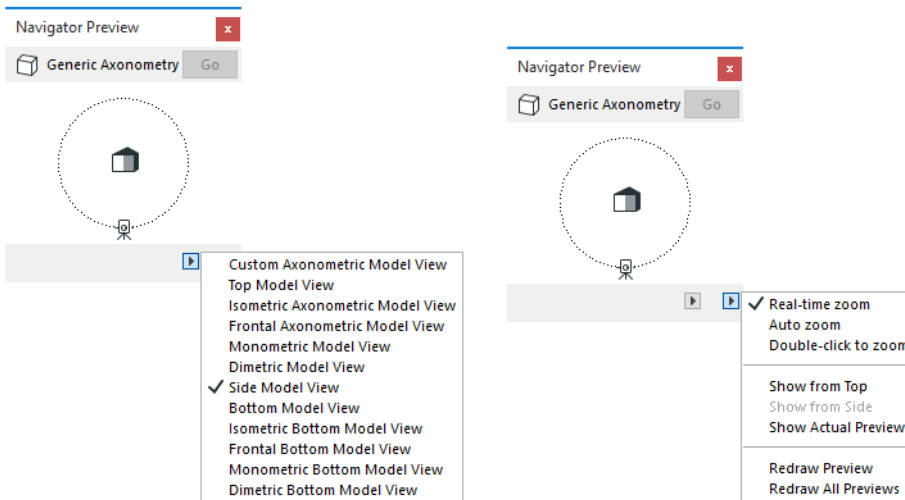
Click and drag the camera's arrow to adjust the camera's tilt.

Use the sliding arrow at the bottom of the window to adjust the camera's view cone.



For **axometric** views, only the small house representing the model is displayed when showing the **Top** view. The controls resemble those of the **Parallel Projection Settings** dialog box.

For more information, see [3D Projection Settings](#).



Choosing **Show Actual Preview** displays the current content of the 3D Window. In this case, all zooming controls are disabled. To refresh the preview of the current 3D view, double-click it or choose **Redraw Preview**. **Redraw all previews** will redraw the 3D Navigator Preview picture of each 3D view.

Control Box

Note: The Control Box is not visible by default. It is available at **Window > Palettes**.

Use the context menu to display this palette in horizontal/vertical or extended/compact form.

The **Control Box** contains a number of drawing aids in the form of icons.

Use the **OK** button to confirm the current operation.

Use the **Cancel** button to abandon the current operation.

The palette's controls, from left to right:

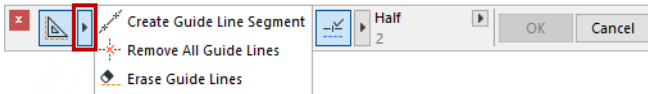
Guide Lines On/Off

Click the toggle icon in the Control Box to turn Guide Lines display on and off.



Guide Lines Options

Click the pop-up menu to access Guide Line-related commands.



- Create Guide Line Segment
- Remove all Guide Lines
- Erase Guide Lines

Relative Construction Methods

The Relative Construction Methods help geometric input by constraining the cursor's movement to a constant angle or distance as you draft or edit elements.

Click the pop-up arrow in the Control Box to access the seven different Relative Construction Method icons.



The **Parallel**, **Perpendicular** and **Angle Bisector** methods constrain the cursor's movement to a constant angle.

- **Perpendicular:** Select an existing edge for a reference line by clicking on it, or draw a new reference line (oriented as needed) using the cursor. Draw the new element perpendicular to the reference line you defined.
- **Parallel:** Select an existing edge for a reference line by clicking on it, or draw a new reference line (oriented as needed) using the cursor. Draw the new element parallel to the reference line you defined.

[See *Parallel and Perpendicular Constraints for more information.*](#)

- **Angle Bisector Constraint:** Define reference vectors; use the same process as with angle dimensioning. Click to start drawing the new element. The mouse is constrained to the bisectorial angle.

[See *Angle Bisector Constraint for more information.*](#)

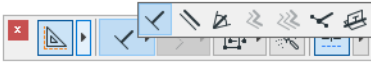
Use the **Offset** and the **Multiple Offset** methods to create a polygonal element that is offset from a reference line.

See [Offset and Multiple Offset Constraints \(Relative Construction Methods\)](#).

- The **Special Snap Vector** method lets you draw a temporary vector that displays your defined special snap points, to aid in accurate placement of elements.

See [Snap Points on Temporary Vector](#).

- The **Align to Surface** method works in 3D only.



Select an existing surface by clicking on it. Place the element on the section line of the input plane, and the surface you defined.

For more information, see [Aligning Elements to a Surface in 3D](#).

Cursor Snap Variants

This set of icons controls the way the current position of the cursor is projected on the rubber band line constrained to a predefined direction. Cursor Snap becomes active when you start drawing a vector.

For more information, see [Projection of Cursor with Mouse Constraints](#).

Enable/Suspend Groups

These two icons act as a toggle and have the same effect as the **Suspend Groups** toggle command. If Suspend Groups is active, grouped elements can be selected and edited one by one.

For more information, see [Group Elements](#).

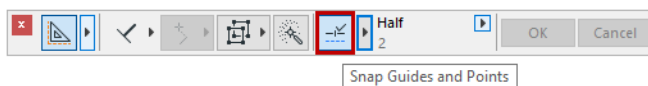
Magic Wand

With the Magic Wand feature, Archicad allows you to trace the outline of shapes for creating elements. Effect of Magic Wand depends on Tool and construction method selections.

For more information, see [Magic Wand](#).

Snap Points (Control Box)

Use this toggle to activate Snap Guides and Snap Points.



For more information, see [Snap Guides and Snap Points](#).

Coordinates Palette

Note: The Coordinates Palette is not visible by default. To show it, choose its name from the **Window > Palettes** menu.

Use the context menu to display this palette in horizontal/vertical or extended/compact form.

From left to right, the Coordinates Palette contains the following controls:

- **User Origin:** Resets the zero point to a user-defined location. Click this button and place the Origin anywhere on the Floor Plan.

Note: To return the User Origin to the Project Origin, double-click the User Origin button.

For more information, see [Origins](#).

- **Set Rotated Grid:** Click Rotated Grid button, then draw a vector on the Floor Plan to specify the angle of the rotated grid. Note: To return to the 'normal' grid, use the Grid Switch button.
- **Switch Orthogonal/Rotated Grid:** Select the Rotated Grid option to switch to the rotated grid or the Normal Grid option to switch to the horizontal/vertical grid.
- **Grid Snap:** Select one of these three options to control Grid Snapping:
 - Grid Snapping **OFF**
 - Snapping to the **Snap Grid**
 - Snapping to the **Construction Grid**

For more information, see [Snap to Grid](#).

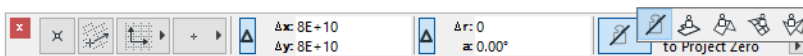
- **Absolute/Relative:** Switch the delta button ON to display and enter relative X and Y coordinates (for Cartesian coordinates) and radial/angle coordinates (for polar coordinates).

Absolute values show the horizontal, vertical, radial and angular distance of the cursor from the Project Origin or a User Origin but never from an Edit Origin.

Relative values show the horizontal, vertical, radial and angular position of the cursor relative either to the Project Origin, to a User Origin before any element is begun, or to an Edit Origin once a drafting or editing process is under way.

In general, viewing Absolute Cartesian and Relative Polar coordinates provides the most information, and is the most useful under normal circumstances.

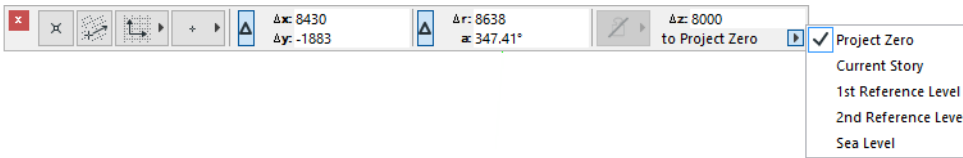
- **Gravity:** Optionally, choose to place elements atop Slabs, Meshes, Shells and Roofs.



See [Gravity](#).

- **Elevation:** The “z” field in the Coordinates Palette displays the elevation at which elements will be placed. The pop-up lets you define the reference level from which this value should be measured: the Project Zero, the current Story (or in 3D, the User Origin), or one of the optional Reference Levels defined in **Options > Project Preferences > Reference Levels**.

See [Reference Levels Preferences](#).

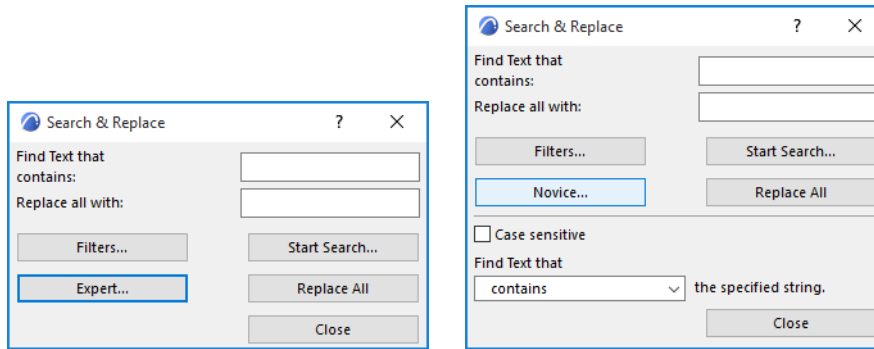


Search and Replace Text Dialog Box

The **Edit > Search and Replace Text** command can search texts in all eligible element types. It works on the Floor Plan and in Section/Elevation/IE, and Detail and Worksheet windows, and on Layouts.

See also [Search and Replace Text](#).

The dialog box it opens has two modes: **Novice** and **Expert**. By default, the **Novice** mode is displayed.



- **Find Text that contains:** Enter the text to search for. Search criteria can be changed in **Expert** mode.
- **Replace all with:** Type replace string to be applied if you click **Replace All**

Clicking the **Filters** button opens a further dialog box in which you can set Element, Layer and Story filters.

Search on:

- **Current Viewpoint:** Click this radio button to restrict searching to the current story.
- **All Viewpoints:** Click this radio button to search on all stories.
- **Visible Layers:** By default, only visible layers are searched. To enable the All layers option, first click “Include uneditable elements.”
- **All Layers:** Click this radio button to search on all layers.

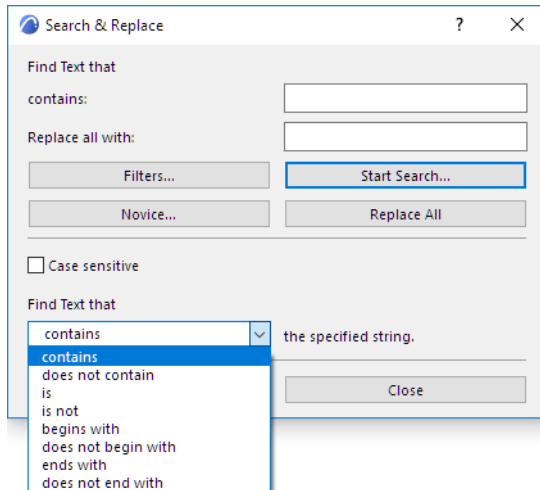
Search in:

- **Include uneditable Elements:** Check this box if you wish to extend search possibilities to three new filters: Dimensions Measured Text; Associative Labels; and to all layers, including invisible ones. (You must mark the filters to include them in the search.)
- **Text blocks:** Check this box to include Text blocks in the search.
- **Labels:** Check this box to include Labels in the search.
- **Dimensions Custom Text:** Check this box to include Custom Dimension Texts in the search.
- **Doors/Windows:** Check this box to include Doors/Windows in the search.
- **Zones:** Check this box to include Zones in the search.
- **Associative Labels:** Check this box to include Associative Labels in the search. (Available only if “Include uneditable Elements” is checked.)
- **Dimensions Measured Value:** Check this box to include Measured Dimension Values in the search. (Available only if “Include uneditable Elements” is checked.)
- **Objects/Lamps:** Check this box to include Objects/Lamps in the search.

When you have made your choices here, click **OK** to return to the main dialog box.

- **Case sensitive:** Check this box for case sensitive searching.

- **Find text that... the specified string:** Select search criterion from the list.



- If you click **Replace All**, all found texts will be automatically replaced.
- If you click **Start Search**, a Window will appear listing all found texts.

Search & Replace Results

- **Found Elements that contains:** The original search criterion
- **Replace Texts that contain:** Type text to be substituted when you replace selected Hit list items
- **With:** Type text to substitute when you replace selected Hit list items.
- **Hits:** This field contains the Hit list. Editable list items are in black, read-only text items are grayed out. You can start to edit a text by double-clicking it. Recently changed items are in *Italic*, they will turn back to plain when you apply changes.
- **Replace:** Click Replace to change all text selected in the list.
- **Zoom in:** Click here to zoom in on the first selected element on the list.
- **Zoom out:** Click here to zoom out.
- **Apply Changes:** Click Apply Changes to actually perform changes made in this list in Archicad.
- By clicking **New Search**, the list will disappear and you can start a new search.

Formatting Palette and Text Editor Controls

The Text Editor and Formatting Palette appears as soon as you click on the Floor Plan (with the Text tool active) and begin typing into the text box.

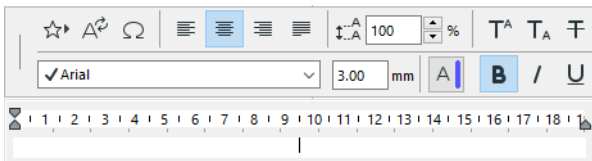
Use these controls to format individual characters, lines and paragraphs that have been typed into your text block, or to switch styles while typing.

The controls of the Formatting Palette and Text Editor apply only to the text you are entering in the Text Editor, or to selections inside the Text Editor.

The position of the Formatting palette can be customized using the “Position of Text Formatting Palette” controls in **Options > Work Environment > Dialog Boxes and Palettes**.

The Info Box is grayed while you are in the Text Editor.

Formatting Palette



The Formatting Palette appears:

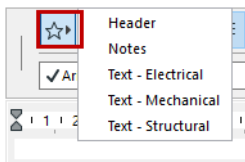
- when you are entering text into the Text Editor
- by double-clicking inside an existing text block with the Arrow tool in Quick Selection mode
- by selecting an existing text block (with the Text tool active) and clicking inside the text

Use the Formatting Palette to apply formatting to selected characters or paragraphs in the text editor, or to insert text symbols.

By selecting and formatting different segments of the text blocks, you can have multistyle formatting within a single text block.

Note: If the selected text contains multiple styles, the Formatting Palette settings reflect the formatting of the first character in the selection.

Choose Favorites: This pop-up contains a list of Text Settings that you have saved as favorites, as well as the default settings. If you have not saved any favorite Text Settings, this button is gray.



To apply text settings saved as Favorites, select text inside the text editor and click on the stored Favorite from the pop-up list.

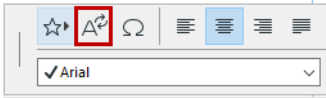
Note: Only the text-specific settings of the Favorite can be applied to selected text in the Text Editor.

To apply Favorite settings to the entire text block, use the Favorites button at the top of Text Tool Settings.

Note: Favorites of the Text tool can be applied to the text-type items in Dimensions and Labels, and to Fill Text.

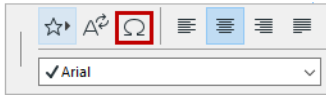
For more information, see [Favorites](#).

Insert Autotext: Use this control to open a dialog box to choose from available Autotext categories and items. The Preview area depicts the selected Autotext.

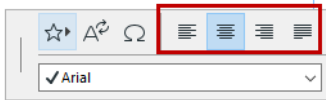


See also [Autotext](#).

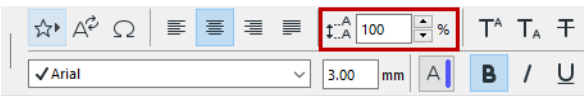
Insert Symbol: Click this button to access a Character Map (Character Palette on Mac) to choose and insert a symbol.



Alignment Buttons: (left, center, right, justify): to set particular alignments for the selected paragraph within your text block.

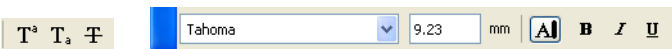


Leading: Use this control to modify the leading value (in percentage) for a selected line in the text block. Enter a number or use the up-and-down arrows to change the value in increments of 25 percent.



Note: In Windows, if you enter a leading value of less than 100 percent, the text in your text editor will not reflect this accurately (it will still appear as if your line spacing were set to 100 percent). However, your text will be displayed correctly on screen.

Superscript/Subscript/Strikethrough: Click either Superscript or Subscript to raise or lower the typed text compared to the text line. Strikethrough will draw a line through the text.



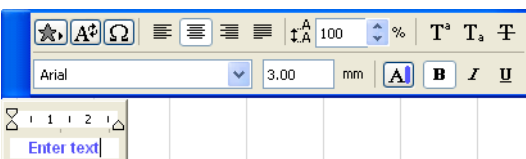
Text format for selected text: Set font, font size, character style or font color for any one or more characters or paragraphs that you select inside the text block.

Note: If you use the “Automatic pen color visibility adjustment for Model Views” feature (**Options > Work Environment > On-Screen Options**), the text in the text editor will reflect this (modified) color.

For more information, see [More Options](#).

Text Editor

The text editor appears (together with the Formatting Palette) every time you activate the Text tool and place a text block in the project. The flashing cursor indicates your place in the text editor.



The background color of the text editor is the same color as the text block’s background.

Setting Indents and Tabs

When creating a new “breaking” text block, indents appear in the ruler of the Text Editor:

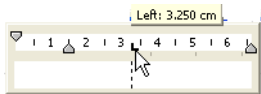
At left, the first-line indent marker (on top) and the left-side indent marker (on the bottom)

At right, the indent that shows the width of the text block.



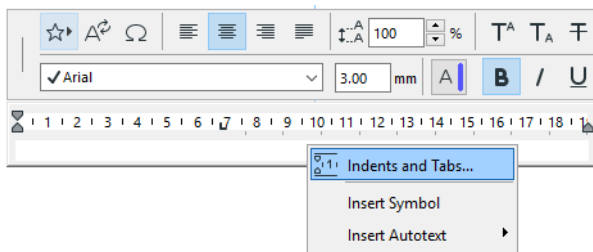
To reposition these indents, just click and drag them to the desired position.

Set a new tab stop by clicking on the ruler. Drag it to a new position as needed. To delete a tab, click and drag it off the ruler entirely.



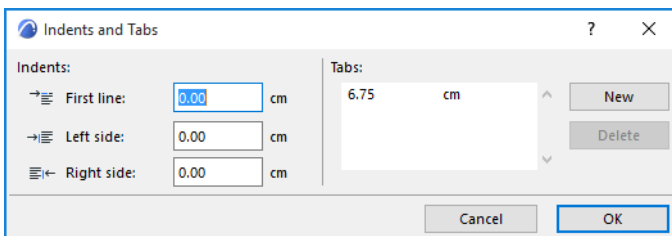
Alternatively, you can use the **Indents and Tabs** commands:

1. Click inside the paragraph whose indents or tabs you want to set or edit.
2. Choose the **Indents and Tabs** command from a context menu opened by right-clicking on the Text Editor’s ruler bar.



The **Indents and Tabs** dialog box appears, where you can enter values for indents of the first line, left side and right side of the text block. You can set new tabs: click the New button and enter a desired value. To modify tabs, select the existing value and enter a different one in the Tabs fields.

Note: Depending on the unit set in **Options > Project Preferences > Working Units**, you may set values in centimeter (cm) or inches (in).

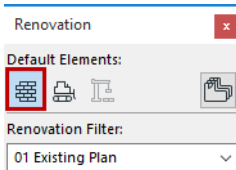


Renovation Palette

The Renovation Palette can be opened from **Window > Palettes > Renovation Palette**.

Use the Renovation Palette:

- to assign a Renovation Status to any number of selected elements
- to set a default Renovation Status for newly placed elements
- to switch the Renovation Filter of the current view
- to limit a selected element to a single Renovation Filter only



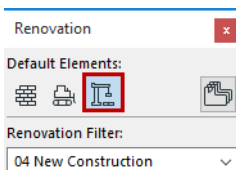
For general information on this feature, see [Renovation](#).

Assign Renovation Status to Selected Element(s)

1. **Select** one or more elements to which you want to assign a different renovation status.
2. From the Renovation Palette, click the icon representing the status you need.
 - **Existing** Element: to be retained
 - **New** Element: constructed during the current renovation process; to be retained
 - **To Be Demolished** Element: to be demolished during the current renovation process

Assign Default Renovation Status

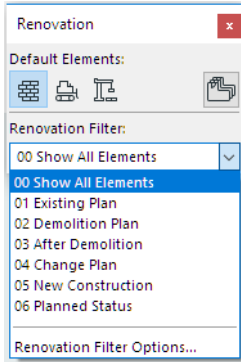
1. With nothing selected, click the icon of the Renovation Status you wish to be the default - for example, "New."



2. Now, all elements you place - regardless of which tool you use - will automatically be assigned the "New" status.

Switch Renovation Filter

From the Renovation Palette, click the desired filter from the Renovation Filter pop-up:



You can also use the Quick Options Palette to switch the Renovation Filter in effect.

Show on Current Renovation Filter Only

Use the icon at the top right of the Renovation Palette to limit a selected element to the Renovation Filter currently in effect.

See [Show an Element in a Single Renovation Filter Only](#).

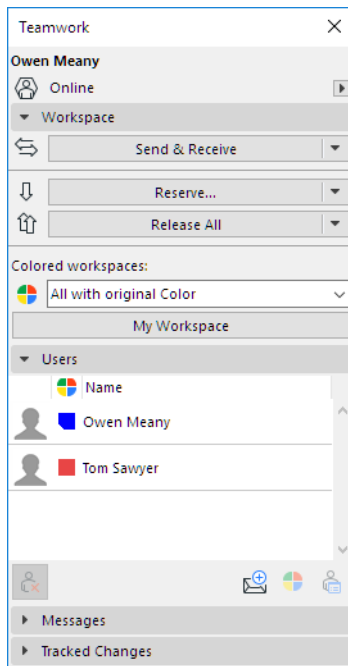
Teamwork Palette

Open this palette either from the Teamwork menu, or from **Windows > Palettes**.

If you are not joined in to a Teamwork project, the controls of this palette are gray.

In Teamwork, availability of the controls of this palette also varies depending on whether you are Online or Offline; and whether you have the rights of a Server or Project Administrator.

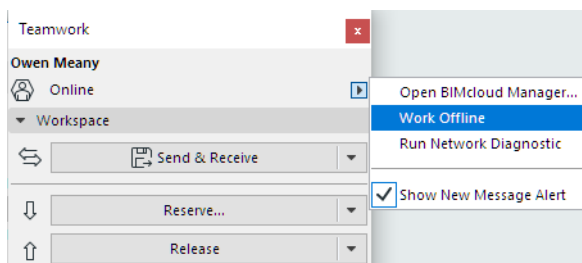
See [Working Online/Offline in Teamwork](#).



The top of the palette shows the User name and the user's online/offline status.

If there is a problem with the server connection, click **Network Diagnostic** to open a dialog box with details and help in solving the problem.

See [Network Diagnostic Information \(BIMcloud\)](#).



To work offline, click "**Work Offline**" from the pop-up.

The Show New Message Alert option is enabled by default. This means that a temporary on-screen alert appears each time you receive a new Teamwork message.

See also [Messaging \(Teamwork\)](#).

The Teamwork palette has four panels: Workspace; Users; Messages; and Project Changes.

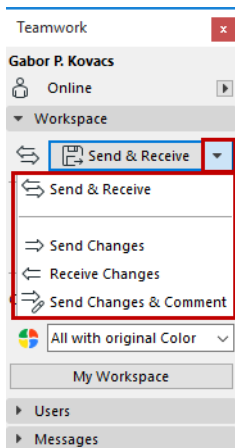
Workspace Panel

Send & Receive: Click this button to both Send and Receive project changes to/from the BIMcloud.

Click the pop-up to access separate **Send and Receive** commands.

- **Send** means that all modifications that you have made since your last send/receive are merged with the project data on the BIMcloud.
- **Receive** means that all modifications in the server’s project data that have occurred since your last “Receive” are merged with your local copy.

Note: No conflicts can arise between edits in your local copy and edits to the server’s project data, because you must reserve elements/data in order to modify them, and you cannot reserve elements/data until they are first synchronized with the server’s project data.



- **Send Changes & Comment:** This option lets you add a comment to the Activities Panel of BIMcloud Manager, in addition to Sending in your changes.

[See Add Comment to Activities \(Teamwork\).](#)

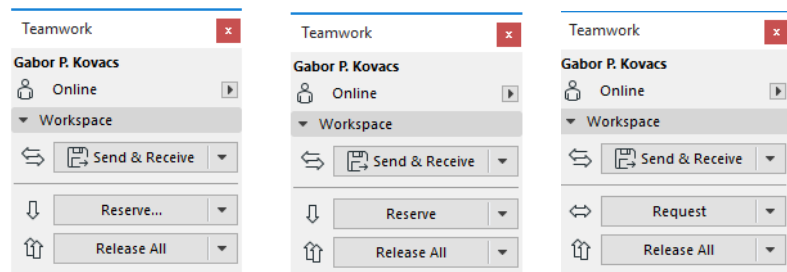
- **Send & Receive/Receive, then Compare:** These commands start the Model Compare process after Receive, so you can visualize what changed in the model as a result of Receive.

[See Compare Teamwork Model Versions \(Before and After Receiving Changes\).](#)

Reserve: This button takes several forms depending on whether you have selected an element, and its reservation status.

- If nothing is selected: The button says “**Reserve...**” and will take you to the Reserve Elements dialog box.

[See Reserve Elements by Criteria \(Teamwork\).](#)



- If you have selected elements that are, at least in part, free for reservation: The button says **Reserve**. After you issue this command, the available selected elements will be reserved for you. If any selected element could not be reserved, you are informed of the results.

See [Reservation Results \(Teamwork\)](#).

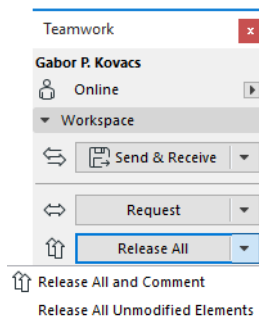
- If you have selected elements which are all unavailable, the command changes to **Request**. Clicking this starts the Ownership Request process.

See [Request Elements or Project Data \(Teamwork\)](#).

Release/Release All: If you have selected items reserved by you, the **Release** button will release them.

Release All is available when nothing is selected; this command will release all of your reservations.

Two additional commands are available from the Teamwork Palette's Release pop-up:

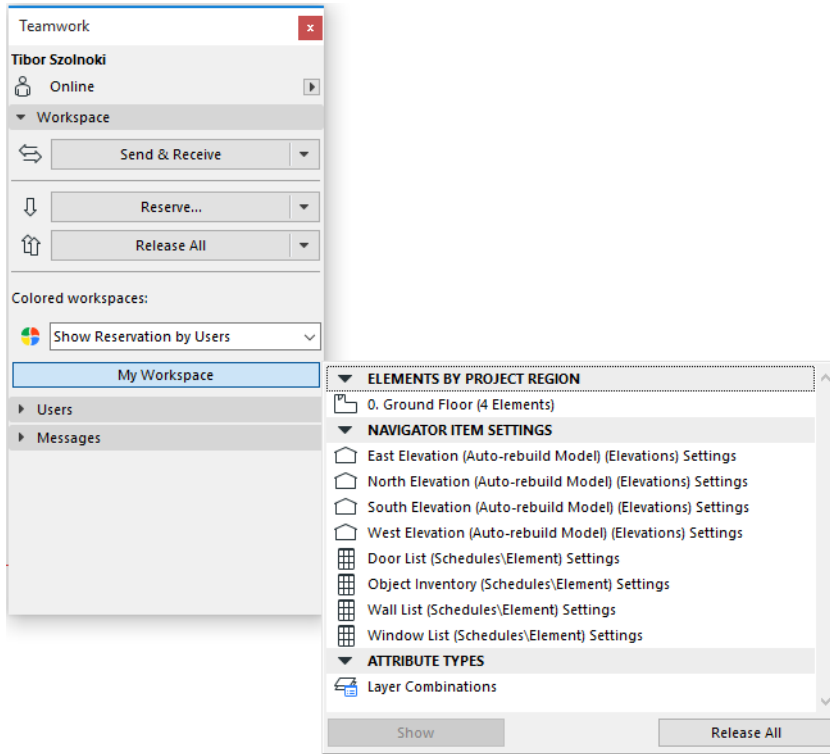


- **Release (All or Selected Elements) and Comment:** This option lets you add a comment to the Activities Panel of BIMcloud Manager, in addition to Releasing your reservations.
- **Release All Unmodified Elements:** This will release those of your reserved items that you have not modified.

Colored Workspaces: Use this drop-down to choose a method for distinguishing project elements by color depending on who owns them.

For details, see [Colored Workspaces](#).

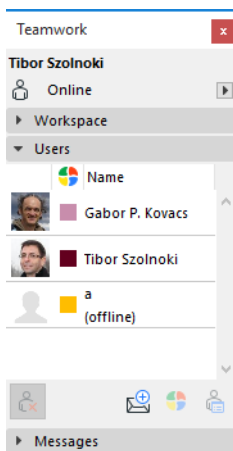
My Workspace: Click this button for an overview of all project items (both elements and non-elements) reserved by you.



Select any item in the list and click **Show** (to zoom to elements) or **Open** (to open the relevant dialog box). Click **Release** to release the selected item.

[See *Review Your Workspace \(Teamwork\)*.](#)

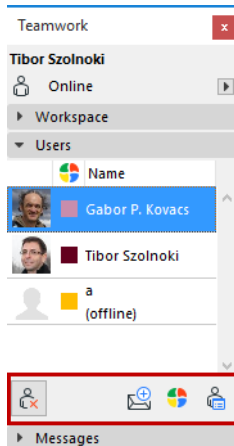
Users



This panel lists the users who are currently joined in to the project. If the user is currently offline, the “person” icon to the left of the name is grayed. The color accompanying each user name can be used to identify his/her elements on your screen.

[See “Show Reservations by Users” at *Colored Workspaces*.](#)

The icon buttons on the Users panel, from left:



Force Leave

For information, see [Force Leave](#).

Send Message

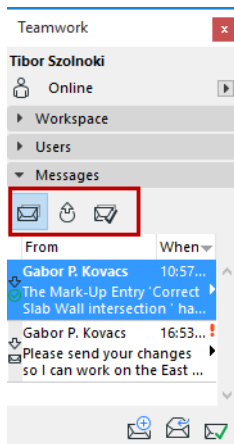
For information, see [Create a New Message \(Teamwork\)](#).

Set User Color

For information, see [Redefine User Colors Locally](#).

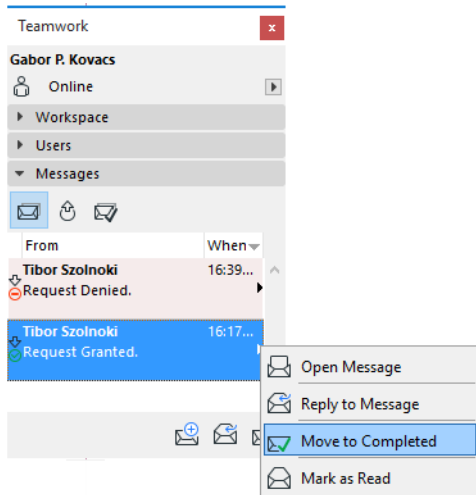
Messages Panel

This panel has three tabs. Click the three icons at the top to view the tab you need:



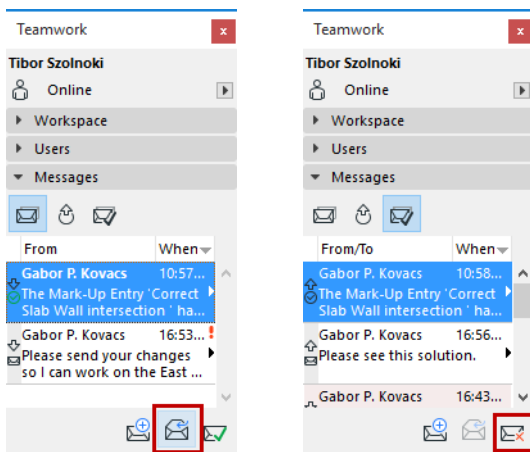
- The **To Do** list shows you items which await action from you.
- The **Pending Requests** list stores request messages sent by you, which are still pending.
- The **Completed** list is a repository of messaging actions that are no longer pending and no longer require action by you.

For any selected message, click the black arrow on the right to bring up a list of possible actions related to the message.



At the bottom of this panel, three icons represent three available actions for the selected mail message:

- **Send New Message:** Click to create a new message.
- **Reply Message:** Click to send a reply to the selected message.
- **Move to Completed:** Click to move this message into the Completed list.
- **Delete Message** (available only on the Completed list): Click to delete message.

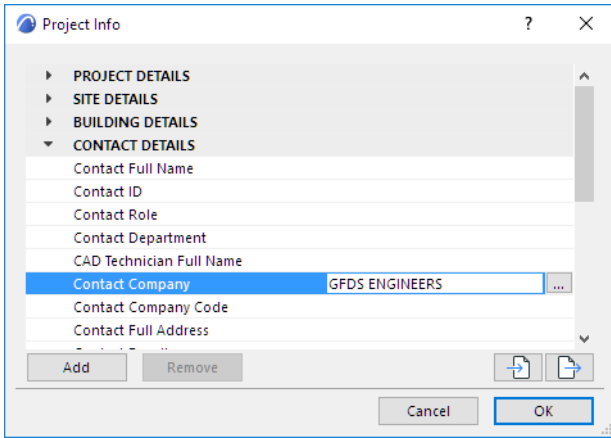


Project Changes Panel

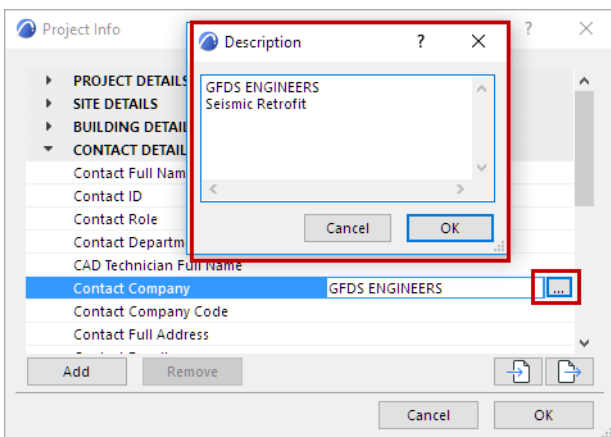
See [View and Manage Tracked Changes.](#)

Project Info

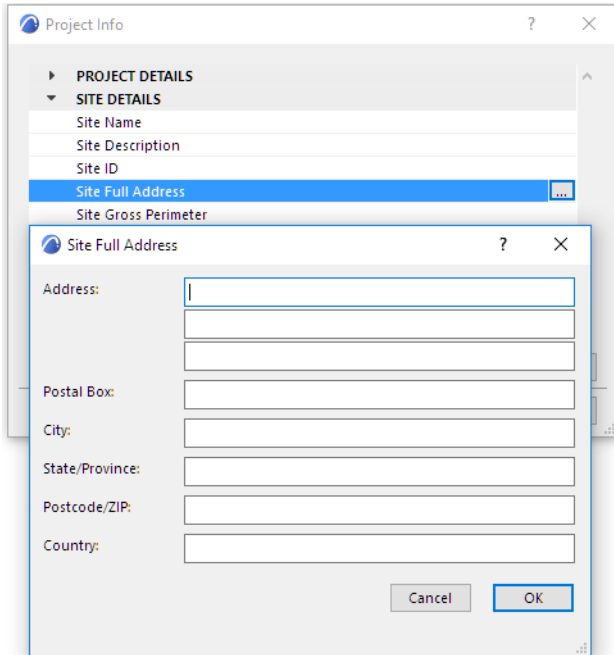
The **Project Info** command from **File > Info** opens a dialog box which enables you to enter information about the currently opened project. Select an item, then enter the desired information in the field.



If you need more space, click the three dots (...) at the right end of the field to access a Description field.



For some fields (e.g. Site Full Address) more complete entry fields are available when you click the three dots:



Defined Project Info items are available for use as AutoText entries in Text Blocks, in both Model Views and Layouts.

For more information, see [Autotext](#).

Use the buttons at right to manage Project Info entries.

- **Add:** Click to add a new custom Project Info entry to the currently selected group.
- **Remove:** Click to delete the selected Project Info item. (Only custom entries can be deleted.)
- **Import:** Click to load another Project Info .xml file. A file directory dialog box appears, letting you choose the desired .xml file.

Note: If you load a new Project Info file into your current project, all existing AutoText entries will be overwritten by the data in the imported Project Info file.

- **Export:** Click this button to save the current project info data as an .xml file. A file directory dialog box appears, letting you choose the path for the .xml file. You can then load this file into any other Archicad project. This can be useful when setting up company-standard Project Info data.

Project Notes

Access this window by choosing the **File > Info > Project Notes** command. Whenever you open it, the date and time will be updated.

It works like a note pad, where you can:

- Enter written notes about the project or comments for fellow designers.
- Keep track of the time spent on a particular job.

Any text entered here is saved with your project.

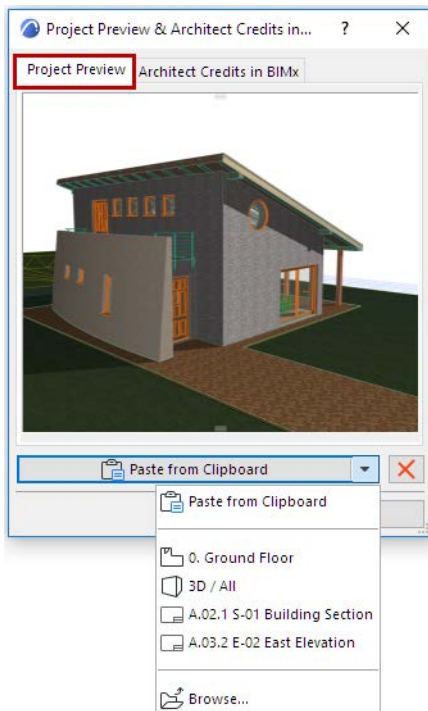
If you are signed into a Teamwork Project, the Project Notes window is not active; instead, **Teamwork > Teamwork Comments and Log** serves the same function.

See also [Editing Commands in Text-Type Windows](#).

Project Preview

Set an optional Project Preview image to help identify the project

- at **File > Open**
 - in the **Start Archicad** dialog box
 - when browsing for a project in file systems
1. Create an image or view to be the Project Preview. Do any of the following:
 - Copy any image to your clipboard
 - Set up the desired project view
 - Create a Rendered view of your project and save the image
 2. Go to **File > Info > Project Preview & Architect Credits in BIMx**.
 3. Click on the **Project Preview** tab.
 4. Use one of these commands to create the desired preview:
 - Paste from Clipboard
 - From the dropdown list of project tabs, click to select a view
 - From the dropdown, click **Browse** and select a saved image
 - Drag & Drop a saved image file



Notes:

- Any size and resolution image can be used.
- If you drag and drop, make sure the image is in JPEG format.

On Windows:

Project Preview image serves as the icon for the project file in Windows Explorer, replacing the standard Archicad icon.

On Mac:

The model's preview picture is visible in the Finder in the following locations:

- Finder window (Icon view)
- Finder window (Cover Flow view)
- Get Info window preview
- Quick Look window appearing for multiple selected elements (Thumbnail view)

Related Topic:

[Architect Credits in BIMx](#)

[Read more](#)

Architect Credits in BIMx

When navigating in BIMx, you have the option to display the logo of the architectural company who designed the project.

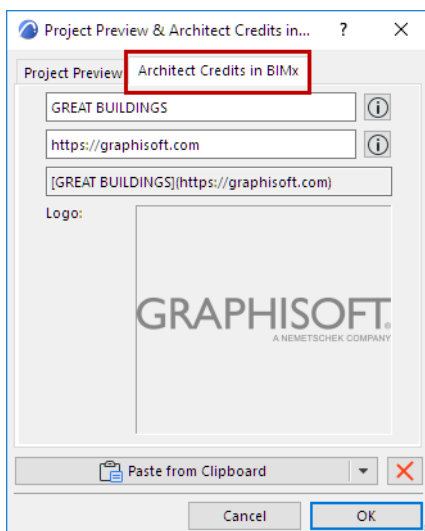
This information must be set up in Archicad, before you save or upload the BIMx Hyper-model.

Define Architect Credits in Archicad

1. Go to **File > Info > Project Preview & Architect Credits in BIMx**.

Note: This dialog also opens automatically from Publishing Properties - see below.

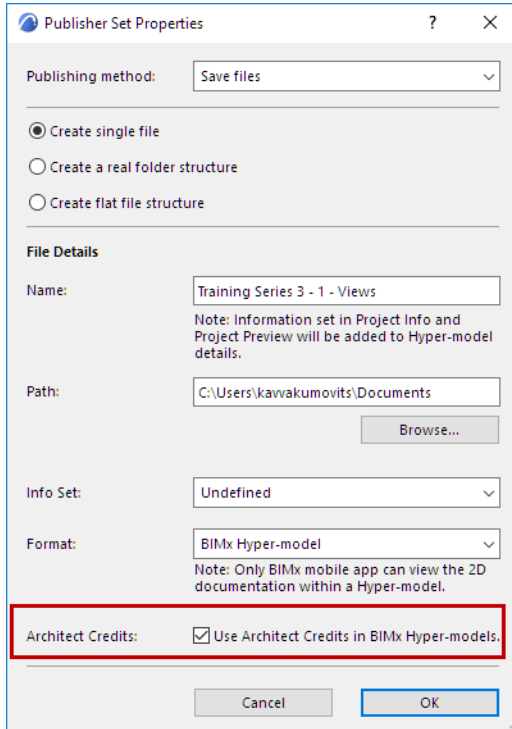
2. Click on the **Architect Credits in BIMx** tab.
3. Click in the "Display text" field to enter the text you want to see in Architect Credits, below the logo. It will work as a link to the website's URL.
4. In the URL field, enter the firm's URL address.
 - The URL must be a fully qualified URL that contains `http://`
 - Example: `http://www.graphisoft.com/`
5. Use Browse, Drag and Drop, or Copy-Paste to add an image file of the firm's logo to the Logo field:



Tip: To make the background transparent, use a PNG file with an alpha channel.

Add Architect Credits to BIMx Hyper-model

In Publisher Set Properties: When you set options to Save or Upload BIMx Hyper-Model, check the Architect Credits box.



The screenshot shows the 'Publisher Set Properties' dialog box. The 'Publishing method' is set to 'Save files'. Under 'File Details', the 'Name' is 'Training Series 3 - 1 - Views', the 'Path' is 'C:\Users\kavvakumovits\Documents', the 'Info Set' is 'Undefined', and the 'Format' is 'BIMx Hyper-model'. The 'Architect Credits' checkbox is checked and highlighted with a red box. The dialog also includes 'Cancel' and 'OK' buttons.

See also [Step 2: Set Publishing Properties](#).

The logo will appear as part of an animation when the Hyper-model is opened in the BIMx application. A user click on the logo will open the corresponding company web page (based on the URL defined in Archicad.)

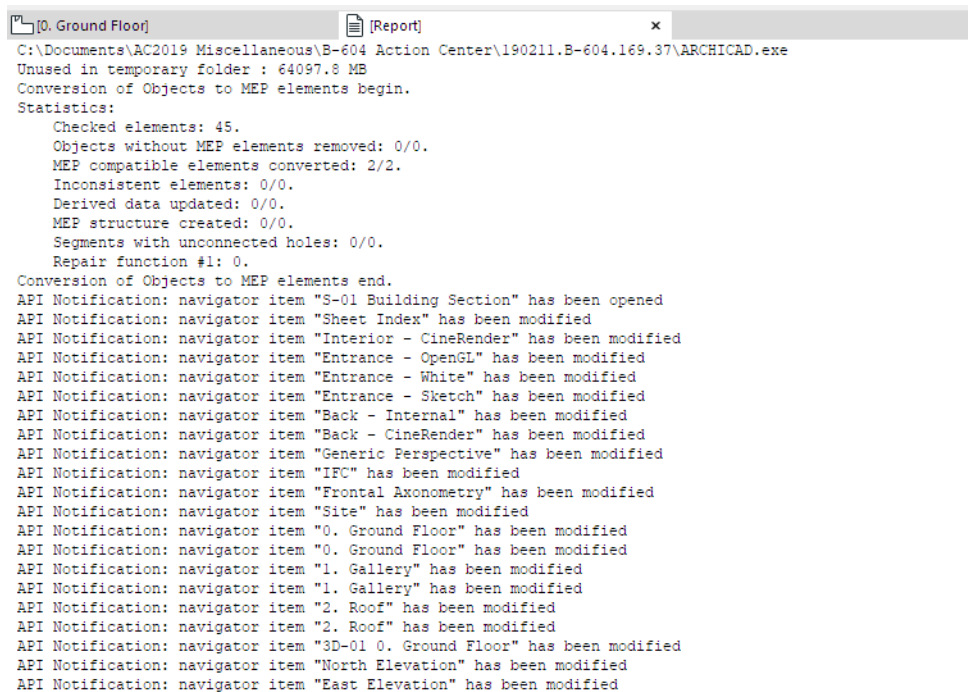
[Read More](#)

Related Topic:

[Project Preview](#)

Session Report Window

The **Report** window displays information about the imaging and listing process.



```

[0. Ground Floor] [Report] x
C:\Documents\AC2019 Miscellaneous\B-604 Action Center\190211.B-604.169.37\ARCHICAD.exe
Unused in temporary folder : 64097.8 MB
Conversion of Objects to MEP elements begin.
Statistics:
Checked elements: 45.
Objects without MEP elements removed: 0/0.
MEP compatible elements converted: 2/2.
Inconsistent elements: 0/0.
Derived data updated: 0/0.
MEP structure created: 0/0.
Segments with unconnected holes: 0/0.
Repair function #1: 0.
Conversion of Objects to MEP elements end.
API Notification: navigator item "S-01 Building Section" has been opened
API Notification: navigator item "Sheet Index" has been modified
API Notification: navigator item "Interior - CineRender" has been modified
API Notification: navigator item "Entrance - OpenGL" has been modified
API Notification: navigator item "Entrance - White" has been modified
API Notification: navigator item "Entrance - Sketch" has been modified
API Notification: navigator item "Back - Internal" has been modified
API Notification: navigator item "Back - CineRender" has been modified
API Notification: navigator item "Generic Perspective" has been modified
API Notification: navigator item "IFC" has been modified
API Notification: navigator item "Frontal Axonometry" has been modified
API Notification: navigator item "Site" has been modified
API Notification: navigator item "0. Ground Floor" has been modified
API Notification: navigator item "0. Ground Floor" has been modified
API Notification: navigator item "1. Gallery" has been modified
API Notification: navigator item "1. Gallery" has been modified
API Notification: navigator item "2. Roof" has been modified
API Notification: navigator item "2. Roof" has been modified
API Notification: navigator item "3D-01 0. Ground Floor" has been modified
API Notification: navigator item "North Elevation" has been modified
API Notification: navigator item "East Elevation" has been modified

```

Open Session Report

Use **File > Info > Session Report**.

The Session Report is also accessible from the Action Center (**File > Info > Action Center**), which alerts you if the Report contains any issue requiring user action.

Enable Session Report (Work Environment)

To create Session Reports, make sure that the **Write Report** option is checked, at:

Options > Work Environment > Model Rebuild Options > 3D and Calculation Process Check Controls

It is checked by default.

Contents of Report Window

The Report Window shows info on:

- Imaging processes
- Calculation processes
- Input/output operations
- Name and version number of newly loaded Add-ons

You can use the report to keep track of and compare imaging time for different images.

New information is always added to the end of the report.

The Report is cleared when you close the Project. If you wish to preserve it, save it as a text file.

Related Topics:

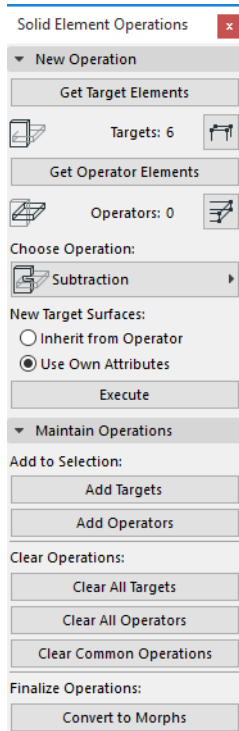
[3D and Calculation Process Check Controls](#)

[Action Center](#)

Solid Element Operations Palette

For more information, see [Solid Element Operations](#).

The **Design > Solid Element Operations** command opens this floating palette.



New Operation Panel

Use the controls in this panel to set up and execute a Solid Element Operation.

- **Get Target Elements:** Click to store the selected objects as Target Elements. They are now stored as Target Elements (even after you deselect them) as long as the Solid Element Operations palette remains open.

Note: If you selected the Target elements in the 3D Window, they may not all belong to the same Story. In this case, a warning will inform you about this.

To select the stored target elements again if you have deselected them, click the Select Stored Targets button.



Note: If you then select another element and click the Get Target Elements button again, it will replace the stored selection.

- **Get Operator Elements:** Click to store the selected objects as Operators. You do not need to perform this selection in the same window as you did for the Targets; you can select the Target on one story and the Operator on another one.

To select the stored target elements again if you have deselected them, click the Select Stored operators button.



- **Choose an operation:** Click to select the desired operation type between Target and Operator elements.
- **New Surfaces of Target will:** Specify whether the new surfaces of Target Elements will use the Target Element surface attributes, or whether they will inherit the Operator Elements' surface attributes.
Inherit Attributes of Operator: Choose to define attributes of the new surfaces as those of the Operator Elements.
Use Their Own Attributes: Choose to define attributes of the new surfaces as those of the Target Elements.
- **Execute:** Click to create a Solid Operation Link between the stored Operator and Target Elements.

Maintain Operations Panel

Maintain Operations: Use the controls in this panel to verify, edit and cancel already existing Solid Operations. These controls affect elements that have been linked via a completed Solid Operation, and have no connection to the settings of the New Operation panel above.

Add to Selection

If you have selected an element that has participated in a Solid Element Operation, use the following two buttons to add more elements to the selection: those which are either the targets or the operators of the selected elements. You can click both buttons in succession; this will select all the targets and all the operators of the linked Solid Element Operation.

- **Targets of Selected Elements:** Click this button to add elements to the current selection which are targets of the selected element.
- **Operators of Selected Elements:** Click this button to add elements to the current selection which are operators on the selected element.

Cancel Operations of Selected Elements

- **All Targets:** Click this button to cancel any links that may exist between the selected operators and their targets (if any).
- **All Operators:** Click this button to cancel any links that may exist between the selected target elements and their operators (if any).
- **All Other Selected Elements:** The current selection may contain elements linked to each other by a solid operation. Click this button to cancel these links.

Classic 3D Navigation Toolbar

If you prefer to use the 3D navigation controls of Archicad 9 and earlier, activate the **Window > Toolbars > Classic 3D Navigation** command. The commands specific to this toolbar are:



- **3D Edit icon:** represented by an Arrow-Wall-Slab combination; with this mode on, 3D construction Tools can be used and the elements edited.
- **Walk:** Walk tool with Camera mode. Move back and forth and turn around.
- **Lateral Move:** Lateral Move tool with Camera mode. Move up and down and laterally to the left or right, but keep looking in the same direction.
- **Turn:** Turn tool with Camera mode. Stand still and look around in all directions.
- **Look To:** Interactively set the target point for the perspective.
- **Lock Target Point:** In the middle of the image, a rectangle will appear. This is the “neutral area”. If you click anywhere between the rectangle and the window border, you will begin moving. Click at the top or bottom for one kind of movement, to the left or right for another, and in the corner areas for a combination of the two.

Click close to the rectangle to move slowly, close to the window’s border to move fast. To move at a constant speed, keep the mouse pressed. Drag the pointer around to increase or decrease speed and change directions. For extreme speed, drag the cursor out of the 3D Window as far as your screen size permits.

If there are individually selected elements in the 3D Window when you start navigating, only these elements will be displayed as you move. This feature can be extremely helpful with large projects. It also allows you to analyze one or several elements on their own, without their environment.

- **Look to Perpendicular of Clicked Surface:** Switch to a view perpendicular to a given point.
- **Reset Roll Angle:** Reset the roll angle to zero to rapidly return from a bizarre view obtained during navigation.
- **Horizontal View:** Return with a single command to a horizontal view of the model.

Dialog Boxes

This section describes Archicad's dialog boxes and palettes in detail. The list on this page is divided into five categories to help you navigate to the dialog box you need.

Other interface item descriptions can be found:

- in [Tool Settings Dialog Boxes](#); and
- in [Controls](#).

Configuration

[Work Environment Dialog Box](#)

[Project Preferences Dialog Boxes](#)

[Add-on Manager](#)

Working in Model Views

[Story Settings Dialog Box](#)

[Edit Elements by Stories Dialog Box](#)

[Floor Plan Cut Plane Settings Dialog Box](#)

[Copy Dialog Box](#)

[Linework Consolidation Settings](#)

[Fill Consolidation Settings](#)

[Model View Options for Construction Elements](#)

[Stair Options](#)

[Detail Level of Door, Window and Skylight Symbols](#)

[Miscellaneous Settings for Library Parts](#)

[3D Document Settings](#)

[Filter and Cut Elements in 3D Dialog Box](#)

[3D Styles](#)

[Hardware Acceleration Options](#)

[3D Projection Settings](#)

[Sunlight Settings](#)

[RoofMaker Settings](#)

[TrussMaker Settings](#)

[Grid System Settings](#)

[GDL \(Geometric Description Language\)](#)

[Scheme Settings Dialog Box \(Interactive Schedule\)](#)

[Index Settings Dialog Box](#)

[View Settings Dialog Box](#)

[Book Settings Dialog Box](#)

[Subset Settings Dialog Box](#)

[Layout Settings](#)

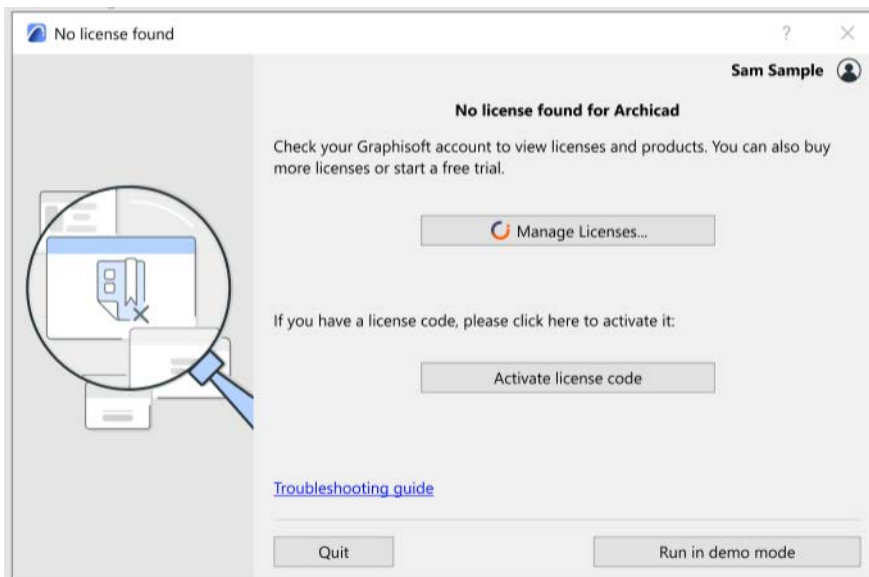
[Master Layout Settings](#)

[Drawing Manager](#)

Output and Imaging[Paste Options Dialog Box](#)[Print 2D Document](#)[Print 3D Document](#)[Print Picture](#)[Print Layout](#)[Plot Setup](#)[Plot 2D Document](#)[Plot Layout](#)[PhotoRendering Settings](#)**Interoperability**[XREF Management Dialog Box](#)[DXF/DWG Translation Setup](#)[Merge DXF-DWG](#)

No license found - when starting Archicad

This dialog box appears when you start Archicad and you are signed in, but your license is not available.



Why don't I have a license?

Possible reasons:

- Your Company account administrator has not assigned a license to your Graphisoft account
- You had an assigned license, but it expired (and your contract manager did not renew it, e.g. due to issues with the payment)
- You had an assigned license, but your contract manager canceled the subscription
- You signed in to a Graphisoft account with the wrong Graphisoft ID, and that account does not have an assigned license

What should I do now?

One of the following:

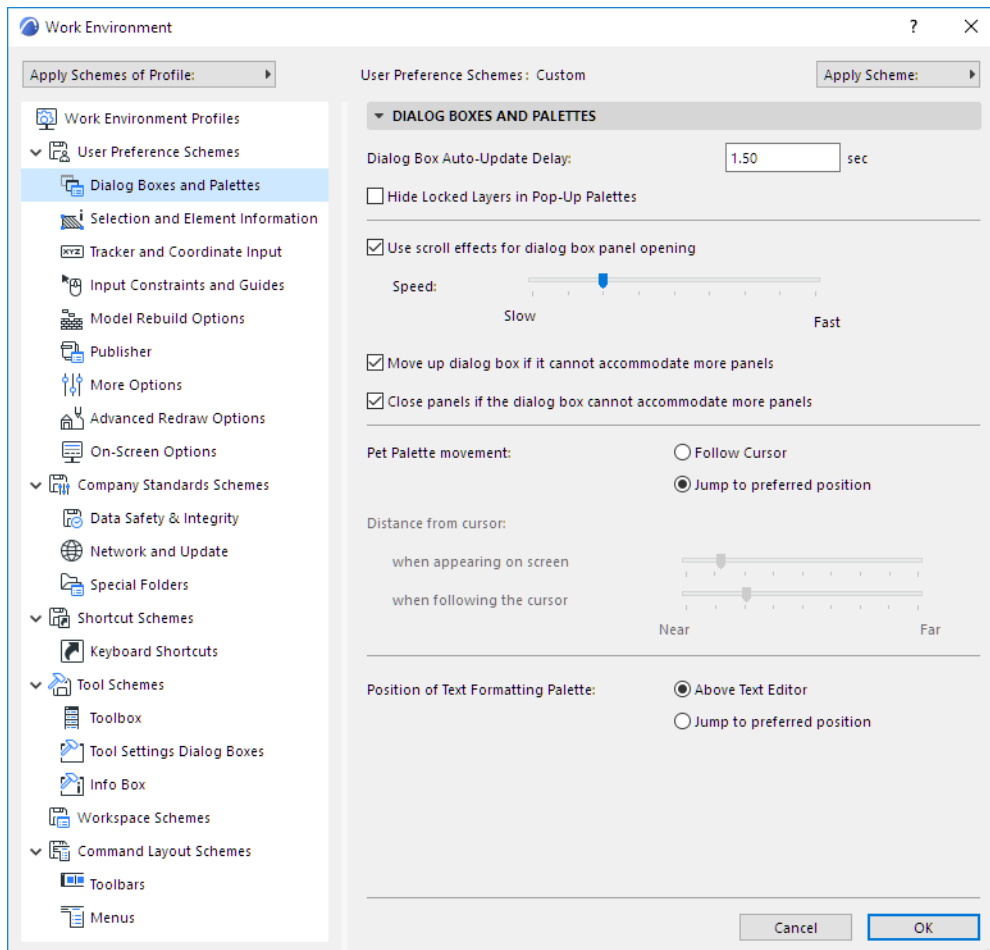
- Click **Manage Licenses** to check you license status in your Graphisoft account, and try to fix it.
- Ask your Graphisoft account administrator or contract manager to assign or buy a license for you
- If you have an EDU or Emergency license, click **Activate license code** to enter it.
- Click the button to **Run in demo mode**

[See also Archicad license types.](#)

Work Environment Dialog Box

As you get used to working with Archicad, you will develop personal preferences for using the program's features and arranging the various palettes, toolbars and menus on screen. You will set most of these options in the many sub-dialog boxes opened from the **Work Environment** dialog box at **Options > Work Environment > Work Environment**.

You can customize any of these settings while you work: just open the needed dialog box, change the setting, and press OK. The setting will take effect.



[Read More](#)

Each setting of the Work Environment dialog boxes is described in the sections below:

[Dialog Boxes and Palettes](#)

[Selection and Element Information](#)

[Tracker and Coordinate Input](#)

[Input Constraints and Guides Dialog Box](#)

[Model Rebuild Options](#)

[Publisher Log File](#)

[More Options](#)

[Advanced Redraw Options](#)

[On-Screen Options](#)

[Data Safety & Integrity](#)

[Network And Update](#)

[Special Folders](#)

[Shortcut Customization Dialog Box](#)

[Toolbox Customization Dialog Box](#)

[Tool Settings Dialog Box Customization](#)

[Info Box Customization Dialog Box](#)

[Toolbar Customization Dialog Box](#)

[Menu Customization Dialog Box](#)

Saving Your Customized Work Environment

If you have customized a number of settings in **Options > Work Environment**, you may wish to save them for future use in other projects, so that your personalized Work Environment is ready for you when you start to work.

Individual settings are organized into six **Schemes**; *saving your settings occurs at the Scheme level*. Work Environment Schemes are saved in a local folder on your computer (not with the project file). You can store, import and export any or all of the scheme files.

[See *Work Environment Schemes* for more information.](#)

You can also combine any of the schemes into a **Profile**, and store the Profile under its own name. Profiles themselves do not contain settings; they are just a collection of schemes. Use **Options > Work Environment > Work Environment Profiles** to export and import profiles to/from folders of your choice.

[See *Work Environment Profiles* for more information.](#)

It is entirely possible to use Archicad successfully without saving schemes and profiles; you can just adjust settings as you work. However, the option to save settings according to named schemes, and then to combine schemes into profiles, is useful for CAD managers, for offices working in teams, and for individual users who might want to alternate among several different sets of customized settings.

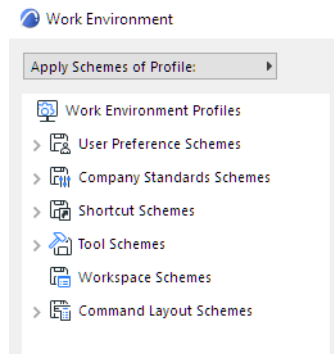
Creating an Office-Standard Work Environment

CAD managers can create office-standard Work Environment Profiles and install them as defaults when installing the Archicad program at multiple workstations.

[For more information, see *Network Installation for BIM Managers*.](#)

Work Environment Schemes

Work Environment settings (**Options > Work Environment**) are divided into six self-contained **schemes**. Each scheme is a thematic collection of settings.



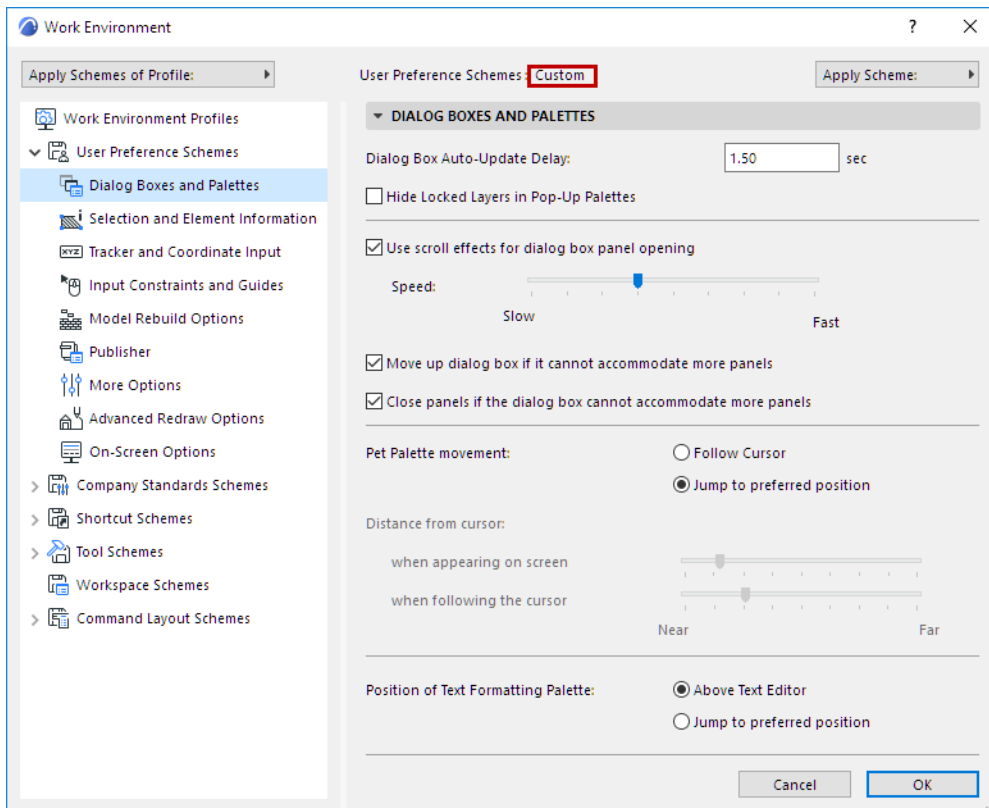
- User Preference Schemes
- Company Standard Schemes
- Shortcut Schemes
- Tool Schemes
- Command Layout Schemes
- Workspace Schemes

The settings belonging to each scheme are set in the dialog boxes that belong to them. See the description of each dialog box for info on all of these settings.

Exception: Workspace Schemes consist of your workspace configuration.

For more information, see [Workspace Schemes](#).

As soon as you make a change to any Work Environment setting, the scheme name at the top of the settings page changes to **Custom**:



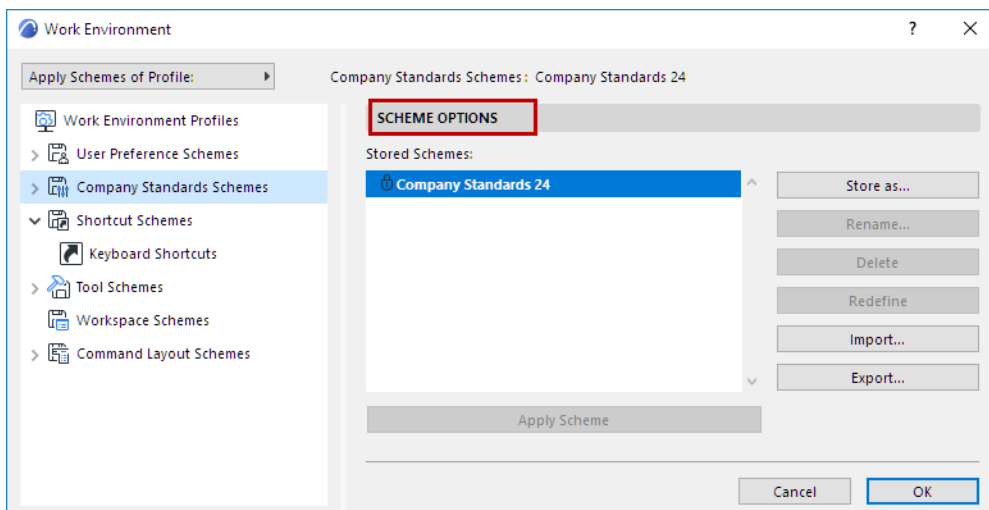
When you finish making changes, press OK to close the dialog box and apply current settings.

The latest Custom settings will remain intact even after you close and restart Archicad.

However, every new settings change redefines the “Custom” scheme according to your latest change. If you want to keep your modifications saved on the long term, it is advisable to store the settings of your Custom scheme in a named scheme, using the Scheme Options dialog box.

Scheme Options Dialog Box

The **Scheme Options** screen appears in the Work Environment dialog box (**Options > Work Environment**) if you have selected one of the six scheme sets from the tree structure at the left side of the dialog box. There is one Scheme Options page for each of the six schemes.



Stored Schemes: This list shows all the Schemes stored for the currently selected Work Environment Profiles. Select the Scheme you wish to apply, rename or delete.

Note: A lock icon next to the scheme name means that the file is read-only; the profile is locked. It is not possible to edit or delete locked schemes.

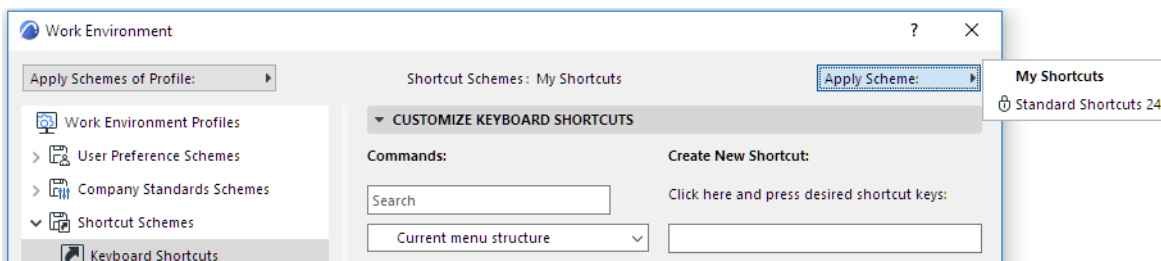
Store a Scheme

Click **Store as** to enter a name and save settings of the currently selected Work Environment component into a Scheme. Each scheme must have a unique name.

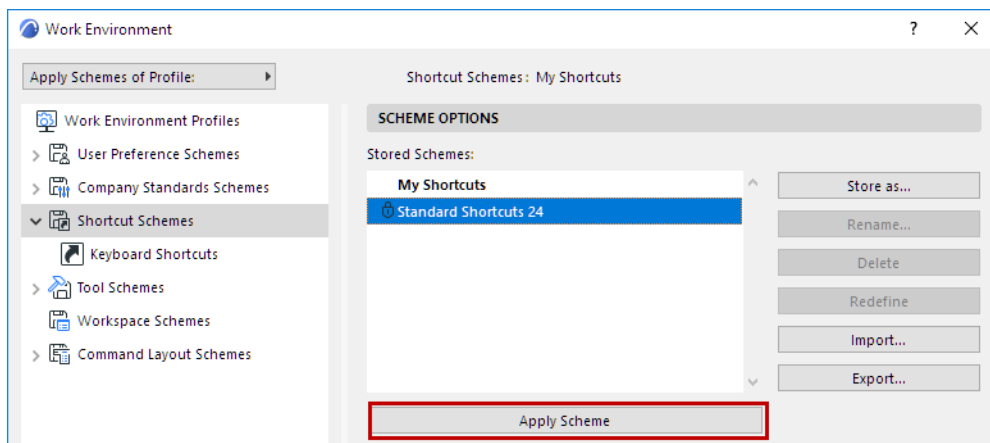
Apply a Scheme

To apply a stored scheme, do one of the following:

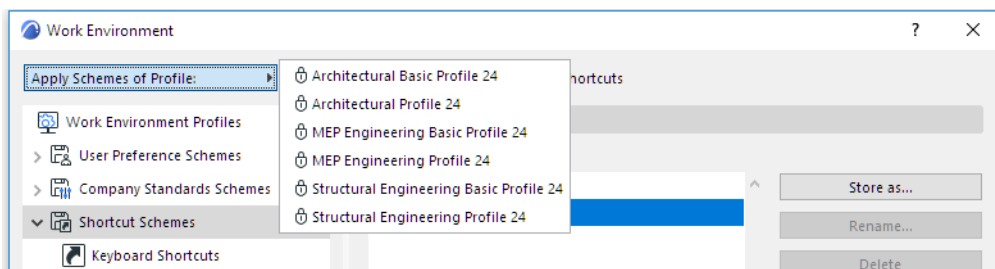
- At the top of any of Work Environment settings pages, choose the desired scheme from the Apply Scheme pop-up:



- On the **Scheme Options** page of Work Environment: choose one of the defined schemes, then either double-click it, or click the **Apply Scheme** button.



- At the top left of any of the Work Environment settings pages, use the **Apply Schemes of Profile** pop-up, choose one of the project's defined profiles, and click it to apply all of its schemes.



Once you apply the scheme and press OK to close the Work Environment dialog box, your Archicad program will adjust itself to reflect the settings you have stored in that scheme.

Rename: Use the Rename Scheme dialog box to give the selected scheme a new name.

Delete: Click this button to delete selected Scheme and the settings stored in it.

Redefine: To redefine a selected scheme according to current settings, use the **Redefine** command. In effect, you will replace the stored settings with the latest Custom settings.

Export/Import Schemes

To make your schemes available to a network or other users, or to transfer them to another computer, use the Scheme Options page to export and import schemes to/from folders of your choice. Schemes can also be exported/imported as a group, when combined into a Profile.

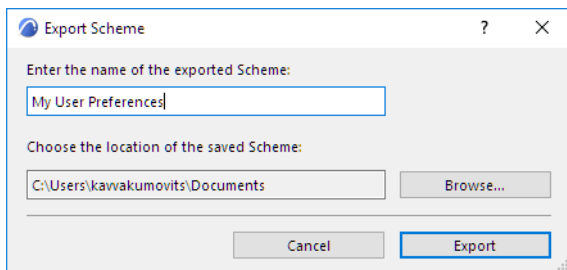
Note: While you can import/export Work Environment schemes among computers using varying operating systems, this is not recommended, because - due to the differences between Windows and Mac - some of the imported shortcuts, menus and Workspace schemes may not correspond to your system standards.

- When transferring Workspace Schemes from Windows to Mac, the Application Frame will always be enabled
- There is one key difference between Mac and PC menus: the Mac has a customizable Archicad (system) menu. Transferring this menu to Windows will result in a new menu to the left of the File menu. This “no name” menu should be kept and not renamed. If you delete it, another will be created automatically.

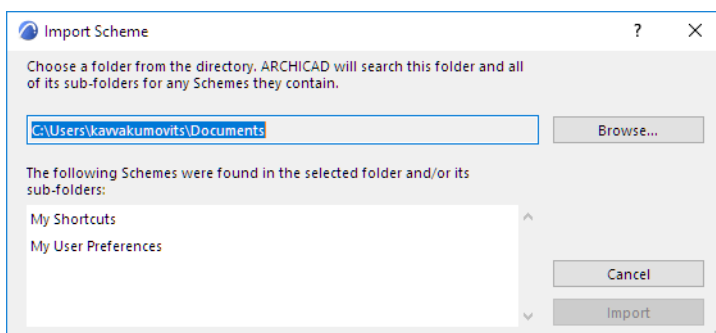
You can also set certain schemes as defaults and have users load these defaults during start-up from a predefined server location.

For more information, see the [Network Installation for BIM Managers](#).

Export: Enter a name for the exported scheme, if you want to export it under a different name. Click the Browse button to choose a location for the scheme, then click Export.



Import: Click **Browse** and locate the folder you need. (Folders can be located on your hard drive or any network location.) When importing schemes, you must point to the folder that contains the scheme(s), not to the actual scheme file itself.



If the folder contains multiple schemes, choose the one you want from the list, then click Import. The browser will list only those schemes that fit the scheme category you are in. (For example, you cannot import a Shortcut scheme into the list of Tool Schemes.)

The imported scheme will be added to the list of stored schemes in your Work Environment dialog box.

Note on Schemes with Identical Names: You can import a scheme that has the same name as a scheme you already have, but Archicad will differentiate between them by displaying the date and time the scheme was created.

Workspace Schemes

These differ from the rest of the Work Environment schemes: the settings they contain are defined not in the Work Environment dialog box; instead, Workspace Schemes settings reflect the way you manually set up your palettes in your workspace.

To customize your Workspace scheme, set up your palettes, toolbars and tab bar manually in the Archicad workspace by doing any or all of the following:

- Show or hide **palettes** as needed (click its name on or off from **Window > Palettes**)
- Show or hide **toolbars** as needed (click its name on or off from **Window > Toolbars**)

Note: While the show/hide status of a toolbar is saved in a Workspace scheme, its content is defined as part of a Command Layout Scheme.

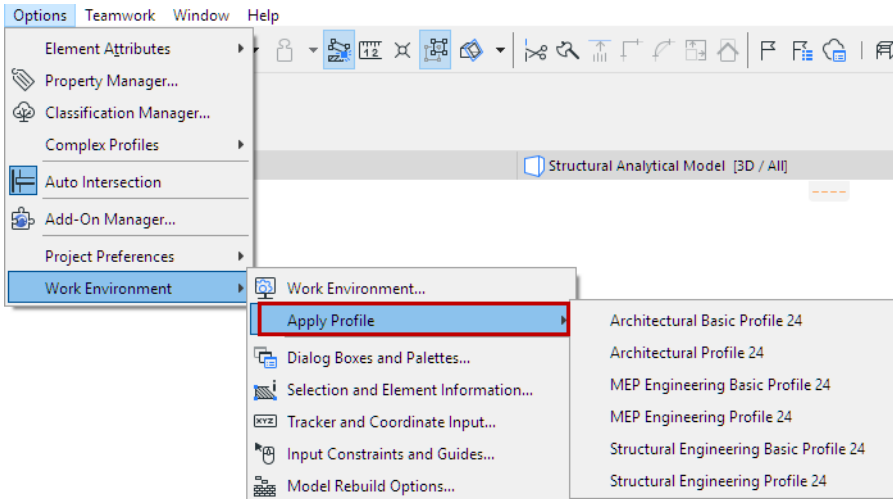
- Show or hide the **Tab Bar, Scrollbar** and **Quick Options Bar (Window > Show/Hide)**
- Change the **position** and **size** of a palette by dragging the palettes to a desired position.
- **Dock** palettes and toolbars as needed

Use the Scheme Options page of Workspace Schemes to manage and apply Workspace Schemes.

Work Environment Profiles

Apply a Profile

Use **Options > Work Environment > Apply Profile** and choose a profile.



Save New Profile with Customized Schemes

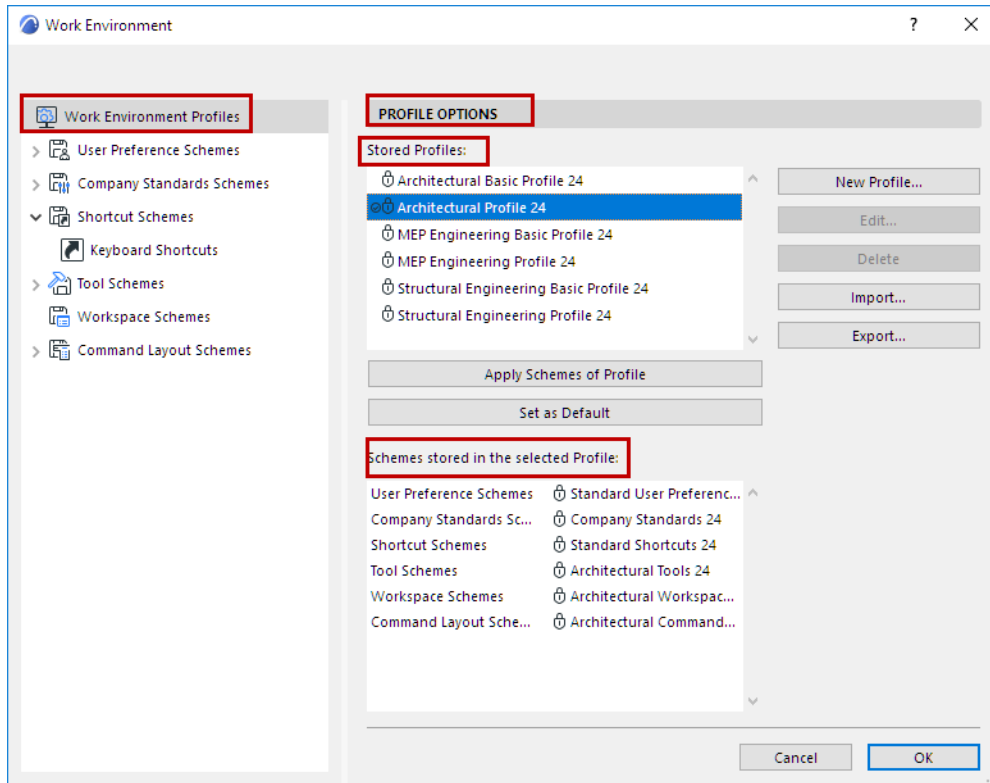
If you have customized and saved multiple schemes, you may want to store and apply them together: in this case, combine any of the schemes into a Profile, and store the Profile under its own name. Using the Archicad interface, you can export and import Profiles to use on other computers. In this operation, profiles are.xml files which refer to the schemes (also.xml files) contained within them.

Notes about Profiles:

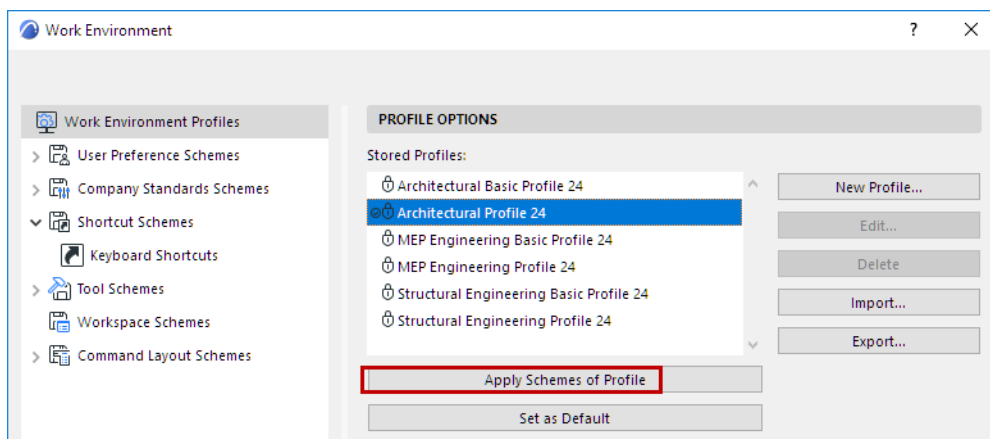
- Profiles themselves do not contain settings; they are just a collection of schemes.
- A profile does not have to contain all six scheme possibilities.
- Once you start working, you don't have to apply entire profiles at once; you can apply schemes one by one.
- You can save a Custom scheme as part of a profile, but the Custom scheme will be given a name when the profile is created.
- Schemes named "Custom" cannot be exported.

Profile Options

To assemble any combination of schemes into a profile, open **Options > Work Environment** and click “Work Environment Profiles” (the top item in the list at the left.) This opens the Profile Options dialog page.



From the list of Stored Profiles, choose the desired Profile. Double-click to apply the profile, or click the Apply Schemes of Profile button.



Stored Profiles displays the list of available Profiles.

Note: A lock icon next to the profile name means that the file is read-only; the profile is locked. It is not possible to edit or delete locked profiles.

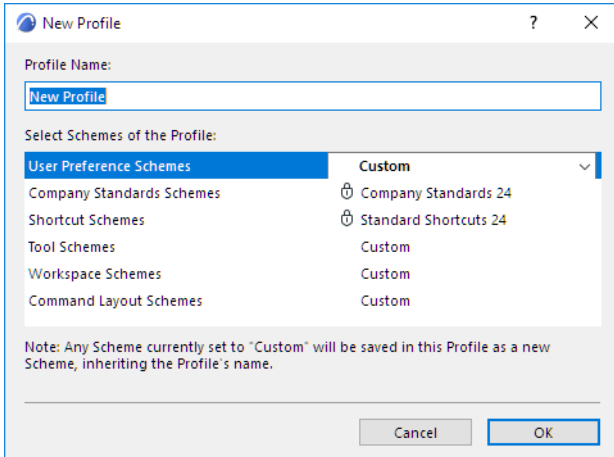
Clicking on any of them displays, in the window below, the **Schemes stored in the selected Profile**.

Set as Default

Click this button to define the selected profile as your default profile. Henceforth, this profile will be applied if you choose “Default Profile” from the Set up Work Environment pop-up when starting Archicad, or with the **New & Reset All** command.

New Profile

Use the New Profile dialog box to define a new profile.

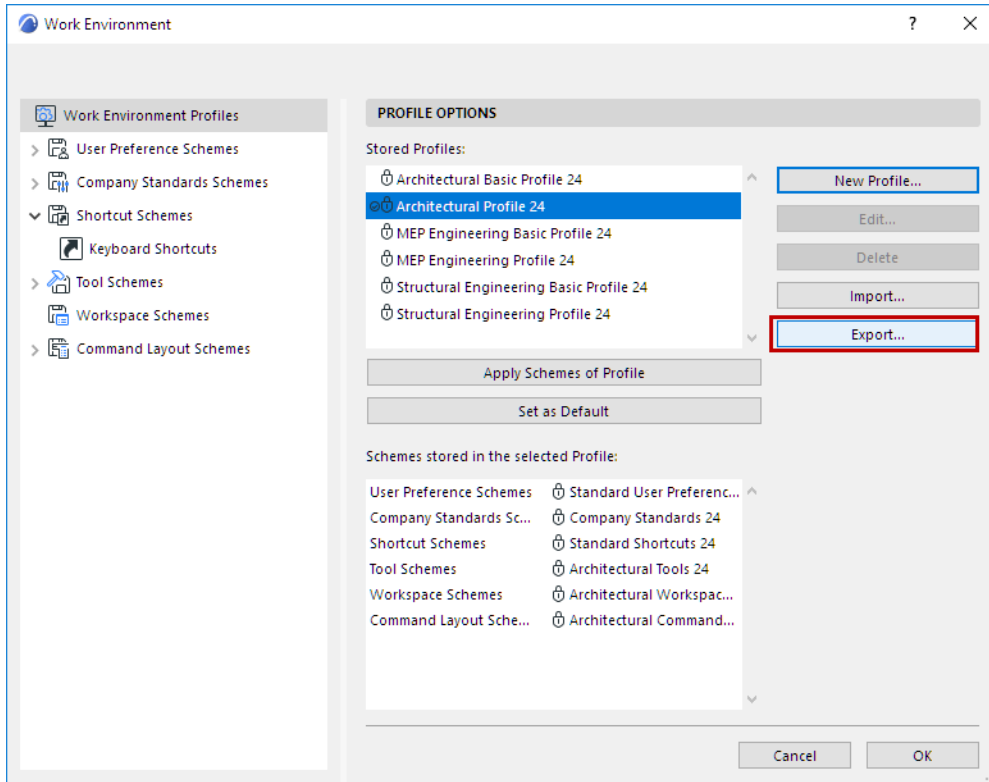


- **Enter the name of the new profile** in the field. Each profile must have a unique name.
- **Select the Schemes of the Profile:** Choose the schemes you want the new profile to contain. Highlight each scheme type, then choose the scheme you want from the drop-down list at right. If you don't want a particular scheme to be defined as part of this profile, choose “Undefined” for that scheme. When you apply the new profile, the “undefined” scheme's settings will remain “as-is” or default. Any scheme in the new profile set to “Custom” will be stored as a scheme (and given the same name as your profile), and will be saved as part of your new profile. If you don't want to include this scheme, set it to “undefined”.

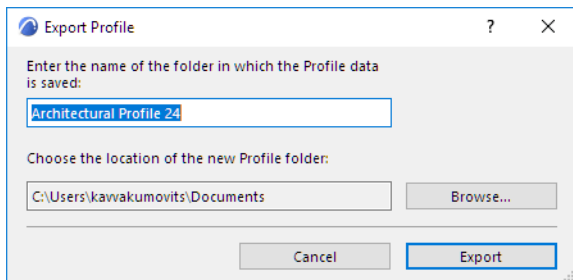
Apply Schemes of Profile

Click this button to apply the Schemes stored in this Profile.

Export Work Environment Profile



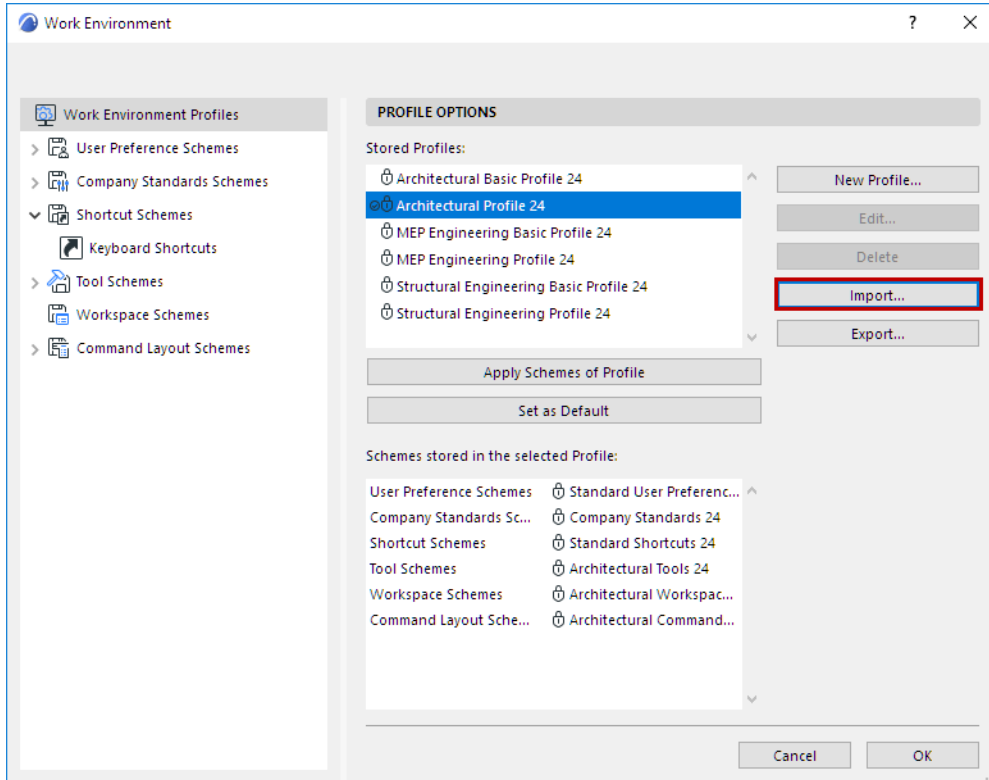
- Select a Profile and click Export. **Enter a folder name in which to save the profile data.** (By default, the folder name is identical to the profile name; but you can choose any folder name, and the folder can contain multiple profiles.)



Click the **Browse** button to choose a location for the profile folder, then click **Export**.

Import Work Environment Profile

Click **Import**.



From the appearing dialog, click **Browse** to locate the profile folder you need. If the folder contains multiple profiles, choose the one you want from the list, then click **Import**.

Note on Profiles with Identical Names: You can import a profile that has the same name as a profile you already have, but Archicad will differentiate between them by displaying the date and time the profile was created.

Note: While you can import/export Work Environment schemes among computers using varying operating systems, this is not recommended, because - due to the differences between Windows and Mac - some of the imported shortcuts, menus and Workspace schemes may not correspond to your system standards.

Dialog Boxes and Palettes

(Options > Work Environment > Dialog Boxes and Palettes)

This screen contains options related to the display of dialog boxes and palettes.

Dialog Box Auto-Update Delay: Enter a value in seconds for the auto-update delay time for edit text dialog boxes. (This is the amount of time it takes for the program to “validate” your typed entry as the desired value; you do not have to click “Enter”.)

Hide Locked Layers in Pop-up Palettes: This option means that locked layers will not be shown in any of the layer pop-up palettes.

Use Scroll Effects for Dialog Box Panel Opening (Windows Only): By default, to prevent crowding of dialogs when opening new panels, previously opened panels will close automatically, and the dialog box will move upwards on the screen. If you check this box it means that dialog box panels will “roll open” (at the speed you set with the **slider**) instead of “snapping open,” which is the effect you get if you uncheck the box.

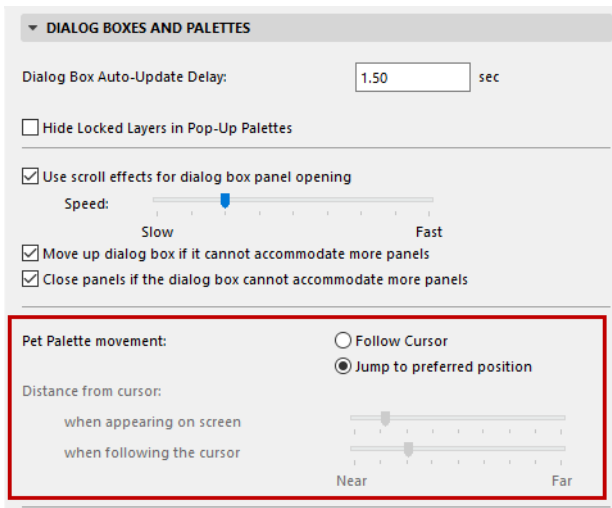
Move up dialog box if it cannot accommodate more panels: Many dialog boxes have five or more panels that might not all fit on the screen if they were opened all at once. This option will move the whole dialog box up to make room for newly opened panels.

Close panels if the dialog box cannot accommodate more panels: This option will close one or more open panels to make room for newly opened panels.

Pet Palette Movement

To define the type of pet palette movement you prefer, open **Options > Work Environment > Dialog Boxes and Palettes**.

Choose a radio button to determine whether and how your pet palettes will move around on the screen as you work.



- **Follow Cursor:** This means that the pet palette will follow your cursor around the screen as you complete your editing operation. (This way the editing icons on the pet palette are available right next to your cursor instead of at a far corner of the screen.)
- **Jump to Preferred Position:** This means that when you bring up a pet palette and release the mouse button, the pet palette jumps to one position and stays there as you complete the editing operation. If you want the pet palette to “stay” someplace else, drag and drop the palette to your preferred position. All subsequent pet palettes will appear at that preferred position. You can still move the pet palette at any time during your editing operation.

Distance from Cursor

Use the two slider controls to set the palette’s following distance from the cursor.

- The first slider sets the palette’s initial distance from the cursor when it first appears on screen.
- The second slider sets a minimum “following distance”: the palette will not follow the cursor until the cursor is at that predefined distance from the palette. (This is useful if you want to prevent the palette from covering up the very area that you want to edit.)

The operations that can be performed by using the pet palette are detailed in [Editing Elements](#).

Position of Text Formatting Palette

- **Above Text Editor:** Check this to locate the Text Formatting Palette above the Text Editor.
- **Jump to preferred position:** Check this to have the Text formatting Palette jump to a preferred position. This means that when you bring up a the Text Formatting palette and release the mouse button, the pet palette jumps to one position and stays there as you complete the editing operation. If you want the palette to “stay” someplace else, drag and drop the palette to your preferred position. All subsequent Text Formatting palettes will appear at that preferred position. You can still move the palette at any time during your editing operation.

For more information on this palette, see [Text Block Formatting Panel](#).

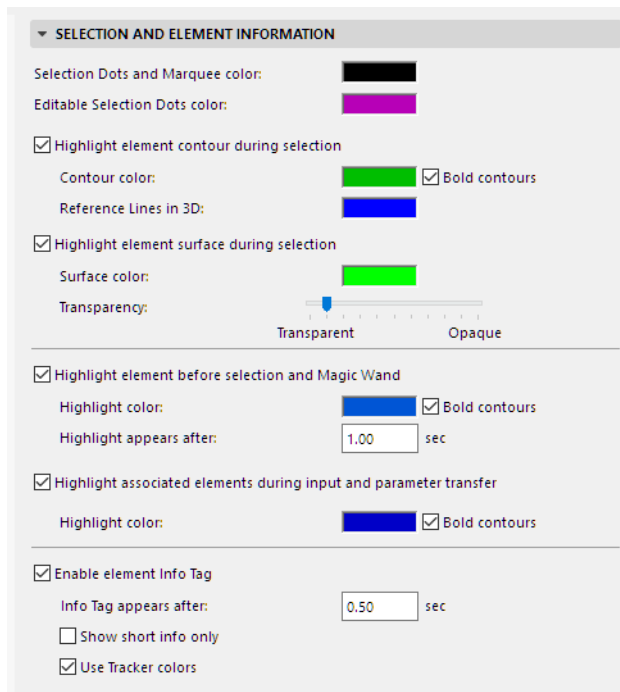
Keep Archicad Palettes visible even when another application is in front (*Available for Mac only*): Check this box if you want to view any open Archicad palettes while another application is in front on your screen.

Show User Photos in Lists

This is a Teamwork-specific option. In Teamwork projects, this option will display the photo of joined-in users on the Teamwork Palette, and on the Users list when sending a message.

Selection and Element Information

(Options > Work Environment > Selection and Element Information)



Selection Dots and Marquee Color: Click in the field to access a color pop-up and customize the color of the dots appearing on selected individual elements, as well as the marquee.

Editable Selection Dots Color: Click in the field to access a color pop-up and customize the color of editable selection dots.

For details about editable selection dots, see [Graphical Editing Using Editable Hotspots](#).

Highlight Element Contour During Selection: Check this box if you wish to have the selected elements' contours and reference lines highlighted.

Note: "Reference lines" include the reference line of a Wall or Beam, the pivot line of a Roof, and the profile(s) and extrusion vector or revolution axis of a Shell.

- **Contour color:** Choose a color for the selected elements' contour from this pop-up palette.
- **Bold contours:** Check this box to display the selected elements' contours in bold font style.
- **Reference Lines in 3D:** Choose a color for the selected elements' reference lines (in the 3D window only).

Highlight Element Surface During Selection: Check this box to apply a highlight to the selected elements' surfaces.

- **Surface color:** Choose a color for the selected elements' surface color.
- **Transparency:** Set the transparency for shaded Hardware acceleration view here. Use the slider to set the grade of transparency/opacity for shaded Hardware acceleration view.

For more information, see [Selection Highlights](#).

Highlight Element before selection and with Magic Wand: Check this box to highlight element contour lines (i.e. detection highlight) before selecting them, and when using the Magic Wand to create an element based on an existing element.

- **Highlight color:** Choose a color for the selection highlight here.
- **Bold contours:** Check this box to display bold contours for the selection highlight.
- **Highlight appears after:** Enter a value here in seconds for the delay time before highlight appears. (Highlight appears immediately if you press Shift.)

Highlight Associated Element During Input and Parameter Transfer: Check this box if you wish to highlight related elements. During dimensioning, labeling, door/window or railing placement, or during parameter transfer operations, the contours of associated elements will be highlighted to help you identify related elements. You need not press Shift.

- **Highlight color:** Choose a color for the related element highlight contours here.
- **Bold contours:** Check this box to display bold contours for the element highlight.

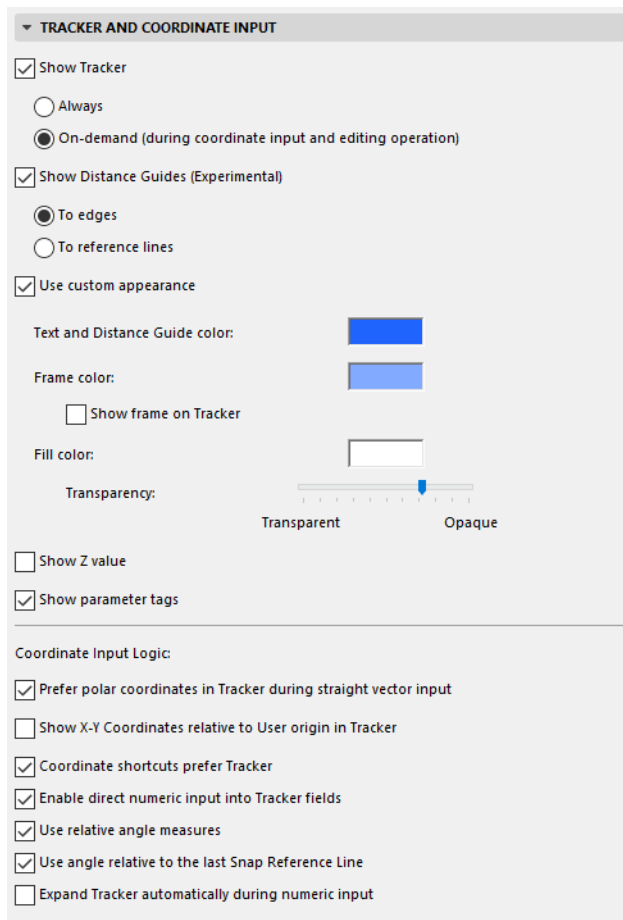
Enable Element Info Tag: Check this box to enable the element information pop-up (e.g. Info Tag).

See also [Selection Highlights](#).

- **Info Tag appears after:** Enter a value here in seconds for the delay time before element info pop-up appears.
- **Show short info only:** Check this box to display only short info in the element info pop-up.
- **Use Tracker colors:** Check this box to use the colors chosen for the Tracker in **Options > Work Environment > Tracker and Coordinate Input**.

Tracker and Coordinate Input

(Options > Work Environment > Tracker and Coordinate Input)



For general information, see [Tracker](#).

Show Tracker: Choose an option for preferred Tracker display:

- **Always:** Select this radio button to always show the Tracker on your plan, both in 2D and in 3D, as you move your cursor.
- **On-demand:** Select this radio button to show Tracker only during coordinate input and editing operations.

Show Distance Guides (Experimental)

Activate Distance Guides as an input/editing aid on the Floor Plan, to display dynamic distances from nearby elements.

See [Distance Guides - Experimental feature](#).

Use custom appearance: Check this box to set custom color settings for the Tracker and Distance Guides, using the controls below.

Note: These color settings also affect the interface when you work with Curtain Walls: the appearance of the “Edit” button and of the Curtain Wall Toolbox (visible in Edit mode).

- **Text and Distance Guide color:** Choose a color from this pop-up for the foreground color (the text) of the Tracker.
- **Frame color:** Check this box to set a separate frame color for the Tracker.

- **Fill color:** Choose a background color for the Tracker.
- **Transparency:** Use the slider to specify the transparency of the Tracker's background.
Note: If you wish, you can apply these colors to the Element Information pop-up (**Options > Work Environment > Selection and Element Information > Use Tracker colors**).

Show Z value: Check this box to display Z value (when applicable) in the Tracker.

Show parameter tags: Check this box to show parameter tags in the Tracker. (These are the texts identifying the data values displayed in the tracker, e.g. "Distance").

Coordinate input logic

This part of the panel refers to the coordinates and angles displayed in the Tracker.

Prefer polar coordinates in Tracker during straight vector input: Check this box to prefer polar coordinates (D/A, or distance, angle) over X-Y coordinates in the Tracker.

Show X-Y Coordinates relative to User origin in Tracker: Check this box if you want to display absolute X-Y coordinates in the Tracker even if you have enabled the "**Relative Coordinates in Tracker**" command from the Standard Toolbar. This way, the Tracker will display Relative polar coordinates and Absolute X-Y coordinates simultaneously.

Coordinate shortcuts prefer Tracker: Check this box if you want coordinate shortcuts to jump to the Tracker (instead of the Coordinates Palette).

Enable direct numeric input into Tracker fields: If this box is checked, numeric input during editing operations will be interpreted as coordinate entries into the Tracker. If this box is not checked, you must enter the Tracker using a coordinate shortcut or the TAB key.

Use relative angle measures: Check this box if you want the angle values in the Tracker to be measured (1) relative to the orthogonal grid (from 1 to 180 degrees), or (2) relative to the previous input segment during polygonal input.

If this box is unchecked, angle values are shown from 0 to 360 degrees.

Use angle relative to the last Snap Reference Line: If you check this box, angle values in the Tracker will be shown relative to the active Snap Reference. (You must first activate a Snap Reference Line for this to work.)

Expand Tracker automatically during numeric input: Check this box to extend the size of the Tracker automatically during numeric input.

Input Constraints and Guides Dialog Box

(Options > Work Environment > Input Constraints and Guides)

Horizontal and Vertical: Check this box to constrain cursor movement to 0°, 90°, 180° and 270° from any Edit Origin by holding down the Shift key.

Parallel to Grid: Check this box to interpret 'Horizontal and Vertical' constraint in the direction of the rotated grid's main axes.

Incremental Angle: Check this box to enable cursor constraint at the defined Fixed Angle from any Edit Origin by holding down the Shift key.

Enter an Angle value here for cursor constraint.

- **Relative to Horizontal:** Check this box to interpret 'Incremental Angle' constraint relative to the horizontal grid's main axes.
- **Relative to Grid:** Check this box to interpret 'Incremental Angle' constraint relative to the rotated grid's main axes.

Guide Line Color: Choose a color for the Guide Lines.

Snap Guide color: Choose a color for the Snap Guides.

Snap Point color: Use the color chooser to set a custom color for the Snap Point, if desired.

For more information, see [Snap Points](#).

Note: Use the Q shortcut (Force Snap Reference) on any edge or point to instantly highlight an it a Snap Guide/Snap Point, without waiting for the predefined delay.

Autohide Snap Points: This option is checked by default. Special Snap Points (if enabled) will disappear on their own after a set interval. If you disable this checkbox, the snap points will remain visible until you either place an element or change to another view or story.

Cursor Snap Range: Enter a value here in pixels for the distance at which the cursor snaps to an element.

Line Drawing: Set the line drawing method here by clicking one of the buttons.

Mac Classic-like: Select this option to use the standard click-drag drafting method.

Example: to draft a Line, click the Mouse button at the first endpoint, drag while holding it down and release at the other endpoint.

CAD-like: Select this option to use the standard CAD click-click drafting method (default).

Example: To draft a Line, click the mouse button at the first endpoint and release it, move to the other endpoint and click to complete the Line.

Choose your Mouse wheel scrolling preference:

- Zoom, plus Alt/Opt to Pan
- Pan, plus Alt/Opt to zoom

Model Rebuild Options

This dialog box (**Options > Work Environment > Model Rebuild Options**) provides controls that affect Background Updating, rebuilding the 3D and Section/Elevation/IE windows, creating reports, and dealing with error alerts.

MODEL REBUILD OPTIONS

Background Updating:

Enabled
Automatically updates inactive model views when computer is idle.

Enabled when device is plugged in
Power save mode recommended for mobile devices. Automatically suspends background updating when device is not connected to external power.

Disabled

Auto-rebuild Model View Control:

Update Auto-rebuild Model viewpoints continuously
Archicad will continuously rebuild Sections/Elevations/Interior Elevations of Auto-rebuild Model type to reflect your edits while you work in the viewpoint.

3D Window Rebuild Controls:

Auto-rebuild 3D Window when activated

3D Settings changes bring 3D Window to front

Section Window Rebuild Control:

Keep zoomed detail on Rebuild from Model

3D and Calculation Process Check Controls:

Show Progress Window

Interrupt with error messages

Write Report

Brief Report Full Report





Selected Elements to be Calculated:

Background Updating

- **Enabled:** This control (on by default) enables Archicad to take advantage of the processor's unused capacity to build up different model views in advance. Background processing also utilizes your hardware more efficiently. Because Background Updating keeps open model windows (or tabs) up to date, it enables you to switch among different views faster.
- **Enabled when device is plugged in:** Power save mode, recommended for mobile devices. Automatically suspends background updating when the device is not connected to an external power source. Background processing will be enabled again once you plug the device in.
- **Disabled:** No background updating will take place.

Feedback in Status Bar

The status bar of your Archicad window provides feedback on background processing (e.g. “Background Updating enabled”). By clicking on the status bar icon, you can access the three above settings and change them.

STATUS ICONS FOR BACKGROUND UPDATING	
	ENABLED
	ENABLED IN POWER SAVE MODE
	DISABLED
	UPDATE IN PROGRESS

Auto-Rebuild Model View Control

- Update Auto-rebuild Model viewpoints continuously

If this box is checked, Archicad will continuously rebuild Sections/Elevations/Interior Elevations of Auto-rebuild Model type to reflect your edits while you work in the viewpoint.

However, in large projects this may cause slowdowns. In this case, try unchecking this box: as a result, Auto-rebuild viewpoints will be rebuilt only when you re-open the window or use the Rebuild command from the context menu.

3D Window Rebuild Controls

- Auto-rebuild 3D Window when activated:** Checking this box means the 3D display of your plan is updated after each modification to the model, as soon as you activate the 3D Window.
 If this box is unchecked, the 3D model does not show the modifications until you choose the **View > Refresh > Rebuild** command. Switching this option off can be useful if you want to keep an already existing 3D image in the window for a longer period, and you want to avoid accidentally rebuilding it.
Note: If you edit elements in an outdated 3D view, deleted elements will be visible but cannot be selected. Moved elements can be selected, but their selection dots are displayed at the real location and you cannot edit them. Modified elements can be selected and edited, but this will trigger the rebuilding of the view.
- 3D Settings changes bring 3D Window to front:** Check this box if you want to activate the 3D window and rebuild its content every time you make a change in either 3D Projection Settings or 3D Styles.
- Keep zoomed detail on Rebuild:** Check this box to preserve the last zoom made in the 3D Window even if you choose **View > Refresh > Rebuild**. If you uncheck it, the 3D Window will recreate the 3D view at the original zoom level.

Section Window Rebuild Control

Keep zoomed detail on Rebuild from Model: If you mark this checkbox, then the zoom level of your Section/Elevation/IE Window will be retained even after you execute a **Rebuild** command.

For more information, see [Updating Sections](#).

3D and Calculation Process Check Controls

Show Progress Window: If this box is checked, the progress of 3D and list processing is indicated by a special timer. This will help you evaluate whether to cancel lengthy 3D redraws or calculations, change some

settings (for example by selecting a smaller area, setting more specific filters or less precise imaging options or reducing the 3D Window's size).

Note: The movement of the indicator is proportional not with the elapsed time, but with the amount of data processed.

Interrupt with Error Messages: If this box is checked, any error messages will appear on the screen (in the Session Report Window) before the imaging or calculation process is completed. If the box is left unchecked, error reports appear only after the process is completed.

Write Report: Use this checkbox to enable or disable the creation of a report on the 3D conversion and list calculation processes. Use the sliding switch to set the amount of detail it should contain.

See also [Session Report Window](#).

Selected Elements to be Calculated: This pop-up field gives you three options for handling a possible conflict between a listing command and items selected in the workspace.

- Select **List All** to calculate all selected elements in the list, even though they do not correspond to the filtering criteria.
- Select **Use Filters** to apply the filters defined for the list scheme to the selected elements. Elements that do not correspond to these filters will be ignored.
- Selecting **Show Alert** will display a warning about the situation and you will be able to choose between the two options above.

For more information, see [Calculation \(Legacy\)](#).

More Options

(Options > Work Environment > More Options)

MORE OPTIONS

- Create Detail/Worksheet Boundary in Detail/Worksheet Window
- Save Navigator Preview Data with Project files
- Assign New Element ID to Each New Element
- Enable Archicad Sounds
- Show New Message Alert in Teamwork
- Show Tracked Changes Alert in Teamwork
- Use large alert cues
- Do not launch new instance of Archicad when placing a view onto a Layout
- When placing model views onto Layouts, adjust each drawing frame to show only zoomed area of model view

When opening another View or Layout of a similar type:

- Open views in a new tab
- Open layouts in a new tab

Note: This option will not affect Floor Plan or 3D windows.

- Show Preview in Tab Bar
- Launch new instance of Archicad when opening a project from the recent list or by dragging onto a running Archicad

EXPERIMENTAL FEATURES

Create Detail/Worksheet Boundary in Detail/Worksheet Window

This box is checked by default, and each Detail or Worksheet Drawing window will include a line that represents the boundary you created when placing the marker. Even if such Boundaries are hidden as part of the Marker (**View > On-Screen View Options > Marker Range**), this line will be shown in the Detail/Worksheet window.

If you do not want such a boundary in the Drawing window, disable this checkbox.

Save Navigator Preview Data with Project files

If you mark this checkbox, the saved Navigator previews will be available when you reopen your project, allowing for faster display of stored previews.

Notes: The Navigator Preview stores a vectorial database for the display of every window preview. A large project with many viewpoints will generate a large amount of such preview data. To avoid saving this data as part of the file, leave this box unchecked.

Assign New Element ID to Each New Element

Using this checkbox, ID numbers will automatically increase when creating a new element, provided that a number is included in the ID field.

For details about ID Management, see the Calculation Guide (a PDF document in the Archicad > Documentation folder, or in the Archicad Help menu).

Enable Archicad Sounds

This box is unchecked by default. If you want to hear the beeping sounds associated with Archicad processing and certain Teamwork interactions (such as new message), check this box.

Show New Message Alert in Teamwork

Check or uncheck this box depending on whether you want a message alert to “float” across your screen every time you receive a message in Teamwork.

Show Tracked Changes Alert in Teamwork

Check to receive alerts of model changes for which you have subscriptions.

[See also *Change Tracking in Teamwork*.](#)

Use Large Alert Cues

In Teamwork: On-screen alerts (Messages, Project Changes) are displayed larger, so they are easier to notice.

Do not launch new instance of Archicad when placing a view onto a Layout

By default, a new instance of Archicad is launched when you place a view from an external project onto a layout in the current project. Check this box if you do not wish to launch this additional instance every time. In this case, the new instance of Archicad will be launched only when you update the placed view.

When placing model views onto Layouts, adjust each drawing frame to show only zoomed area of model view

Check this box if you want the placed drawing’s frame to be adjusted to the zoomed area of the model view by default.

Once the zoomed view is placed, you can later opt to readjust the frame to fit other parts of the model view, either manually using pet palette commands, or using the **Fit Frame to Drawing** radio button in Drawing Settings.

[See also *Drawing Settings*.](#)

When opening another View or Layout of a similar type:

Use these checkboxes to set preferences for opening views and/or layouts in either a new or an existing tab (or window, if you are not using the Tab Bar):

- Open views in a new tab (or window)
- Open layouts in a new tab (or window)

These options do not apply to Floor Plans and 3D windows, since multiple Floor Plans and 3D windows cannot be open simultaneously.

Show Preview in Tab Bar

Hovering over any tab displays a preview based on the tab’s last opened view.

Launch new instance of Archicad when opening a project from the recent list or by dragging onto a running Archicad

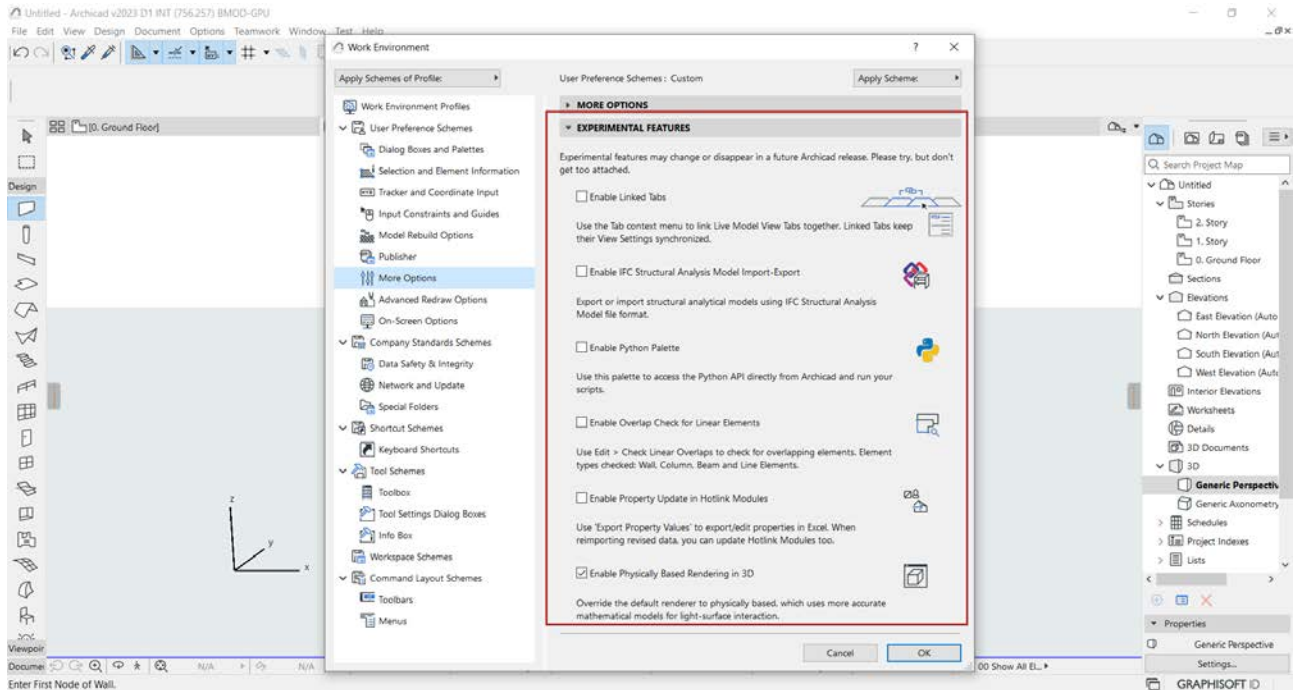
Click this option to always start a new copy of Archicad when you open a project from the “Recent list,” or drag a project file onto a running Archicad.

Experimental Features

Go to Options > Work Environment > More Options > Experimental Features.

One or more “experimental” features may be shown here that you can try out. Based on user feedback, these may or may not become part of the core Archicad functionality in future versions. Try them out!

Check the box to turn on any experimental feature.



Provide feedback

To provide feedback, try the experimental feature, then uncheck the feature checkbox in the Work Environment. You are prompted to send your comments. We appreciate your feedback!

Related Topics

[IFC Structural Analysis Model Import/Export - Experimental Feature](#)

[Check for Linear Overlaps - Experimental Feature](#)

[Update Properties in Hotlink Modules - Experimental Feature](#)

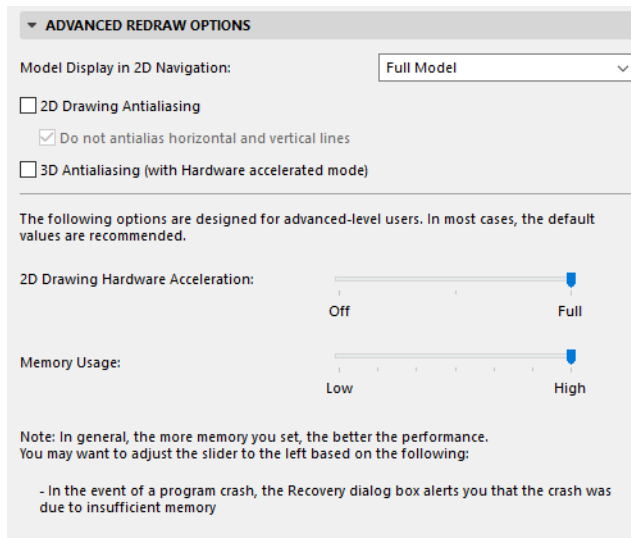
[Physically based renderer in 3D Window - Experimental feature](#)

[Distance Guides - Experimental feature](#)

Advanced Redraw Options

(Options > Work Environment > Advanced Redraw Options)

These options are designed for advanced-level users or IT managers familiar with processing and memory issues. For most users, it is sufficient to leave these settings at their default values.



Model Display in 2D Navigation

Choose from three options for displaying project elements during panning operations. (If you are experiencing fits and starts while navigating in a large, complex model, switching among these options can speed and smooth the process.)

- **Simplified Model:** The full model is shown in simplified version (for example, hatches are temporarily hidden).
- **Structural Contours Only:** Only the outlines of construction elements appear during a panning operation.
- **Full Model:** All elements are shown in full.

When you finish with the panning operation, all elements are fully displayed again.

2D Drawing Antialiasing

Turned on by default. This option makes line-type 2D Drawing elements appear much smoother.

- *Mac only:* **Do not antialias horizontal and vertical lines.** Check this box if you want to exclude horizontal and vertical lines from the antialiasing process.

3D Antialiasing (with Hardware antialiasing mode)

Check this box if you want to achieve antialiasing in 3D, if you are using the Hardware acceleration 3D engine.

2D Hardware Acceleration Level

This option is enabled and cannot be turned off (technology serving this is obsolete and will be eliminated in upcoming versions)

Memory Usage

Use this slider to adjust the level of memory usage by Archicad. Provided that you have enough memory available, and depending on your processor, increasing this value (by moving the slider to the right) may result in improved performance.

The Memory Usage slider defines ways of working in Archicad that are not strictly related to the 2D display, yet might have an effect on the speed of 2D display. By default, the slider is set to Full. In this situation, when Archicad is opened, part of the memory is reserved for indexing elements. This reserved memory enables Archicad to accomplish various tasks - most of them affecting 2D speed - faster.

If memory usage is very high (as when working with large files) then it is advisable to move the slider to the left. In this case there will be less memory reserved for the indexing function, and more memory reserved for regular Archicad functions. Note that if you are working with small files, the difference between the two ways of working is not noticeable. We recommend that you leave the slider on the right, in its default position, and only move it to the left when Archicad experiences serious memory shortage problems.

On-Screen Options

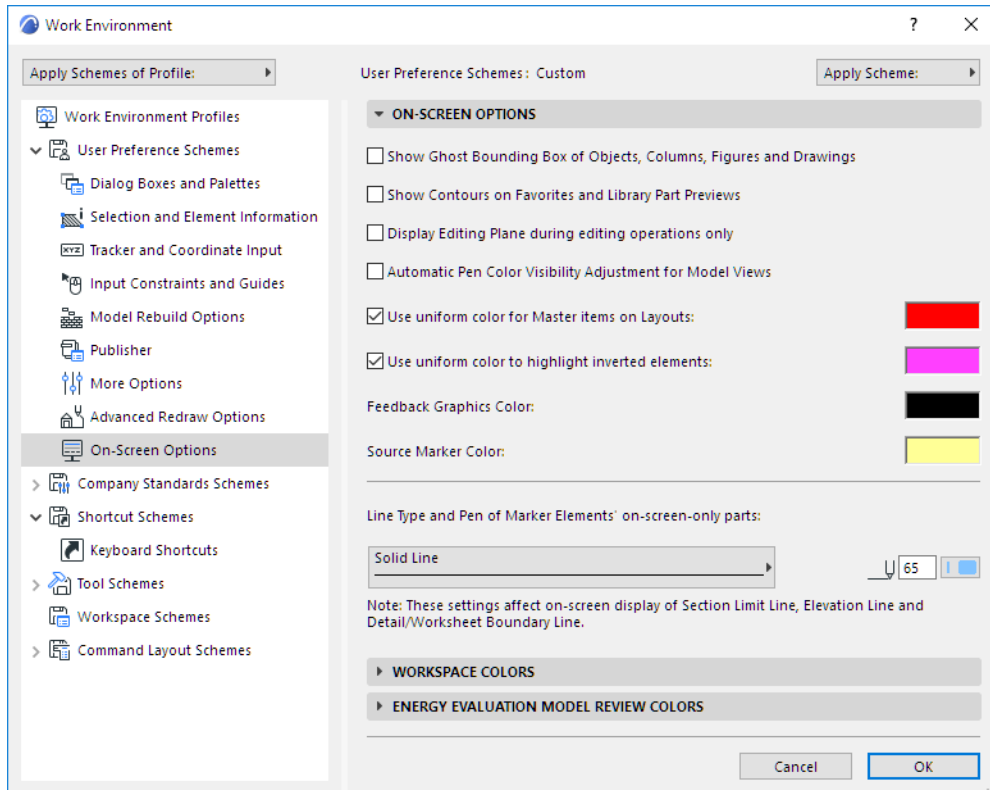
(Options > Work Environment > On-Screen Options)

This dialog box has three panels:

On-Screen Options

Workspace Colors

Energy Evaluation Model Review Colors



On-Screen Options

Show Ghost Bounding Box of Objects, Columns, Figures and Drawings

If this box is checked, the bounding box of Door, Window, Object, Lamp, Column, Figure and Drawing elements that you wish to place will follow the cursor.

Show Contours on Favorites and Library Part Preview

Check this option if you prefer to see contours on 3D Preview for Favorites and Library Parts.

Display Editing Plane During Editing Operations Only

If you check this box, the 3D Window Editing Plane will appear only during the time you are actively editing in the 3D window (provided that the “Editing Plane Display” toggle at **View > Editing Plane Display** is turned on.) When you are done with the operation, the Editing Plane disappears.

Automatic Pen Color Visibility Adjustment for Model Views

When this box is checked, then black pens will be shown as white on your Archicad screen if the luminance value of a particular background color falls below a threshold value - that is, if your background is sufficiently dark.

Use uniform color for Master items on Layouts

Check this to display all Master items using a single color on Layouts. To choose the color, double-click the color field.

If you leave this box unchecked, the element's own colors will be applied.

Use uniform color to highlight inverted elements

Check this box to display inverted or invalid elements using a single color.

To choose a color, double-click the color field. This highlight is an on-screen attribute only; it will not be seen on printed/published output.

Inverted or invalid elements may include

- Inverted construction elements (those whose top elevation falls below their bottom elevation, as a result of changing story levels affecting top linked elements)
- Stairs whose top-linked story height has changed, yet the stair parameters are defined in a way that the stair cannot be adjusted to the new story height automatically
- Some construction elements (e.g. Beams, Columns) in projects using the Legacy Intersection Preferences. If "Legacy Intersection and Surface Methods" is checked at Project Preferences, certain geometries (based on new tool functions in later versions of Archicad) are not supported.

[See *Legacy Preferences*.](#)

Feedback Graphics color

Use the color chooser to set a custom color for bounding boxes and rubber band effects shown during editing operations.

Source Marker Color

Use the color chooser to set a custom color to highlight source markers. (To turn Source Marker Highlight on or off, use **View > On-Screen View Options**.)

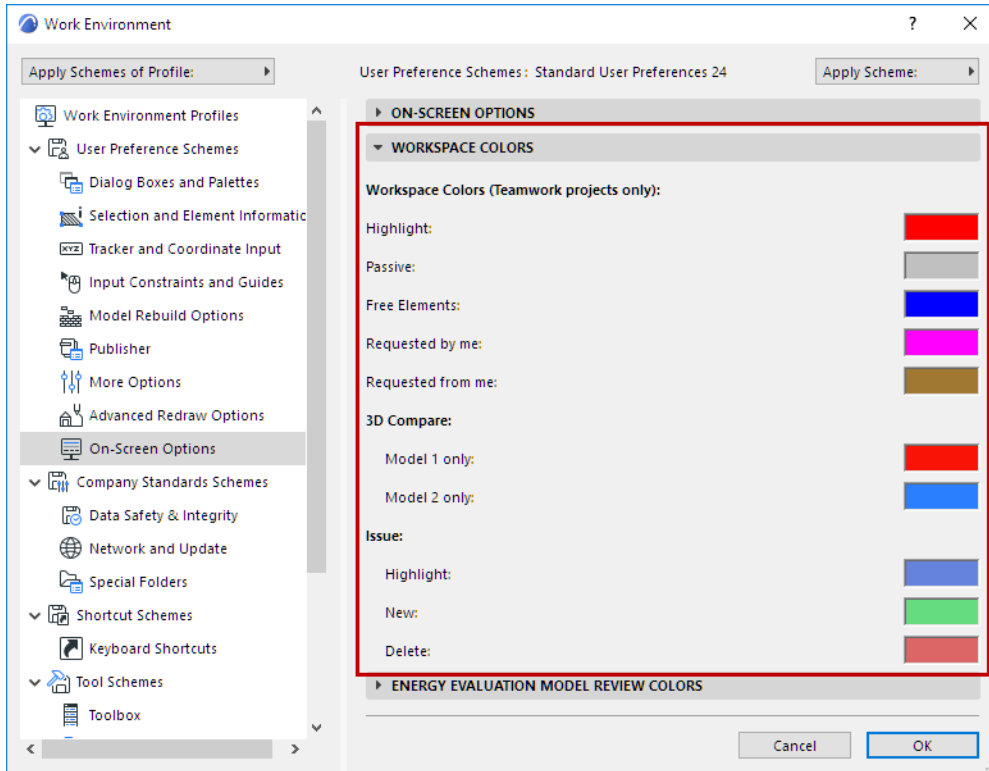
For more information, see [Source Marker Highlight](#).

Line Type and Pen of Marker Elements' on-screen-only parts

Use these controls to set a line type and color for the on-screen parts of all marker elements. These are:

Section Limit Line, Elevation Line, Section and Elevation Distant Line, Interior Elevation Line and Limit Line, and Detail/Worksheet Boundary Line.

Workspace Colors



Teamwork

Choose preferred color for highlighting elements depending on their ownership, in Teamwork projects only.

See [Colored Workspaces](#).

3D Compare

Set colors to display model differences in the 3D Compare window.

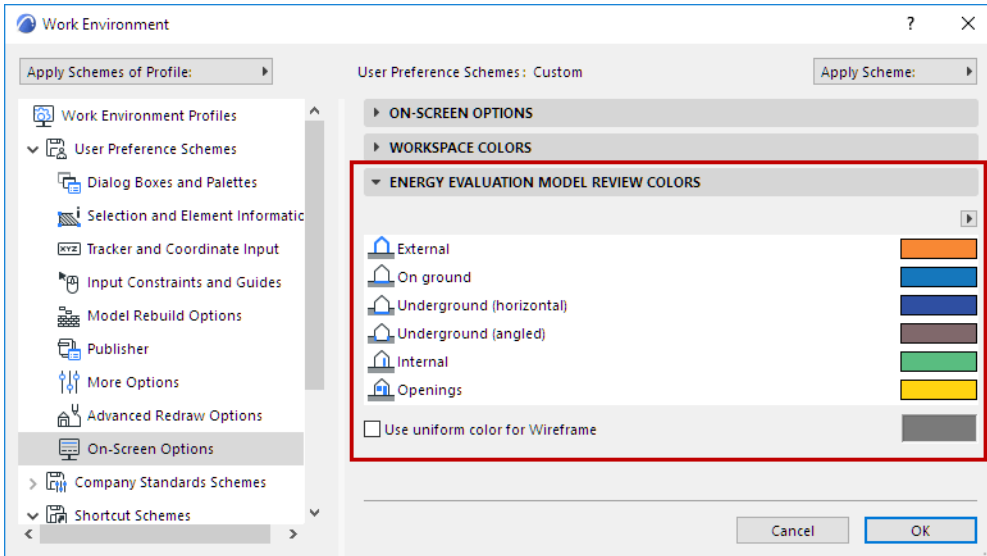
See [Workspace Colors for Model Compare](#).

Issue

Set colors to display Issue elements in the model depending on their category.

See [Issue Handling](#).

Energy Evaluation Model Review Colors



Choose your preferred color for highlighting model structures according to their function for the Energy Evaluation process. (Building structures and openings are shown in these colors when you click **Show in 3D** in the Model Review palette of Energy Evaluation.)

[See *Customize Energy Model Review Colors*.](#)

Data Safety & Integrity

(Options > Work Environment > Data Safety & Integrity)

Use the features of the **Data Safety & Integrity** dialog box (**Options > Work Environment > Data Safety & Integrity**) to minimize the risk of data loss and file corruption. This is especially important if your power lines suffer from voltage spikes or failures, or if your computer is prone to software conflicts.

DATA SAFETY

Saving to Recovery File

Ultra Safe (save each step)

Safe (save every 5 minutes)

Minimum (save every 20 minutes)

Location:

Use Default Location (recommended)

Choose Location

Note: Changes take effect only after restarting Archicad.

Undo Limit: steps

Make Backup Copy

Teamwork 'Send' reminder:

Show after every steps or every minutes

DATA INTEGRITY

Hotlink Update:

Release All when closing Teamwork Project with Send Changes

Automatically Check Status of All Hotlinks in Hotlink Module Manager

Automatically reserve available elements when editing; requires BIMcloud User License

TEAMWORK MESSAGING

Teamwork Messaging:

Limit Number of Completed Messages to:

Oldest Completed Messages over this limit will be Automatically Deleted.

Saving to Recovery File

The **Saving to Recovery File** function controls a number of options protecting you against losing work. If you are forced to restart your computer or the program because of a crash, then the next time you launch Archicad, a dialog box gives you the option of resuming the interrupted work.

When **Saving to Recovery File** is triggered, the **Bulldozer** cursor may appear momentarily on the screen while the program saves any changes made to the Project since the last **Saving to Recovery File** (or manual **Save**). Even with large Projects, **Saving to Recovery File** should complete its cycle within a few seconds.

Saving to Recovery File Options:

- Ultra Safe: The default and recommended option. Data is saved to a recovery file after each step.
- Safe: Saves data every 5 minutes.

- Minimum Saves data every 20 minutes.

There are three limitations to Saving to Recovery File functions. The following items must be saved manually:

- Opened Library Parts
- Completed PhotoRendering Images
- All types of Calculation Lists

If Archicad is restarted after a crash, it will detect the Recovery File data and prompt you to open it. The Recovery File is also available for opening in the Start Archicad dialog.

Recovery Folder

Stores the saved documents at the specified location:

- **Use Default Location**, or
- **Choose Location** and then **Browse** among folders on your computer or across a network.

Changes to the settings of the Recovery Folder will take effect after the program is restarted.

Undo Limit

Set the number of undoable steps.

See also [Undo/Redo](#).

Make Backup Copy

Mark this checkbox to create a backup copy of the previously saved version of your work, by appending the “.bpn” extension to its name.

For creating backups of Shared Projects, see also [Teamwork](#).

Teamwork ‘Send’ Reminder

Check this box to have Archicad automatically display an on-screen reminder to Send your changes while you work in Teamwork.

Enter how often the notification should appear:

- after a certain number of actions in the project, or
- after a certain number of minutes

(whichever comes first).

See [Automatic Reminder to Send Changes](#).

Data Integrity

Hotlink Update

Controls whether Archicad should check the status of Hotlink Modules whenever you begin document output (e.g. Publish).

- **Ignore Hotlinks:** Choose this option to ignore any modification made on hotlinked files. This is the fastest opening option for a file containing Modules.
- **Check Hotlinks and ask for update confirmation:** When outputting a project, Archicad looks for all hotlinked files and checks their modification date. An alert will appear asking you to confirm whether you want to update modifications or not. This is the default setting in the program.
- **Check and update Hotlinks automatically:** When outputting a project, Archicad updates all Modules that are hotlinked to modified files.

Should there be any problem in the course of the automatic updating process, choose the first option (Ignore Hotlinks), open your file again, and then update the hotlinks manually through the **Hotlink Module Manager** dialog box.

Note: The option you choose here will be preserved in your Preferences file, but will not be saved with the Project.

Release All (Teamwork)

The Release All with Send Changes preference affects whether Archicad will automatically release all your reservations whenever you Close a Teamwork project using the Send option.

[See *Keep or Release Your Reservations with Send & Save*.](#)

Automatically Check Status of All Hotlinks in Hotlink Module Manager

With this default setting enabled, Archicad will check the status of Hotlinks each time you open Hotlink Module Manager. In projects with many hotlinks, the auto-check process might slow things down. In this case, uncheck the box.

Automatically reserve available elements when editing

Note: Not available in BIMcloud Basic.

This Teamwork feature is enabled by default. Whenever you work on an element in Teamwork, it is reserved by you automatically, whenever possible.

If Auto-reserve should slow down your work, you will be prompted to uncheck this feature. You can then Reserve elements as needed, by clicking the Reserve button in Teamwork Palette or from context menus.

Teamwork Messaging

Use this control to manage the number of stored Teamwork messages. Check the **Limit Number of Completed Messages to** box and enter the maximum number of messages you wish to store. Once you reach that limit, your oldest messages will be deleted.

Network And Update

Use this screen (**Options > Work Environment > Network And Update**) to define options related to

HTML Output Options

Check for Updates and/or Update Previews

Teamwork network options

Network volumes (Mac only)

NETWORK AND UPDATE

HTML Output Options

Simple

With Styles

Default Style Sheet:

ListStyle.css

Style Options: External

Check for Updates

Check for Updates:

Updates Only Weekly

Archicad and Library

Archicad only

Library only

Teamwork

Always compress network traffic

Compressing data in a LAN environment might reduce performance. Use this option when using low bandwidth connections. (e.g. Wireless or VPN connection)

Use a BIMcloud Delta Cache to optimize bandwidth usage; requires BIMcloud User License

Address:

HTML Output Options

Choose a style to use when saving data in HTML format, e.g., from a list-type Window in text format.

Simple: Choose this to display the HTML file without styles.

With Styles: Choose this to display the HTML file using the Style Sheet formatting below. (Click Browse to find and use a different style sheet.)

Style Options: Choose External if you want the resulting HTML file to include a link to the chosen Style Sheet. Choose Internal if you want to include the Style Sheet itself as part of the HTML file.

Check for Updates and/or Update Previews

Note: You initially set these update preferences when you installed Archicad. If you want to change update preferences, use the following controls.

Check the box to run an automatic check for Archicad and/or Library update (Hotfix). The Action Center (**File > Info > Action Center**) then alerts you to any available updates.

- Click **Check Now** to check for updates immediately.
- Set the components (both Archicad and Library, or either one) for which to Check for Updates.

- Use the pop-up to set how often you want to run Check for Updates (**Daily**, **Weekly**, or **Monthly**.)
- Set the type of updates you want to get notified about: **Updates only**, or **Update Previews** also.

What is an Update Preview?

A few weeks before a regular Archicad update, Graphisoft releases a preview version. It is a fully functional update, tested by Graphisoft, that works just like the regular update. Users can install and test the Update Preview, and can report any issues or bugs, so we can fix them before the regular update is released. Please use it at your own risk.

Notes on “Check for Updates” function:

- Requires internet connection
- Does not download anything to your computer automatically. It only alerts you (in Action Center) that an update is available.
- Provides Graphisoft with automatic information about your Archicad version. This information is limited to data needed for update; it includes no personal information.

Teamwork network options

Always compress network traffic

This is checked by default. However, compressing data in a LAN environment might reduce performance. Use this option when using low bandwidth connections. (e.g. Wireless or VPN connection).

Use BIMcloud Delta Cache to optimize bandwidth usage

Enter the address of the BIMcloud Delta Cache of your office. Archicad will download data from a BIMcloud using this BIMcloud Delta Cache.

Address format: <http://proxy.address:port>

Note: BIMcloud Delta Cache is used only for projects located on a BIMcloud. Not available in BIMcloud Basic.

Network volumes (Mac only)

Prevent macOS from automatically mounting server volumes (Mac Only)

Check this if you have a slower network connection.

Handle AFP Connections as SMB Connections (Mac Only)

Enable this checkbox to ensure that Archicad uses the SMB file server protocol (not AFP) when locating external files (e.g. Drawings). Both protocols can still be used, but this checkbox ensures the use of SMB. (MacOS 10.14 supports the APFS file system, which supports the SMB protocol but not AFP).

Special Folders

On this screen (**Options > Work Environment > Special Folders**), you can relocate the Temporary, Cache and Template Folders to a hard disk volume that is different from the default location.

Note: If you choose a custom location which is not available, Archicad will use the displayed system default location.

When using remote network volumes, Archicad's operations may be affected by the network performance capabilities.

Note: If your startup volume does not have the recommended amount of free space when you launch Archicad, you will have the opportunity (through an alert box) to select another volume during the startup process.

Changes to the folder location settings will take effect after the program is restarted.

▼ SPECIAL FOLDERS

Temporary Folder

Location:

Use Recommended Location

Choose Location

Note: Changes take effect only after restarting Archicad.

Cache Folder

Location:

Use Recommended Location (System Default)

Choose Location

Note: Changes take effect only after restarting Archicad.

Template Folder

Location:

Use Recommended Location (System Default)

Choose Location

The **Temporary Folder** stores data required for the quick operation of Archicad.

The **Cache Folder** stores 3D projections, PhotoRendered images and other project data. The cache data is used when the program generates the image files automatically in the background and stores them in the chosen **Cache Folder**. Archicad uses the Cache Folder for quick display and updating of images on the screen.

- On Mac, very large hard disks are recommended for advanced imaging, because the **Cache Folder** can expand greatly when rendering a very complex Project with shadow casting.
- On Windows, Archicad keeps this data in the virtual memory. Set the limitation of the page file to the highest possible value when rendering a very complex Project with shadow casting.

The **Template Folder** contains template (.tpl) project files shipped with Archicad. Template files do not contain any virtual building elements, but certain attributes and options have been pre-set.

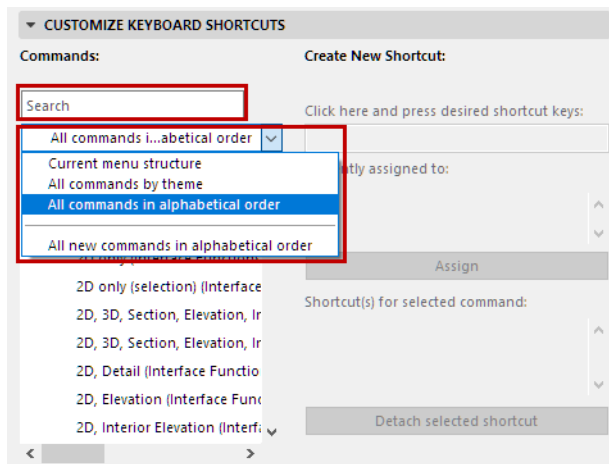
Shortcut Customization Dialog Box

Open **Options > Work Environment > Keyboard Shortcuts**.

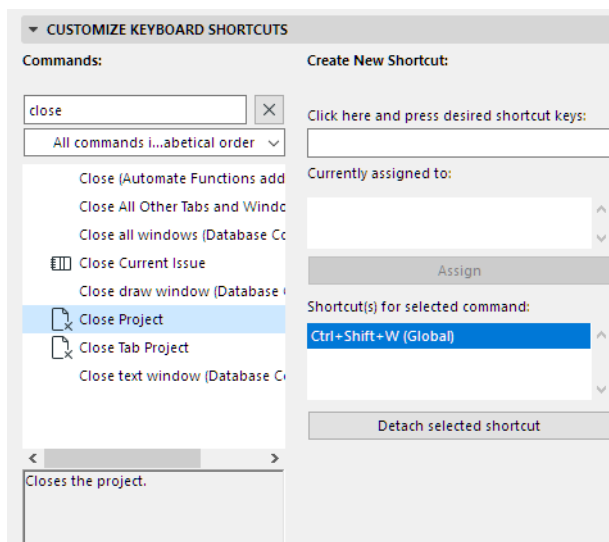
From the list window, locate any function or command:

- by typing its name in the Search field
- or click a list format from the drop-down list.

[Watch the Video](#)



Click on any command or function to view its brief description in the field at the bottom of the list.

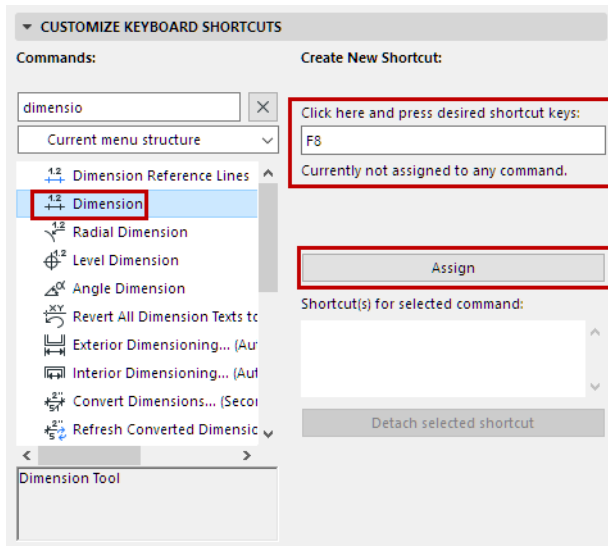


If the selected function/command already has a shortcut assigned to it, it is displayed in the **Shortcut(s) for selected command** window on the right.

Assign Shortcut

1. If the command already has a shortcut: Click **Detach selected shortcut** to remove the existing shortcut.
 - If you don't detach it, you can still assign another shortcut; this way the function/command will have multiple shortcuts.

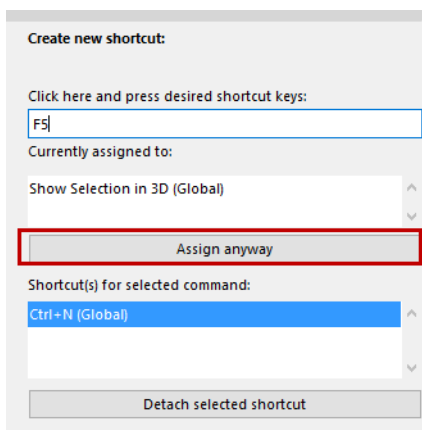
2. Now type the keys that will constitute the new shortcut. (Use the field where it says “Click here and press desired shortcut keys.”)
 - Use any single letter or number character, or a function key (e.g. F2) with or without modifier keys (i.e., Ctrl, Shift, and/or Alt).



3. Click **Assign**.
4. Click **OK** to close the dialog box and apply the selected shortcut scheme according to its current settings.

Shortcut Collision

If the key combination you entered has already been assigned to another function or command, this will be noted in the field: **Currently assigned to.** - In the following image, we typed F5 as a new shortcut, but this key has already been assigned.



In this case, you can try typing a different shortcut.

Note: No “collision” is detected if the shortcut has been assigned to commands from two different window types.

[See *Window Types \(Text or Non-Text\)*.](#)

To assign the shortcut despite the collision, click **Assign anyway**. This action assigns the shortcut and *detaches* the shortcut from the function/command to which it was previously assigned.

Window Types (Text or Non-Text)

The set of commands available in Archicad varies slightly, depending on whether the active window is Text-type (Schedules, Lists, Report, Project Notes or GDL script window) or Non-Text type (all other windows).

Use these two radio buttons in the Keyboard Shortcuts dialog box to choose the window type in which the shortcut will work:

- **Only in the currently active window type:** The shortcut will work only in the type of window that is currently active (i.e. the one you were working in at the time you opened the Work Environment dialog box). If you choose this option, you can assign the same shortcut to two different commands as long as the two commands are from different window types.

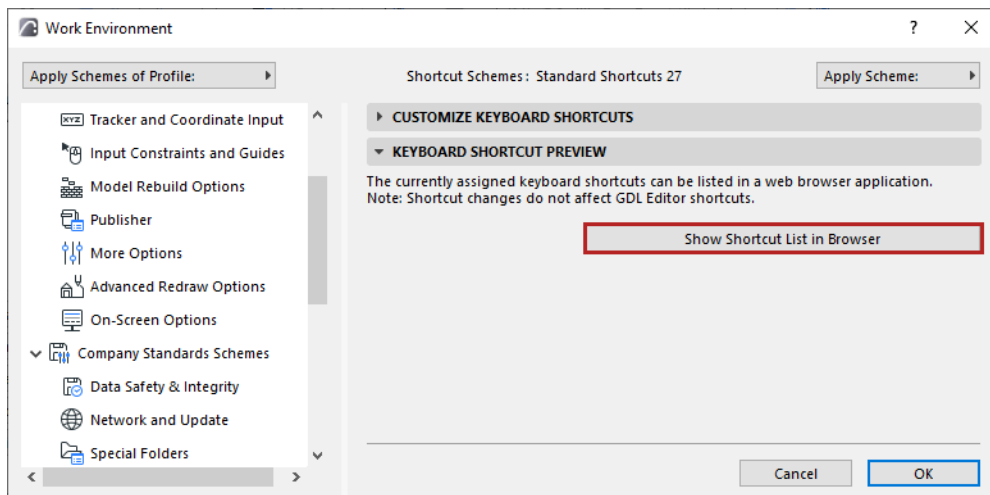
Note: If you have defined a shortcut to apply only in the current window type, you can henceforth edit that shortcut only by opening the Work Environment dialog box from the window-type of that shortcut.

- **In All window types:** The shortcut will work in all windows of Archicad.

Click **OK** to close the dialog box and apply the selected shortcut scheme according to its current settings.

Keyboard Shortcut List

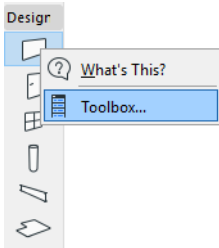
To see and/or print a list of currently assigned shortcuts for the chosen scheme, click **Show Shortcut List in Browser**.



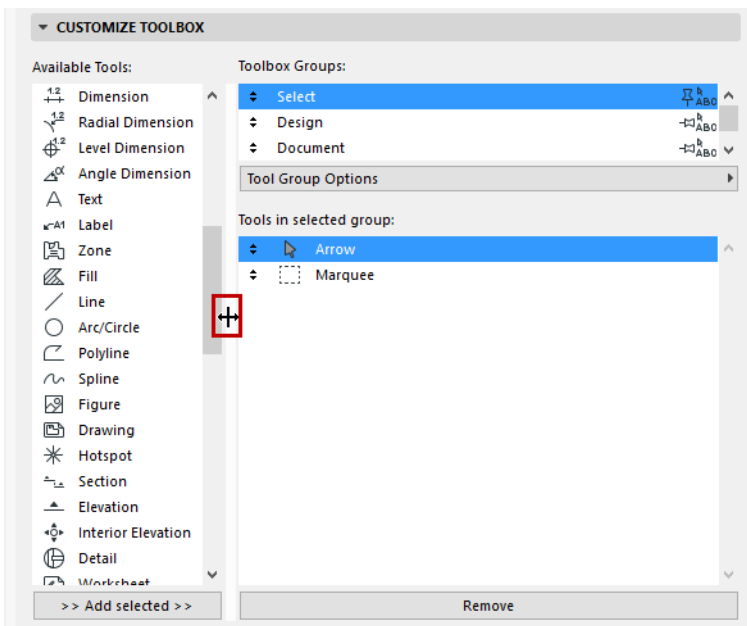
Toolbox Customization Dialog Box

Go to **Options > Work Environment > Toolbox**.

Another way to access this dialog box is to open the Toolbox context menu by right-clicking anywhere in the Toolbox, then click the icon representing the Toolbox customization page.



Use the splitter bar as needed to change the width of the fields in this dialog box.



The **Available Tools** window lists all the Archicad tools.

The **Toolbox Groups** window lists the existing tool groups.

- To change the order of the tool groups, click and drag on the up/down arrows at the left of each group.
- By clicking the pushpin icon, you define whether the selected toolbox group will remain permanently open (vertical pushpin), or open and close as space permits (horizontal pushpin).
- By toggling between the arrow/ABC icons, you define whether the selected toolbox group will be displayed by its icon only (arrow icon), or by both its name and icon (ABC icon).

The **Tools in Selected Group** window lists the tools that are currently part of the group you have selected.

Add a tool to the selected tool group by selecting it from the Available Tools list, then click the **Add Selected** button at the bottom, or simply drag it to the right. It is possible to place the same tool into more than one Tool Group.

Remove a tool from the selected tool group using the **Remove** button.

How to Open and Close Tool Groups Within the Toolbox

By default, the Toolbox shows all tools in all Tool Groups simultaneously. If the Toolbox becomes too unwieldy, use the open/close arrows at the top of each Tool Group to temporarily open and close these groups as needed.

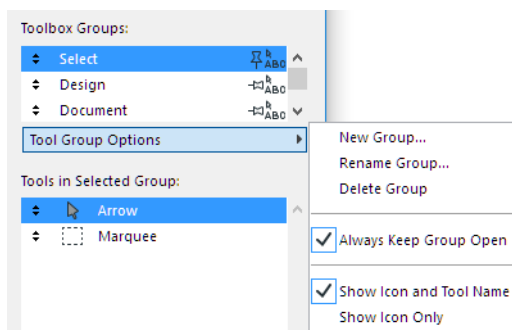
If the open/close arrows are not displayed, disable the “Always open” control in Tool Group Options.

[See *Tool Group Options*, below.](#)

Tool Group Options

- **New Group:** Use this command to create a new tool group.
- **Rename or Delete Group:** These commands are applied to the selected tool group.
- **Always Keep Group Open:** This option will keep the selected Tool Group open on your screen, and no open/close arrow is available on its panel.

To be able to open or close each Tool Group in the Toolbox separately: disable the “Always Keep Tool Groups Open” option.



- **Show Icon and Tool Name, or Show Icon Only:** This option determines how the tools in the selected Tool Group are displayed.

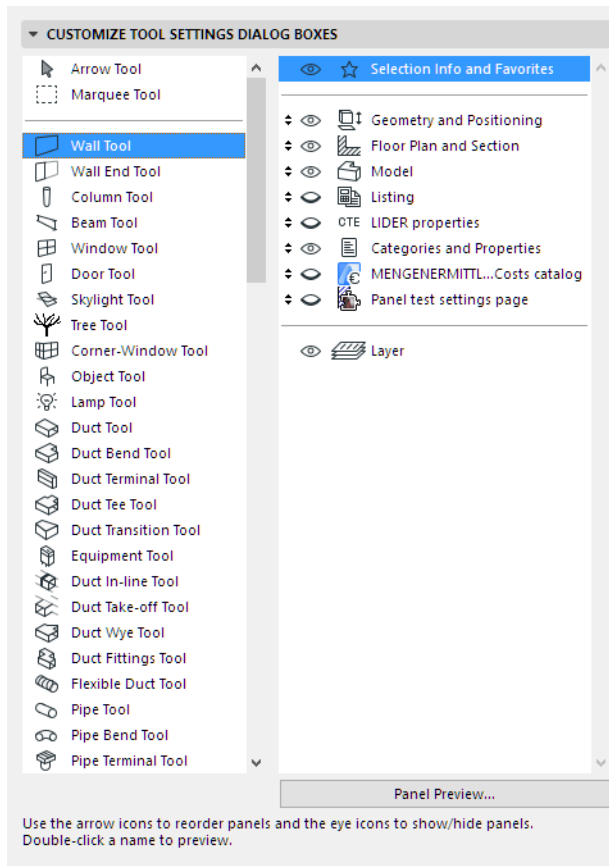
Changes to the Toolbox are applied when you press **OK** and close the dialog box. The current open/close status of your toolbox panels will be saved as part of the Tool Scheme, together with all the other settings customized in this dialog box.

Note: It is possible that the Toolbox size, as saved in your Workspace Scheme, is not big enough to accommodate all the toolbox panels you saved as open. In this case, one or more panels of your Toolbox will be displayed as closed.

Tool Settings Dialog Box Customization

Go to **Options > Work Environment > Tool Settings Dialog Boxes**.

Use the splitter bar as needed to change the width of the fields in this dialog box.



The left window contains a list of all Archicad tools. The right window lists the panels that appear on the selected tool's Settings Dialog Box.

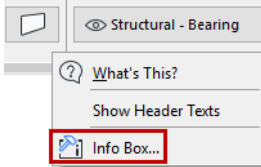
- To see what the selected panel looks like, double-click its name or click the **Panel Preview** button.
- To change the order in which the Tool Settings panels appear, use the arrow icons.
- To show or hide Tool Settings panels, click to open or close the relevant “eye” icon. (Multi-select to set show/hide preferences more quickly.)

When you are satisfied, click **OK** to apply the changes and close the dialog box.

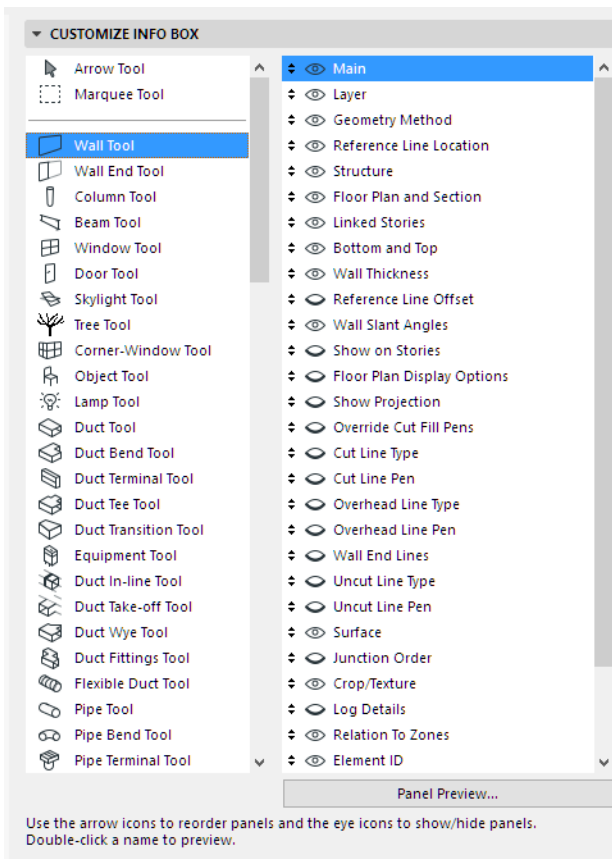
Info Box Customization Dialog Box

Go to **Options > Work Environment > Info Box** to open this dialog box.

Or open any Info Box's context menu, then click on Info Box:



Use the splitter bar as needed to change the width of the fields in this dialog box.



The left window contains a list of all Archicad tools. The right window lists the panels that appear on the selected tool's Info Box.

- To see what the selected panel looks like, double-click its name or click the **Panel Preview** button.
- To change the order in which the Info Box panels appear, use the arrow icons.
- To show or hide Info Box panels, click to open or close the relevant “eye” icon. (Multi-select to set show/hide preferences more quickly.)

Note that you can use a scroll-mouse to scroll through the contents of your Info Box.

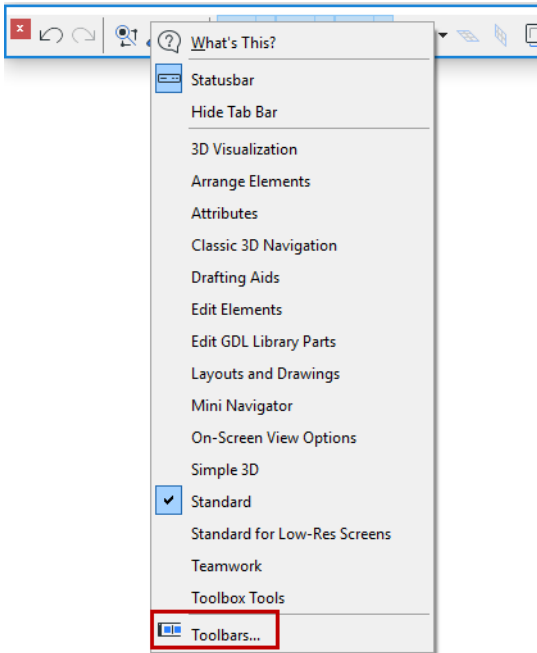
When you are satisfied, click **OK** to apply the changes and close the dialog box. To customize the location of the header on your Info Box, right-click anywhere in the Info Box to access its context menu commands.

You can store your customized Info Box settings as part of a Tool Scheme in your Work Environment.

Toolbar Customization Dialog Box

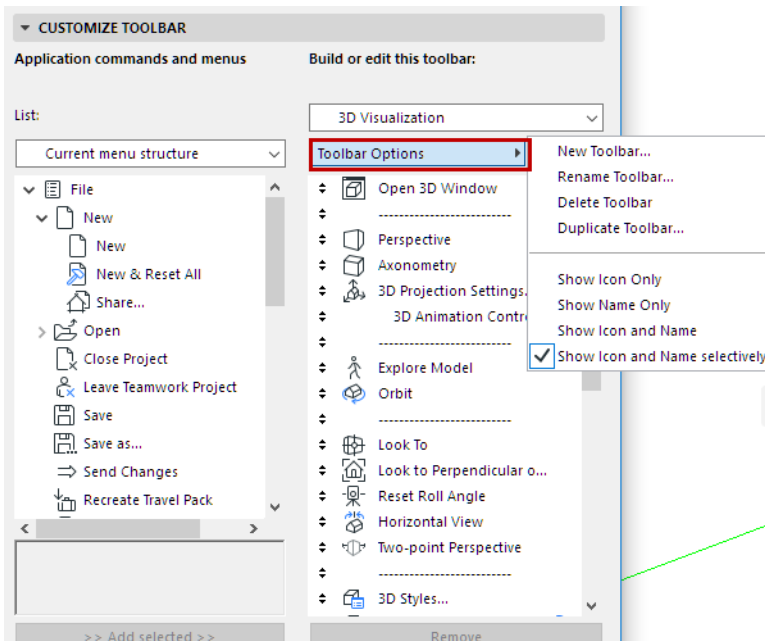
Open this dialog box at **Options > Work Environment > Toolbars**

Or open any Toolbar's context menu, then click on **Toolbars**.



Create New Toolbar

To create a new toolbar, use the **New Toolbar** command from the **Toolbar Options** pop-up menu.



Display Command Icon and/or Name

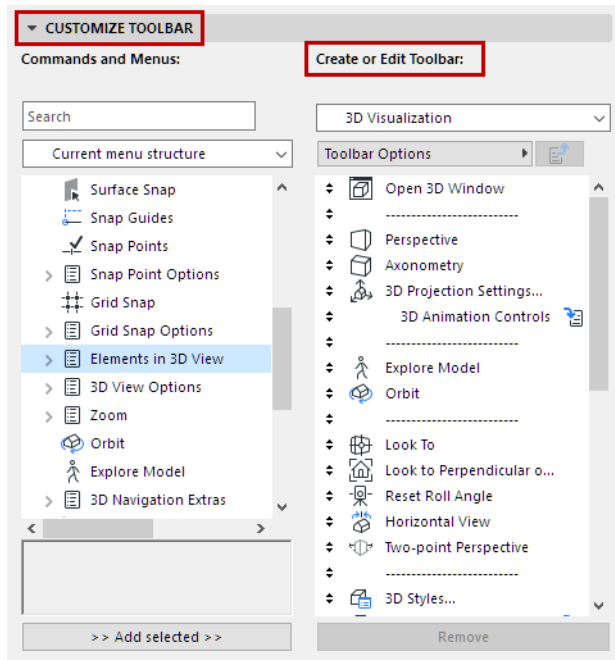
From **Toolbar Options**, set a view mode for the selected toolbar: for every command in the toolbar, you can show its Icon, or its Name, or both Icon and Name. You can also choose “Icon and Name selectively”.

Note: Icon and Name Selectively means that the Icon will be shown for every command, but the name will be shown only selectively, depending on what you set in the Further Options panel.

See [Further Options Panel](#), below.

Customize Toolbar

Add, remove and rearrange the menus and commands of any toolbar.



The interface is identical to the Menu Customization dialog.

See [Customize Menu Panel](#).

Add Menu to Toolbar

You can add not only commands, but also menus to a toolbar.

The menu can be an existing one, or you can create a customized menu (at [Customize Menu Panel](#)), then add it to the toolbar.

To set the behavior of a menu in a toolbar: See [Sticky pop-up menu](#).

Further Options Panel

The settings in this panel apply to selected commands or menus in the toolbar.

Show command name in case of “Selective” option:

This checkbox is relevant only if you have chosen “Icon & Name Selectively” from Toolbar Options. In this case, select a command in the list for which you want to show both Icon and Name, then mark this checkbox. Any command for which you do not check this box will be displayed with its Icon only.

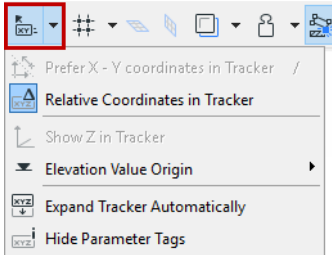
Sticky pop-up menu

This option is available if you have added a menu to the toolbar.

If you want the menu contents to appear as a pop-up of the previous command in the toolbar: check the Sticky pop-up menu box. The effect: the commands of the menu will appear only inside the pop-up of the previous Toolbar command.

If you do not check the box: the menu appears on the toolbar under its own name, with the commands inside the pop-up.

For example, the **Tracker** command on the Standard toolbar contains an associated (sticky) pop-up menu, represented by a black pop-up arrow, which in turn accesses **Tracker-related** commands.



Menu Customization Dialog Box

Use the **Options > Work Environment > Menus** dialog box to customize the contents and arrangement of all of the menus that exist in Archicad. Any command or menu can be placed into or removed from any menu; the order of the commands within any menu is entirely customizable.

Exception: Context menus are not customizable.

Use the splitter bar as needed to change the width of the fields in this dialog box.

Customize Menu Panel

Commands and Menus

The left side of the dialog box lists all of Archicad's commands and menus.

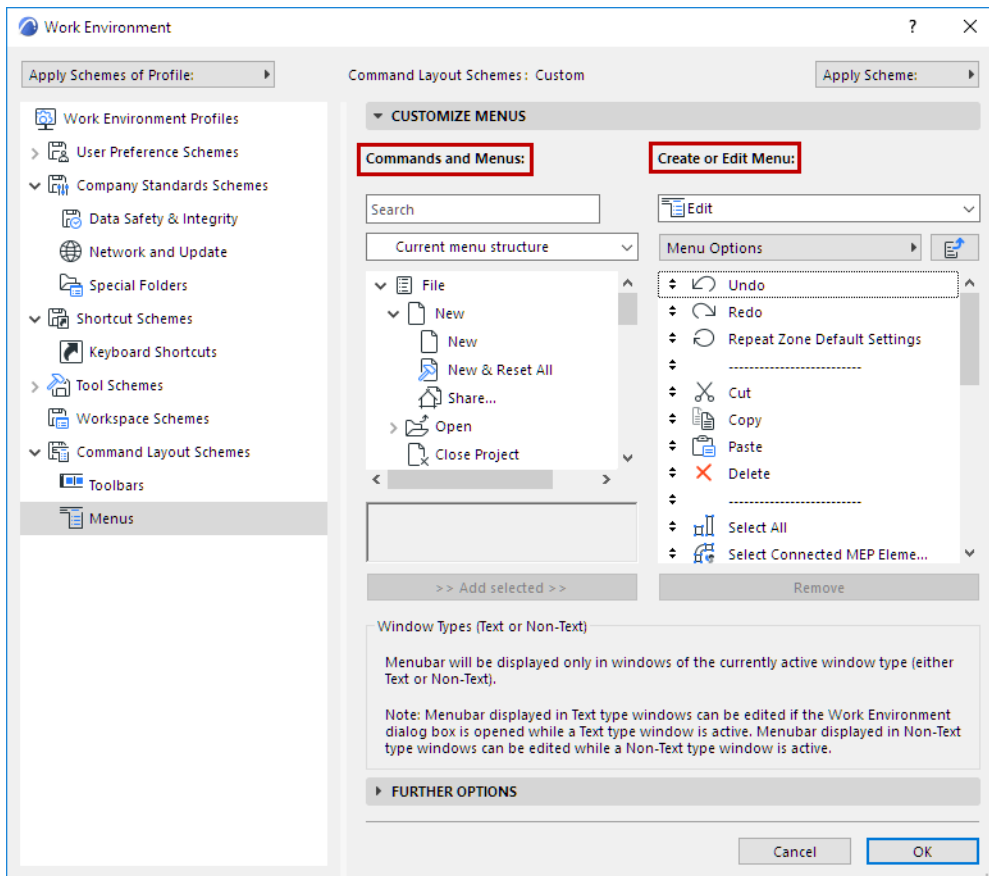
From the list window, locate any function or command:

- by typing its name in the Search field
- or click a list format from the drop-down list.

[Watch the Video](#)

Notes:

- You can display this list in one of five formats: Current menu structure; All commands by theme; All commands in alphabetical order; All menus in alphabetical order; All New commands in alphabetical order. The last item in the list is a Separator bar.
- If you are in one of the two **All commands** formats, you can access them (i.e. click and drag them to the right) only if you have chosen a particular menu (as opposed to the menubar) in the "Build or edit the menu" list on the right.

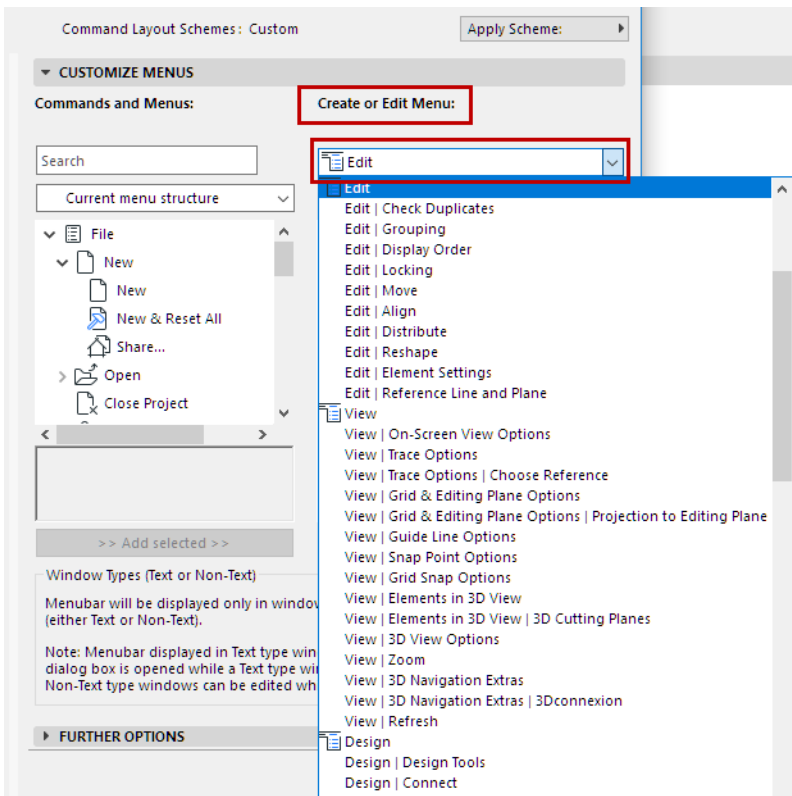


Create or Edit Menu

Choose a menu to build or edit from this pop-up list.

Note: The menu you are editing will be displayed only for windows of the currently active window type (either Text or Non-Text).

For more information, see [Window Types \(Text or Non-Text\)](#).



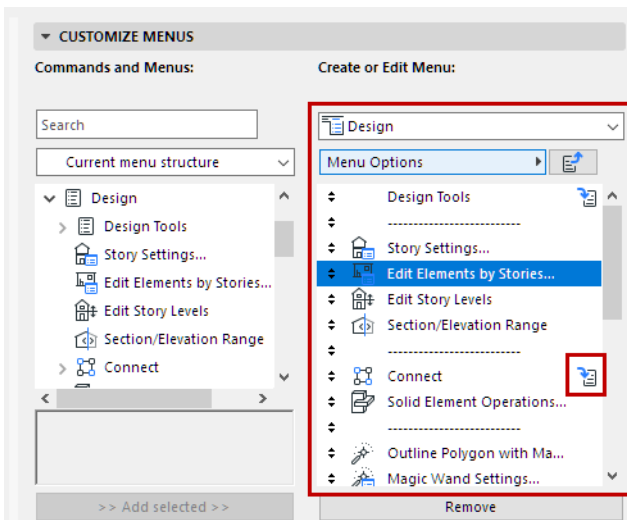
- The pop-up provides a list of menus only (no commands), together with the menu path
- A menu appears in this list as many times as it is used in the interface
- Menus not yet inserted in your Work Environment are grouped at the bottom

Menu Content List

The right-hand list window displays the content of the selected menu, including

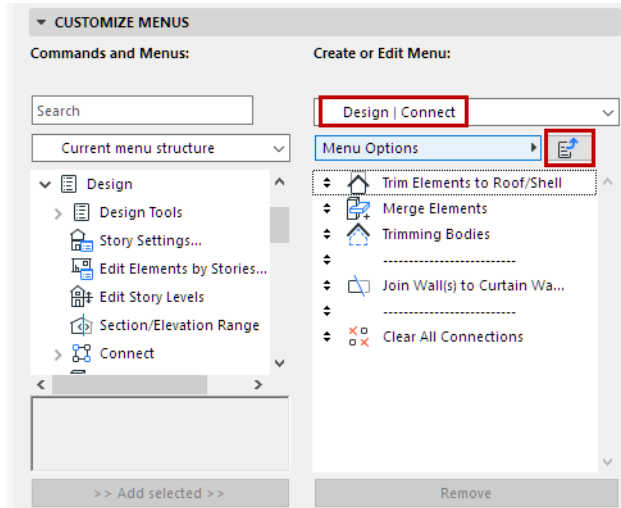
- Commands (dragged in from the “All commands” lists at left)
- Menus (dragged in from the “All menus” list at left).

A menu within a menu is indicated by an icon.



- Click to enter the sub-menu

- Click “Up one level” icon to move up a level in the hierarchy

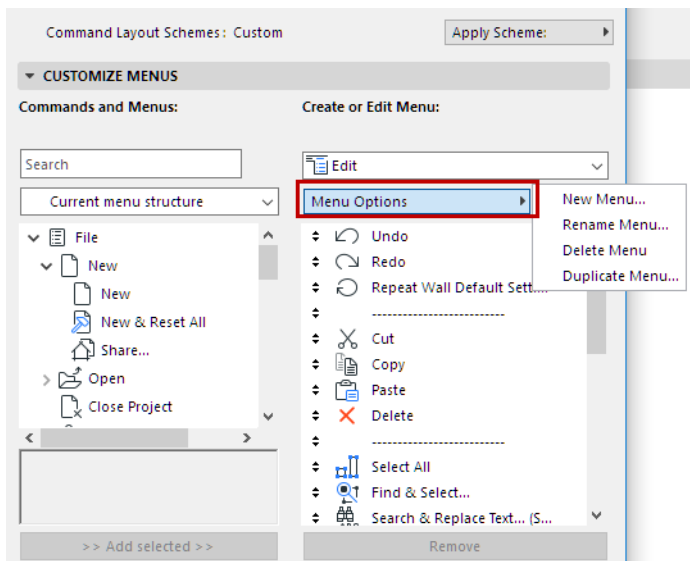


Add/Remove/Change Menu Commands

- Use the **Add selected** and **Remove** buttons on commands or menus
- Rearrange the menus in the main Archicad menubar by dragging the up-down arrows
- Change order of commands/menus by dragging the up-down arrows
- Use separator bars to group items
- To insert a new separator bar: drag it from its location at the bottom of the left-hand list

Menu Options

For the selected menu:



New menu creates a submenu within the selected menu.

Further Options Panel

Alternating menu items

This checkbox is available after you have selected a menu or command in the right-hand column.

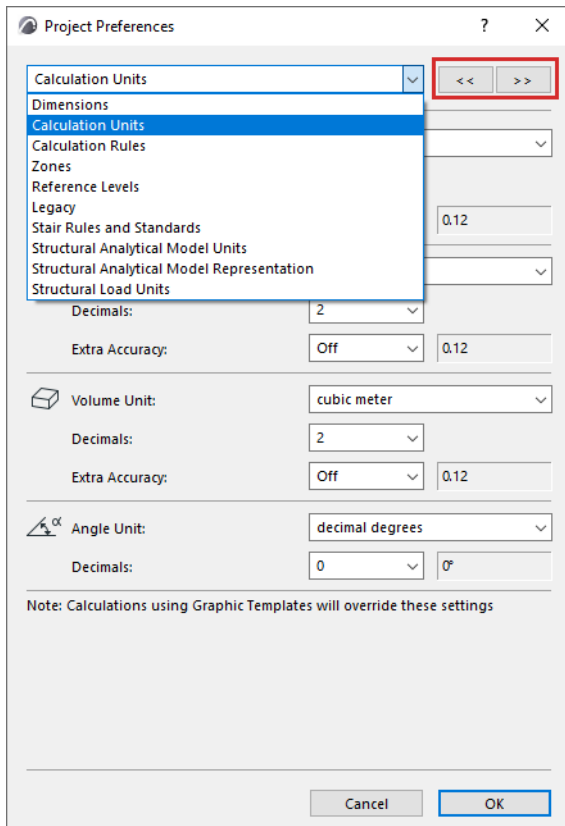
Checking this box means that the selected item (for example, **New & Reset**) will alternate with the item above it (**New**). (The item set as alternating will be indented underneath its pair.) Only one of the two alternating items will be visible at any given time in the menu.

Important: For items to alternate, their shortcuts must correspond as follows: the shortcuts are the same except that one item's shortcut has an additional modifier. For example: The shortcut for **New** is **Ctrl+N**; the shortcut for **New & Reset** is **Ctrl+Alt+N**.

To alternate between them, press the unique modifier key (in this case, **Alt**).

Project Preferences Dialog Boxes

This dialog box (**Options > Project Preferences**) contains several screens. Navigate among them using the drop-down menu at the top, or the “Previous” and “Next” arrows.



The settings you make here are Project Preferences: specific to the Project you are working on and are saved with it. If another user opens the Project on his or her own computer, the same settings will be applied.

Topics in this section:

[Dimensions Preferences](#)

[Calculation Units Preferences](#)

[Calculation Rules Preferences](#)

[Zones Preferences](#)

[Reference Levels Preferences](#)

[Legacy Preferences](#)

[Stair Rules and Standards: Project Preferences](#)

[Structural Analytical Model Representation \(Preferences\)](#)

[Structural Analytical Model Units](#)

Dimensions Preferences

Dimensions preferences affect the dimension units as they are **displayed in the project**:

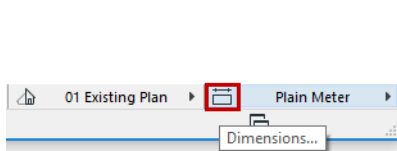
- Dimensions
- Zone Stamps
- Fill text
- Marker texts
- Symbol labels (not Autotext, which is displayed according to Calculation Units Preferences)
- in Element Schedules: Core element parameters entered by you (e.g. Wall height, Slant angle)

Dimensions preferences do *not* affect calculated values (e.g. Wall length) in Interactive Schedule, Listing, or the List Areas add-on. Those calculations are affected by the Calculation Units page of Project Preferences.

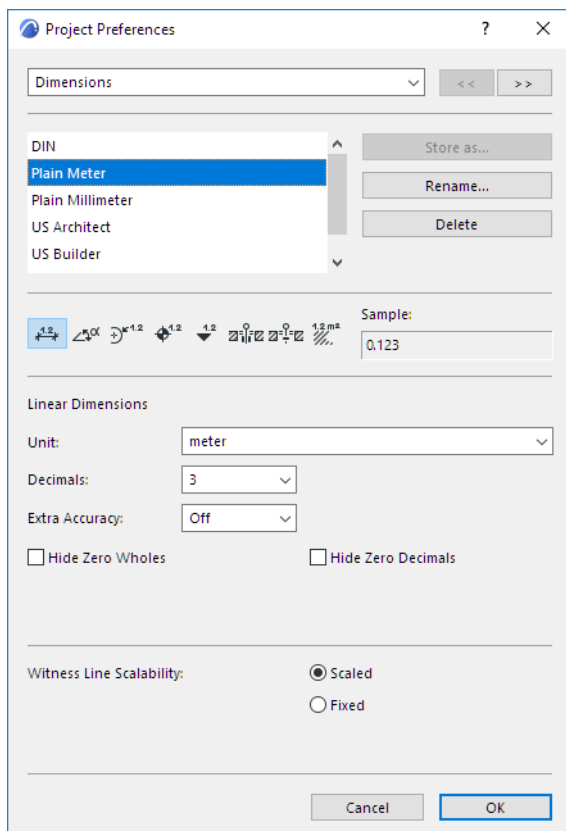
(See [Calculation Units Preferences](#).)

To access Dimensions Preferences, do one of the following:

- Click the Dimensions icon in the Quick Options bar



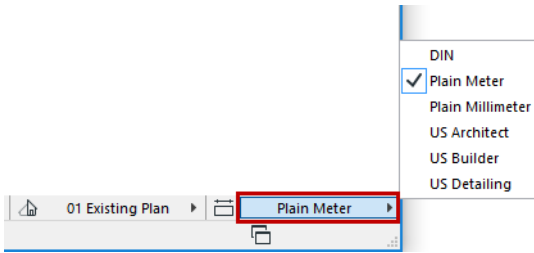
- Go to **Options > Project Preferences > Dimensions**



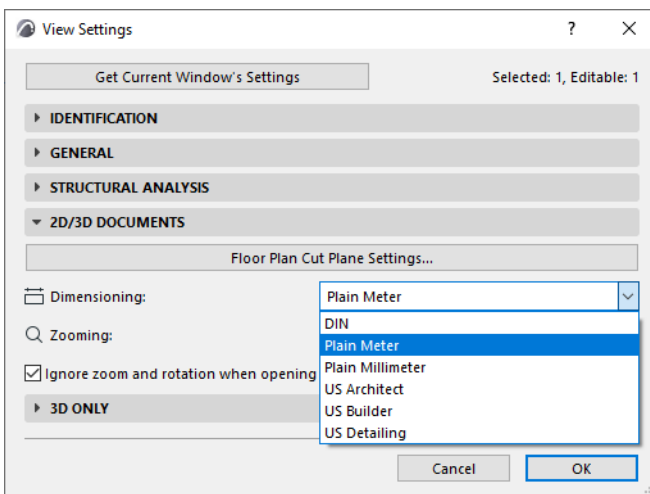
Choose Dimension Standard

Do one of the following:

- Use the Dimensions pop-up from the Quick Options Bar at the bottom of the window

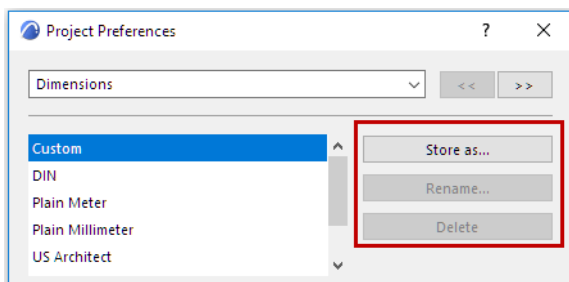


- Use the list on the Dimensions page of Project Preferences
- Use the Dimensioning pop-up in View Settings (2D/3D Documents Panel)



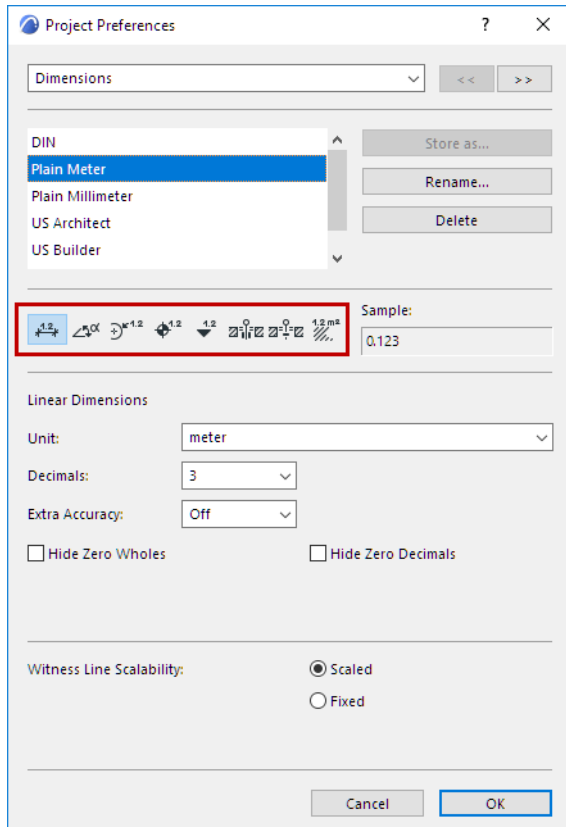
Store Custom Dimension Standard

Store As: If you have customized a dimension setting in this dialog box, the Standard name changes to "Custom." Click "Store as" to name this customized Standard and make it available to your project as a named Standard.



Choose Dimension Type

Click one of the 8 icons to choose the dimension type whose options you wish to set:



- Linear
- Angular
- Radial
- Level
- Elevation
- Door/Window/Skylight
- Sill Height
- Area Calculations (this also affects area as displayed on Zone Stamps)

Unit Display

As you set each control, consult the **Sample** field for a preview of how that Dimension Type will be displayed in the project.

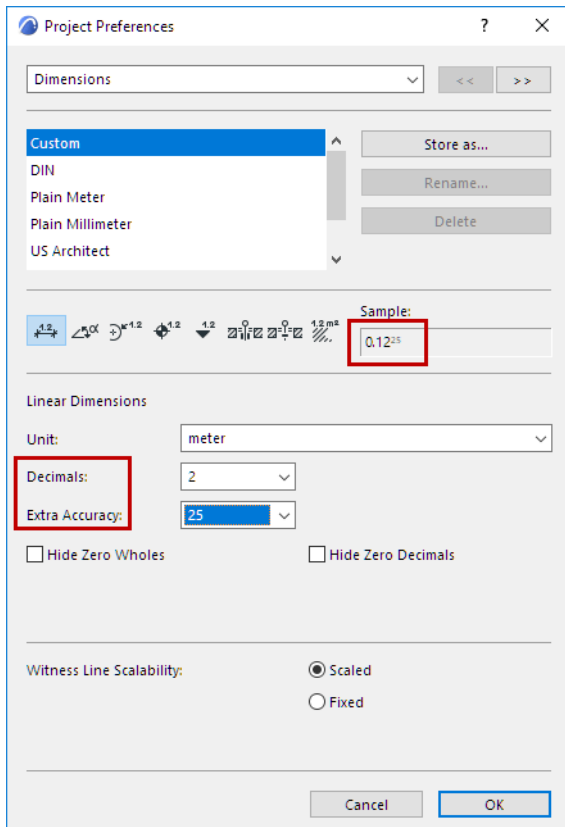
The available dimension options will vary depending on the unit (metric or imperial) you choose, and on the dimension type whose preferences you are setting.

Unit: Define the **Unit** for each of the Dimension types.

Decimals: Define the number of **Decimals** you want to display in each dimension value.

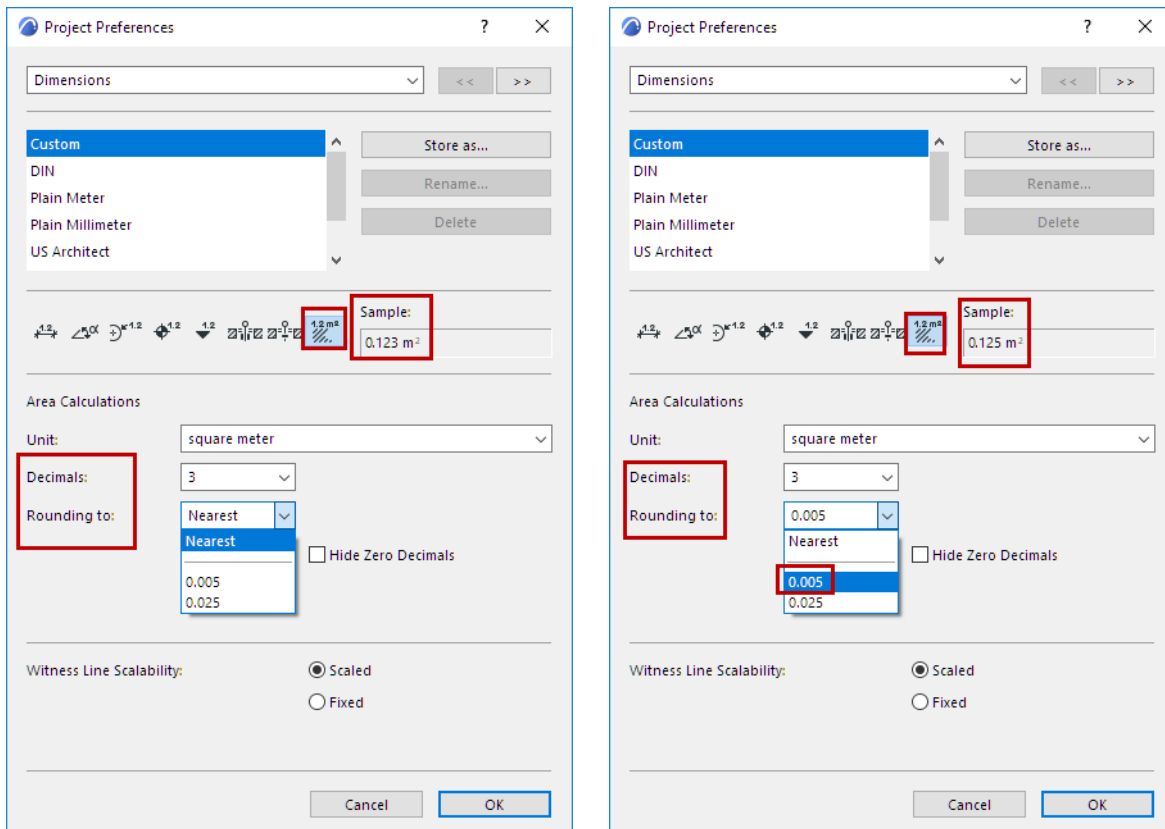
Extra Accuracy: Choose this if you want to display additional decimal values as a superscript.

For example, here the number of decimal places is set to two; but Extra Accuracy is enabled to 25: thus, the next decimal value is also displayed, as a superscript, rounded to increments of 25. See the result in the Sample field of the dialog box.



Rounding to: *Available for Area-type dimension.* Choose a rounding option from the pop-up field.

- **Nearest:** Value is rounded to the number of decimals you set.
- **Other:** Value is rounded to the number of decimals you set, but the final digit(s) are an increment of 5 or 25.



Hide Zero Wholes: Check this box to suppress whole zero values.

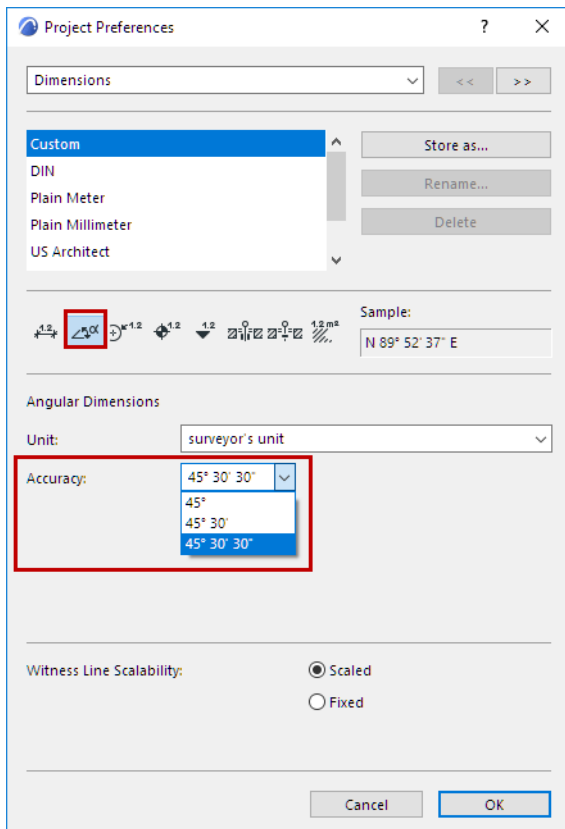
- For example: 0.25 meters is displayed as 25

Hide Zero Decimals: Check this box to hide any zeros at the end of the decimal dimension, regardless of the number of decimal places you set in the Decimals pop-up above.

- For example: 25 meters is displayed as 25.00

You cannot use the Hide Zero Decimals options concurrently with Hide Zero Wholes.

Accuracy: Available for an Angular Dimension Unit of degrees/minutes/seconds or surveyor's unit. Click this pop-up field to select accuracy level.



Witness Line Scalability: Choose either fixed or scaled to make your witness lines keep a fixed scale, or else vary them depending on window scale. (Witness lines are the perpendicular lines between a dimension and the dimensioned element.)

Imperial Unit Display

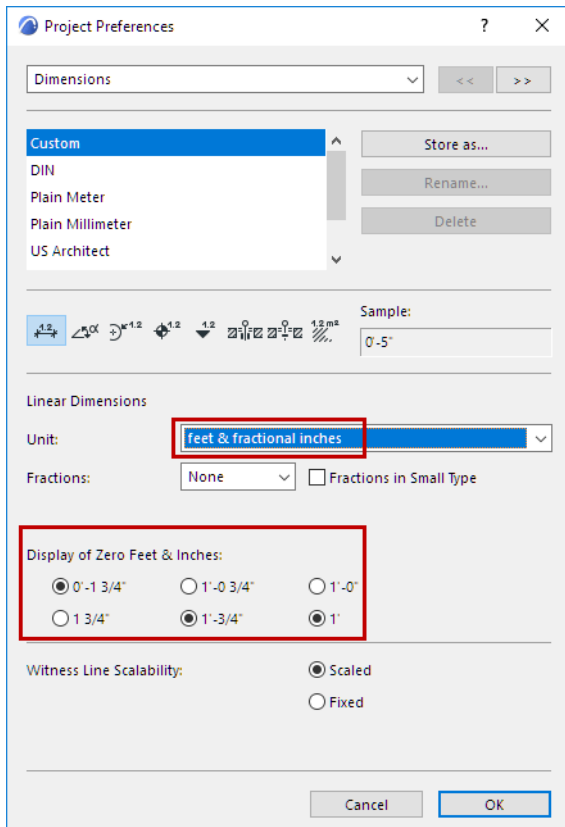
If you choose an imperial unit, the following other options may be available.

Note: You cannot mix units for the same dimension type in a Project. However, you can display two types with the **Secondary Dimensions** command.

Fractions: Click this pop-up field to define the fractions displayed for the selected Measurement Unit, if you are using a unit that includes fractional inches.

Fractions in Small Type: Toggles printing of fractional inches and feet in small characters on and off in length dimensioning.

Display of Zero Feet & Inches: If using Feet & fractional inches, this section has three pairs of radio buttons; if using fractional inches, there is one pair of radio buttons. For each pair, choose the option you prefer for displaying dimensions that measure either zero feet or zero inches.



Calculation Units Preferences

To access this dialog, go to **Options > Project Preferences > Calculation Units**.

This dialog allows you to set the units to be displayed

- in Interactive Schedules, for calculated values
- Autotext Labels
- in lists created with the Calculation function
- the List Areas add-on (available in selected markets)

Also use this dialog (Setup Rules) to set additional rules for computing the values of parameters listed in Interactive Schedules.

Note: If you are using the Calculation function, these settings will affect Text format lists only (not those based on a Graphic Template, which sets its own number format).

Project Preferences

Calculation Units

Length Unit: meter

Decimals: 2

Extra Accuracy: Off 0.12

Area Unit: square meter

Decimals: 2

Extra Accuracy: Off 0.12

Volume Unit: cubic meter

Decimals: 2

Extra Accuracy: Off 0.12

Angle Unit: decimal degrees

Decimals: 0 0°

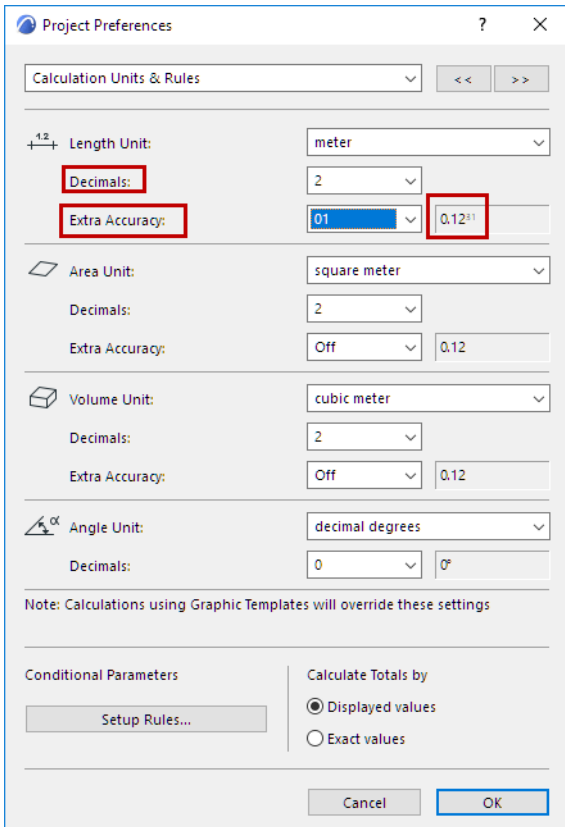
Note: Calculations using Graphic Templates will override these settings

Cancel OK

As you set each control, consult the preview.

- **Unit:** Click this pop-up field to select from various standard Measurement Units.
- **Decimals:** Click the pop-up field to select the number of decimals for the chosen Unit.
- **Extra Accuracy:** Choose this if you want to display additional decimal values as a superscript. The superscript value will be rounded off to increments of the number you select (e.g. 25).

Note: Extra Accuracy values will be displayed as superscripts only in Interactive Schedules. In Calculation lists and some add-ons, these extra values will be shown as extra decimals (rather than superscripts).



For example, here the number of decimal places used in calculating length (for the Wall Perimeter field) is set to two; but Extra Accuracy is turned on to 01, so that the next decimal value is also displayed, as a superscript, and rounded to 01.

Wall Schedule						
Full Element ID	Wall Type	Perimeter [m]	Volume [m3]	Thickness [m]	Height [m]	Area [m2]
SW - 001	Generic Wall/Shell	43.67 ⁸⁵	20.12	0.300	3.100	6
SW - 002	Generic Wall/Shell	36.18 ²	16.43	0.300	3.100	5
SW - 003	Generic Wall/Shell	43.67 ⁸⁵	16.62	0.300	3.100	6
SW - 004	Generic Wall/Shell	36.18 ²	16.23	0.300	3.100	5
SW - 005	GENERIC - STR...	42.65 ⁴⁷	19.08	0.300	3.100	6
SW - 006	GENERIC - STR...	32.55 ⁹³	14.99	0.300	3.100	5
SW - 007	GENERIC - STR...	42.47 ⁹	19.36	0.300	3.100	6
SW - 008	GENERIC - STR...	32.55 ⁹³	14.71	0.300	3.100	5

Note on Volume Unit

Your choice of metric vs. imperial unit for measuring volume will affect the units used by the **Density** value and for the **Embodied Carbon** property of the project’s Building Materials.

[See Building Material Classification and Properties.](#)

Note on Angle Unit

If your chosen Angle Unit is degrees/minutes/seconds, click the **Accuracy** pop-up field to the right to select the precision.

Calculation Rules Preferences

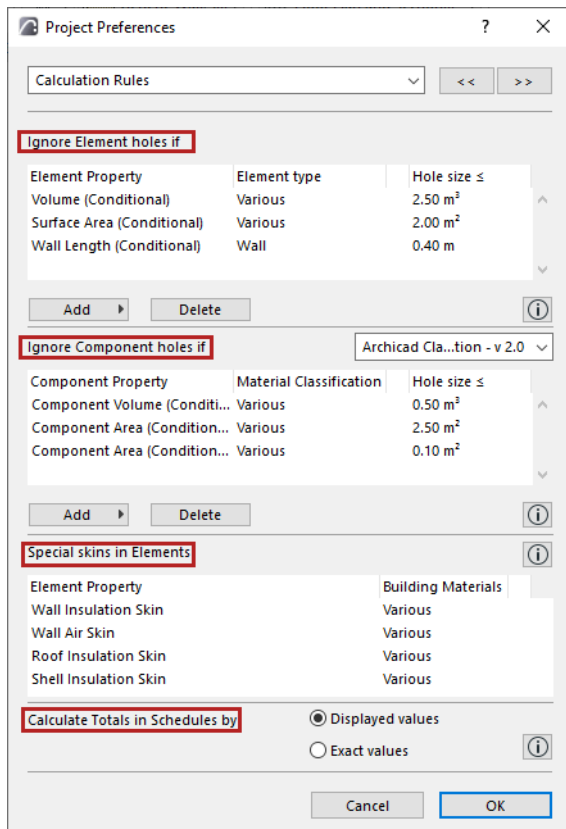
Open this dialog at **Options > Project Preferences > Calculation Rules**.

Use these controls to set how to calculate certain elements and components in Interactive Schedules. If you use a “Conditional” parameter in your schedule, you can specify the conditions in this dialog box.

For example: if you want to list Slab Surface Area so that small holes are ignored but big holes are subtracted:

- Add the “Surface Area (Conditional)” parameter to Scheme Settings of the Interactive Schedule
- Use the **Calculation Rules** dialog box to specify the conditions for such parameters (e.g. the size of holes to ignore or to subtract).

See details below.



Ignore Element holes if

These conditions are relevant when listing element properties (volume, area or Wall length) which are “conditional”. The holes in such listed elements are either subtracted or ignored, depending on hole size:

For each conditional property:

- Set a hole size limit. Holes bigger than this limit are subtracted. (Smaller holes are ignored.)
- Choose one or more element types. The condition applies only to the element types you checked.

Click **Add** to add another conditional property (or the same property) to the list, and customize the conditions for various element types, as needed.

Ignore Component holes if

These conditions are relevant when listing component properties (volume or area) which are “conditional”. The holes in such listed components are either subtracted or ignored, depending on hole size:

For each conditional property:

- Set a hole size limit. Holes bigger than this limit are subtracted. (Smaller holes are ignored.)
- Choose a Building Material classification. (The condition applies only to components that are classified accordingly.)

Note: Available classifications depend on the selected Classification system.

[See *Classification Manager*.](#)

Special Skins in Elements

When listing “insulation” or “air fill” quantities:

- Define which Building Materials, used for the components of composite elements, should be recognized as “insulation” or “air fill”.

For example, if your standard insulation layer in roofs is called “GENERIC - Internal Cladding”, check that box from the Roof Insulation Skin pop-up.

Calculate Totals in Schedules by

This option determines how subtotals are added up in Schedules.

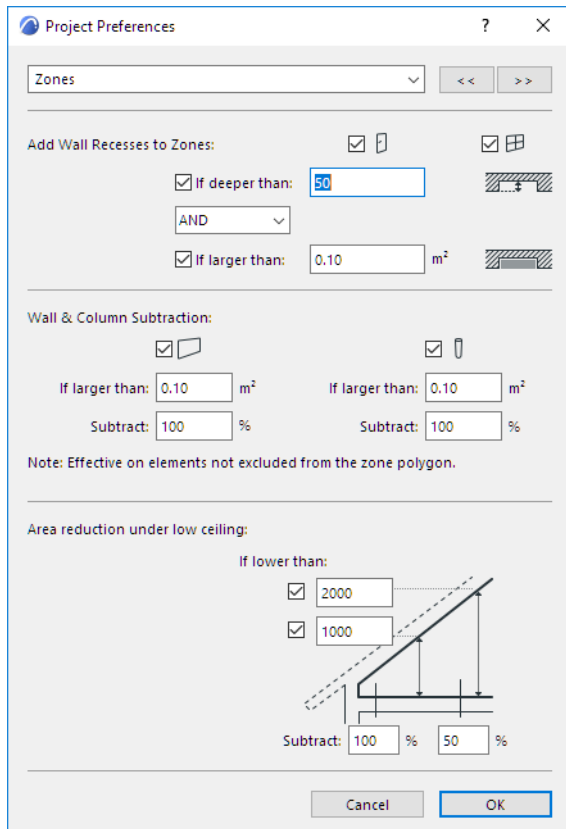
- **Displayed values** (default): Each value (as displayed on the interface) is rounded off according to your Rounding/Extra Accuracy settings. The sum of these rounded values produces the grand total.
- **Exact values:** Choose this for a more accurate value: the program adds up the exact values (not displayed on the interface) before it rounds them off according to your Rounding/Extra Accuracy settings to produce the grand total.

Related Topic

[Interactive Schedule](#)

Zones Preferences

To access this dialog, go to **Options > Project Preferences > Zone Preferences**. These options define how zones are calculated in Zone Lists.



The settings you make here are Project Preferences: specific to the Project you are working on and are saved with it. If another user opens the Project on his or her own computer, the same settings will be applied.

Note: To see the effect of any changes in Zone calculation settings, you must execute the **Design > Update Zones** command.

See [Updating Zones](#).

Add Wall Recesses to Zones

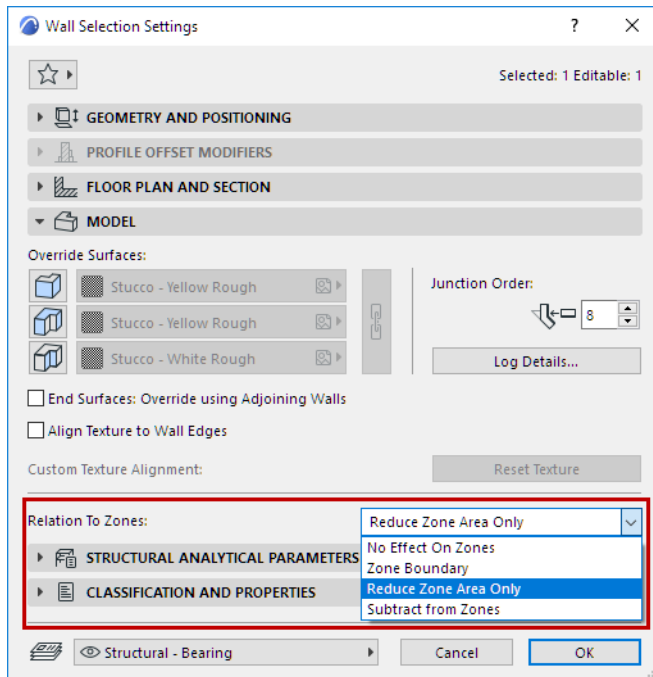
These options define how to take large door/window recesses into consideration when calculating Zones. Door/Window recesses which meet the criteria defined here will be included as part of Zones when Archicad calculates Zone area.

Check the **Door** and/or **Window** icons if you want to consider Door and/or Window recesses in zone calculations.

- **If deeper than:** Check this box to enter a minimum depth for the recess to be considered, then enter this minimum depth into the text box.
- **If larger than:** Check this box to enter a minimum surface area for the recess to be considered, then enter this minimum area into the text box.
- Click **AND** or **OR** from the pop-up menu to further define the recess size (minimum depth and/or minimum area) to take into consideration.

Walls & Column Subtraction

- **If larger than:** Enter the minimum area of Walls, Curtain Walls and/or Columns whose areas should be subtracted when calculating the size of Zones containing them. (Any elements *smaller* than this minimum limit will be included in the zone size.)
- **Subtract:** Enter how much (in percent) of the Wall/Column area to subtract from the Zone area.
Note: These options will affect only elements that are set to “Reduce Zone Area Only” or “Subtract from Zones” in their Settings dialog box.



Area reduction under low ceiling

Use these controls if you want to exclude areas under low ceilings as part of a zone's area.

You can set one or two height limits.

- **First checkbox:** Check this box to define a height limit to take into account for calculating Zone sizes.
- **Second checkbox:** Check this box to define a second height limit to take into account for calculating Zone sizes.
- **Subtract:** Use these text boxes to define the percentage of the area underneath ceiling that should be subtracted from the zone area. You can enter up to two different subtraction percentages, one for each height limit you defined.

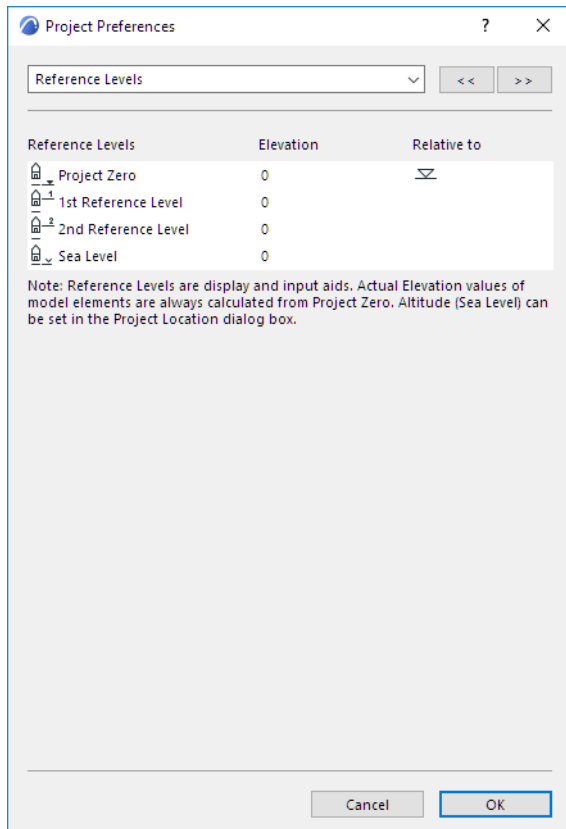
To see how much of a zone's area has been reduced due to this Area Reduction setting, update the Zone, then select it and view the data in the Zone Area Calculation Panel.

Related Topics

[Calculating Zone Area and Zone Volume](#)
[Zone Area Calculation Panel](#)

Reference Levels Preferences

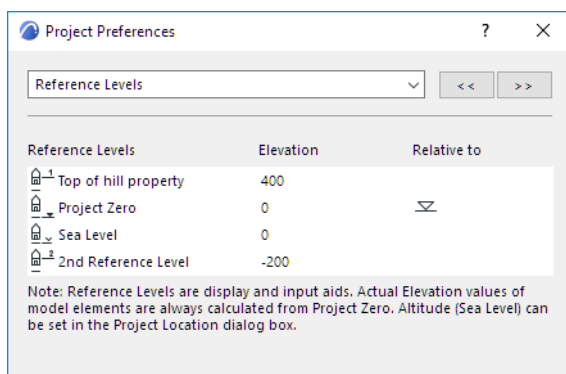
You can define optional Reference Levels that will help you position elements. This is done at **Levels Preferences (Options > Project Preferences > Reference Levels)**.



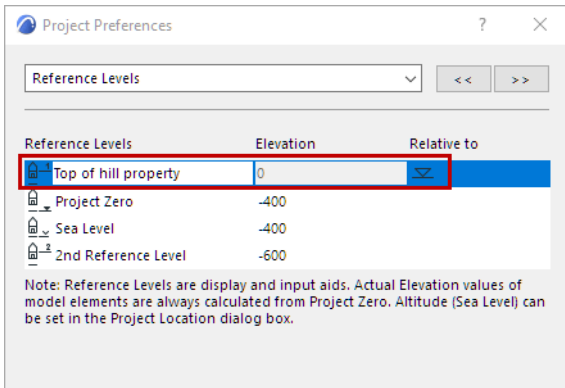
All of these settings are effective for Coordinates Palette, Info Box and all Dialog Boxes.

Select a level's name and modify it as you like, then type the value you need in the **Elevation** field. In this example, we defined the top of the property, at 400 meters.

The Sea Level takes its value from the Altitude, which is set at **Options > Project Preferences > Location Settings**.



You may find it easier to calculate levels (in this dialog box) by switching the zero value to another level than Project Zero. You can do so by simply clicking in the **Relative to** field next to the level's name.



Note: You cannot rename the Project Zero or Sea Level reference levels.

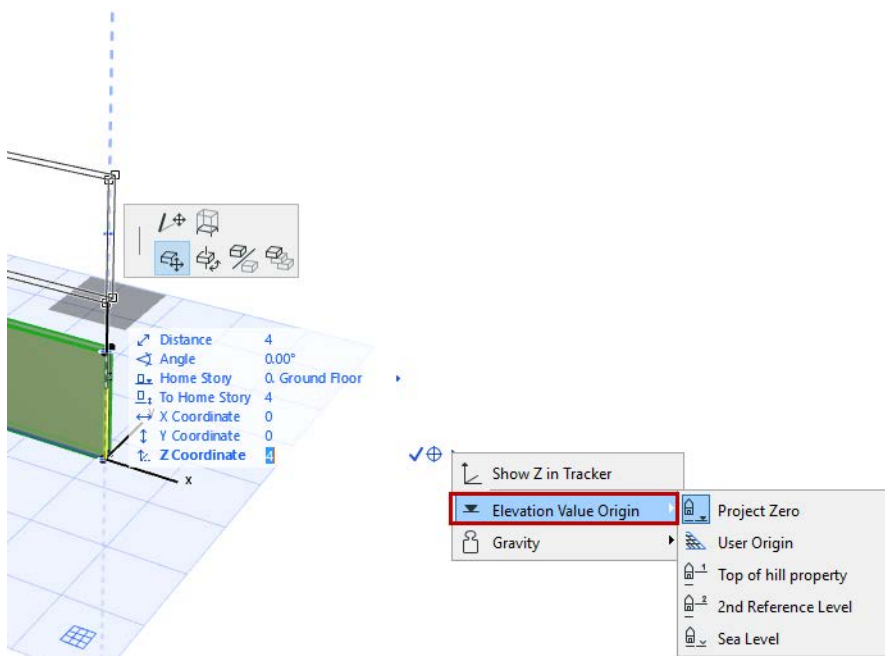
Note: Reference Levels are display and input aids only. Changing their value has no effect on placed elements, whose actual elevation is always calculated from Project Zero. The exception is Grid lines in Section view, whose position changes with the Reference Level.

The newly defined Reference Levels are now available in the Bottom Elevation [to Reference level] fields of the Geometry and Positioning Panels of construction and object-type elements.

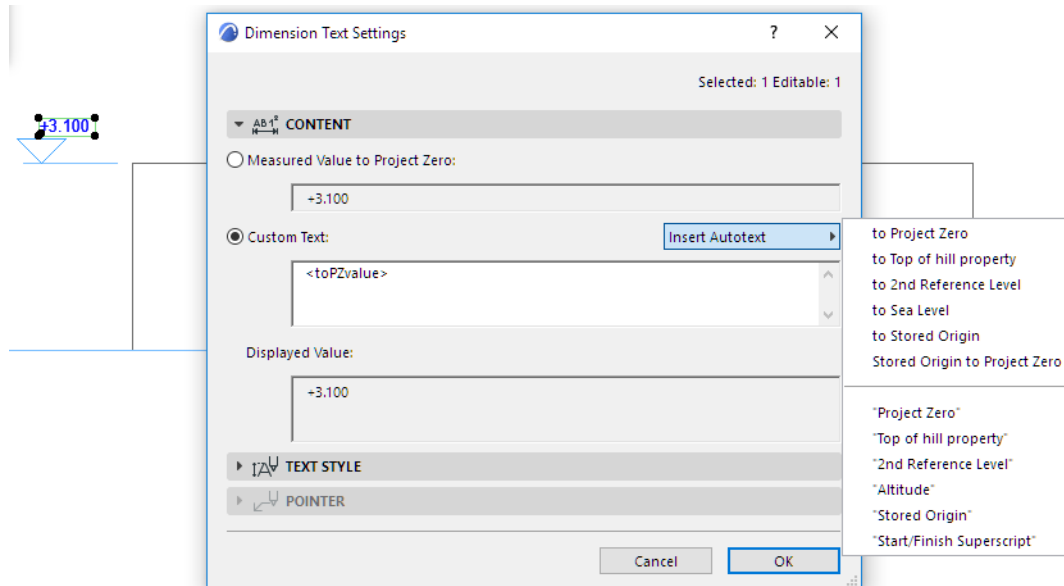
[See also Place a Construction Element in Archicad.](#)

Reference levels can also be used for showing elevation (Z) values in the Tracker.

[See Tracker.](#)



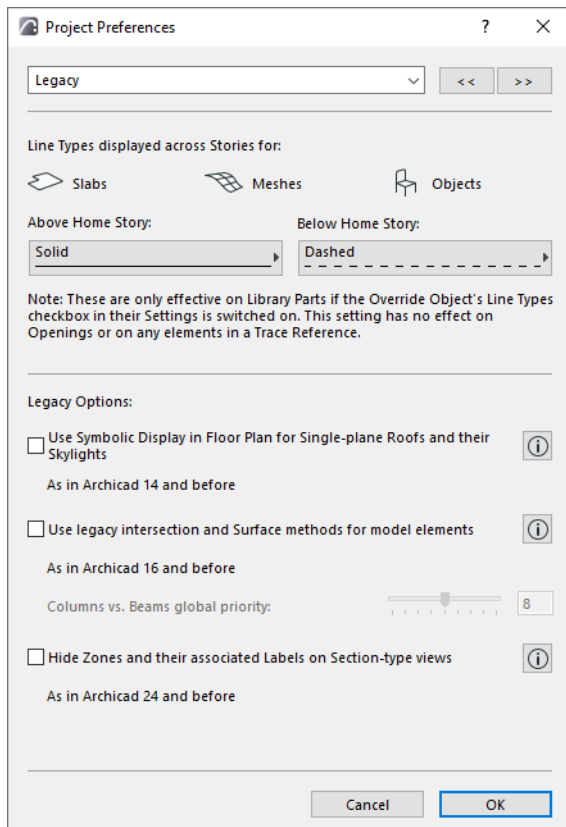
Reference Levels can also be used as Autotexts, such as Autotext used in Elevation Dimensions:



Related Topics:

[Autotext](#)

Legacy Preferences

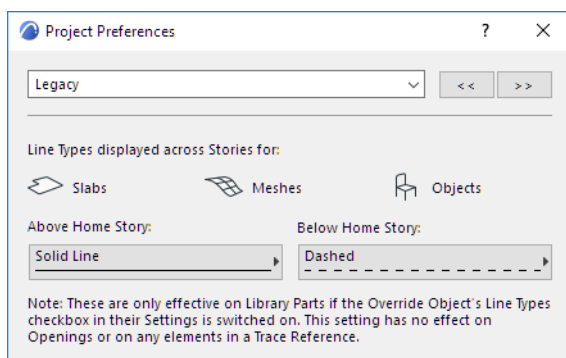


To access this dialog, go to **Options > Project Preferences > Legacy**.

The settings you make here are Project Preferences: specific to the Project you are working on, and saved with it. If another user opens the Project on his or her own computer, the same settings will be applied.

Line Types Displayed Across Stories

Select a line type with which to display the outlines of **Slabs** and **Meshes** that appear on stories above and below the Home Story, as well as certain **Objects**.



Note: This setting has no effect on Walls, Roofs or Shells, their Openings, or any elements in a Trace Reference.

Note: These line types are only effective on Objects if the **Override Object's Line Types** checkbox in their Settings dialog boxes is on.

Use Symbolic Display in Floor Plan for Single-plane Roofs and their Skylights (as in Archicad 14 and before)

Check this box if you want your Single-plane roofs and any Skylights within them to be displayed as they were in Archicad 14 and earlier versions. Click the Info button at right for an illustration of the difference in displays.

- **True 3D Projection:** This display mode is available as of Archicad 15. (All Multi-plane Roofs and newly placed Skylights will be shown this way.)
- **Symbolic:** This display mode corresponds to the way Roofs and Skylights were displayed in Archicad 14 and earlier.

(For more information, click the Info button at the right of this option.)

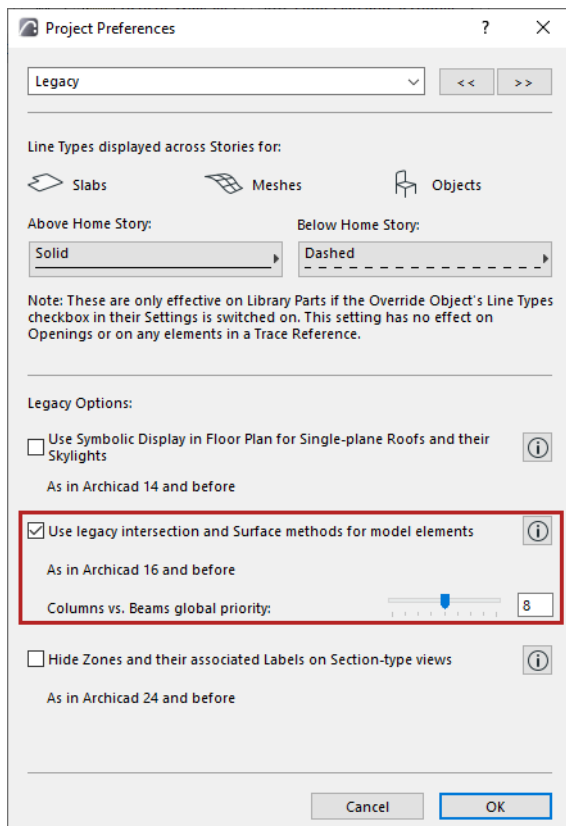
Use legacy intersection and Surface methods for model elements (as in Archicad 16 and before)

Note: Legacy intersection mode may be discontinued in a future Archicad release.

Archicad 17 introduced smart intersection features based on element Building Materials.

When you open an older-version project (Archicad 16 and earlier), you can opt to use these latest intersection methods (Legacy mode OFF). To continue using the old methods instead, you can stay in Legacy mode (Legacy mode ON) instead.

The sections below describe the difference between these two modes.



Legacy Mode ON: Use Old Intersection Logic

There are two ways to enable Legacy intersection mode in your Archicad project:

- Activate the checkbox on the Legacy page of Project Preferences

- Choose **Stay in Legacy Mode** from the dialog which appears when you open an Archicad project that is already in Legacy mode.

Legacy intersection mode (Legacy mode ON) creates the following limitations. Also, please be aware that Legacy intersection mode may be discontinued in a future Archicad release.

- Automatic and Building Material **priority-based intersections** will not take effect in 3D. Connection methods used in Archicad 16 and earlier versions will be used. You may prefer this when working on an older project, to ensure that your existing intersections remain intact.
- The **Building Material** attribute is available – but the priority-based connections will not take effect in 3D (except for Walls, where they will work as in Archicad 16 and earlier versions). Instead, use the slider below to set a global priority for Columns that intersect Beams. (See description below).
- **Surfaces** assigned to Building Materials are not displayed. Only element-level surfaces can be shown.
- **Surface Painter** is not available
- **Surface Schedules** are not available
- Expanded geometry features for **Beams** and **Columns** (introduced in Archicad 23) are NOT available. These include multi-segment, end cuts, tapered and vertically curved geometry. If you are in Legacy mode, and if you try to use an unsupported geometry method, the resulting element is shown with a highlight. ([See Use uniform color to highlight inverted elements.](#))

This table specifies how Legacy mode (checkbox ON) affects different construction elements.

Element Type	Behavior in Legacy Mode (Checkbox ON)
Wall	Will not intersect with non-Wall elements. Wall-Wall intersections shown with simplified 3D feedback.
Slabs and Morphs	Priority from Building Material has no effect.
Columns	Priority from Building Material has no effect. When intersecting with Beams, uses the priority setting under Legacy Options. Multi-segment, End Cuts, Tapered Beams not supported.
Beams	Profiled Beams ignore priorities of individual components, and will use Junction Order from the element settings. Multi-segment, End Cuts, Tapered Beams not supported.
Roofs and Shells	Composite elements ignore priorities of individual skins, and will use Junction Order from the element settings.
Openings (via Opening Tool, AC23 and later)	Visible only as 2D symbols on Floor Plan. Not visible in 3D.

Legacy mode OFF: Use New Intersection Logic

If you open an older-version project (Archicad 16 and earlier), you can opt to use the latest intersection methods (Legacy mode OFF).

- Newly placed elements with collisions modeled using the new methods will clean up according to the new logic.
- Newly placed Beams, Columns and Openings use the full feature set available as of Archicad 23.
- Existing, modeled connections that do not involve collisions will not clean up automatically.
- This way, even when you migrate an older-version project, you will be able to enjoy the benefits of the new intersections features, while maintaining the existing model as it was.

Columns vs. Beams global priority (Legacy mode ON)

Available if Legacy mode is ON.

This slider defines the relationship between all Columns and Beams in the project. When Beams cross Columns, the element of lower priority will be cut in 3D, i.e., the intersection part will be removed. The element of higher priority will remain intact. This applies to volume calculations as well.

Note: Column priority numbers are always even and can only be defined as a general setting, in this dialog box. Beam Junction Order is set for each beam separately in Beam Tool Settings (Model panel).

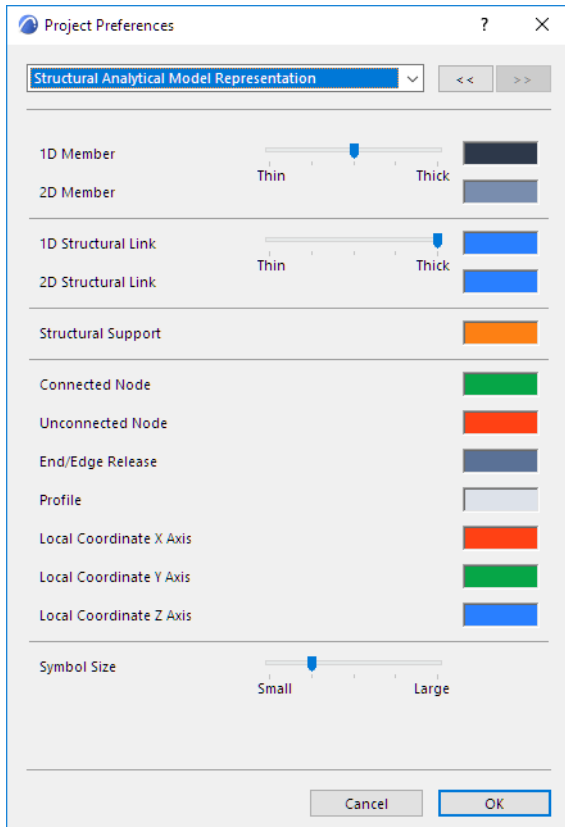
Hide Zones and their associated Labels on Section-type views

As of Archicad 25 Update, Zones are visible on Sections, Elevations and Interior Elevations. In addition, Zones have associated Labels. To maintain the appearance of older projects, enable this checkbox: Zones and their Labels will not be visible on these views (as in earlier Archicad versions).

Structural Analytical Model Representation (Preferences)

Use this page of the Project Preferences dialog to set colors and symbol sizes for the Members and visible components of the Structural Analytical Model.

- **Options > Project Preferences**
- **View > Structural Model Appearance > Structural Analytical Model Representation**



Related Topic:

[Representation of Structural Analytical Model](#)

Structural Analytical Model Units

To access this dialog, go to **Options > Project Preferences > Structural Analytical Model Units**.

The screenshot shows the 'Project Preferences' dialog box with the 'Structural Analytical Model Units' section selected. The dialog has a title bar with a question mark and a close button. Below the title bar is a dropdown menu showing 'Structural Analytical Model Units' and navigation buttons '<<' and '>>'. The main area contains seven settings, each with a unit selection dropdown and a 'Decimals' dropdown set to '2':

- Point Translation Stiffness: MN/m - meganewton per meter
- Point Rotation Stiffness: MNm/rad - meganewtonmeter per radian
- Line Translation Stiffness: MN/m/m - meganewtonmeter per meter
- Line Rotation Stiffness: MNm/rad/m - meganewtonmeter per meter
- Surface Translation Stiffness: MN/m/m² - meganewtonmeter square meter
- Translation Resistance Force: MN - meganewton
- Rotation Resistance Moment: MNm - meganewtonmeter

At the bottom of the dialog are 'Cancel' and 'OK' buttons.

Related Topic:

[Structural Analytical Model](#)

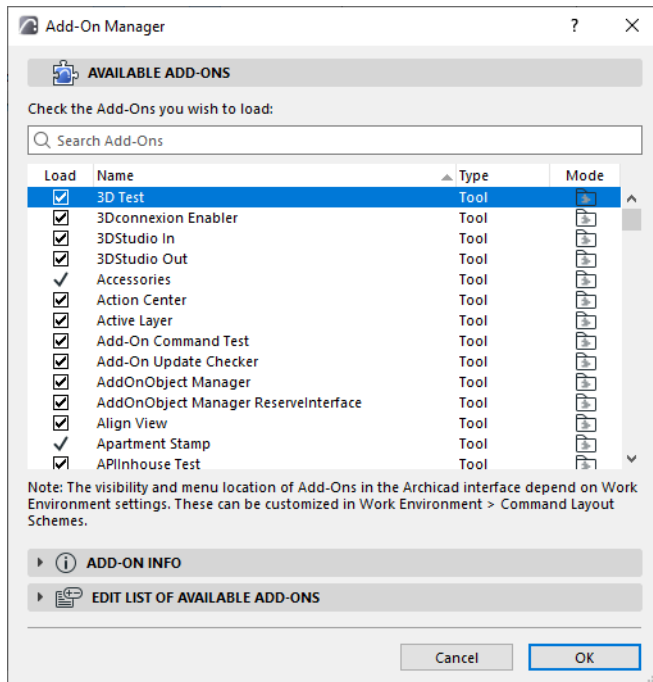
Add-on Manager

Use the **Options > Add-on Manager** command to open the Add-on Manager dialog box.

For more information on this topic, see [Add-ons and Goodies](#).

Click the header fields to sort data by that column. Use the splitter bars to edit column width.

Available Add-ons Panel



Here you can load or unload Add-ons by switching the checkboxes on and off, then check OK to close the dialog box and execute the changes.

If you see a checkmark in front of the name of an Add-on without a box, this means that the Add-on must always be loaded with Archicad.

The **Type** column refers to the type of the Add-on, which can be Tool or Imaging.

- **Tool** type add-ons which are loaded by default appear as menu commands in the Standard Work Environment profile. You can customize their location like any other menu command. Other Tool-type add-ons, after being loaded manually, will be placed in a menu according to the location of that menu's Add-on Anchor Point.
- **Imaging** type add-ons add rendering or animation options to View menu items.

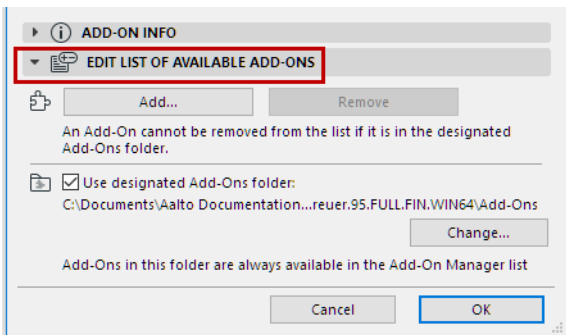
The icons in the **Mode** column show whether the Add-on was loaded individually or as part of an Add-on folder set to be available automatically at each startup.

Add-on Info Panel

The **Add-on Info** panel contains information on the interface location and operation of the Add-on selected in the Available Add-ons panel.

Edit List of Available Add-ons Panel

In the **Edit List of Available Add-ons** panel, you can add more Add-ons, remove items from the list and redefine the location of the dedicated Add-on folder.



The **Use designated Add-ons folder** checkbox will activate the folder enabled by default (the Add-ons folder in your Archicad Folder) or another dedicated folder that you can choose with the Change button by browsing your computer.

If you are using such a designated folder, the add-ons made available from this folder cannot be removed from the list of available add-ons. However, they can be unloaded manually (by unchecking its name in the Load column) for the current Archicad session. In this case, the rest of the add-ons which are still loaded from this folder will appear as individual add-ons in the **Mode** column.

If you load an Add-on that has the same name and ID as a loaded one, it will replace the previous one.

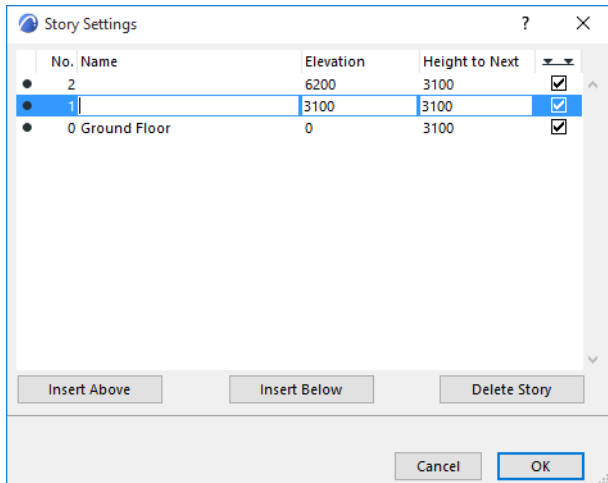
- All settings of the **Add-on Manager** will be preserved when you quit and restart the program, even if you choose **New and Reset**.
- If you open a Project that refers to currently unloaded Add-ons, an alert will appear, listing the missing Add-ons.

Some of the Add-ons cannot be loaded until you restart the program, as they contain references to the project database. If loading such Add-ons with the Add-on Manager, an alert will appear. If current modifications do not concern these Add-ons, you can ignore the alert or choose to save the recent changes and reload the Project with the required loaded Add-ons.

Story Settings Dialog Box

To open this dialog box, do one of the following:

- Use the context menu of a Story tab
- Use the **Design > Story Settings** command
- Choose **Story Settings** from the context menu of a selected Story, or the Stories folder, in the Navigator.

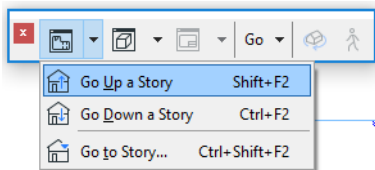


Use the splitter bars in the header to edit column width.

In the upper section of the dialog box, a scrollable list shows the current stories in your design. When starting a new project, you may have several stories already created by default. Every project must contain at least one story.

To activate a story, select it in the list and click OK. The Floor Plan will show this selected story.

Note: You can also navigate among stories using the Navigator's Project or View Map, or the story-related commands in the Mini-Navigator toolbar.



Insert Above/Insert Below: Click either button to create a new story above or below the story currently selected in the list.

When the name of a Story is selected, you can edit its Name, Elevation (to Project Zero), its Height to the Story above, and whether to show the Story Level line on sections. The Number field is not editable manually.

Multi-selection is available, to edit these values simultaneously for multiple stories at once.

Changes to story **height** and **elevation** are interpreted differently:

- If you modify the **height** of a story, this will affect the elevation of all stories above (if the story number is positive) or below (if the story number is negative). In practical terms, you stretch or shrink the whole building by making the height of a story larger or smaller. The height of all other stories remain as they were.

- If you modify the **elevation** of a story, this will affect the heights of the stories immediately above and below. In practical terms, you move the level of a single story while keeping the rest of the building static. The elevation of all other stories remain as they were.

Note: Changing story heights may result in inverted elements: their top elevation will fall below the bottom elevation.

See [Inverted Elements](#).

Show Level on Sections (icon at far right in Story Settings): Check this box to display Level Lines of this Story in Section view.



For more information, see [Show/Hide Story Level Lines in Section/Elevation/IE](#).

Delete Story: Click this button to delete the selected story.

Warning: If you delete a Story, all elements that had that story as their Home Story will also be deleted!

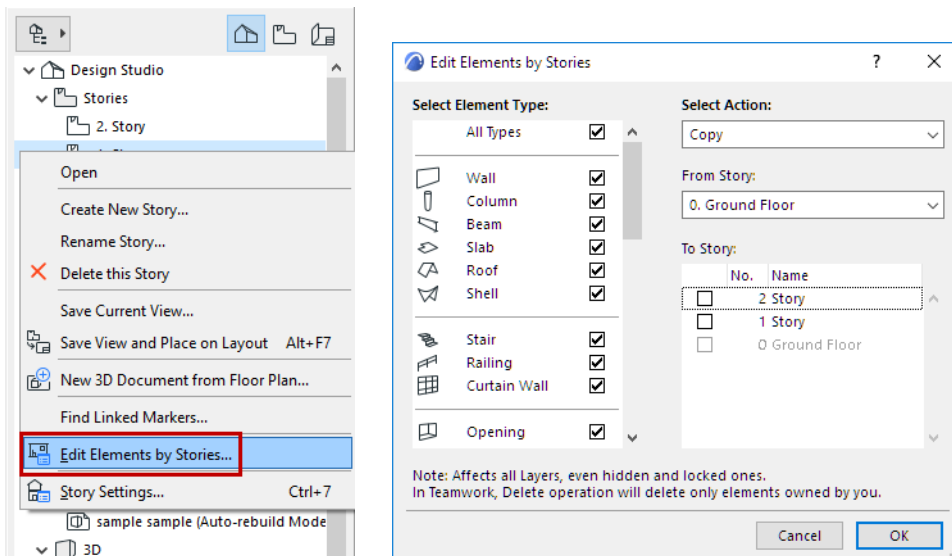
Warning: The undo queue is cleared entirely whenever you delete or add a story, or change story height.

Edit Elements by Stories Dialog Box

Use this dialog box to cut-paste or copy-paste elements from one story to another, or to delete elements on a given story.

To open this dialog box, do one of the following:

- Select any Story item from the Navigator's Project Map, then click **Edit Elements by Stories** from the context menu.
- Design > Edit Elements by Stories
- Edit > Move > Edit Elements by Stories



Select Element Type

At left is a list of element types. By default, they are all checked.

Note: Not all tools are included in this list, because not every element type can be copied from one story to another. For example, Section and Elevation lines cannot be copied among stories, and these tools are not included in the list.

Adjust these so that the element types you want to cut/copy/delete are checked story.

Select Action

Choose either Copy, Cut or Delete. (There is no separate Paste command. If you choose "Copy" or "Cut", you will then be able to paste those elements to any story.)

From Story:

From the drop-down list, choose the Story whose elements you want to edit (i.e. cut, copy or delete).

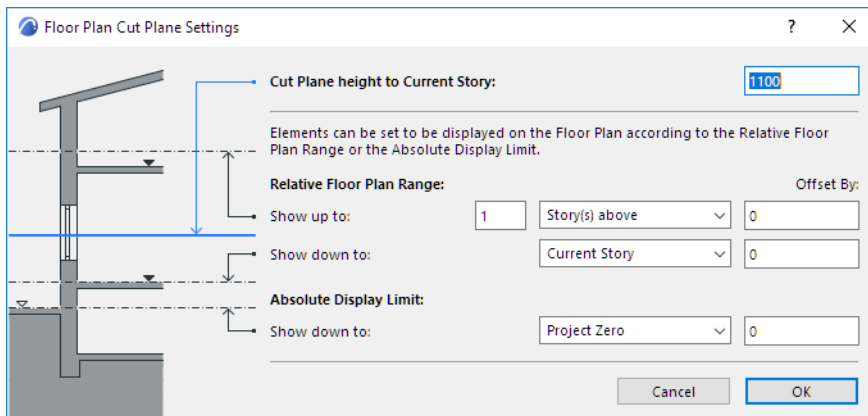
To Story: (not available if you selected "Delete")

From the drop-down list, choose the Story to which you want to paste the cut or copied elements.

Click **OK**. This will carry out the action, close the dialog box and activate the selected story on the Floor Plan.

Floor Plan Cut Plane Settings Dialog Box

Use **Document > Floor Plan Cut Plane** to open the Floor Plan Cut Plane Settings dialog box.



Cut Plane height to Current Story

Enter the height at which elements will be cut for display on the Floor Plan.

The other options in this dialog box - **Relative Floor Plan Range** and **Absolute Display Limit** - will be applied only to construction elements whose “Show Projection” setting (in the Settings dialog box) is set to use one of these two definitions. These definitions set a limit on how much of the element is to be displayed, relative to the current story.

Relative Floor Plan Range

Use these controls to define the Relative Floor Plan Range for construction elements that use this definition.

Show up to: Enter the number of stories you want to display the element above the current story.

Offset by: Optionally, enter a positive or negative elevation value by which to offset the “Show up to” display limit.

Story pop-up: Choose either **Current Story** or, if you want to display the element on additional stories above the current one, **Story(s) above**.

Show down to: Enter the number of stories you want to display the element below the current story.

Offset by: Optionally, enter a positive or negative elevation value by which to offset the “Show down to” display limit.

Story pop-up: Choose either **Current Story** or, if you want to display the element on additional stories below the current one, **Story(s) below**.

Absolute Display Limit

Use these controls to define the Absolute (lower) Display Limit for construction elements that use this definition. This limit is in effect regardless of the currently active story.

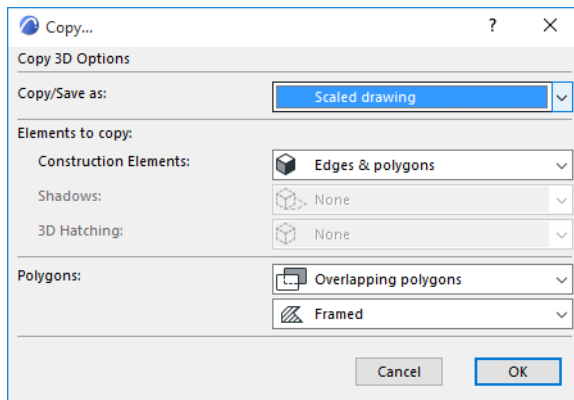
Show down to: Enter the fixed lower limit to which elements will be projected. (By default, this is Project Zero.)

Offset by: Optionally, enter a positive or negative height value by which to offset the “Show down to” absolute display limit.

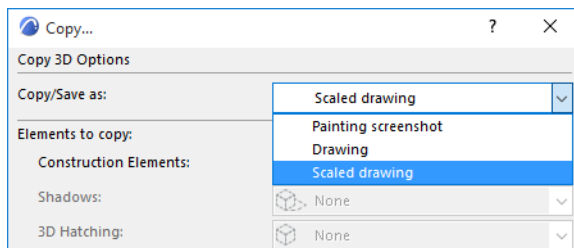
Copy Dialog Box

This dialog box appears if you have copied a Marquee area from the 3D window (using the 2D Marquee option of the Marquee tool), when using the Vectorial 3D Engine.

For details, see [Copy/Paste Marquee Area from Project Window](#).



Copy/Save as:



- **Painting Screenshot** will copy the contents of the **3D Window** as a unified picture (Figure).
- When choosing **Drawing**, the resulting image will consist of the parts described in the dialog box. The size of the picture obtained will be equal to the size of the image in the **3D Window**. This is important when pasting the copied parts into another application such as SuperPaint or Photoshop.
- The **Scaled drawing** option creates an image that keeps the sizes of the Floor Plan. If you paste a **Side view** of the Project on the Floor Plan using the **Scaled drawing** option, the sizes will match the sizes of the plan.

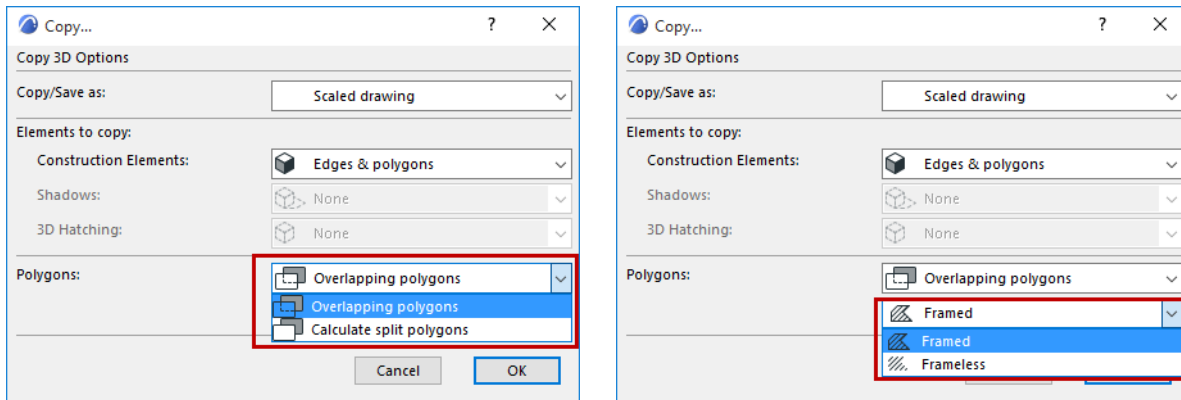
With the **Elements to Copy** pop-up menus you can define which elements of the 3D View you want to copy to the Clipboard.

The Shadows pop-ups are available only if the Sun Shadows is “On” in 3D Styles (**View > 3D View Options > 3D Styles.**)

Copying the shadows of top views can be useful if you want to place accurate, measurable shadows on the Floor Plan. Shadows and 3D hatching of orthogonal views add a further touch of realism to elevation drawings.

The 3D Hatching pop-ups are available only if the Vectorial Hatching option is “On” in 3D Styles, and if the 3D view contains at least one element that has 3D hatching applied.

In the lower part of the dialog box you can define whether to save/copy invisible **Polygons** (for example, the polygon elements of a window behind a wall) or only visible ones. You also have the option of copying the polygons with or without frames.



You will see the **Remove Redundant Lines** checkbox if you want to copy the selection in **Drawing** or **Scaled drawing** mode from a wireframe or a hidden line picture. If you check this checkbox, lines exactly on top of each other will be omitted in the copied file.

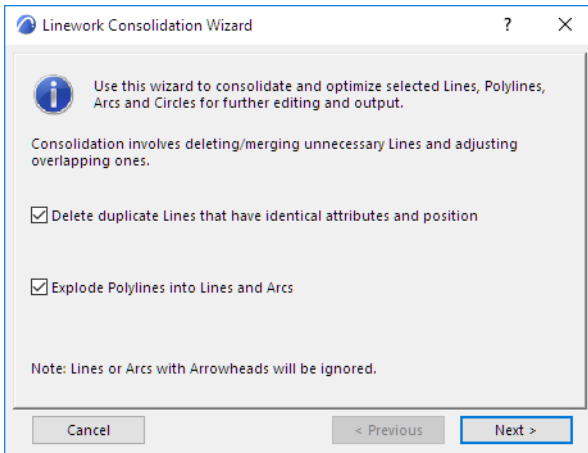
Hint: If you want to copy the elements of the **3D Window**, e.g., for an elevation view, it is recommended to use the **Calculate split polygons** and **Frameless** options.

Linework Consolidation Settings

To access this dialog box, you must first select at least two line-type elements. Use **Edit > Reshape > Linework Consolidation...** command.

For general information on this feature, see [Consolidate Lines and Fills in Drawing Windows](#).

The first time you execute this command, you will bring up the Linetype Consolidation Wizard.

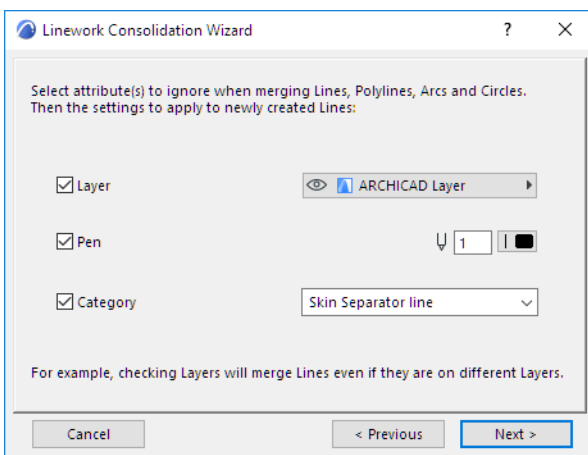


The wizard takes you through the Consolidation options one by one. Note that each line consolidation option builds on the previous one; the program will execute the consolidation in the order in which they appear in the Wizard.

Delete Duplicates: Duplicate lines are those which have identical attributes and positions. One line will remain, the rest will be deleted.

Explode Polylines into Drawing Primitives: Polylines will be exploded into multiple single lines.

Linework Attribute Merge Settings



Check any or all of the following attributes:

- Layer
- Pen
- Category

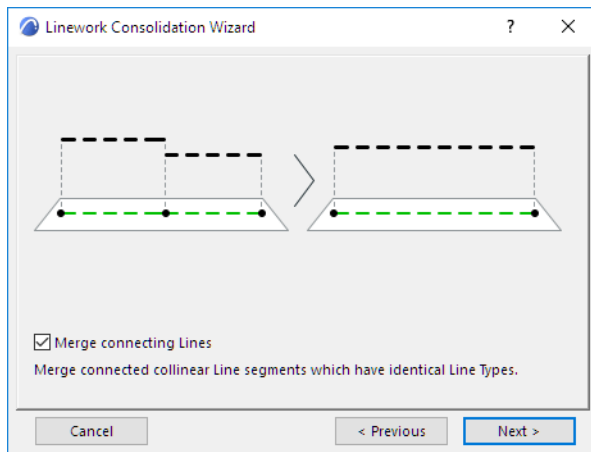
Checking an attribute here means that the program will not take these attributes into account when merging connecting or overlapping lines.

- For example, if you check “Pen”, then the program will merge overlapping or connecting lines even if the lines do not use the same Pen (e.g. one line is blue and the other is red).

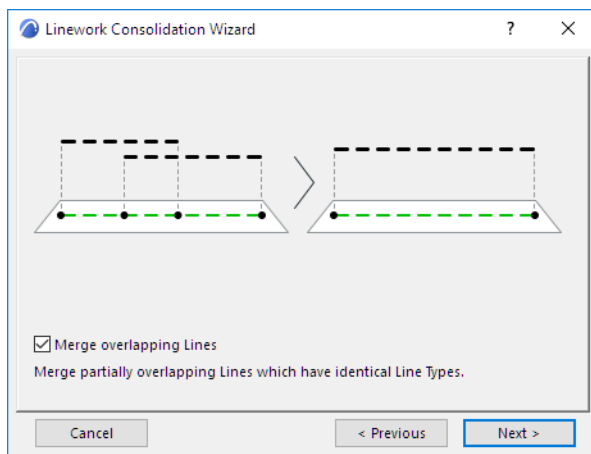
For each checked attribute in this dialog box, the Layer/Pen/Category controls are activated, so that you can assign a layer/pen/category for the lines which result from the merge process.

- For example, after checking Pen, you must decide whether the new line, merged from the original red and blue lines, should henceforth be red, blue, or any other color.

Merge Connecting Lines: Co-linear line segments which are connected and which have identical line types will be merged into a single line.



Merge Overlapping Lines: Lines which partially overlap and which have identical line types will be merged into a single line.

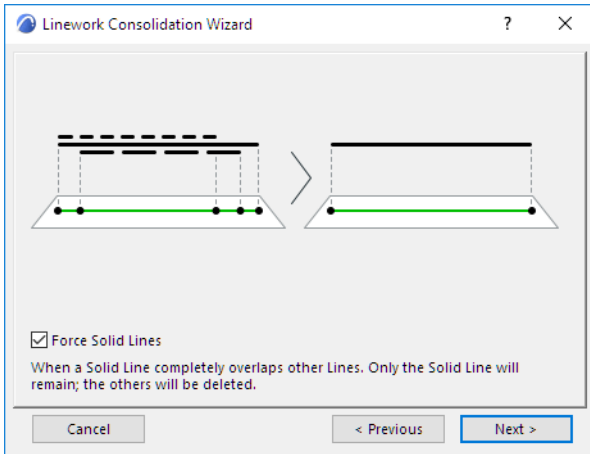


Merge Settings

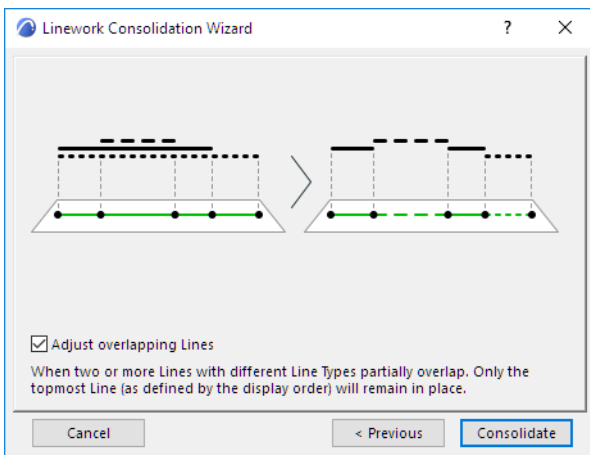
If you have selected either or both of the two Merge options (Merge Connecting Lines/Merge Overlapping Lines), use the next screens to define the attributes of the lines which result from the merge process.

(If you are using the single-page Linework Consolidations Settings instead of the Wizard, click Merge Settings to access these options.)

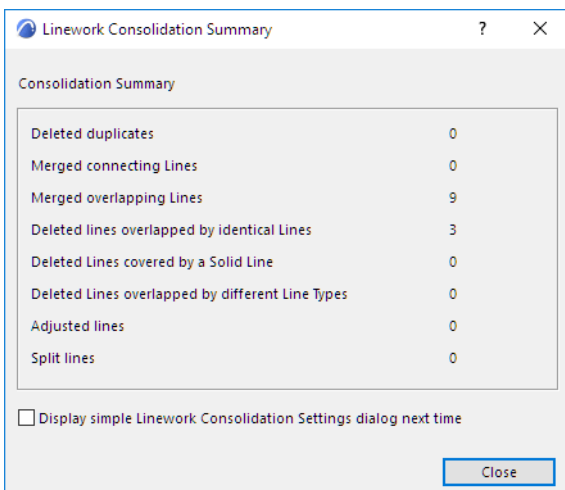
Force Solid Lines: This option is useful if several lines overlap each other, and one of them is a solid line which completely overlaps the other line(s). Checking this box will delete the other line(s), with the single solid line remaining in place.



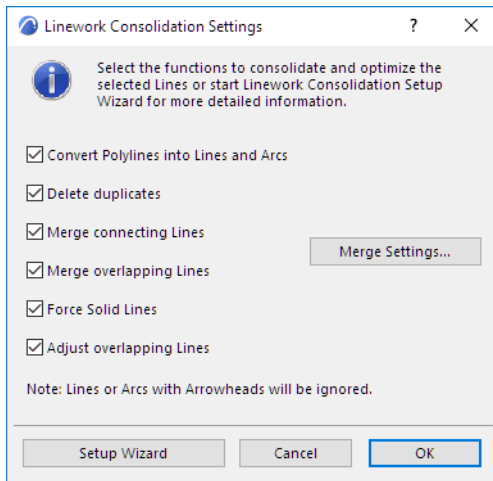
Adjust Overlapping Lines: This option will apply if two or more lines of different line types partially overlap each other. If you check this box, only the topmost line (as defined in the display order) will remain in place of the overlapping part.



Click **Consolidate** to carry out the Linework Consolidation and to display a Summary of the result.

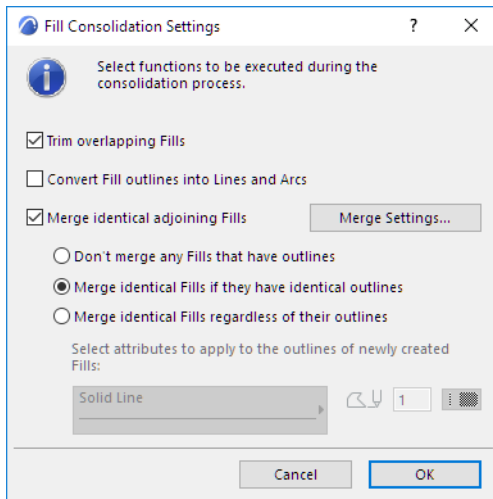


On the last screen of the Wizard, check “Display simple Linework Consolidation Settings next time”. This way, the next time you use the **Edit > Reshape > Linework Consolidation** command you will access the simplified Settings dialog box.



Fill Consolidation Settings

To access this dialog box, you must first select at least two fill-type elements. Use the **Edit > Reshape > Fill Consolidation** command.



For general information on this feature, see [Consolidate Lines and Fills in Drawing Windows](#).

Trim Overlapping Fills: Any selected fills that partially or entirely overlap each other will be trimmed so that only the top fill remains; any other fill (or part of a fill) that is covered up beneath the top fill will be deleted.

Note on Overlapping Fills with Transparent backgrounds: If the top-level fill has a transparent background, any solid-background fill beneath it will remain intact, because it will be visible.

Exception: If there are multiple transparent fills atop each other, only the top-level fill will remain in full; any fills or parts of fills underneath, if they have transparent backgrounds and are covered by the top-level fill, will be deleted.

Convert Fill outlines into Lines and Arcs: Any fill contours will be exploded into simple line segments/arcs.

Merge identical adjoining Fills: Any adjoining fills (regardless of display order) which have identical fill patterns/colors will be merged into one.

Note: In **Merge identical adjoining Fills**, “identical” means that fills must have identical fill patterns, identical foreground/background colors, and identical orientations. Fills that have different pattern orientations (project origin/fill origin/distorted) will not be merged.

The resulting fill will carry the ID of the largest of the original fills.

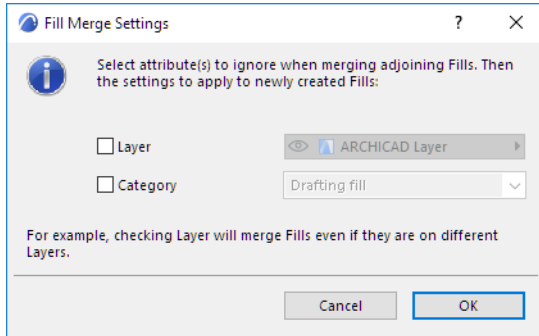
Outlines of Merged Fills: If you checked “Merge identical adjoining fills,” three different options are activated.

- **Don’t merge any Fills that have outlines:** Such fills are not merged at all.
- **Merge identical Fills if they have identical outlines:** Only those fills will be merged which also have identical outlines.
- **Merge identical Fills regardless of their outlines:** All identical fills are merged, whether they have outlines or not, and whether the outlines are the same or not. If you check this option, additional controls are activated to let you define the line type and pen of the outlines that will result after consolidation.

Note: If you checked “Convert Fill outlines into Lines and Arcs”, these options are not available, since the fill outlines will no longer exist after the consolidation process.

Fills that have different pattern orientations (project origin/fill origin/distorted) will not be merged.

Merge Settings: If you have selected “Merge identical adjoining fills,” the Merge Settings button is activated. Click to access Fill Merge Settings to define the attributes of the lines which result from the merge process.



Check either or both of these attributes:

- Layer
- Category

Checking an attribute here means that the program will not take these attributes into account when merging adjoining fills.

- For example, if you check “Category”, then the program will merge adjoining fills even if the fills are not assigned to the same Fill Category (e.g. one fill is a “Drafting” fill, the other is a “Cover” fill).

For each checked attribute in this dialog box, the Layer/Category controls are activated, so that you can assign a layer/category for the fills which result from the merge process.

- For example, after checking Category, you must decide whether the new fill, merged from the original Drafting and Cover fills, should henceforth be a Cover fill, a Drafting fill or a Cut fill.

Related Topics:

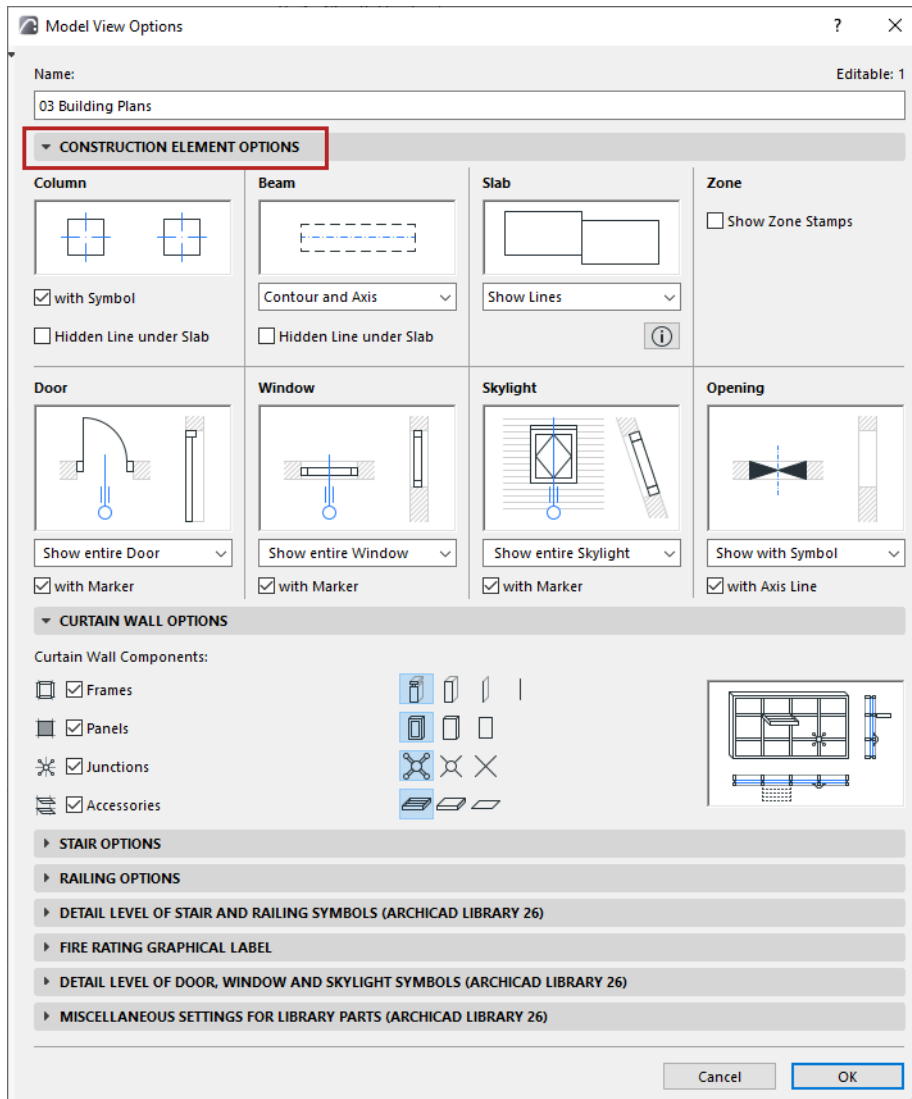
[Fill Categories](#)

Model View Options for Construction Elements

Construction Element Options

To access this panel, go to **Document > Model View > Model View Options**.

For general information, see [Model View Options](#).



A preview window next to each settings group indicates the result.

Column

- Check **with Symbol** to display the symbol chosen in Column Settings (**Floor Plan and Section panel > Floor Plan Symbol**) for each column.
- **Hidden Line under Slab**: Check this to display Columns underneath Slabs, using Hidden Line. (Define Hidden Line attributes in Column Settings.)

Beam

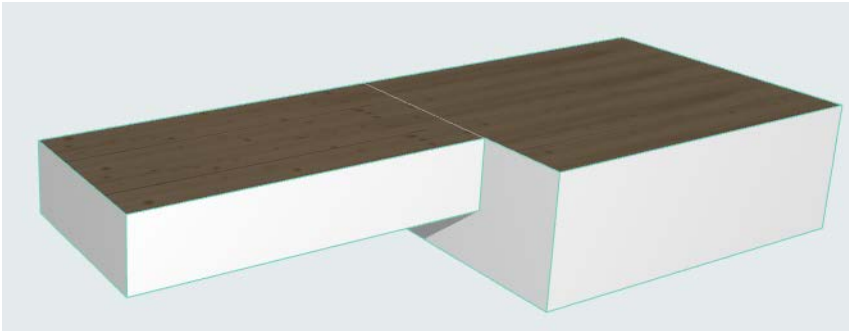
- Contour and Axis
- Axis Only

- Contour Only
Note: You can override the Axis/Contour settings for individual Beams in Beam Settings (Floor Plan and Section Panel).
- **Hidden Line under Slab:** Check this to display Beams underneath Slabs, using Hidden Line. (Define Hidden Line attributes in Beam Settings.)

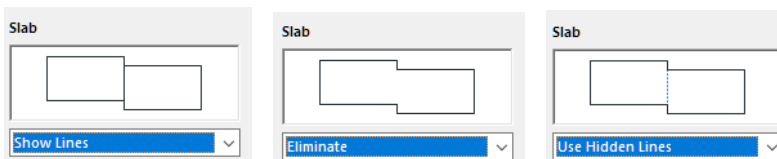
[Watch the Video](#)

Slab

For adjoining Slabs (with identical top surfaces in the same plane):



Choose an option to display or hide the dividing line.



- Show Lines
- Eliminate Lines
- Use Hidden Lines

[Watch the Video](#)

Door/Window

Set these options separately for Doors and Windows.

- **Show entire Door/Window:** Doors/windows will be displayed with the appropriate symbol on the Floor Plan and in Section-type views.
with Marker: Check this box to display doors/windows with Markers (if assigned in the Settings dialog box.)
- **Show Opening Only:** The place of the doors/windows (but not the doors/windows themselves) will be indicated by openings in the walls, in both Floor Plan and Section views.
with Contours: The door/window openings shown on the Floor Plan will include contours.
- **Hide on Floor Plan:** No placed doors/windows or openings will be shown on the Floor Plan. (In Section-type views, openings only will be shown.)

Skylight

- **Show Entire Skylight:** Skylights will be displayed on the Floor Plan and in Section-type views, in accordance with the Floor Plan Display preferences of the host Roof or Shell.
with Markers: Check this box to display Skylights with their Markers (if any Markers are assigned in the Skylight Settings dialog box.)
- **Show Opening Only:** The place of the Skylights (but not the Skylights themselves) will be indicated by an opening in the Roof/Shell, on both Floor Plan and Section views.
- **Hide on Floor Plan:** No placed Skylights or Skylight openings will be shown on the floor plan. (In Section-type views, openings only will be shown.)

Opening

- **Show with Symbol:** Openings displayed on the Floor Plan and in Section views, together with their symbol.
- **Show Opening Only:** No Symbol is displayed
- **Hide:** Openings hidden in all views (including Floor Plan, 3D and Section)

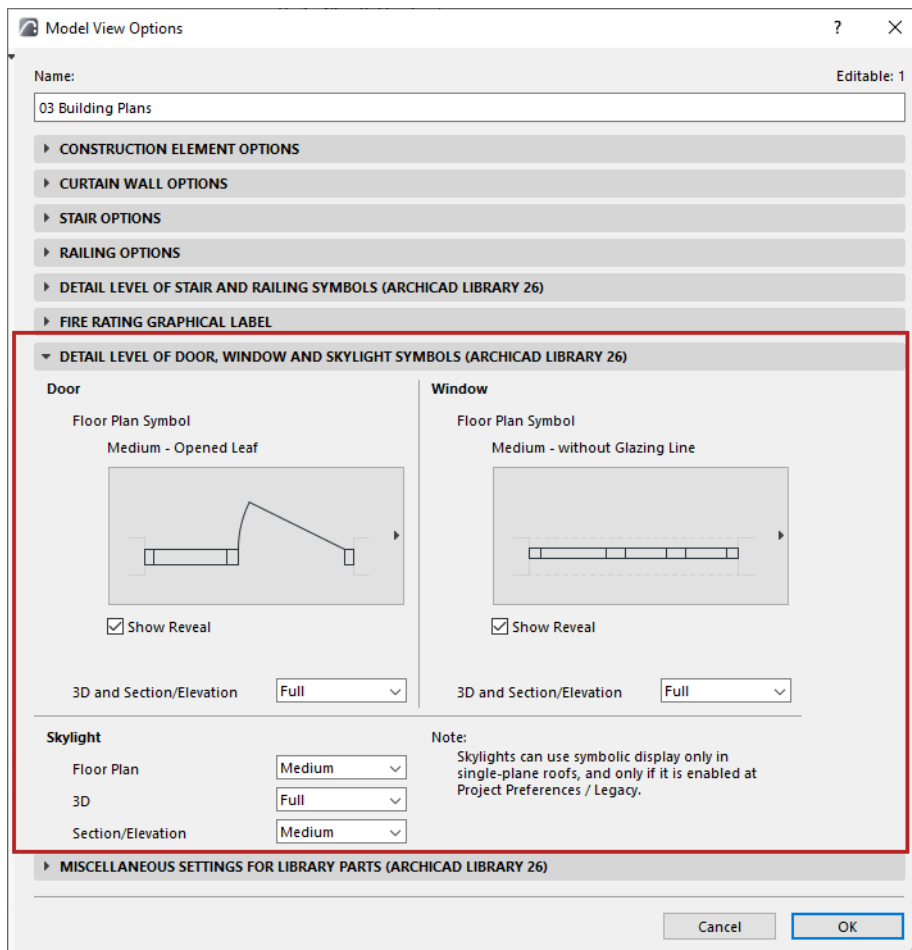
Zone

- **Show Zone Stamp:** Check this box to display Zone Stamps in your project. (Off by default. Zones are placed with associated Labels instead.)

Detail Level of Door, Window and Skylight Symbols

To access this panel, go to **Document > Model View > Model View Options**.

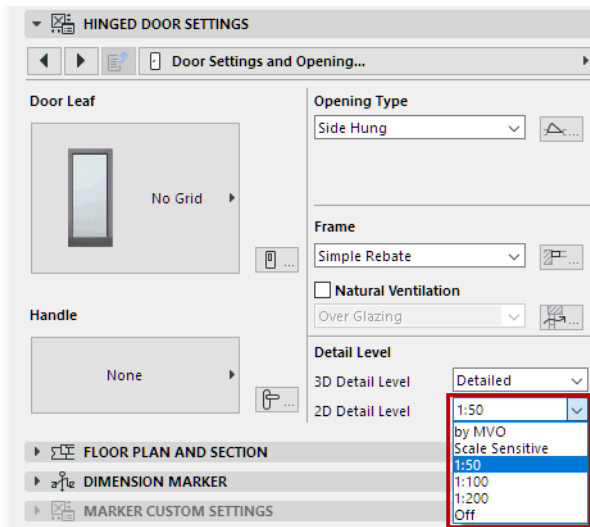
- This panel and its contents depend on the libraries you have loaded. (You may have multiple panels, one for each loaded library in your project.)
- Library objects located in that specific library will be globally displayed according to the Model View Options set here.



Override Detail Level for Particular Door/Window/Skylight

Model View Options affect all Doors/Windows in the project by view.

To override the MVO-defined Detail Level settings for any particular Door/Window, use the Door (Window) Settings and Opening tab page of the Door or Window's Custom Settings panel.



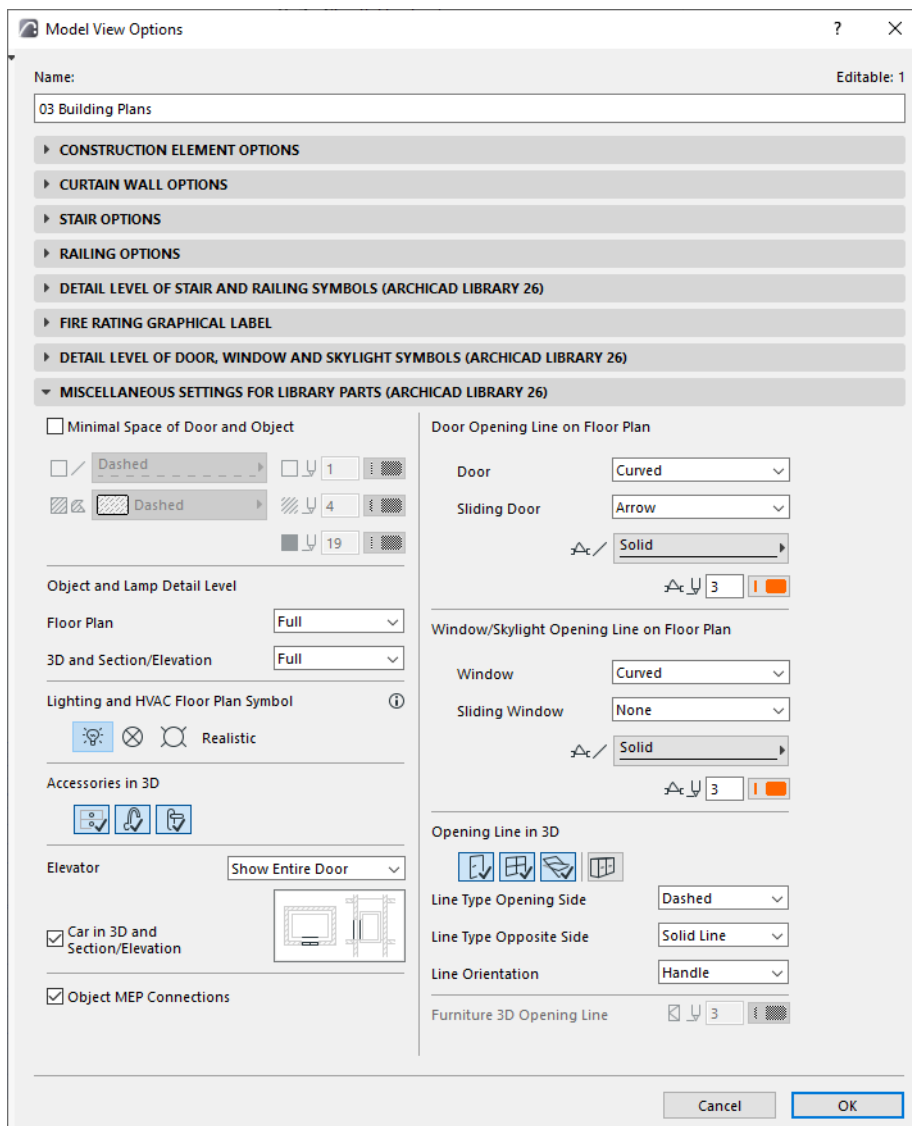
See [Door/Window Settings and Opening](#).

Miscellaneous Settings for Library Parts

To access this panel, go to **Document > Model View > Model View Options**.

- This panel and its contents depend on the libraries you have loaded. (You may have multiple panels, one for each loaded library in your project.)
- Library objects located in that specific library will be globally displayed according to the Model View Options set here.
- Options will be applied only to objects whose GDL script is programmed to allow for this.

In the INT version of Archicad, the panel looks like this:



Show Minimal Space of Door and Object

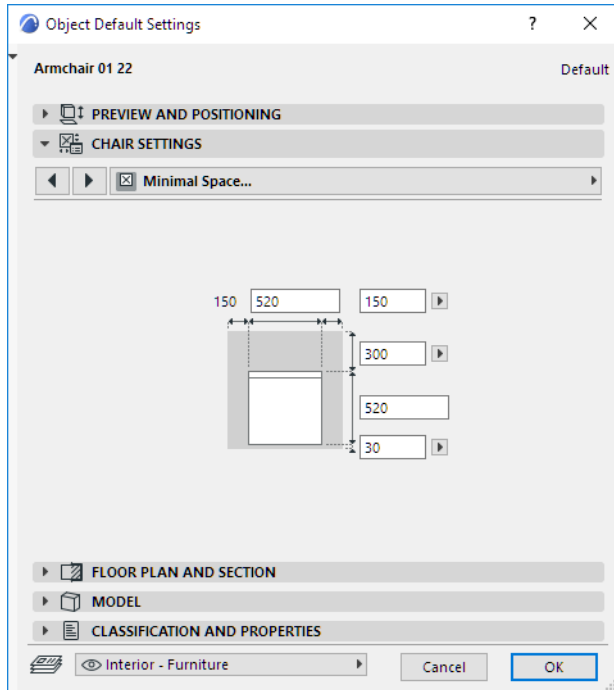
Check this box to show the objects' and doors' minimal space on the Floor Plan by displaying a contour and fill corresponding to the minimal space parameter. Choose options for the contour's line type and pen, and for the fill type/pen/background.

Minimal Space contour/fill display is shown for all doors and objects which have a defined Minimal Space parameter.

- For Doors, set these parameters in the Minimal Space tab page of Door Settings (Custom Settings panel).

[See *Minimal Space*](#).

- For Objects, use the Minimal Space parameters in Object Settings.



Object and Lamp Detail Level

Set the Detail Level on Floor Plan:

- Full, Medium, or Low

and in 3D Projections:

- Full, Simplified or Schematic

Objects and Lamps use the detail level settings defined here by default (**By MVO**). For a specific object/lamp, if the control is available, you can override its detail level in Element Settings (use the 2D or 3D Representation controls)

[See *2D and 3D Representation of Objects*](#).

Lighting and HVAC Plan Symbols

Choose one of the options for displaying the Floor Plan Symbols of Lighting and HVAC objects (e.g. ceiling lamp, smoke detector, air diffuser) in the project.

- Realistic
- Electric
- Reflected Ceiling Plan

Accessories in 3D

Click the toggle buttons to show or hide these types of accessories in 3D:

- Object Knobs in 3D

- Object Taps in 3D
- Door Handles and Knobs in 3D

Elevator Car

Set options for the display of the Elevator Object's Car component:

- **Show/Hide Car** in 3D and Section/Elevation
- **Show/Hide Elevator Doors** in Floor Plan/Section, or display just the opening

Object MEP Connections

Elements compatible with MEP Modeler include Connections to enable connection to any object in an MEP route.

Use this checkbox to show or hide these Connections.

Doors/Windows/Skylights Opening Line on Floor Plan

- Opening Line style: **Curved** or **Straight**
- For Sliding Doors and Windows: **None** or **Arrow**
- Line Type and Pen for the opening line

Opening Lines in 3D Projections

Click the toggle buttons (Door, Window, Skylight) to show or hide their opening lines in the 3D window.

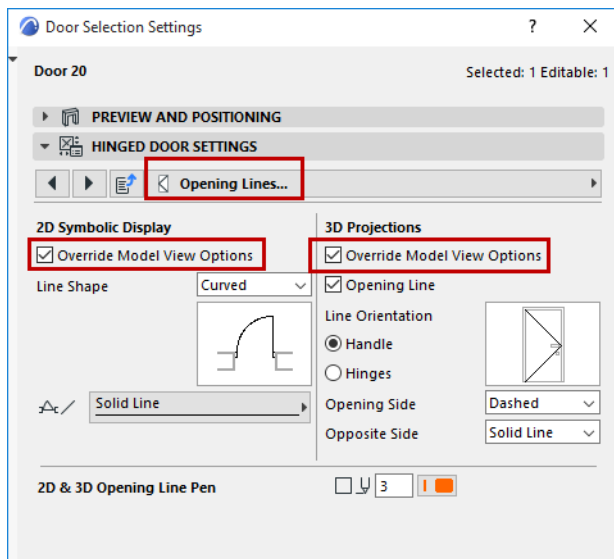
- Choose **lines** (dashed, solid, dotted) for the opening lines on the opening and opposite sides.
- Choose either **Handle** or **Hinges** to set the opening lines' orientation.

Note: This option is scale-dependent (generally, opening lines are not displayed at small scales).

Override Model View Options for Particular Door/Window

Model View Options affect all Doors/Windows in the view. To override Door/Window opening lines settings for any particular Door/Window, use the Opening Lines tab page of the Custom Settings panel.

[See *Opening Lines \(Door/Window\)*.](#)



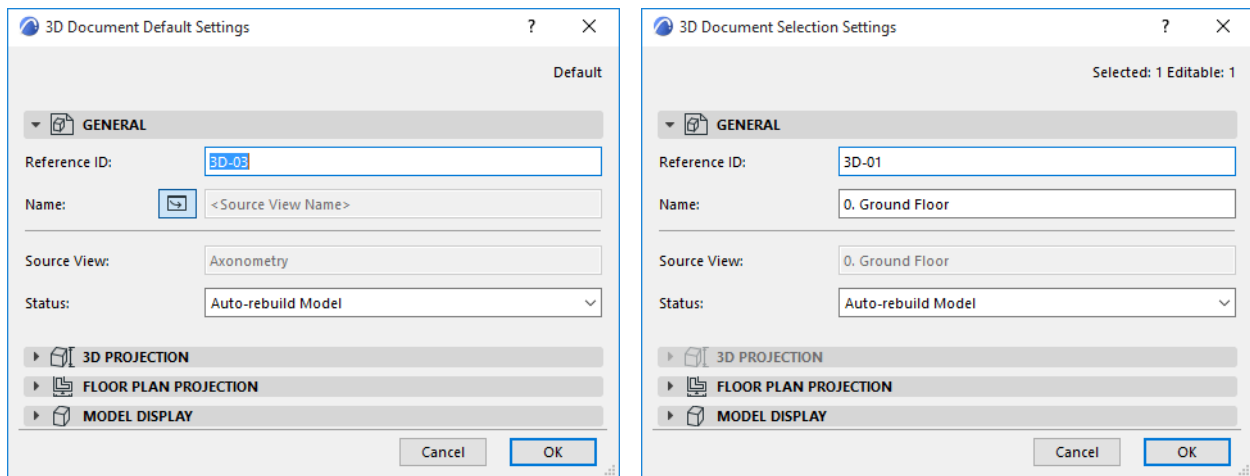
3D Document Settings

The 3D Document Settings dialog box is available from:

- the context menu of any 3D Document item listed in the Navigator
- the Settings button at the bottom of the Navigator Project map, if a 3D Document is selected
- the context menu of a 3D Document window
- the **Document > Documenting Tools** menu

For general information on this topic, see [3D Document](#).

3D Document General Panel



Reference ID: An ID is assigned by default; you can change it by typing any other value into the field.

Name: Appears in the Navigator palette and the title bar of the 3D Document window.

- In 3D Document Default Settings, you can activate the **Source View Name** icon: this will automatically assign the source view as the name of the newly created 3D Document.

Note: This will only set the name of the source view at the time of creation; it will not automatically follow story name changes. Later, you can change it to any custom text.

Status: Define the link between the 3D Document and the Model.

- **Auto-rebuild Model:** Rebuild the 3D Document automatically every time it is opened or brought to the front of the screen, if the model has changed.
- **Manual-rebuild Model:** The 3D Document is rebuilt from the model only by using the **View > Refresh > Rebuild from Model** command. (The same command is available from the 3D Document window's context menu, and from the Project Map item's context menu.)

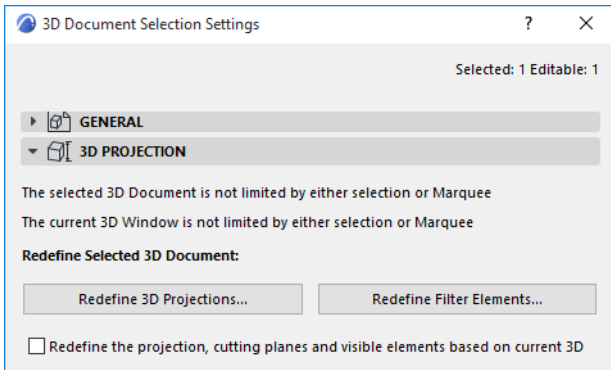
Note: There is no Drawing-type 3D Document.

3D Document 3D Projection Panel

This panel is only available for a selected 3D Document that is based on the 3D window.

This panel provides information on:

- the 3D window *source* of the selected 3D Document (is it limited by selection/marquee, or not?); and
- the *current* 3D window (is it limited by selection/marquee, or not?)

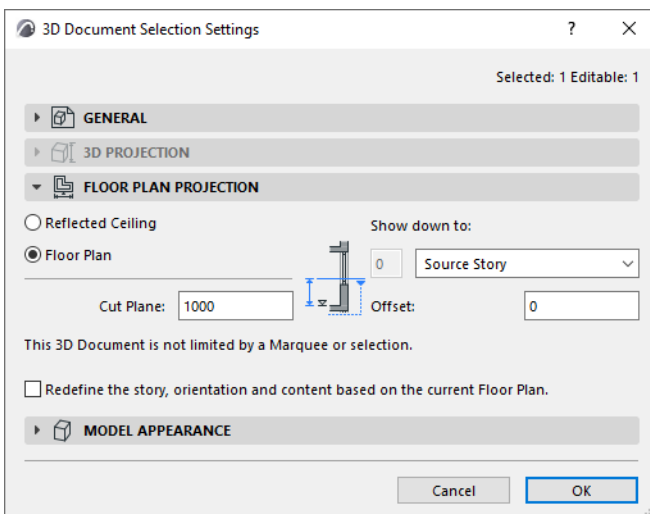


Use the next three controls to redefine the selected 3D Document:

See [Redefine the 3D Document](#).

3D Document Floor Plan Projection Panel

This panel contains options for 3D Documents based on a Floor Plan Projection.



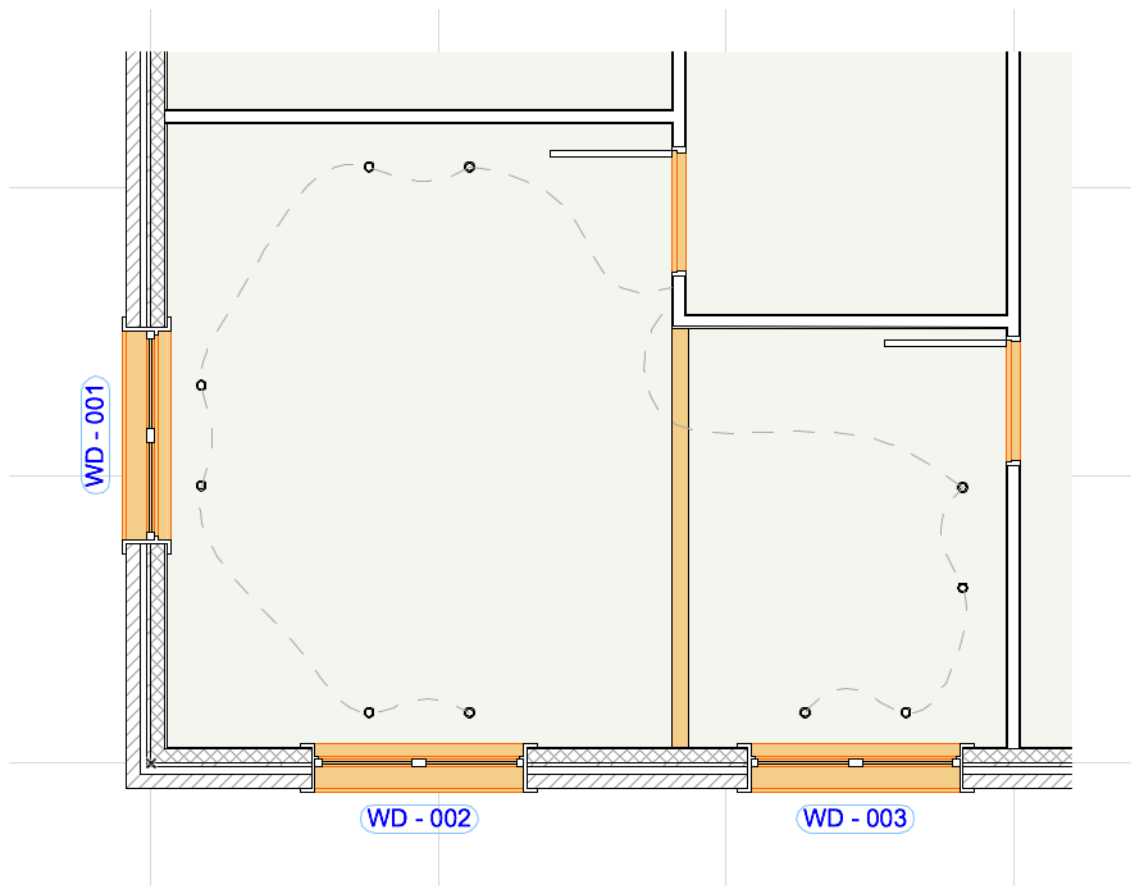
Source View: Shows the name of this 3D Document's Source Floor Plan view.

Direction: Choose an option to define which direction to project the view from the cut plane:

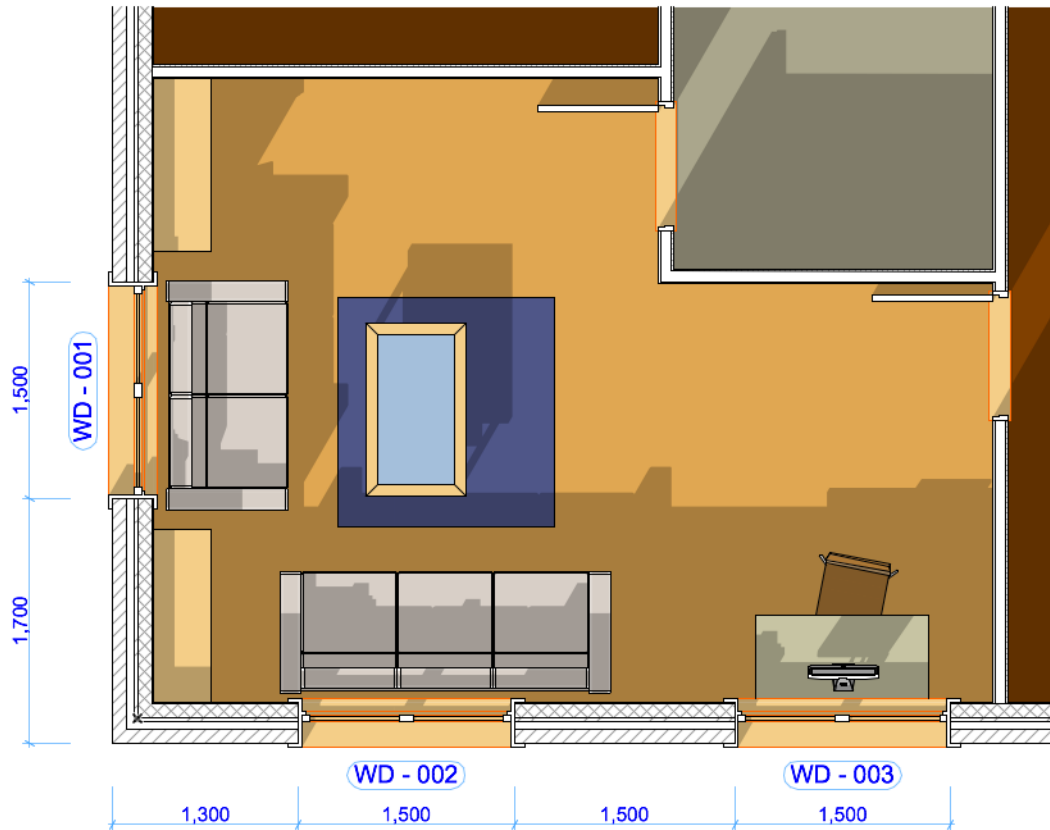
- **Reflected Ceiling:** "Upward" projection from the cut plane. Will display the mirror image of the overhead part of the model.

Define the extent of the projection: for example, **show up** to 1 Story above.

Optionally, define a positive or negative **offset** from the projection's story limit.



- **Floor Plan:** “Downward” projection from the cut plane (down to the floor): for example, **show down** to 1 Story down.
Optionally, define a positive (upwards) or negative (downwards) **offset** from the projection’s story limit.

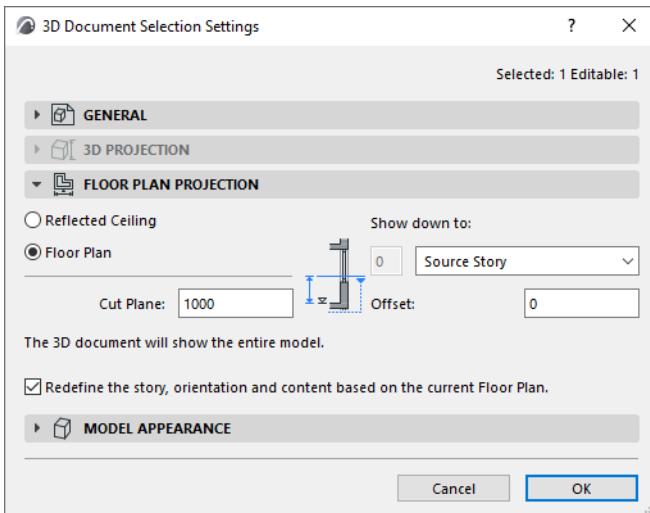


Cut Plane Height: Define the height of the cut plane from which to create the Floor Plan Projection. Cut Plane Height is measured from the Floor Plan story that is the source of the 3D Document.

Selection/Marquee Information: The dialog box provides information on the Floor Plan *source* of the selected 3D Floor Plan Document. It indicates whether a marquee or selection was present on the Floor Plan at the time the 3D Document was generated: this limited the content of the resulting 3D Document.

Note: The Floor Plan source window is not necessarily the same as the currently open Floor Plan window.

If you have checked the **Redefine** box below, you are given information on whether the resulting, redefined 3D Document will be cropped, or whether it will show the entire model.



3D Document Model Appearance Panel

[See Model Appearance \(Section-Type Views\).](#)

Important: These settings apply to the 3D Document and are independent of its source window.

Hidden Edges

Check to display the edges of elements that are otherwise hidden in this view.

Set Line Type and Line pen for the Hidden Edges.

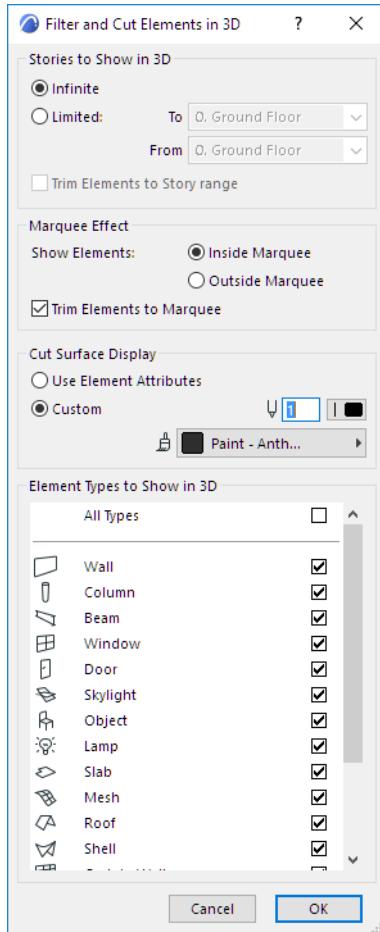
Show on Cut Plane: Controls the visibility of hidden lines that would fall in line with the cut plane or the display range boundaries.

Element Contours at 3D Document Boundary (for 3D Document based on the Floor Plan)

Check to add a contour line where elements are cut by the 3D Document boundary.

Optionally override the line type and color.

Filter and Cut Elements in 3D Dialog Box



Access this dialog box from:

- View > Elements in 3D View > Filter and Cut Elements in 3D
- View Settings (3D Only panel)
- 3D Visualization toolbar's button to open this dialog box.



- 3D Document Settings to redefine filtering criteria for a 3D Document.

Note: You may wish to display 3D Zones as solid bodies in shading mode, and all other elements types in wireframe mode. To achieve this effect (a default setting in earlier Archicad versions), apply the predefined "Show 3D Zones as Solid" Layer Combination.

See [3D Document Settings](#).

Stories to Show in 3D: Define the stories of the project to be displayed in the 3D window:

- **Infinite:** Show all elements.
- **Limited:** Specify a continuous range of stories (from/to stories).
- **Trim Elements to Story range:** Check this box to clip the model at the story range boundaries. If this box is not checked, then any element that falls at least partially inside the story range will be displayed.

Marquee Effect: If you have used the Marquee tool to create a selection area prior to generating the model, use these options to define which part of the project will be shown in the 3D window:

- **Inside Marquee:** Elements inside the Marquee will be displayed.
- **Outside Marquee:** Elements outside the Marquee will be displayed. This option allows you, for example, to exclude internal elements of a house when you need only external views of the 3D model. The Marquee can affect multiple stories, not just the current story.

If there is a Marquee selection made in the Floor Plan, the settings of the Filter Elements in 3D dialog box are observed, but the caption of the 3D window includes the Marquee mention and the range of included stories.

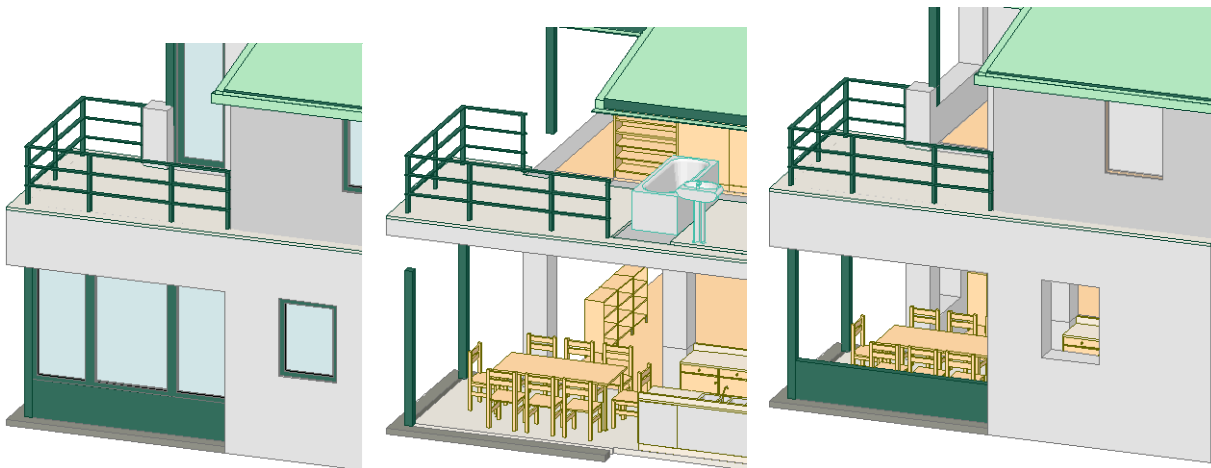
Cut Surface Display: Use these controls to define the surface and pens for the interior surfaces and edges of 3D Cutaways.

- **Use Element Attributes:** Click this button to use existing Surfaces & Pencolors/Penweights for highlighting the edges and surfaces of elements cut in 3D Sections. The cut surfaces of construction elements will have the same colors/surfaces as the uncut edges (as defined in their respective settings dialog boxes).
- **Custom:** Click this button to choose Custom Surface & Pencolor/Penweight for highlighting the edges and surfaces of elements cut in 3D Sections.

Surface pop-up palette: Click this pop-up palette to select a custom Surface for exposed interior surfaces of solid elements.

Cut surfaces of Library Parts will be painted according to the surface set for the Library Part. If the Library Part has internal surface descriptions, the first one in the script will define the surface for cut surfaces. The 3D Cutting Planes feature does not show the various layers of composite elements. To obtain sections showing these (if required for working drawings) use the Section/Elevation/IE tools.

The **Element Types to Show in 3D** checkboxes allow you to turn the 3D Window display of construction elements on and off. With Doors and Windows switched off, openings are interpreted as empty holes. Checking the **All Types** box selects all element types.



3D Styles

A 3D Style is a collection of settings (3D Engine, shading, background, etc.) that determine model display in the 3D Window.

The 3D Style is saved with View Settings. Every time you edit a 3D Style, all views using that 3D Style are updated instantly.

Note: To set 3D Window Size, use the separate command from 3D window-related menus (Tab Bar, or context menu of 3D window or tab).

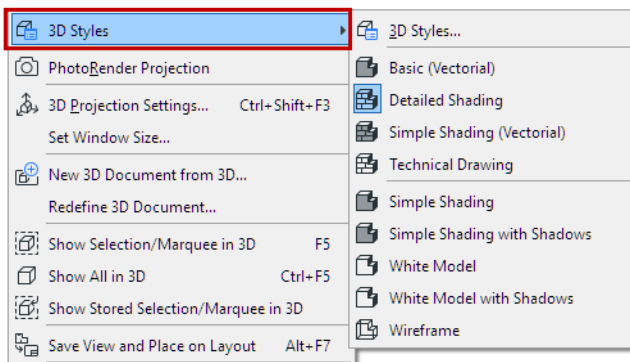
[See 3D Window Size.](#)

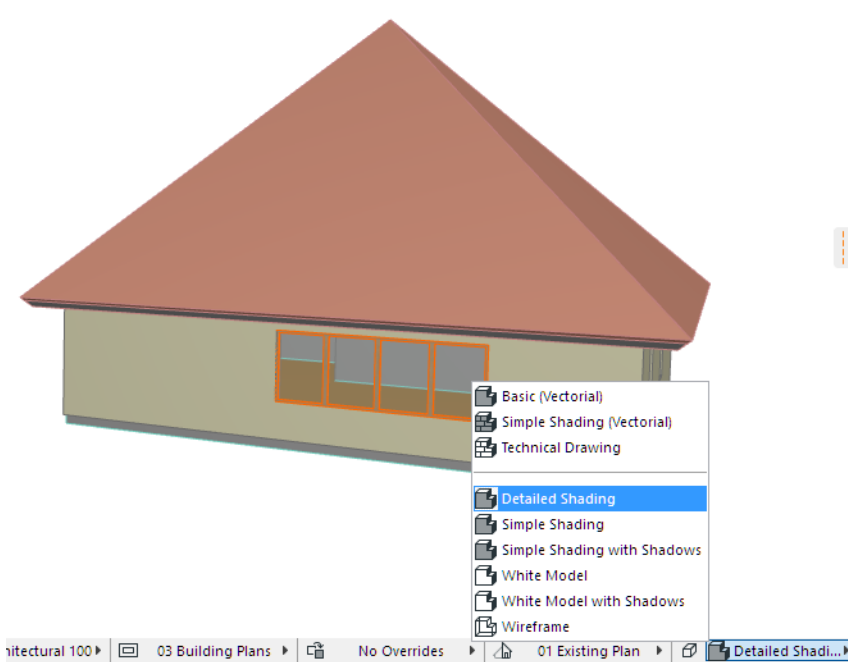
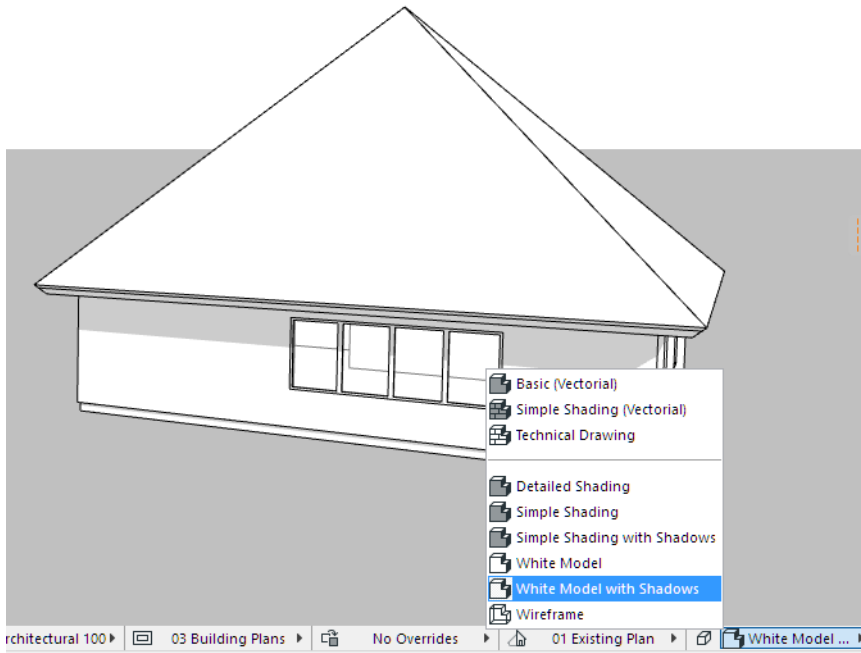
Apply 3D Style

To switch the 3D style of a 3D window, use the 3D Styles pop-up from:

- context menu of the 3D window
- Quick Options Bar or Quick Options Palette
- 3D Window Tab in the Tab Bar

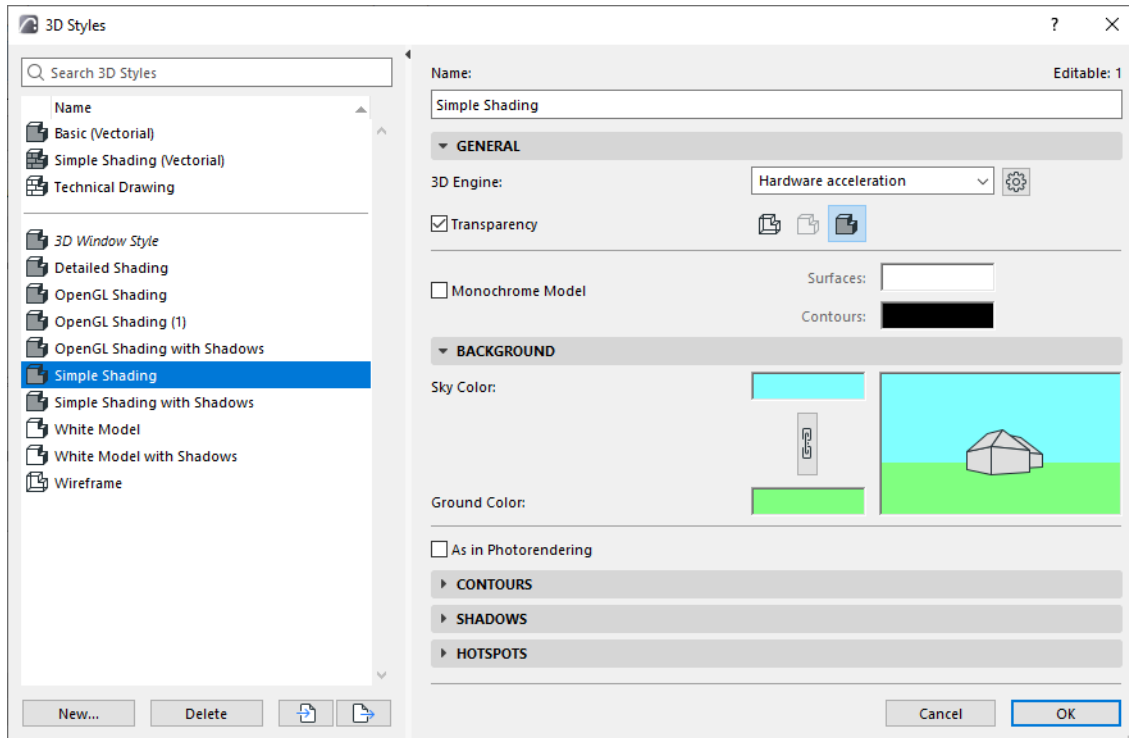
In the pop-up list, a separator divides the styles based on their 3D engine (Vectorial or Hardware acceleration).





Define or Modify 3D Styles

Use **View > 3D View Options > 3D Styles** to open this dialog box.



The project's 3D Styles are listed on the left.

- Select any Style to edit its settings using the controls on the right.
- If needed, multi-select Styles to edit their settings simultaneously.
- Click **New** to create and name a new 3D Style.
- Use **Import** and **Export** to transfer 3D Styles (XML files) between Archicad projects.

General Options

3D Engine

Choose a 3D engine to generate the 3D window:

- **Hardware acceleration:** Can display texture images (from Surfaces) and takes advantage of high-performance Hardware acceleration graphics cards. This is the default engine and recommended if your video display card supports Hardware acceleration. It is optimized for fast, smooth navigation and useful in a live presentation environment. Copy/paste or export will result in bitmap images.

Some effects, for example vectorial 3D hatching, are *not* available with Hardware acceleration. With Hardware acceleration, you cannot save the 3D window contents as a vectorial 2D drawing.

Click the Settings button at right to set additional options with Hardware acceleration.

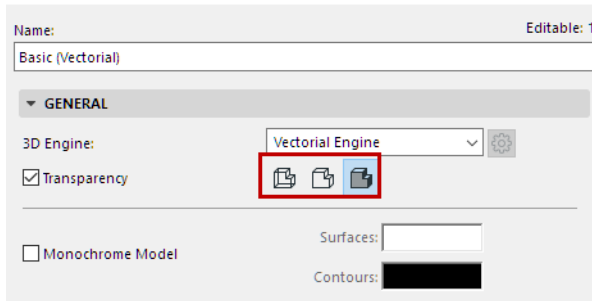
[See Hardware Acceleration Options.](#)

- **Vectorial 3D engine:** Produces a non-photorealistic 3D view, without texture images. Shows vectorial patterns (from Surfaces) and can be printed at any scale. You can save the contents of the 3D window as a vectorial 2D drawing. Copy/paste or print to PDF will result in 2D primitives such as lines and arcs. Usually results in slower navigation of the model on most machines.

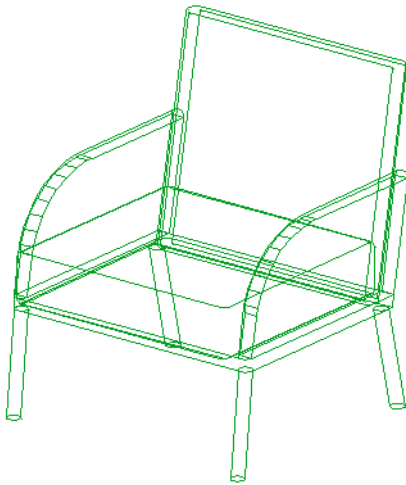
3D View Mode

Choose Wireframe, Hidden Line or Shading for this 3D Style.

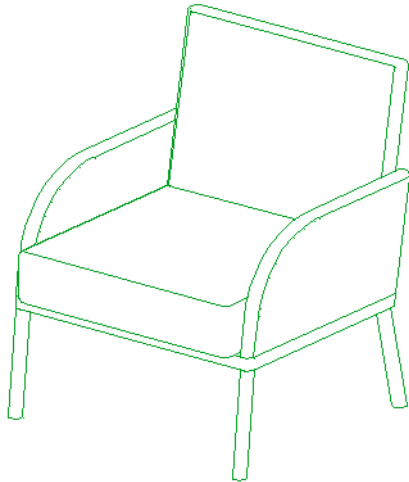
Note: When using the Hardware acceleration engine, only the Wireframe and Shading modes are available.



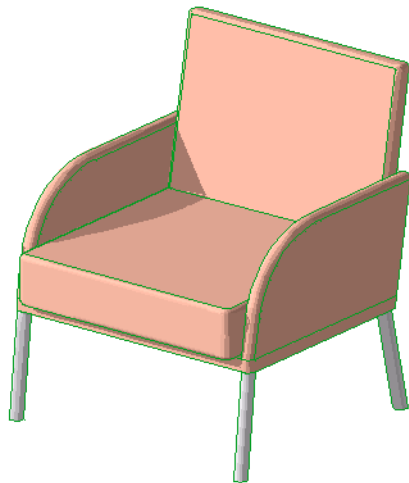
Wireframe mode shows all the edges and lines of your project. Note that if you have many elements one behind the other, this type of view can be hard to interpret.



In **Hidden Line** mode (available with Vectorial Engine only), lines which are blocked by solid objects are removed. This type of view is the best choice for easy interpretation, if you do not want a shaded view. It is also suitable for hard copy output on a printer. Hidden Lines can be saved in many file formats, either as 2D or 3D files.



In **Shading** mode, the 3D model appears with all of its visible surfaces shaded according to the light direction set in the 3D Projection Settings dialog box.



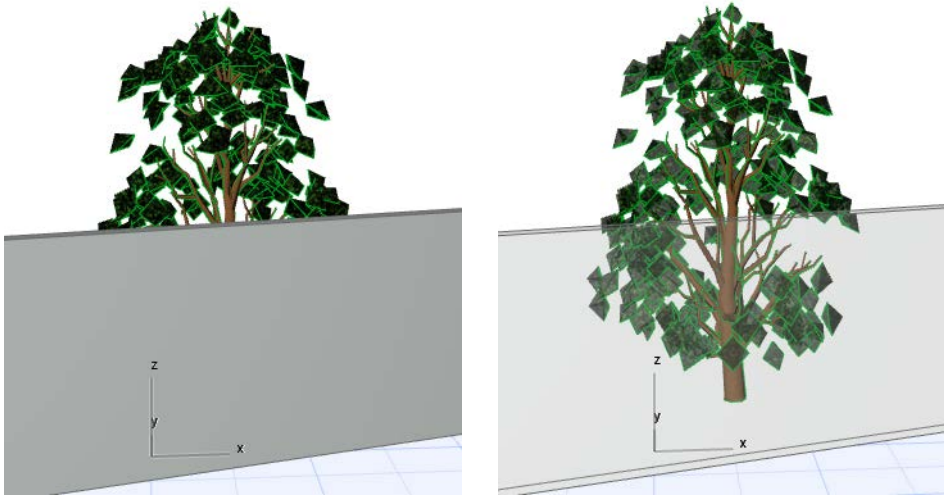
Shading is recommended for:

- Fast on-screen feedback
- Easy checking of surface colors
- Any presentation for which photorealistic quality is not a requirement

Note: If you have defined a Layer as Wireframe, its elements will always appear in Wireframe mode, even if you have chosen another 3D view mode here.

Transparency

Check this to produce transparent views of elements that have the appropriate Surfaces setting when generating a shaded view. Below, the same glass wall appears with Transparency Off and On.



When you save a view, the **Transparency** setting will be saved with the view.

Note: When used with the Vectorial Engine, Transparency is not effective with Postscript printers.

Monochrome Model

Check **Monochrome Model**, then set uniform colors for all of the Surfaces and Contours.

Otherwise, the model's surface colors depend on the elements' Building Materials.

[Watch the Video](#)

Background

Consult the Preview to see the effects of each setting:

Click in the **Sky Color** and **Ground Color** fields to choose background colors for the 3D Window.

Click the chain icon for identical colors in both.

- **As in PhotoRendering:** Check this to use the background specified in PhotoRendering Settings.

[See *Cineware Basic Settings: Background Panel*](#).

Contours

Consult the Preview to see the effects of each setting:

Show Contours

Check this to make the contours of model elements visible in Shaded images. Use the pop-up at right to define contour quality.

- The **Best** option produces a perfect result, but it may be rather slow for large and complex 3D models.
- The **Draft** option produces a fast result, but it doesn't eliminate all the lines that the Best method would.

Use the sliders to define line weights for

- **Contours.** Contour settings are applied when all its attached polygons are visible in the view.
- **Silhouettes** (Vectorial Engine only). Silhouette edge settings are shown when only a single attached polygon is visible (the rest are hidden from the camera).

Vectorial Hatching (Vectorial Engine only)

Display Vectorial Hatching patterns in Hidden Line and Shaded images, provided that you have selected a **3D Hatching** in the Surfaces dialog box.

For more information, see [Surfaces](#).

Note: Vectorial 3D Hatching is also available for Section/Elevation/IE Windows and for 3D Documents, but that setting has to be made individually in each Section/Elevation/IE and 3D Document Settings dialog box.

Shadow Contours (Vectorial Engine only)

Available if Sun Shadows are turned on (see below).

Shadows

Sun Shadows

Check this box to enable Archicad's shadow-casting capability.

With the Vectorial Engine, the following additional Sun Shadow controls are available:

- **Cast Shadow Intensity:** Define using the slider.
- **Shadow Contours:** Check to display shadow contours.
- Use the pop-up to define where Sun Shadows are generated:
 - on all surfaces, or
 - on a single level only (that is, a horizontal plane). This option is useful for shadow studies in top view. If you choose this option, enter a value here to specify the absolute elevation of the horizontal plane on which the Vectorial Shadows will be generated.
- **Shade surfaces using Sun:** If this is turned off, all identical surfaces are draw with the same color, regardless of orientation.

GDL Object Hotspots

- **Enable 2D Hotspots:** Enable Hotspots defined for 2D editing. (This will give you more editing points in the 3D Window.)
- **Show 3D Hotspots only**

Note: If the GDL Object has no 3D hotspots at all, the 2D hotspots will be always displayed, as well as their copies projected to the zzyzx height value, provided that such a parameter has been defined for the object.

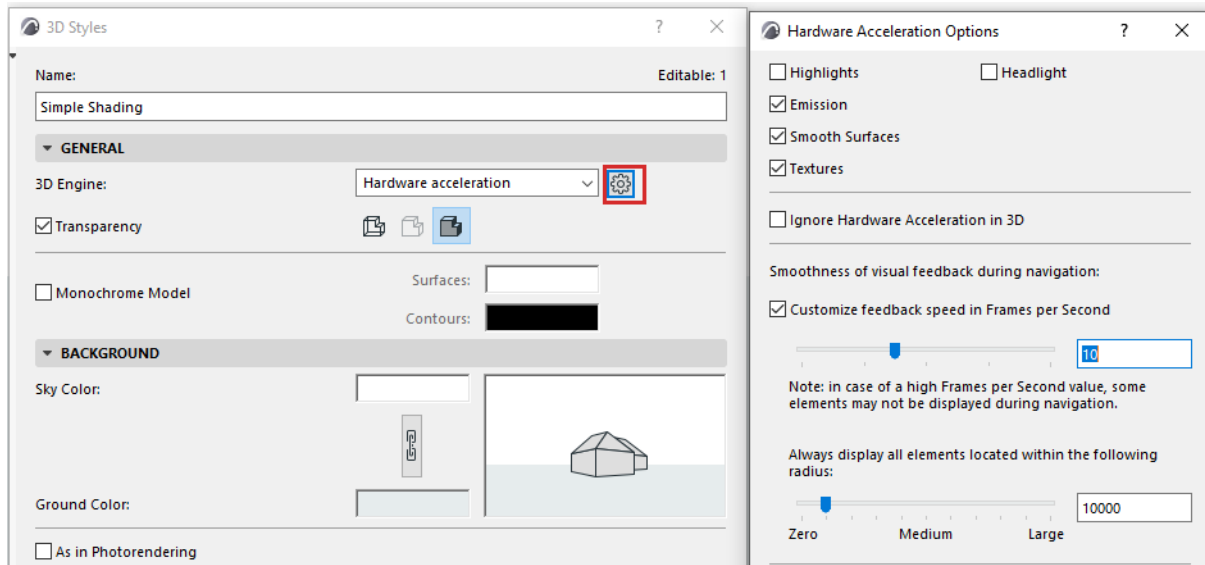
[Watch the Video](#)

[Watch the Video](#)

[Watch the Video](#)

Hardware Acceleration Options

This dialog box is available from the Settings button in **3D Styles** if you have chosen Hardware acceleration as your 3D engine.



Check the boxes of the effects you want to use when generating 3D images with the Hardware acceleration Engine:

Headlight: Controls the light coming from the direction of the camera (on/off). Headlight On provides extra light and associated shadow effects in the model. Turn Off to make flat surfaces look flatter.

Highlights: Display specular reflections as highlights.

Emission: Use if you have chosen or created light emitting Surfaces and want this quality to be displayed.

Smooth Surfaces: Use to control the degree of smoothing. Curved surfaces, which are normally approximated by flat planes, are represented by more lifelike surfaces when this option is on.

Textures: Display bitmapped pictures on the corresponding surfaces.

Ignore hardware acceleration in 3D: Archicad will bypass the display card's Hardware acceleration support and use the system-level software emulation of Hardware acceleration, available both on Windows and Mac.

Note: Software emulation is a slower method. Use it if you are experiencing difficulties with Hardware acceleration display.

Smoothness of Visual Feedback during Navigation

Customize feedback speed in Frames per Second: If you check this box, the slider below is activated. Use the slider to set feedback speed in frames per second.

Note: In case of a high Frames per Second value, some elements may not be displayed during navigation. See below.

Always display all elements located within the following radius: This slider fine-tunes your feedback speed preference: regardless of the effect of the Frames per Second set above (which may result in some elements not being shown during navigation), you can set a minimum radius here, within which all elements will be shown during navigation. (Set the radius either numerically or using the slider.)

Automatic Correction for Optimized Visual Feedback

Archicad uses an algorithm to maintain feedback speed when navigating in complex models using Hardware acceleration. The algorithm may cause certain “secondary” elements to temporarily disappear:

- “Secondary” elements are displayed based on their distance from the viewcone. Close-in elements are prioritized, and those furthest away are – if necessary – the first to be dropped.
- “Primary” elements never disappear during navigation, regardless of FPS. Key elements for finding your way are always displayed.

This algorithm applies ONLY if 3D navigation falls below a set Frames per Second (FPS) rate, as defined by the user, and only until FPS reaches an acceptable threshold.

[Read More.](#)

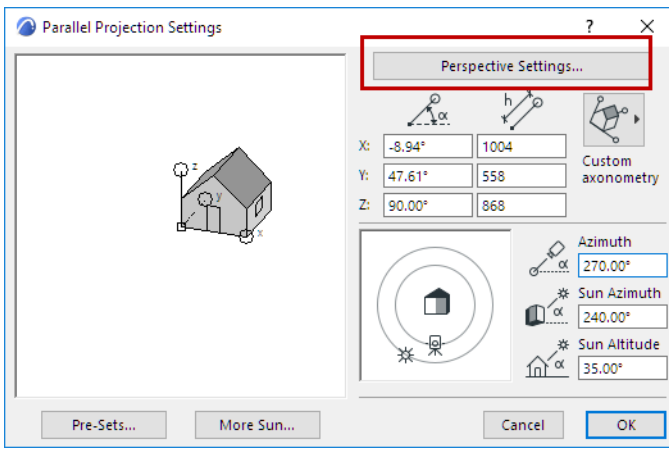
3D Projection Settings

Use the **View > 3D View Options > 3D Projection Settings** command or the **3D Visualization toolbar's** button to open the two modes of this dialog box.



For general information, see [3D Projections](#).

The name and contents of the dialog box depend on the projection type: **Parallel Projection Settings** or **Perspective Settings**. You can easily switch from one settings dialog to the other with the button at the top right corner.



The projections that you define in the 3D Projection Settings dialog boxes remain valid only until the next time you open the dialog box and modify them.

Related Topics:

[Parallel Projection Settings](#)

[Perspective Projection Settings](#)

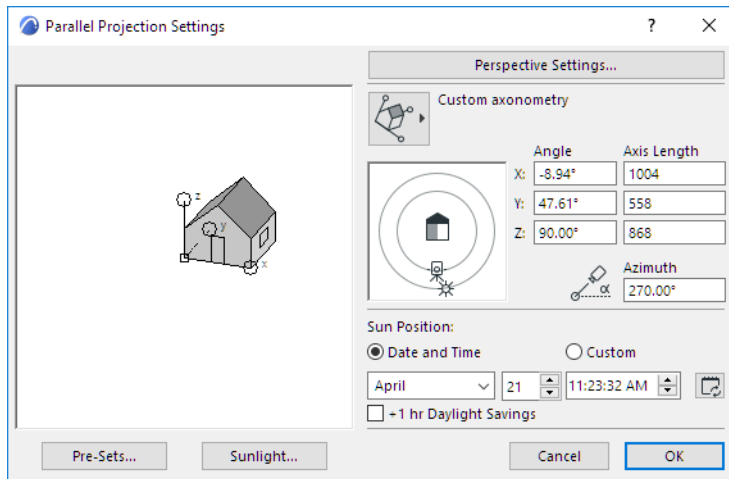
[Location Settings](#)

[Sun Position](#)

[Sunlight Settings](#)

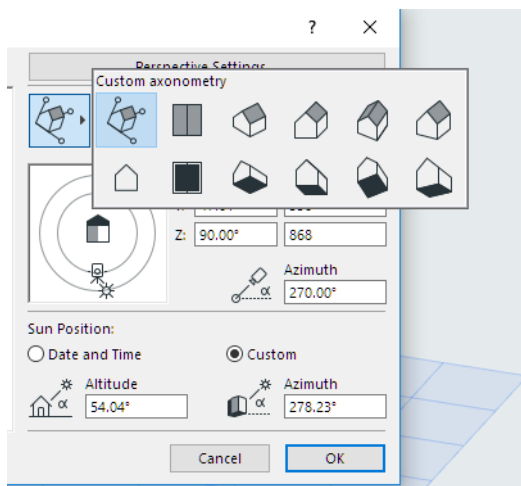
Parallel Projection Settings

Use the **View > 3D View Options > 3D Projection Settings** command or the **3D Visualization toolbar's** button to open this dialog box.



Use the controls in this dialog box to set up 3D views as parallel projections.

Click this pop-up button to select from 12 preset Projection Types, including predefined axonometrics, side, top and bottom views, and your own custom-defined axonometry:

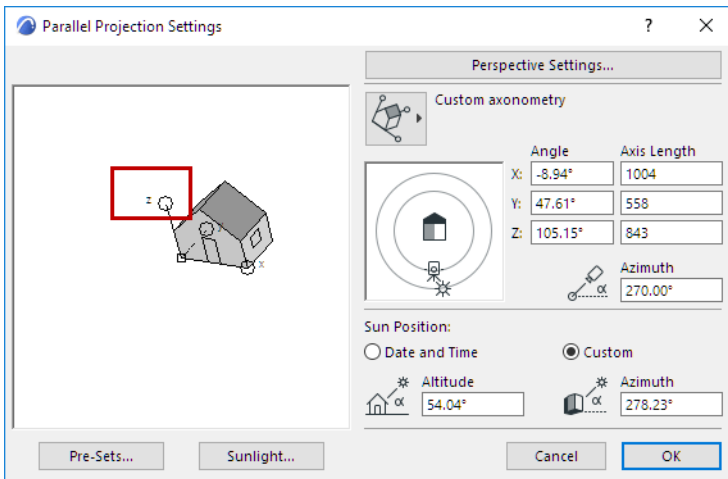


For each Projection type, the editable boxes show the defining angles and scaling ratios of the X, Y and Z coordinate axes.

Projection Preview

This preview window shows the position of the coordinate axes for the current projection type. Click or drag to reposition the axes, or enter values for the angles and ratios of the three coordinate axes.

Use the Shift key to lock the angle while dragging. (If you only want to rescale, it is easier to enter a value in the edit box.)

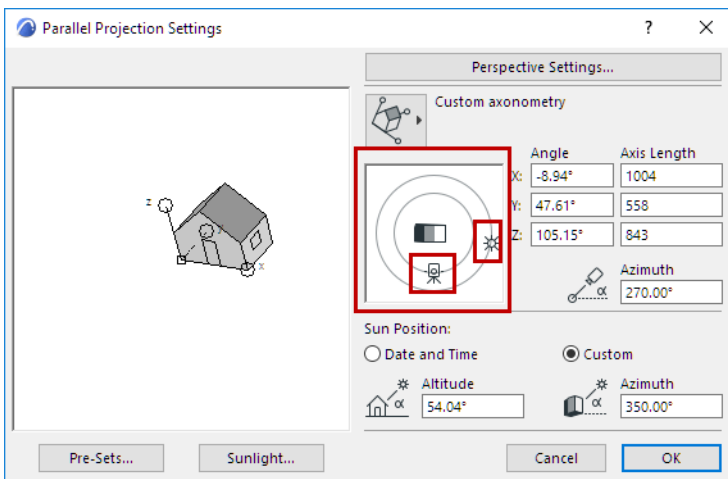


Set Sun and Camera Positions Graphically

The dial shows Camera Azimuth and Sun Azimuth with respect to the target.

Click the **Sun** or **Camera** icon and drag it to the desired location. The effect of your changes is shown by the rotation of the house in the preview area and by the values in the edit boxes (Angle, Axis Length, Azimuths).

Note that the Sun position and Sun Azimuth can be overridden by the Sun Position values given below.



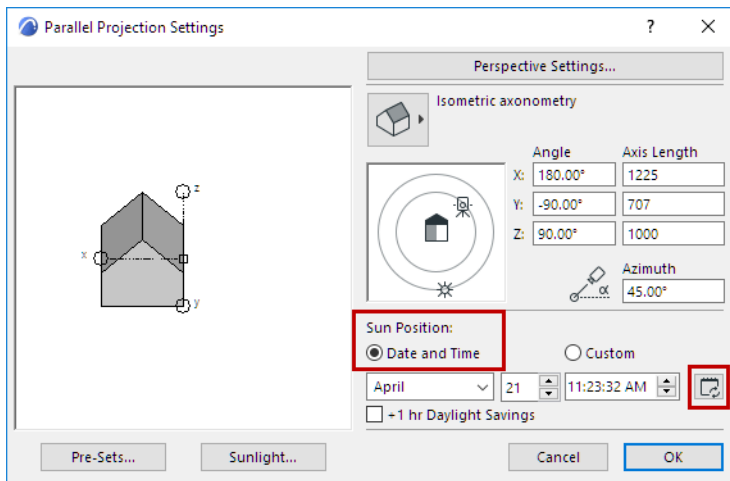
Sun Position

Sun position values can be defined either automatically (based on a given Date and Time at the defined Location Settings), or with **Custom** values.

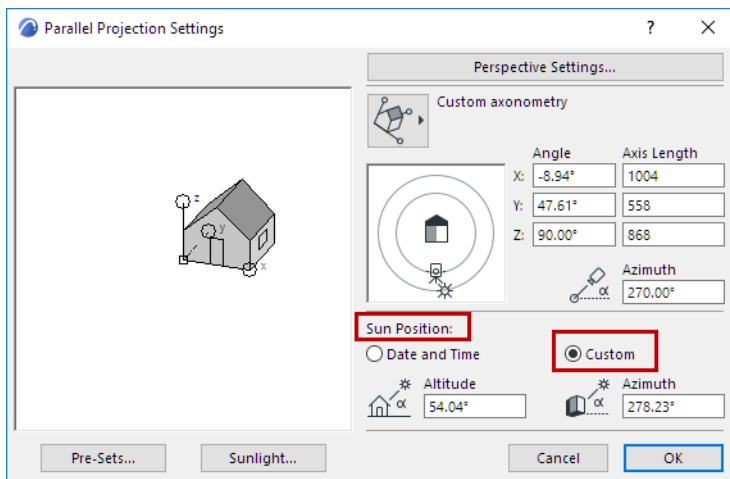
- **Date and Time:** Enter a date and time. In combination with the Project Location, this data will automatically define the sun position, which in turn is used to calculate shadows in shaded or rendered images.

Note: Based on these data, Archicad uses the Solar Position Algorithm from the National Renewable Energy Laboratory (USA) to calculate the solar zenith and azimuth angles.

- Use today's date: Click the Current Date and Time icon to apply the current date (as defined by your computer's operating system).



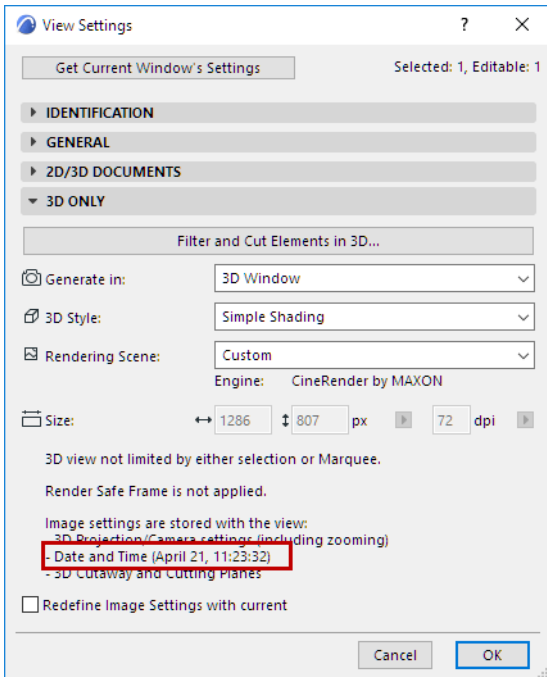
- **+1 hr Daylight:** Check this box to adjust the hour forward for Daylight Savings Time.
- **Custom:** Choose this, then enter the values for Sun Altitude and Azimuth manually (or adjust the sun icon by hand in the preview above).



Note: The **Sun Altitude** is measured in degrees, above the Horizon. A negative Altitude angle indicates that the Sun is below the horizon (night). This may produce completely black images.

Date and Time in View Settings

The Date and Time settings are stored in the View Settings of saved 3D perspective and axonometric views:



Pre-Sets: Button: Opens a dialog where you can specify options for and insert frames in Fly-Through Paths.

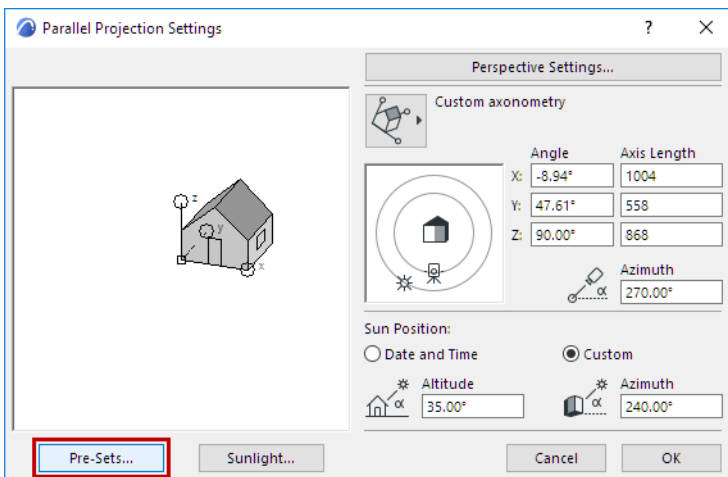
For details, see [Pre-Set Projections](#) below.

Sunlight Button: Opens a dialog to define Sunlight-related settings.

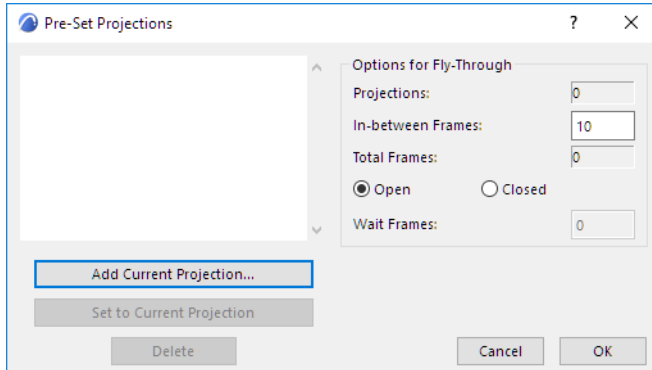
See [Sunlight Settings](#).

Pre-Set Projections

Open this dialog from Parallel Projection Settings.



Here, you can store and retrieve projection settings.



You can also use this sequence of axonometric views for a future animation. Animations based on axonometric views are not fly-throughs in the same sense as perspectives, even though they are created with the same command. Using in-between frames, you will get an animated metamorphosis from one projection to the other.

- The **Add Current Projection** button places the current view on the list. The views on the list will be keyframes in the Fly-Through. You can also name each keyframe in the dialog box that appears after pushing this button.
- If no projection is selected, **New** will be added to the bottom of the list. If a projection is selected, the new projection will be inserted before the selected one.
- The **Set to Current Projection** button changes the highlighted keyframe in the list to the one you set prior to opening the **Pre-Sets** dialog box.
- The **Delete** button clears a keyframe from the list.
- Under **Options for Fly-Through** in the right section of the dialog box you can also set the number of in-between frames that Archicad will create by interpolating between every parameter (e.g., axis scaling ratio, camera angle, etc.) of consecutive keyframes.

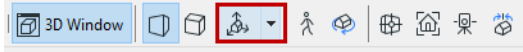
For more information, see [Fly-Through](#).

- The **Open** and **Closed** radio buttons control whether the animation will loop continuously from the last defined keyframe back to the first keyframe during the Fly-Through.
- **Wait frames:** This number refers to the number of still frames (time units) during which the fly-through motion will remain frozen at the given camera before moving on to the next one.

Enter the number of still frames to be included in the Fly-Through Path.

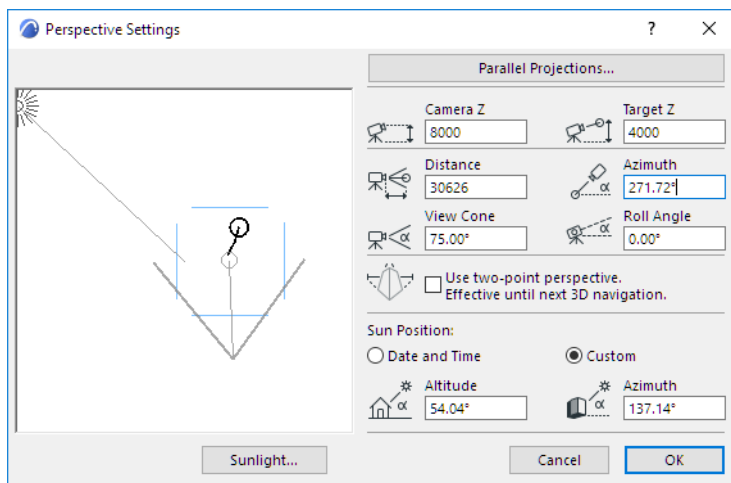
Perspective Projection Settings

Use the **View > 3D View Options > 3D Projection Settings** command or the **3D Visualization toolbar's** button to open this dialog box.



Use this dialog box to set all the relevant parameters for perspective 3D views, such as viewpoint location and elevation, target location and elevation, width of the viewing cone, and sun position.

Note: Cameras placed in the Floor Plan window also define perspective views, but they are configured in the Camera Tool Settings dialog box, not in this dialog box. To apply the camera-defined view to the 3D model, select the camera and activate the 3D Window.



The preview area on the left side of the dialog box shows your Floor Plan as it currently appears in the window. To see another part of the Project, leave the dialog box, choose another detail using the Navigator or zooming and panning operations, then open it again.

The solid line in the preview area represents the line of vision between the viewpoint and the target point. You can click and drag both the target point and the viewpoint independently of one another. Click anywhere in the preview area and the line of vision will be rotated around the target point to your click position.

The angle defining the **View Cone** is displayed in the preview area and provides reliable information if the line of vision is horizontal. If the line of vision is looking up or down, the angle can be used only for estimating the resulting view.

After zooms, pans, or numeric distance settings, the viewpoint and/or target point may occasionally disappear from the preview area. To retrieve them, simply Shift-click where you want the viewpoint and Alt/Cmd-click where you want the target point to be in the Window. They will immediately move to the points where you clicked.

You can also specify the following values numerically. Effects of these settings will be displayed graphically in the preview area.

- **Camera Z:** Enter a value here to specify Camera height from Project Zero. (In this case, the position of the target will remain unchanged.)
- **Target Z:** Enter a value here to specify Target height from Project Zero.
- **Distance:** Enter a value here to specify the horizontal distance between Camera and Target.

- **Azimuth:** Enter a value here to specify Camera Azimuth to Target.

Point Perspectives

- **Use Two-Point Perspective:** Check this box to view the 3D model from a two-point rather than three-point perspective, while maintaining the original position and viewing angle. This perspective remains until you navigate away.

As a result, all 3D vertical edges become vertical in perspective projection. This emulates the vertical correction function of shift lenses in SLR cameras.

- To get a top view in perspective, set the viewpoint elevation higher than that of the target point elevation, drag the target to the desired location on the plan, then set the distance to zero by dragging the viewpoint until the numeric field displays a minimal value near zero. (You cannot type zero in the **Distance** field.)

The elevation of the viewpoint and the target point, as well as the **View Cone** and **Roll Angle** of the Camera, must be set numerically by using the remaining edit boxes.

View Cone: Enter a value here to specify the opening angle of the Camera View Cone.

Hint: Wider angles ($a^\circ \Rightarrow 60^\circ$) can fit more objects in the Window, but can also distort the image.

Roll Angle: Enter a value here to specify Camera Roll Angle.

Set Sun Position Graphically

In the preview area, the dashed line with sun icon shows the direction of light for shaded and rendered representations. The sun icon can be dragged around the perimeter of the plan, but it cannot be positioned closer. Its distance is considered to be infinite.

The other controls are identical to those in Parallel Projections.

See [Sun Position and Sunlight Settings](#).

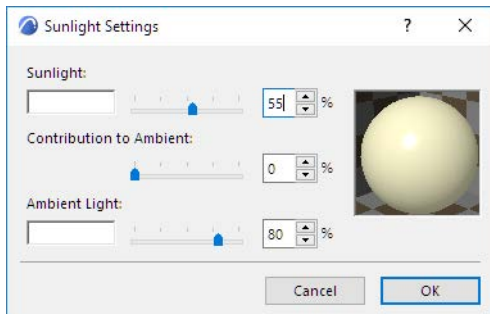
Sunlight Settings

The Sunlight Settings dialog box is accessible from:

- the Camera Tool Settings dialog box, by clicking the **Sunlight...** button
- the **View > 3D View Options > 3D Projection Settings** dialog boxes (Parallel Projections or Perspective Settings), by clicking the **Sunlight...** button.

Sunlight Settings are relevant for:

- PhotoRenderings
- Sun Studies
- 3D Projections (and views created from these)



Light Parameters

Set parameters for each component of the light: **Sunlight** (directional normal) and **Ambient Light**.

- **Sunlight** is the directional light falling on the project.
 - Click inside the color sample box to choose a Sunlight color.
 - Choose a sunlight color intensity between 0 and 100%. This Intensity value multiplies the Brightness component of the Sunlight color.

Note: The Sunlight color and intensity settings are considered by the Cineware Rendering Engine.

 - **Contribution to Ambient:** Choose a percentage for how much of the Sunlight should be represented in non directional, ambient light.
- **Ambient light** is general, non-directional light that contributes to the overall brightness of the image, and can brighten otherwise overshadowed surfaces.

Note: Ambient light parameters affect only the 3D Window and PhotoRenderings created with the Basic Rendering Engine!

 - Color: Click inside the color sample box to choose a color for the Ambient Light.
 - Choose an Ambient light color intensity between 0 and 100%. Vary this value to obtain special effects in renderings.

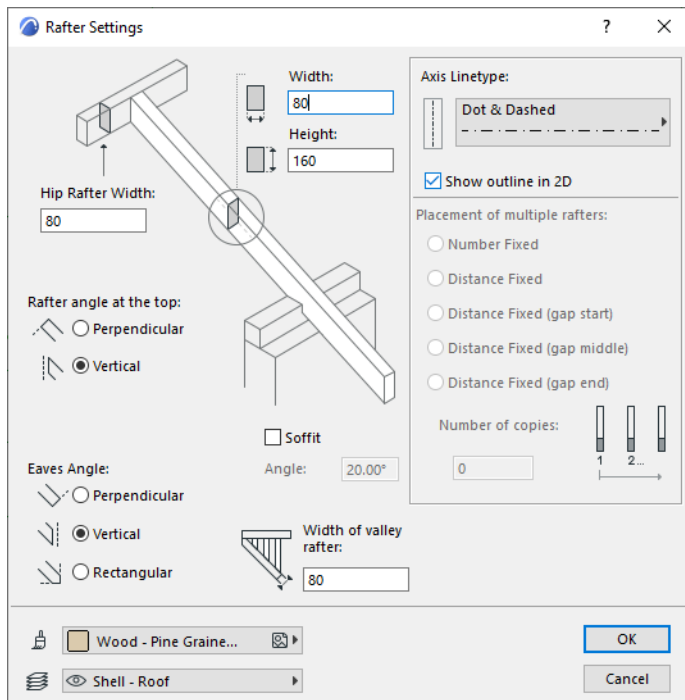
RoofMaker Settings

(Design > Roof Extras > RoofMaker)

Rafter Settings Dialog Box

Select a roof on the floor plan. Choose **Create a Rafter** from the **Design > Roof Extras > RoofMaker** menu (or select the first icon from the RoofMaker toolbox).

The Rafter Settings dialog box appears in which rafter parameters can be set. The preview picture in the dialog box reflects the changes in the settings as you make them, to let you experiment with several possibilities before placing the rafter.



Width: Enter the width of the rafter cross-section here.

Height: Enter the height of the rafter cross-section here.

Hip Rafter Width: Define the width of the hip rafter (the connecting element at the top of the rafter.)

Rafter angle at the top: Choose either radio button (**Perpendicular** or **Vertical**) to define the angle at which the rafter joins the connecting element at its top end.

Eaves Angle: Choose a radio button (**Perpendicular**, **Vertical** or **Rectangular Cut**) to define a how to cut the bottom end of the rafter.

Soffit: Click this box to attach a soffit to the lower end of the rafter. The soffit will have the same cross section as the rafter.

Soffit Angle: Enter a value to set the inclination of the soffit. A zero angle means the soffit is horizontal.

Note: The preview will not reflect the angle you set for the soffit, but the value will correctly displayed on screen and in all outputs.

Surface: Choose a surface for the rafter from the pop-up palette.

Width of Valley Rafter: Enter the width of the valley rafter.

Layer: Choose a layer for the Rafter element.

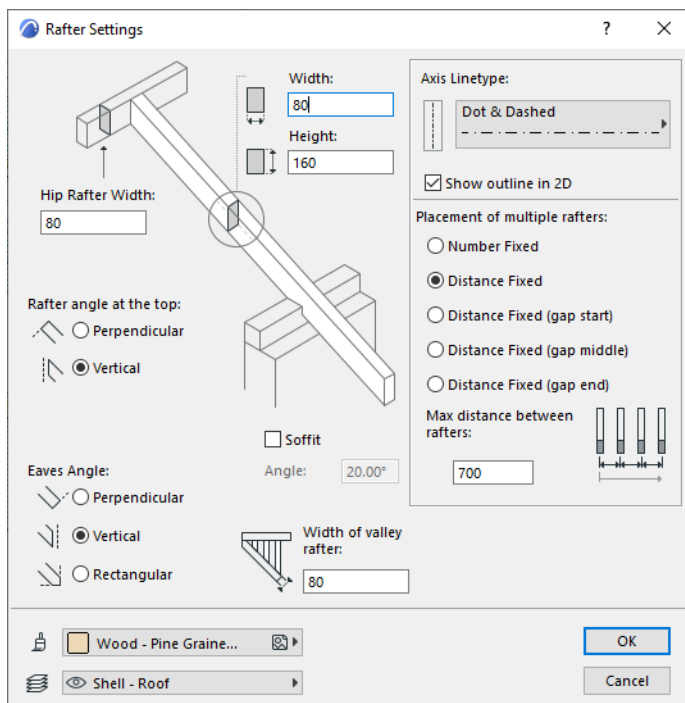
Axis linetype: Choose a linetype for the axis of the rafter.

Show outline in 2D: Check this box to display the full outline of the rafter, including its axis, on the Floor Plan. Although roof plans usually show only the axis of any inclined element (rafters, hip rafters, valley rafters), switching on the outline view temporarily gives you better control of the connections between elements. Clicking OK returns you to the floor plan.

Multiple Rafters Dialog Box

Select a roof on the floor plan. Choose **Create Multiple Rafters** from the **Design > Roof Extras > RoofMaker** menu (or select the appropriate icon from the RoofMaker toolbox).

The controls of this dialog box are identical to those of the Rafter Settings Dialog Box, but the controls related to multiple rafters are also active.



Placement of multiple rafters: The placement method is defined by either a maximal distance between the axes of the rafters (**Distance Fixed**), or by a fixed number of rafters to be placed at an equal distance (**Number Fixed**).

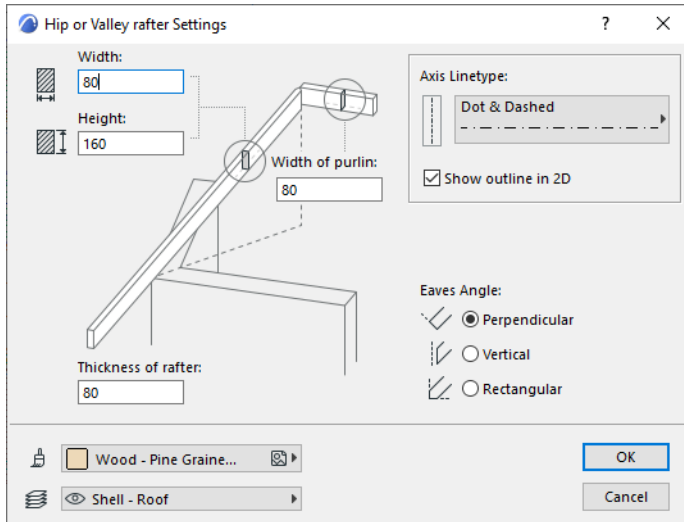
If you choose **Distance Fixed**, enter a value below for **Max. distance between rafters**.

If the user-defined range might not allow the regular placement of the objects. In such a case, use the **Distance Fixed - gap start/middle/end** radio buttons to fine-tune where the gap between rafters will be introduced (either at the start, the middle or the end).

Number of copies: This field is active if you have checked the Number Fixed radio button. Enter the desired number of rafters.

Clicking OK returns you to the floor plan.

Hip or Valley Rafter Settings



Select either one or two roofs on the floor plan. Choose **Create a Hip or Valley Rafter** from the **Design > Roof Extras > RoofMaker** menu (or select the appropriate icon from the RoofMaker toolbox).

Width: Enter the width of the rafter cross-section here.

Height: Enter the height of the rafter cross-section here.

Width of Purlin: Enter the width of the purlin here.

Thickness of Rafter: Enter the rafter thickness here.

Axis linetype: Choose a linetype for the axis of the rafter.

Show outline in 2D: Check this box to display the full outline of the rafter, including its axis, on the Floor Plan. Although roof plans usually show only the axis of any inclined element (rafters, hip rafters, valley rafters), switching on the outline view temporarily gives you better control of the connections between elements.

Eaves Angle: Choose a radio button (**Perpendicular**, **Vertical** or **Rectangular Cut**) to define a how to cut the bottom end of the rafter.

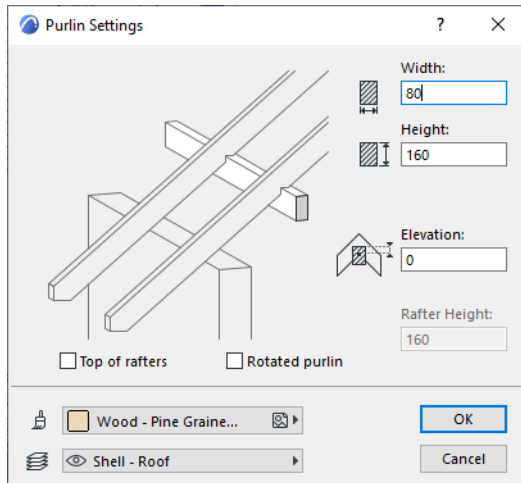
Surface: Choose a surface for the rafter from the pop-up palette.

Enter values for the **Width of purlin** and **Thickness of rafter** fields.

Clicking OK returns you to the floor plan.

Create a Purlin (or Beam)

Select a roof on the floor plan. Choose **Create a Purlin** from the **Design > Roof Extras > RoofMaker** menu (or select the appropriate icon from the RoofMaker toolbox).



Width: Enter the width of the purlin cross-section here.

Height: Enter the height of the purlin cross-section here.

Elevation: Enter an elevation value for the purlin so that it can be higher or lower than the reference plane.

Top of Rafters: Check this box to place the purlin on top of the rafters. If the purlin is placed on top, you must also set the rafter height at which to place the purlin.

Rotated Purlin: Check this box to place the purlin perpendicular to the rafters.

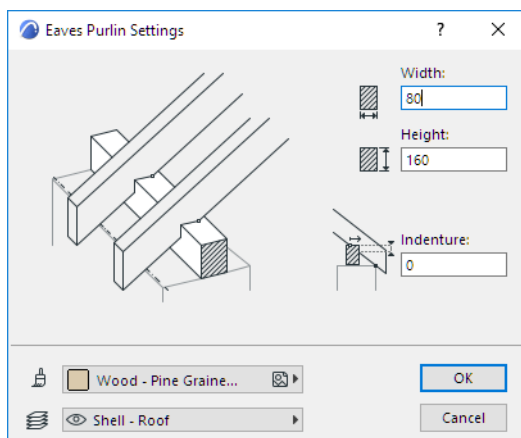
Rafter height: Enter a value for the height of the rafters.

Surface: Choose a surface for the purlin from the pop-up palette.

Clicking OK returns you to the floor plan.

Create an Eaves Purlin (or Plate Beam)

Select a roof on the floor plan. Choose **Create an Eaves Purlin** from the **Design > Roof Extras > RoofMaker** menu (or select the appropriate icon from the RoofMaker toolbox).



Width: Enter the width of the purlin cross-section here.

Height: Enter the height of the purlin cross-section here.

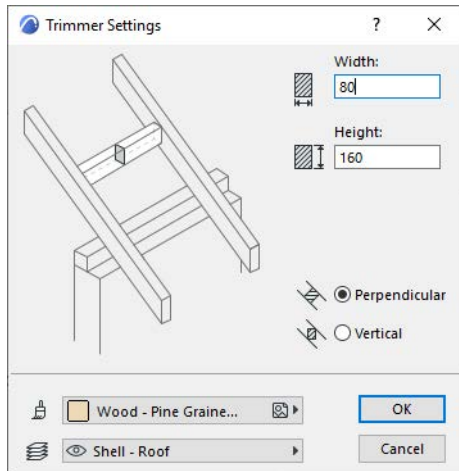
Elevation: Enter an elevation value for the purlin so that it can be higher or lower than the reference plane.

Surface: Choose a surface for the purlin from the pop-up menu.

Clicking OK returns you to the floor plan.

Create a Trimmer (or Blocking)

Select two rafters on the floor plan. Choose **Create a Trimmer** from the **Design > Roof Extras > RoofMaker** menu (or select the appropriate icon from the RoofMaker toolbox).



Width: Enter the width of the trimmer cross-section here.

Height: Enter the height of the trimmer cross-section here.

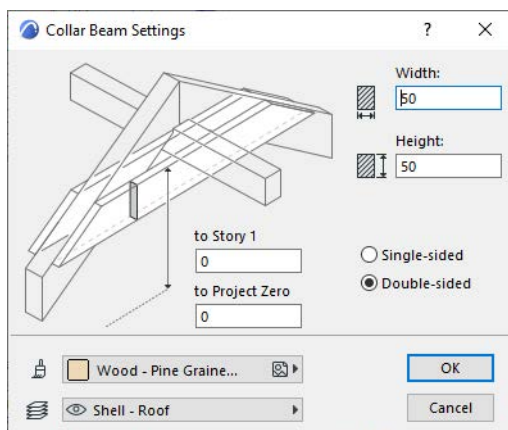
Perpendicular: Check this to place the trimmer perpendicularly to the rafters

Vertical: Check this to place the trimmer vertically.

Surface: Choose a surface for the trimmer from the pop-up menu.

Clicking OK returns you to the floor plan.

Create Collar Beams



Select two rafters on the floor plan. Choose **Create Collar Beams** from the **Design > Roof Extras > RoofMaker** menu (or select the appropriate icon from the RoofMaker toolbox).

Width: Enter the width of the collar beam cross-section here.

Height: Enter the height of the collar beam cross-section here.

to Story 0: Enter a value here for the height of the top of the beam from the current story

to Project Zero: Enter a value here for the height of the top of the beam from Project Zero.

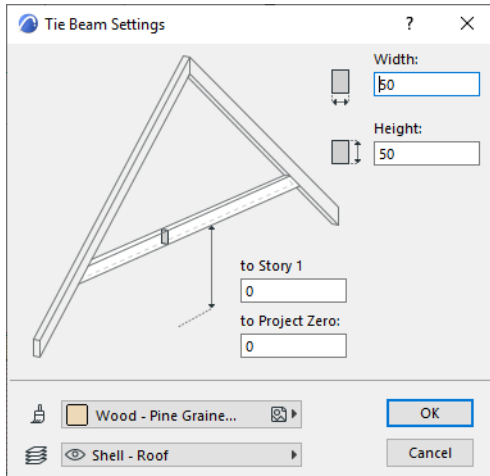
Single-sided: Click this button to create a single-sided beam.

Double-sided: Click this button to create a double-sided beam (on both sides of the rafter).

Surface: Choose a surface for the purlin from the pop-up palette.

Clicking OK returns you to the floor plan.

Create Tie Beams



Select two rafters on the floor plan. Choose **Create Tie Beams** from the **Design > Roof Extras > RoofMaker** menu (or select the appropriate icon from the RoofMaker toolbox).

Width: Enter the width of the tie beam cross-section here.

Height: Enter the height of the tie beam cross-section here.

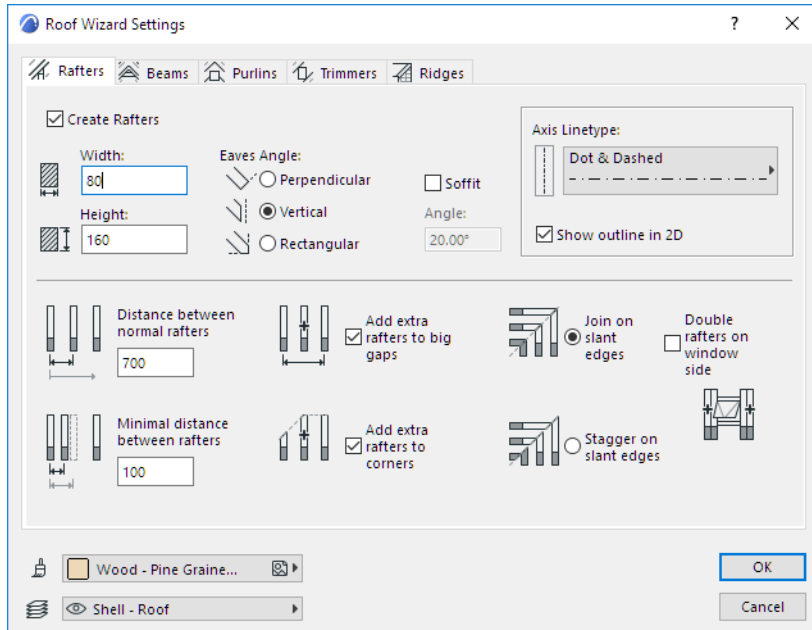
to Story 0: Enter a value here for the height of the bottom of the beam from the current story

to Project Zero: Enter a value here for the height of the bottom of the beam from Project Zero.

Surface: Choose a surface for the purlin from the pop-up menu.

Clicking **OK** returns you to the floor plan.

Roof Wizard Dialog Box



For more information, see [Create Special Roof Objects with RoofMaker](#).

Rafters Tab Page

Create Rafters: Check this to create Rafters and enable related controls on this tab page.

Width: Enter the width of the rafter cross-section here.

Height: Enter the height of the rafter cross-section here.

Eaves Angle: Choose a radio button (**Perpendicular**, **Vertical** or **Rectangular Cut**) to define a how to cut the bottom end of the rafter.

Soffit: Click this box to attach a soffit to the lower end of the rafter. The soffit will have the same cross section as the rafter

Soffit Angle: Enter a value to set the inclination of the soffit. A zero angle means the soffit is horizontal.

Axis linetype: Choose a linetype for the axis of the rafter.

Show outline in 2D: Check this box to display the full outline of the rafter, including its axis, on the Floor Plan. Although roof plans usually show only the axis of any inclined element (rafters, hip rafters, valley rafters), switching on the outline view temporarily gives you better control of the connections between elements.

Distance between normal rafters: Enter a value here to set the standard distance between rafters.

Minimal distance between rafters: Enter a value here to set the minimum distance between rafters.

Use the checkboxes for special rafter placement options:

- Add extra rafters to big gaps
- Add extra rafters to corners

Choose one of the two radio buttons to set how rafters behave at slant edges:

- Join on slant edges
- Stagger on slant edges

Double rafters on window side: Check this box if you want to double the rafters on the side of window openings.

Surface: Choose a surface for the rafter from the pop-up palette.

Clicking **OK** returns you to the floor plan and creates the rafters according to the current settings.

Beams Tab Page

Create Beams: Check this to create beams and enable related controls on this tab page.

Check a radio button for the type of beam needed:

- Collar Beams

Note: With the Roof Wizard, only double collar beams can be created.

- Tie Beam

Width: Enter the width of the beam cross-section here.

Height: Enter the height of the beam cross-section here.

Set the Elevation of the beam.

- **to Story 0:** Enter a value here for the height of the beam from the current story.
- **to Project Zero:** Enter a value here for the height of the beam from the Project Zero.

Note: For collar beams, the elevation value is the top of the beam; for tie beams, the elevation value is the bottom of the beam, as reflected in the preview picture.

Surface: Choose a surface for the rafter from the pop-up palette.

Clicking **OK** returns you to the floor plan and creates the beam(s) according to the current settings.

Purlins Tab Page

- Create Ridge Purlins
- Create Eaves Purlins

Check either or both of these boxes to create ridge purlins (use controls on the left side of the dialog box) and eaves purlins (use controls on the right side).

Width: Enter the width of the purlin cross-section here.

Height: Enter the height of the purlin cross-section here.

A purlin is placed either below the opposing rafters (supporting them), or between the rafters (supported by them).

Purlin between rafters: Check this box to place the ridge purlin between rafters.

Indenture: Check this box to set the depth at which the eaves purlin cuts into the rafter it supports

Surface: Choose a surface for the purlin from the pop-up palette.

Clicking **OK** returns you to the floor plan and places the purlins according to the current settings.

Note: The eaves purlin is always placed on the same level as the pivot line of the corresponding roof surface.

Trimmers Tab Page

Click the **Create Trimmers** checkbox to activate the controls on this tab page.

Width: Enter the width of the rafter cross-section here.

Height: Enter the height of the rafter cross-section here.

Perpendicular: Check this to place the trimmer perpendicularly to the rafters

Vertical: Check this to place the trimmer vertically.

Surface: Choose a surface for the purlins from the pop-up palette.

Clicking **OK** returns you to the floor plan and creates the purlins according to the current settings.

Ridges Tab Page

Create Hip Rafters and Valley Rafters: Check this box to use the controls in this dialog box to create Hip Rafters and Valley Rafters.

Width: Enter the width of the rafter cross-section here.

Height: Enter the height of the rafter cross-section here.

Axis linetype: Choose a linetype for the axis of the rafter.

Show outline in 2D: Check this box to display the full outline of the rafter, including its axis, on the Floor Plan. Although roof plans usually show only the axis of any inclined element (rafters, hip rafters, valley rafters), switching on the outline view temporarily gives you better control of the connections between elements.

Eaves Angle: Choose a radio button (**Perpendicular**, **Vertical** or **Rectangular Cut**) to define a how to cut the bottom end of the rafter.

Drop hip rafter to match rafter's top: Check this box to lower the hip rafter to match rafters' top with its own top. (This may be necessary if the pitch angle of the standard rafters differs from that of the hip/valley rafters.)

Surface: Choose a surface for the rafters from the pop-up palette.

Clicking **OK** returns you to the floor plan creates the rafters according to the current settings.

TrussMaker Settings

(Design > Design Extras > TrussMaker)

For general information, see [TrussMaker](#).

Choose a construction type using the three large buttons on the left of the screen of the TrussMaker settings dialog box:

- **Timber Construction:** click this button to construct wooden trusses.
- **Hollow Section:** click this button to construct rectangular or circular hollow trusses.
- **Rolled Steel Profile:** click this button to construct steel trusses using a variety of profile options.

After setting a construction type, use the controls on the three tab pages to define the truss.

TrussMaker Attributes tab

This tab page defines the appearance of the truss in floor plan, section and 3D views and is identical for all three construction types.

Floor Plan and Model Attributes

Enter a pen number and pen color, line type and surface setting for the line of the truss.

Enter a vertical offset for the truss:

- **to Story:** this value is the elevation of the truss from the current story.
- **to Project Zero:** this value is the elevation of the truss from Project Zero.

Resolution of Arcs: Enter a value to set the segmentation of any curves that you used to define the outline of the truss.

Note: The value you enter in the Resolution of arcs field is relative to the complete circle, not the arc segment.

TrussMaker Truss Profiles tab

Use this tab page to assign distinct parameters (e.g., thickness, diameter or profile) to each part of the truss.

Note: To define the different parts of the truss, you must assign different pencolors when drawing the truss outline on the floor plan.

To select a part to work on, either click its pencolor icon, or corresponding button/field, or its outline in the **Truss Preview**. The thick outline in the preview and a black diamond next to the pen color indicate which part of the truss you are working on. The available parameters depend on the construction type of the truss you are creating.

Width: Enter a value to set the width of the element.

Height: Enter a value to set the height of the element.

Thickness: Enter a value to set the thickness of the element.

Cross Section: Displays the cross section of the selected profile type.

In addition to the above, the **Hollow Section** type of truss has a number of additional settings.

Profiles: Click a radio button to choose either a rectangular or circular profile.

If you choose a circular profile, such truss elements are defined by their diameter and resolution rather than height and width.

Thickness: Enter a value for the thickness of the truss part.

For a **Rolled Steel Profile** only:

Click the **Profiles** and **Positioning** pop-ups to choose a shape and placement option for your Rolled Steel Profile.

Steel Junctions tab

This tab page is only available for the **Rolled Steel Profile** construction type. These controls allow you to use customizable junction plates at the points where the truss elements meet.

Choose a radio button to set whether your truss will include junction plates.

- **Without junction plates:** Check this radio button if you do not want to use junction plates. If you choose not to use junction plates, the corresponding options are grayed in the dialog box.
- **With junction plates:** Check this radio button to use junction plates.

When joining two identical profiles, choose a radio button to set whether your truss will include junction plates:

- Don't use junction plate
- Use junction plate.

Choose a radio button to set the geography of the steel junction profile:

- Use center of gravity
- Use midpoint of bounding box.

In the Junction Plates section, T stands for the cross-section value, L for the length of the junction plate from the intersection point of the profile axis and M for the minimal overlapping length of the profile and the junction plate. Depending on the geometry (the angle between the profiles), either L or M will be used to determine the actual junction plate length - whichever results in a longer junction plate.

T: Enter a value to set the thickness of the plates (cross-section).

L: Enter a value for L (length of junction plate).

M: Enter a value for M (minimal overlapping length between profile and junction plate).

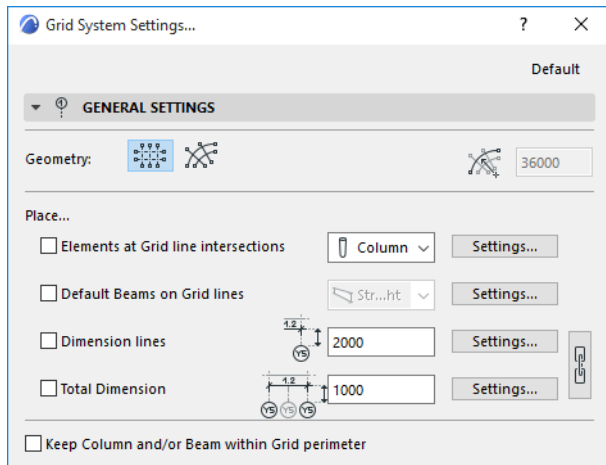
When you have finished customizing your truss, click the **Save** button at the bottom right of the dialog box. You will then be prompted to name the new truss object and place it in your active library.

Grid System Settings

For general information, see [The Grid Tool](#).

This dialog box is available from the **Design > Grid System** command.

Grid System General Settings Panel



Geometry: Select between orthogonal and curved grid systems. If you select a curved system, then enter a value for the radius of the outermost gridline.

Place...: Check any of the next four checkboxes to place additional elements at each gridline intersection:

- **Elements at Grid line intersections:** Check this to place elements at grid line intersections. Click the pop-up to select the element type (Column or Object) and click the Settings button to open that element's settings dialog on top of this dialog.
- **Default Beams on Grid lines:** Check this to place a Beam element along the grid lines. Click the Settings button to open the Beam settings dialog on top of this dialog.

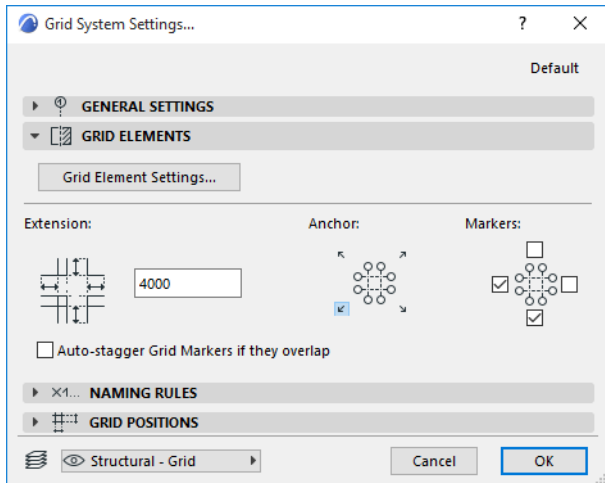
Note: If you are placing a curved grid system, you have the option to place either straight or curved beams along the gridlines.

- **Dimension lines:** Check this to place a dimension line between each and every grid line. Enter a value for the offset between the dimension line and the first crossing grid line. Click the Settings button to open the Dimension settings dialog on top of this dialog.
- **Total Dimension:** Check this to place a dimension line between the two side grid lines. Enter a value for the offset between the dimension line and the first crossing grid line. Click the Settings button to open the Dimension settings dialog on top of this dialog.

Click the chain icon to have identical attributes for dimension lines and the total dimension.

Keep Column and/or Beam within Grid perimeter: Check this to shift the columns and the beams so that will appear inside the perimeter of the Grid system.

Grid System Grid Elements Panel



Grid Element Settings: Click this button to open the Grid Element to set properties for every Grid element in this Grid System. (All Grid elements will look the same.)

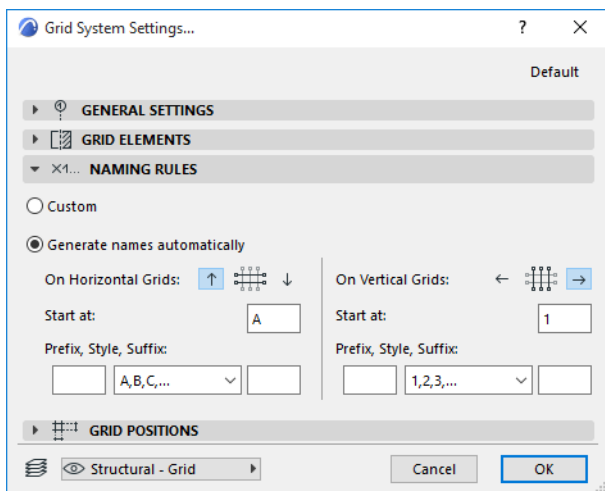
Extension: Enter a value for the distance that the Grid line extends beyond the last crossing Grid line

Anchor: Click one of the four positioning buttons to define an anchor by which to place the Grid System.

Markers: Check any or all of the four sides of the Grid System to place Markers. (If you define a rotation during Grid System placement, then these sides will be rotated as well.)

Auto-stagger Grid Markers if they overlap: Check this to automatically stagger Grid markers sideways to avoid overlapping, if the markers would otherwise overlap.

Grid System Naming Rules Panel

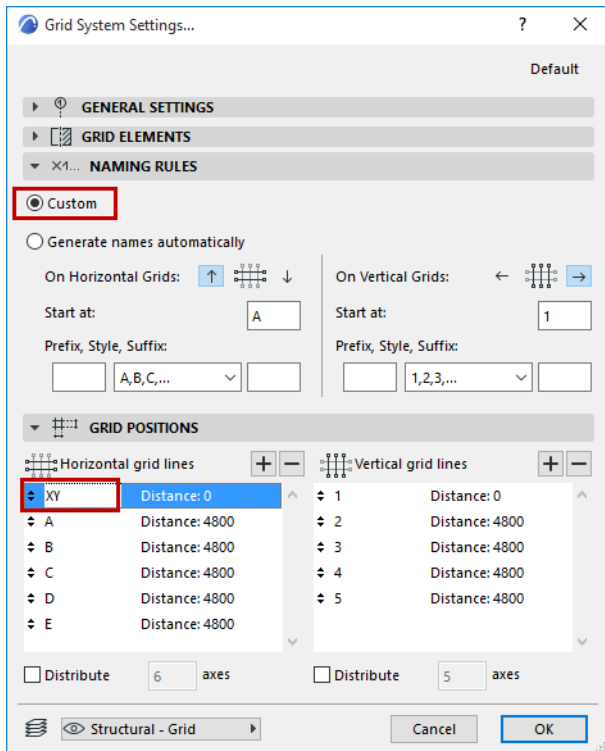


Use this panel to define the logic by which to name the Grid lines of this Grid System. You can define separate naming styles for the two grid directions.

- For a curved grid system: circular Grid lines and radial Grid lines.

Note that assigning naming rules to “horizontal” and “vertical” gridlines, as well as the “up” and “down” definitions, are relative to the Grid System’s orthogonal position (if you rotate the Grid System when placing it, the assigned names will be rotated accordingly.)

Custom: Select this to enter names manually for any grid element, using the fields in the Grid Positions Panel: if you click Custom in the Naming Rules Panel, then the Grid line names become editable in the Grid Positions Panel.



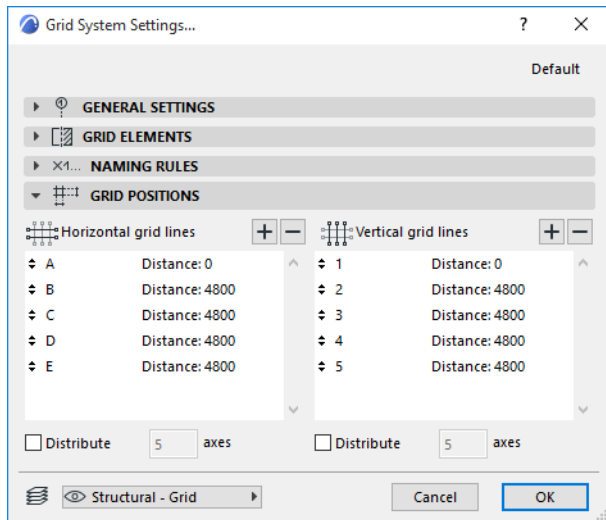
Generate names automatically: Select this to auto-generate names for the Grid lines in this Grid System.

Ascending direction: Select one of the two directions (click the up or down arrow) to define in which direction the automatically generated names should be assigned.

- **Start at:** This field shows the starting value of the Grid element naming.
- **Prefix:** a static text displayed in front of each auto generated name.
- **Suffix:** a static text displayed following each auto generated text.
- **Style:** Choose a style for the auto generated Grid element names: either numbers (1,2,3,...), letters (A,B,C,... or a,b,c,...), or Roman numbers (I,II,III,IV,...)

Grid System Grid Positions Panel

Use this panel to define the number and position of horizontal and vertical Grid lines for this Grid System.
(For a curved grid system: circular Grid lines and radial Grid lines.)



Note that “horizontal” and “vertical” gridlines parameters are relative to the Grid System’s orthogonal position, and rotate along with the Grid.

Add a Grid line: Click the Plus sign to add a Grid line.

Remove a Grid line: Click the Minus sign to remove a Grid line.

Set distance: Click the distance field of any of the listed Grid lines to define its distance from the previous Grid line. The Distance of the first Grid line in the list is always zero.

Distribute: If you do not know the distance you will need between your Grid lines, check the **Distribute** box for either or both Grid line directions. This will enable you, when you place the Grid system, to set the beginning and endpoints of the Grid system in the given direction, divided into the number of Grid lines defined in the list.

GDL (Geometric Description Language)

The dialog boxes and commands described below are used when editing GDL objects in Archicad.

[GDL Object Editor](#)

[About GDL Object Subtypes](#)

[Subtype Hierarchy Dialog Box \(GDL Master Window\)](#)

GDL Object Editor

The GDL Object Editor contains all controls needed to edit a GDL object.

For a detailed description of scripting commands and object-level editing, see the [GDL Reference Guide](#) (available from the [Archicad Help menu](#)).

Open GDL Object Editor

To open the GDL Object Editor, do one of the following:

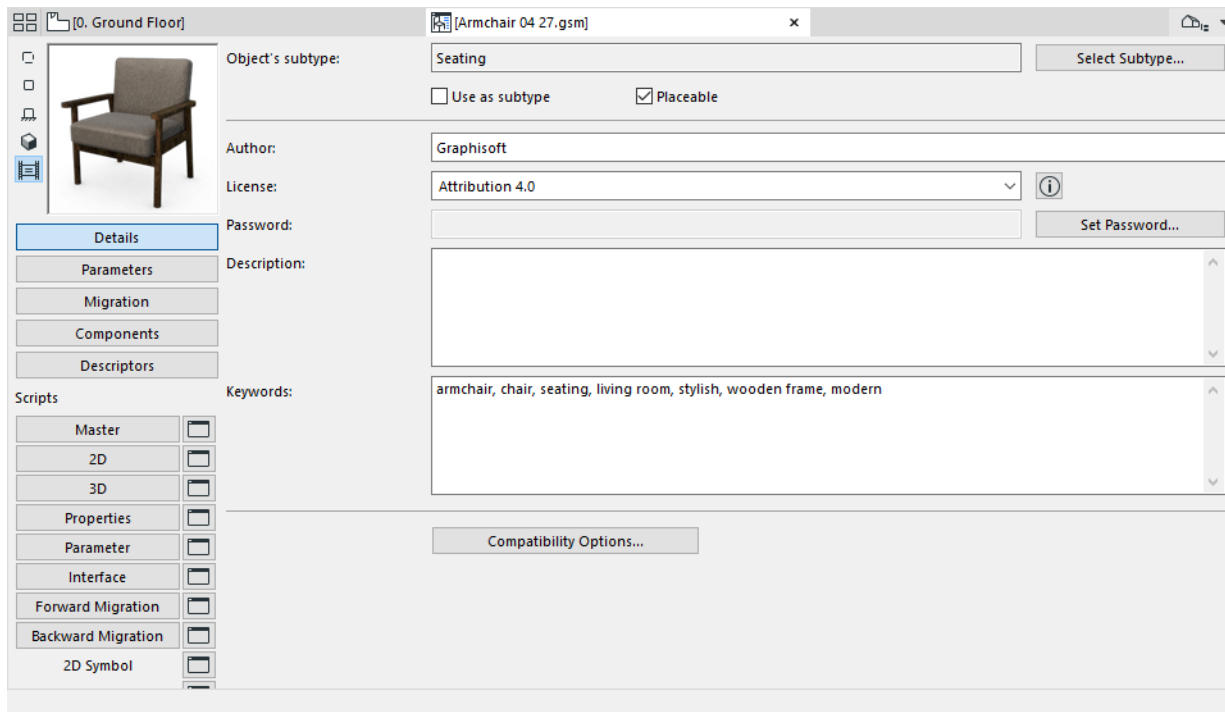
- For a placed object: select the object, then use **File > Libraries and Objects > Open Object** (or choose this shortcut command from the Edit GDL Library Parts toolbar)



- Browse an object with **File > Libraries and Objects > Open Object**, then browse for the desired object.
- Browse an object by subtype with **Open Object by Subtype** from the Edit GDL Library Parts toolbar.



The window that opens is the GDL Object Editor.



Use the buttons on the left side to open different tables/lists for editing (e.g. Details, Parameters, Components) within the GDL Object Editor.

“Read-Only” Objects from a Library Container File

To edit an object placed from a library container file, you must first extract the container. Otherwise, the Object Editor Window is “Read-Only”.

For more information, see [Library Container File](#).

Topics in this section:

The controls of the GDL Object Editor are described in the following.

[Object Preview Picture](#)

[Details](#)

[Parameters](#)

[Migration](#)

[Components and Descriptors](#)

[Script Windows](#)

[2D Symbol](#)

[Preview Picture](#)

[2D and 3D Views](#)

[GDL Editor Shortcuts](#)

Object Preview Picture

These controls are available from the GDL Object Editor.

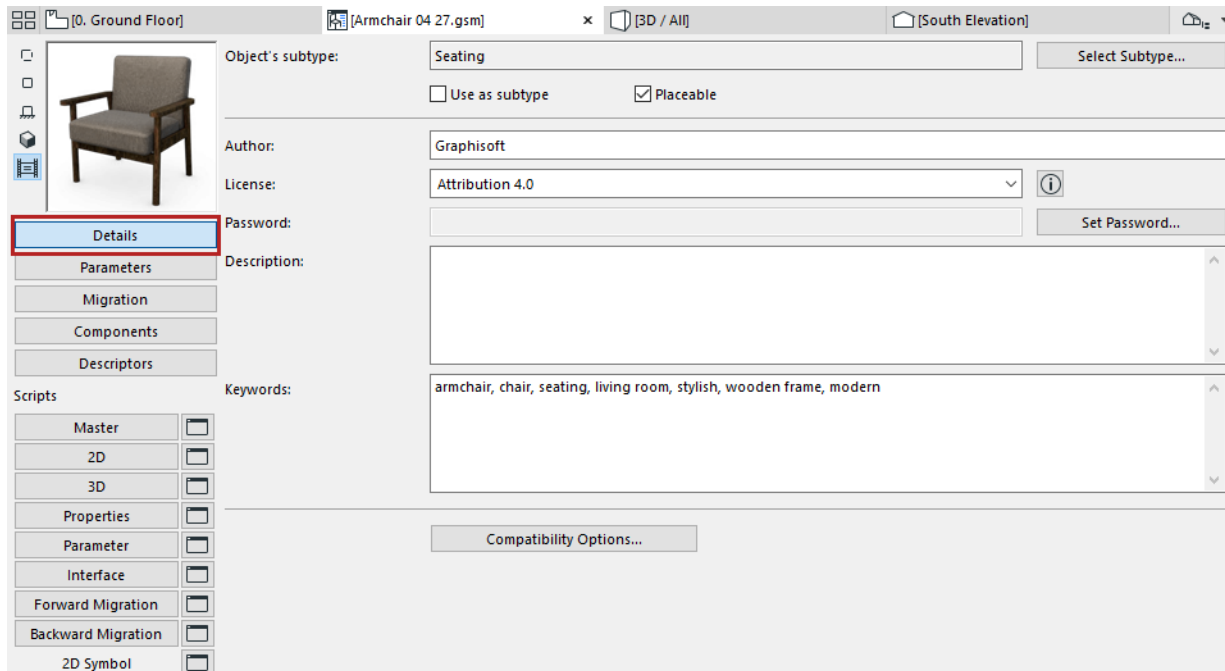


- Click the buttons to choose a display option for the preview: 2D Symbol, 2D View, Front, 3D, and Preview Image.
- With the Front or 3D display: click in the preview window to rotate the image by 90 degree increments.

For info on how to generate a preview, see [Preview Picture](#).

Details

These controls are available from the GDL Object Editor.



Object's subtype: To the subtype, click **Select Subtype**. This opens the **Subtype Hierarchy** dialog. Changing the Object's subtype modifies its behavior and adds the parameters and functionality needed for the new subtype. For example, you can change a plain object into a light source by changing its subtype to Light. [For more information, see Subtype Hierarchy Dialog Box \(GDL Master Window\).](#)

Use as subtype: Check this box to save the object as a template to be included in the list of available Subtypes. The new template will appear in the **Subtype Hierarchy** dialog box, and any new objects that you associate to this subtype will take on all of the parameters of the subtype.

The **Placeable** checkbox is enabled by default.

If you uncheck it, the Object will behave as a macro. In this case,

- the Object will not appear in any of the tool settings dialog boxes
- you will not be able to place it directly in the project
- it cannot be included in an archived project file
- you can refer to its scripts from other Objects

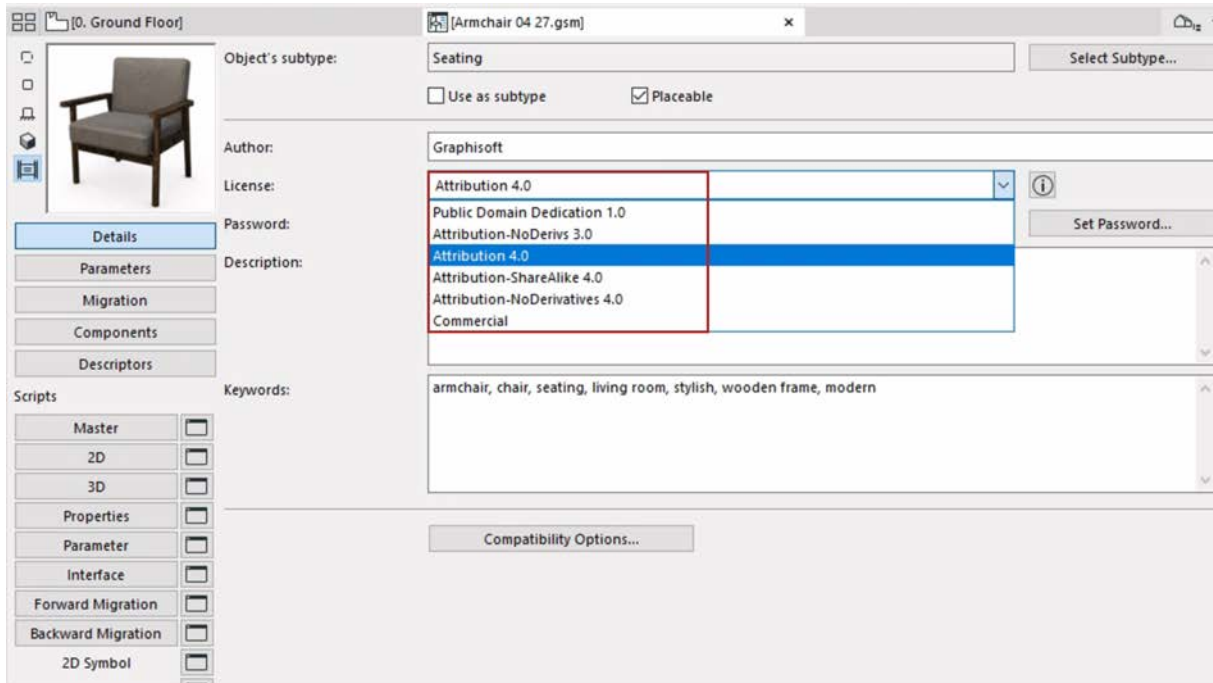
Object Information Fields

The next few controls are relevant if you wish to share the library part to BIM Components, or if you plan to make your own object commercially available. Also, any text you enter will be considered when you use the Search function.

[See Search for Library Part.](#)

- Enter an **Author**. (If you leave this blank, the name you used to register on BIM Components will be used as the author, after the object is shared.)

- Assign a **License**. For a complete description of a selected license option, click the Info button from the dialog box.



- Commercial. Use this kind of copyright if you are selling the object over the internet. An object with Commercial copyright *cannot* be shared to BIM Components.

See [BIM Components](#).

- **Password** (available if the copyright is “Attribution-NoDerivs0” or “Commercial.”). Password-protected objects can be freely viewed and downloaded, but the user cannot open the object to edit or copy its script.
- **Description**. This text will appear when you click the Info button in the Settings dialog box, as well as in BIM Components, if the object has been shared there. Also, Description text will be considered when you use the Search function.

Compatibility Options (Developer Use Only)

Consult the GDL Center for detailed information.

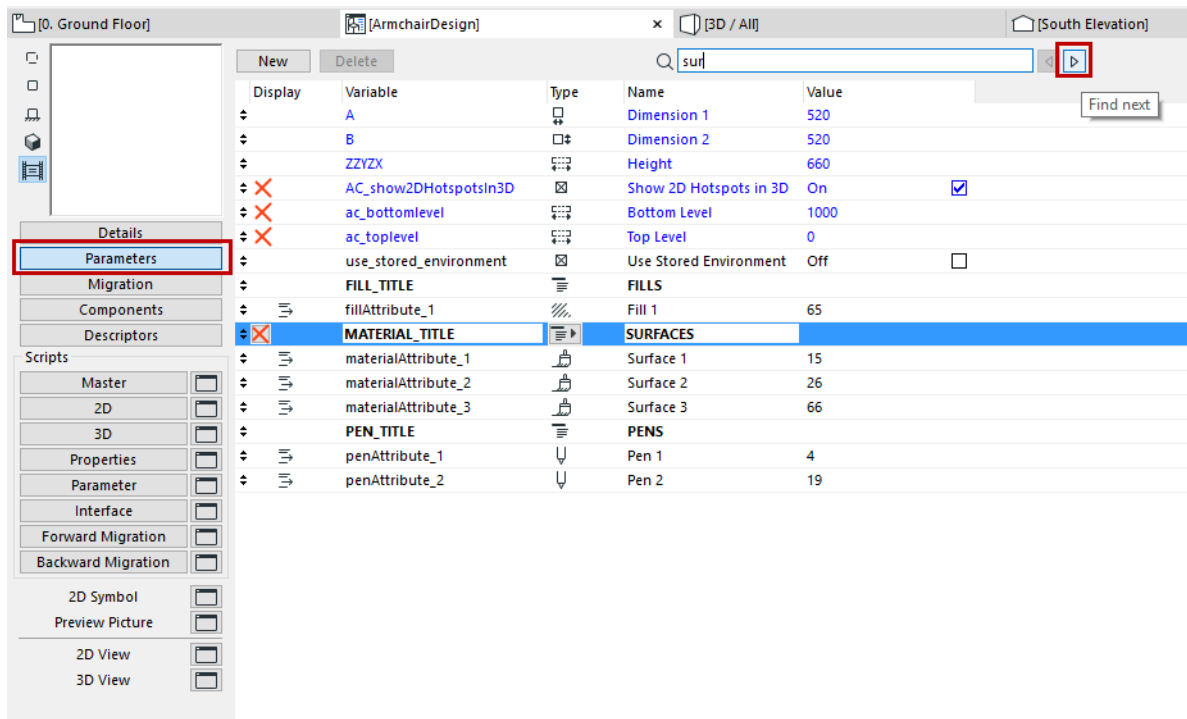
See gdl.graphisoft.com.

Parameters

Display the Parameters of the current Library Part in the GDL Object Editor.

Search Parameters

Enter a text string in the search field to locate any part of that string in the Variable, Name, or Value columns. The cursor jumps to the first found instance. Use the Find next/Find previous buttons to jump to additional found instances.



By assigning values to the set of variables in the Parameters list, you will define the attributes of a particular GDL Object. The values assigned here will be the default values found in the Settings dialog box when you choose a tool that uses GDL Objects. These locally defined Parameters can also be called by the different Scripts of the Object.

The parameters that appear in **blue** are the obligatory parameters (such as A and B dimensions) that belong to the edited Object's subtype.

Optional parameters are available for use in the Object's GDL description. The values specified for them will become the default additional values displayed when you open the appropriate Settings dialog box from the Toolbox.

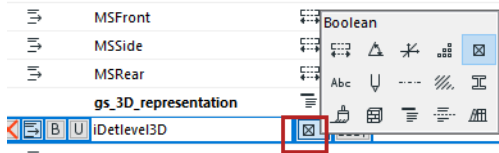
The optional parameters are typically used to define such things as the number of window or door panels, the usual surfaces of a part, etc.

Create or delete parameters with the **New** and **Delete** buttons.

Highlight the parameter to edit. For obligatory parameters, you cannot edit the variable name or type icon. For optional parameters you can edit all the parameters.

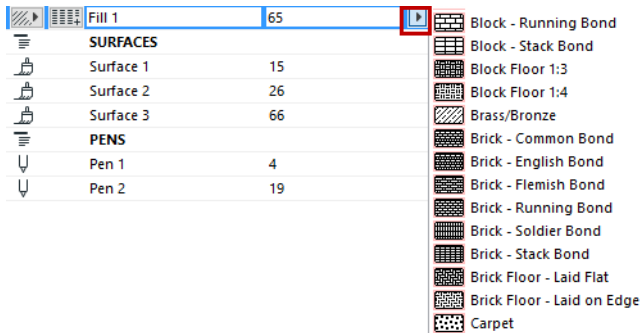
Set Parameter Type

The **Type** icons control how the values are interpreted by Archicad.



Icon	Parameter name	Description
	Length	Specifies physical dimensions of the GDL Object (and makes conversions to whatever units you are using.) Must be a positive number.
	Angle	Specifies angular values in decimal degrees for such GDL operations as rotation transformations and defining arcs.
	Real Number	Specifies decimal non-dimensional values (any real number) for uses like including a magnification factor in a GDL calculation.
	Integer	Specifies whole numbers, dimensionless values for uses like defining the number of identical components in a GDL description or the number of times to loop through a part of the GDL Script.
	Boolean	Use value of either 1 or 0. Use it to activate or deactivate an action or an item in a GDL Object. Enter value directly, or click the checkbox at the right end of the parameter line.
	Text	Enter up to 255 text characters in the parameter's value box.
	Pen	Specifies a pen attribute as a GDL parameter
	Line type	Specifies a line type attribute as a GDL parameter.
	Fill type	Specifies a fill attribute as a GDL parameter.
	Profile	Specifies a profile attribute as a GDL parameter
	Surface	Specifies a surface attribute as a GDL parameter
	Building Material	Specifies a Building Material attribute as a GDL parameter
	Separator	Virtual parameter, used to make the Parameter List more organized and easier to view.
	Title	Virtual parameter, used to make the Parameter List more organized and easier to view.
	Dictionary	Technical parameter, used to store multiple data

Note: For Surface, Line Type, Fill Type parameters: Use the pop-up to choose a corresponding Archicad attribute.



Note: Depending on how you write your GDL descriptions, the parameters you use can affect both the 2D and 3D aspects of the Library Part.

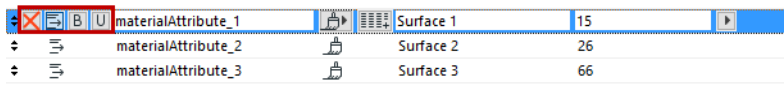
Array



You can restrict the possible values of a parameter by defining an array. Click it to toggle to array and open the **Array Values** dialog box.

List formatting for Parameters

Select a parameter and use any of the list formatting icons.

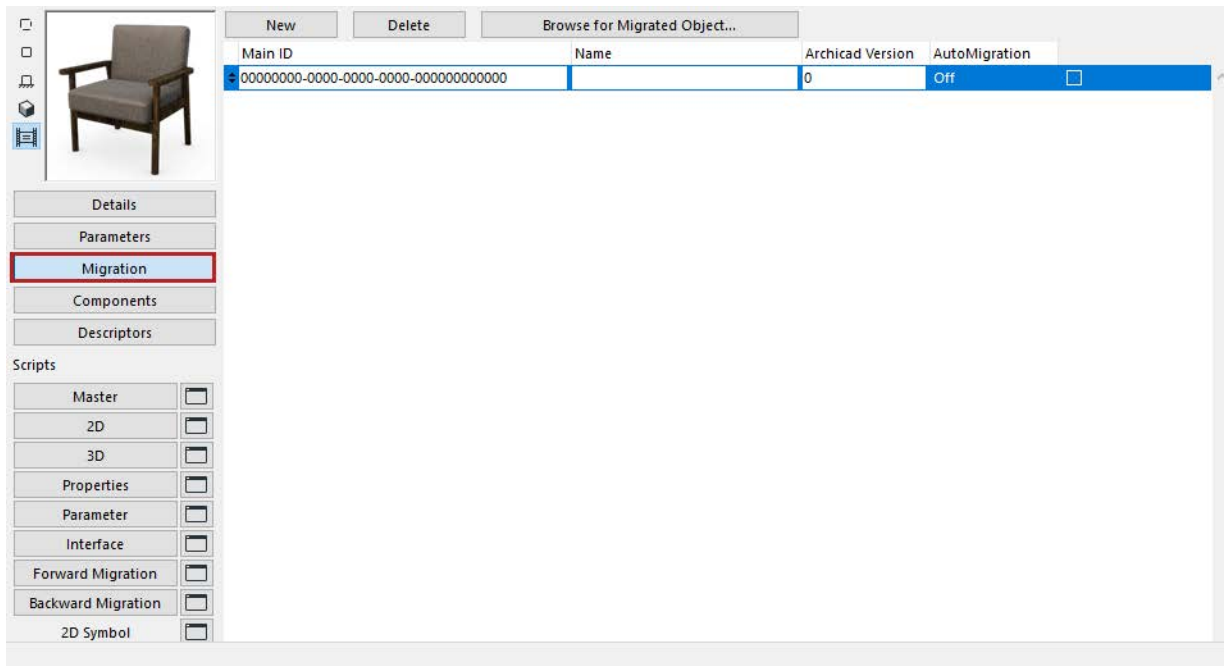


- **Hide Parameter:** Removes parameter from the Parameters panel in the Object Settings dialog box.
- **Subordinate:** Creates a hierarchy among parameters. The superior parameter can be of any type; however, a Title type parameter cannot be subordinated.
- **Bold:** Display parameter name in bold type.
- **Unique:** The current parameter will NOT be transferred if you transfer the rest of this object's parameters to another object.

Note: When applying a Favorite to an Object, the "unique" parameters are applied together with the Favorite - they are not excluded.

Migration

Use this page of the GDL Object Editor to define the older-version objects that should be migrated to the currently opened (newer-version) object.



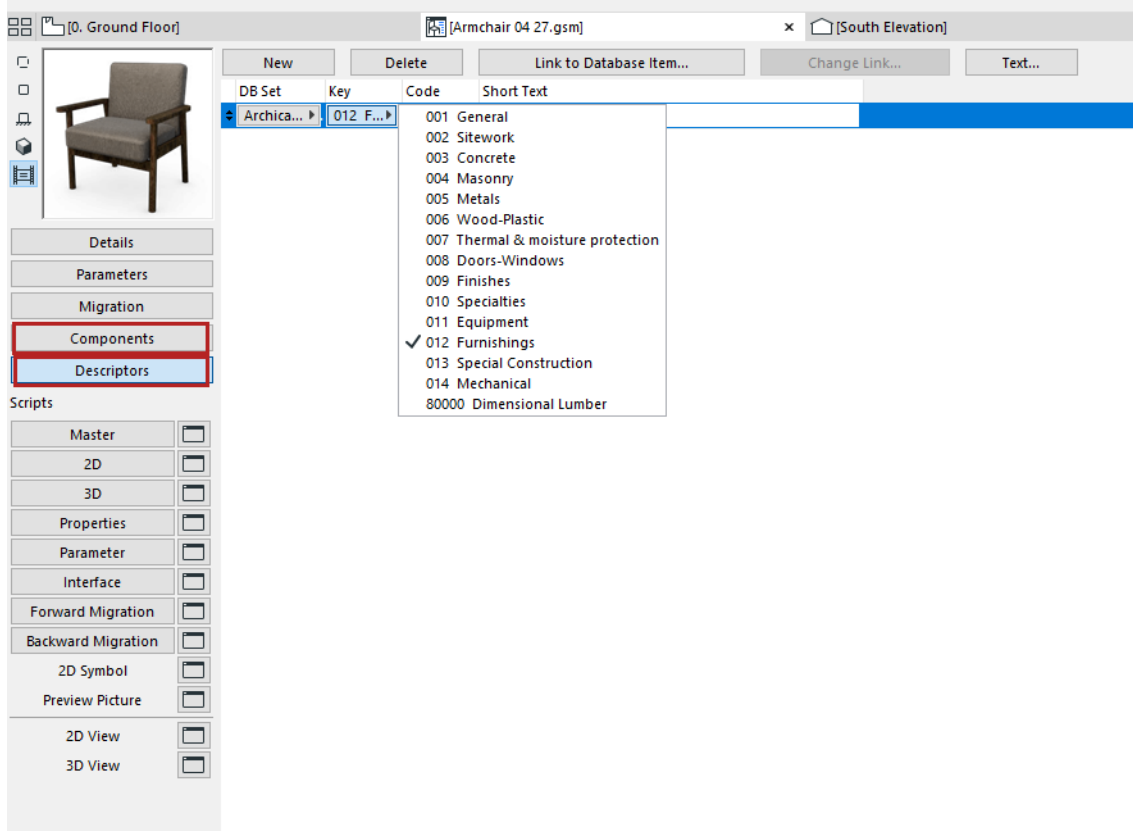
- Fill in the main **GUID** of any old object that can be migrated to this new, currently opened object.
- Fill in a **version** number for the old object (typically the version of a previous Archicad): when the project is saved back to the previous version (and if “Convert library objects backward” is set in Options from the Save As dialog box), this new object will be migrated to an old object in the migration table which has been assigned the corresponding version number. The Backward Migration script will consider all such rows until the first successful run.

Note: if you enter 0 for the version number, backward migration will skip this object.

Components and Descriptors

Display the Components or Descriptors of the current object in the GDL Object Editor.

Components and Descriptors define the object's composition for use in listing and calculation features.



Click **New** to create a component or descriptor.

DB Set: Select Database from this pop-up menu.

Key: Select a Key.

Note: Available if your loaded library contains Key description.

Code: Define a Code for new Component/Descriptor or modify the Code of a selected one.

Quantity: Define or modify the Quantity of the selected Component

Unit: Select predefined Unit for the selected Component.

Note: Available if your loaded library contains Unit description.

Proportional with: Select a predefined expression, or select Custom, then enter any GDL expression in the edit box above.

For details about Component Editing, see the Archicad Calculation Guide.

Link to Database Item: Link a Component or Descriptor to an external database residing in the loaded Library. While a Component or Descriptor is linked to the selected database, it cannot be edited.

Change Link: Re-link to another database

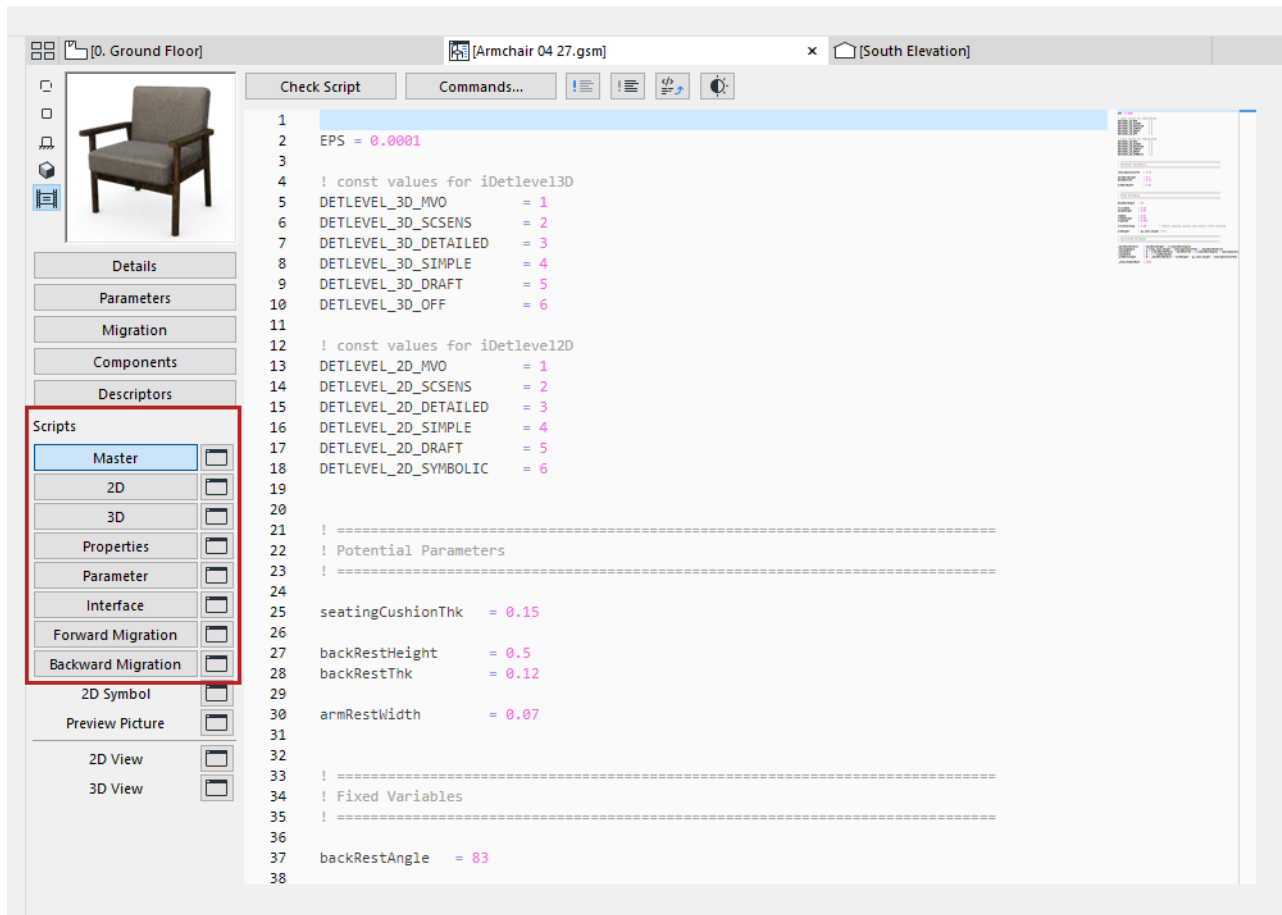
Text: Open the Descriptor Full Text dialog box displaying the whole Descriptor text. The first line will be the Short Text.

For details about Descriptor Editing, see the Archicad Calculation Guide.

Script Windows

Use the scripting windows of the GDL Object Editor to edit object scripts.

- If you click the script name (e.g. Master), you edit right in the GDL Editor window
- If you click on the window icon next to the script name, you open a separate scripting window



- The **Master Script** will be executed each time any of the other scripts are executed. The primary benefit of this script is that you can globally define variables or operations, or access external data before executing other Scripts.
- In the **2D Script** window, you can create a parametric 2D symbol using the transformations and elements available for the two-dimensional space in GDL. This script will be used to generate the 2D Full View of the Object.
- In the **3D Script** window, you can find and edit the 3D description of the GDL Object. The script will be used to generate the 3D View of the Object.
- With the **Property Script**, you can attach descriptions and technical data to the Object using its variables and the related GDL commands and expressions. Component quantities (weight, price, paint, etc.) can be calculated according to the expressions in the Properties Script and included in the lists. To use the custom Properties Script as the default in the element's settings dialog box, click the **Set as Default** button at the top of the screen.

Note: When there is a Property Script defined, the listing commands will use this script, even if you previously created Components and Descriptors in the corresponding sections of the dialog box.

- Using the **Parameter Script**, you can define the choices available as a value of a given parameter. For example, you can restrict the choice of the surface of a table top to wooden surfaces, or define the different panel styles of a given door by their name.
- The **Interface Script** allows you to define a custom user interface for your GDL Objects, including the graphic explanation of additional parameters. If an Interface Script exists, the settings dialog box of the given library part will include a **Custom Settings** panel. To use the custom Interface Script as the default in the element's settings dialog box, click the **Set as Default** button at the top of the screen.
- **Forward/Backward Migration Scripts:** Click the Forward Migration or Backward Migration buttons to define how to migrate the parameters of an old instance forward to the current open element (or backward to an older element).

Autocomplete

While you type, use the suggested auto-complete (all GDL commands and parameters are there) to make your work easier.

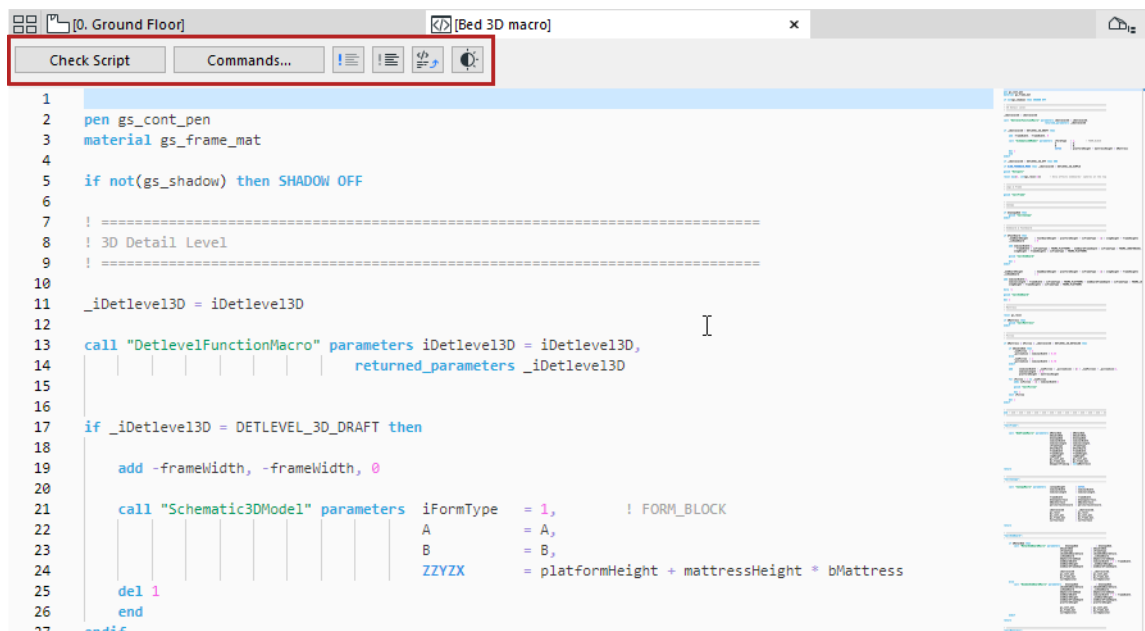
At the top of all script windows, some commonly used functions:

Note: Some of these commands are also available from the Archicad **Edit** menu (if a script window is active).

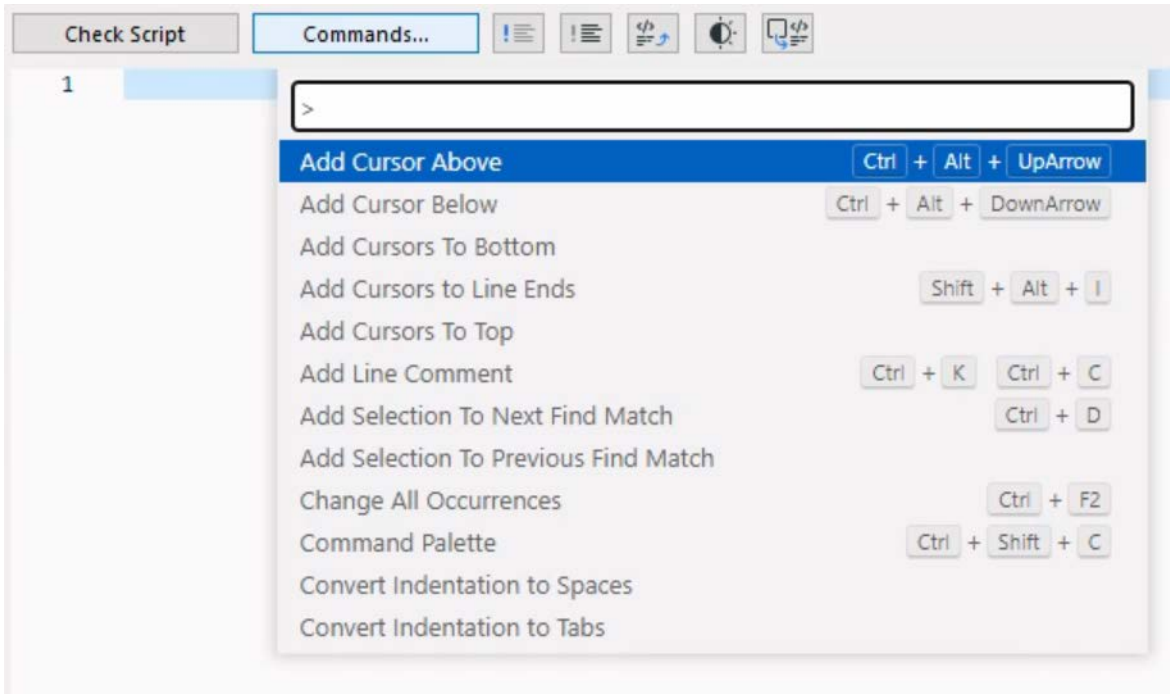
See also [Editing Commands in Text-Type Windows](#).

Check Script

Check syntax of the GDL script in the active window. To check all scripts of the current Library Part, use this command in the Master script window.



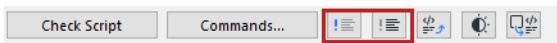
Commands



Click to access common script editing commands, including:

- Code folding: This feature allows you to collapse and expand sections of code, making it easier to navigate through large files.
- Multiple cursors: This feature allows you to place multiple cursors in different parts of your code, so you can edit multiple lines at the same time.
- Find and replace

Comment/Uncomment



Transform selected lines into comments and vice versa.

Open Macro in another script window

In the script window, select the macro text (preceded by “call”) and click **Open Object**.

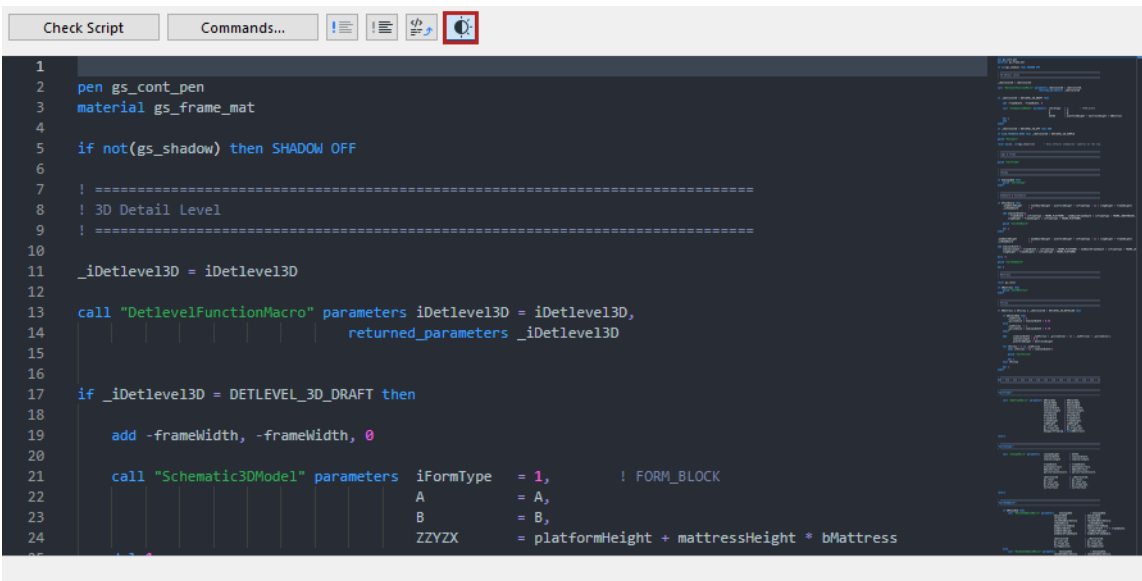


This opens the macro script for editing.

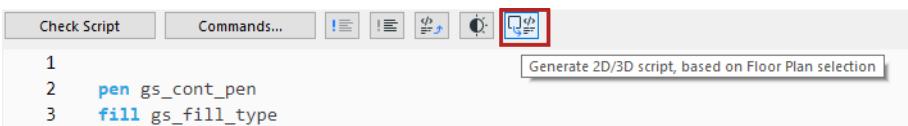
If nothing is selected, this command opens the Open Object dialog.

Dark Mode

Toggle dark mode on-off.



Generate Script based on Floor Plan selection

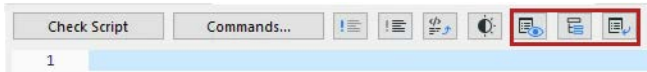


Use this while working in the 2D or 3D GDL script window.

Select an element in the Floor Plan window.

Return to GDL Editor window (2D or 3D script window) and click the command to "Generate 2D-3D script".

Interface Script Window



- Preview - See a preview of the Interface you are creating.
- Hierarchical Pages - Turn on hierarchical tab page structure.
- Set as Default - Use the current Interface settings as default.

2D Symbol

Click from the GDL Editor to open a new editing window.

Each GDL Object you place in your Project is shown on the plan by a 2D Symbol. You can edit an existing symbol or create a new one with the standard Archicad selection and 2D tools: **Arrow**, **Marquee**, **Text**, **Fill**, **Line**, **Arc/Circle**, **Spline** and **Hotspot**. You can also paste elements from the 3D Window.

The final symbol that appears on the Floor Plan will be created from the elements of this window and the 2D Script, if there is one.

2D Symbol Display (Layers)

To control the visibility of the 2D Symbol fragments, select the 2D Symbol button, then go to **Options > Element Attributes > Layer settings**.

Note: Fragments that are hidden are not available for storing new graphic elements. If you select a hidden Fragment in the dialog box of the Tool you are using for editing the symbol, you will get a warning asking you to change your selection or Show the selected Fragment.

Fragments also let you easily replicate parts of the Symbol. You can draw elements in the 2D Script Window with the various graphic tools, and refer to them by layer number in a GDL 2D Script.

Preview Picture

Available from the GDL Object Editor. The preview is the default representation of your Object in tool settings dialog boxes.

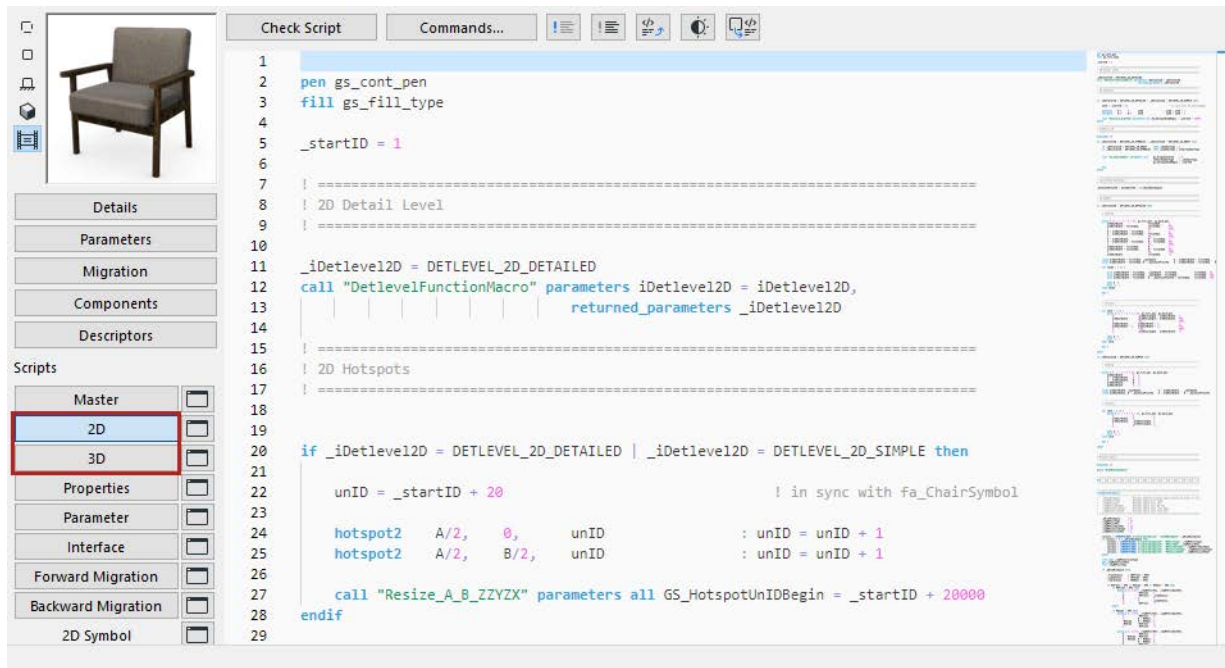
To generate the preview of a new Object:

1. Place the Object in the project.
2. Select the object and open the 3D Window to see this selection.
3. Go to **View > 3D View Options > 3D Projection Settings**. Choose a projection type and set up a view angle that show the typical features of your Object. Click OK.
4. Open **Set Window Size** (from the 3D Window tab or context menu). Set the window's height and width to the same value to obtain a square shaped window.
5. Go to **Document > Creative Imaging > PhotoRendering Settings**. Define a pixel size that you will be able to insert in the preview without any distortion, and choose a neutral background for the Object (for example, white color for both the Sky and the Ground).
6. With the Object still selected, choose **Document > Creative Imaging > PhotoRender Projection**. A Model Picture is generated.
7. Save the generated image.
8. Go back to the Floor Plan. With the Object still selected, choose **File > Libraries and Objects > Open Object**.
9. Click Preview Picture to open the corresponding window, then drag the generated image file into the window.
10. Save the GDL Object.

You can also paste other data from the clipboard, for example a photograph of the real-life object.

2D and 3D Views

Available from the GDL Object Editor.



2D View

Shows the object's Floor Plan symbol.

If there is a 2D Script defined, the 2D View will be calculated according to the 2D Script, even if the 2D Symbol window contains drawing elements. However, the 2D Script can refer to 2D Symbol fragments.

3D View

Shows the object's 3D symbol.

In this window you can see the 3D view of the GDL Object converted from the 3D Script that refers to the current Parameter values. Use the **View > 3D View Options > 3D Styles** dialog box or the 3D Navigation Palette to adjust its view settings.

Matching 2D & 3D Descriptions

The 2D symbol and the 3D description are automatically linked to each other in the GDL Object. There are no built-in safeguards against geometric incompatibilities between the two components. The most common incompatibility error occurs when the 2D symbol and the 3D shape have different positions or orientations with respect to the origin.

To avoid this problem, follow these steps:

1. Activate the **3D View** of the GDL Object.
2. Choose Top View in **View > 3D View Options > 3D Projections Settings > Parallel Projections**, and set the angle to 270°.
3. Choose **Edit > Select All**, then **Edit > Copy**. Activate the GDL Object's 2D Symbol window and choose **Edit > Paste**. This will place the top view of the object (as a wireframe or hidden line) into the symbol window. It should match the dimensions and position of the 2D symbol.

4. Choose Undo once you have verified that the symbol matches the 3D view. This will remove the 3D lines. You can use this feature to have Archicad create the 2D symbol for you. You will only need to add additional hotspots, lines, etc. The procedure is the same for **Doors** and **Windows**, with the following modifications:

- Choose **Side View** instead of Top View (the angle should still be 270°).
- A mirrored view will be placed. Select the placed lines and mirror them across the reference line of the wall.

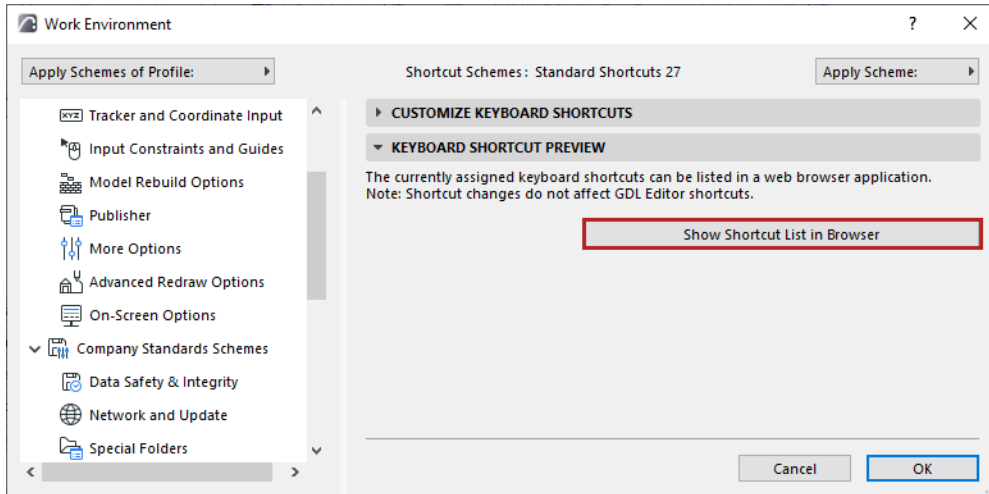
Note: When you place a wireframe view, redundant lines are removed. Using Hidden Lines will give you an accurate view, but underlying lines will be hidden.

You can also ensure compatibility between the 2D plan symbol and the 3D model of a GDL Object by writing a parametric description in GDL for the symbol.

GDL Editor Shortcuts

To see a list of GDL Editor shortcuts:

1. Open Options > Work Environment > Keyboard Shortcuts.
2. Click **Show Shortcut List in Browser**.



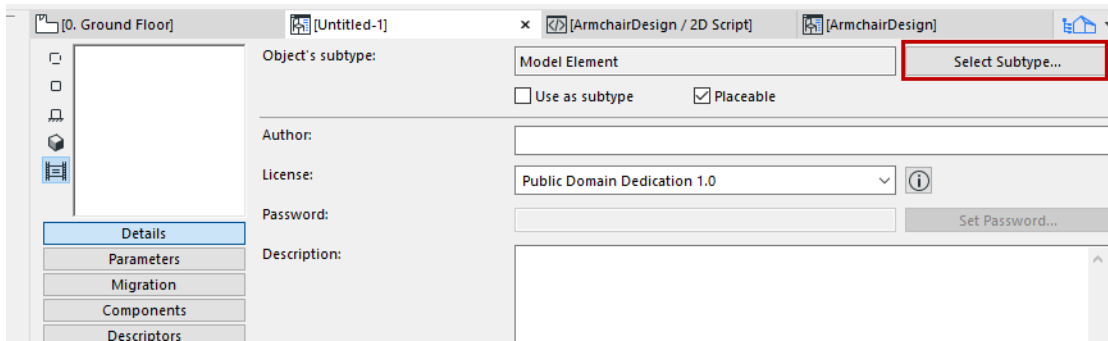
3. A webpage opens with a list of all shortcuts.
4. Scroll down to see the GDL Editor shortcuts.

GDL shortcuts are not editable.

Subtype Hierarchy Dialog Box (GDL Master Window)

See also [About GDL Object Subtypes](#).

This dialog box is accessible using the **Select Subtype** button in the GDL Master window (**File > Libraries and Objects > New Object**)



or is available on the Edit GDL Library Parts toolbar (**Window > Toolbars > Edit GDL Library Parts**). You can add it to your Archicad menu structure using the Work Environment dialog box.



The items listed by name in the Subtype Hierarchy are color-coded.

- **Green** indicates that the item is placeable.
- **Black** indicates that the item cannot be placed on its own and can be only used as a macro reference.
- **Bold** type indicates that the item is a subtype template and is placeable.

If you highlight an item in the hierarchy, you can review its **Properties** in the bottom section.

Below the **Name** field, the **GUID** refers to the highlighted Subtype's Global Unique Identification code.

For a complete description of how GUID works, see [this article](#).

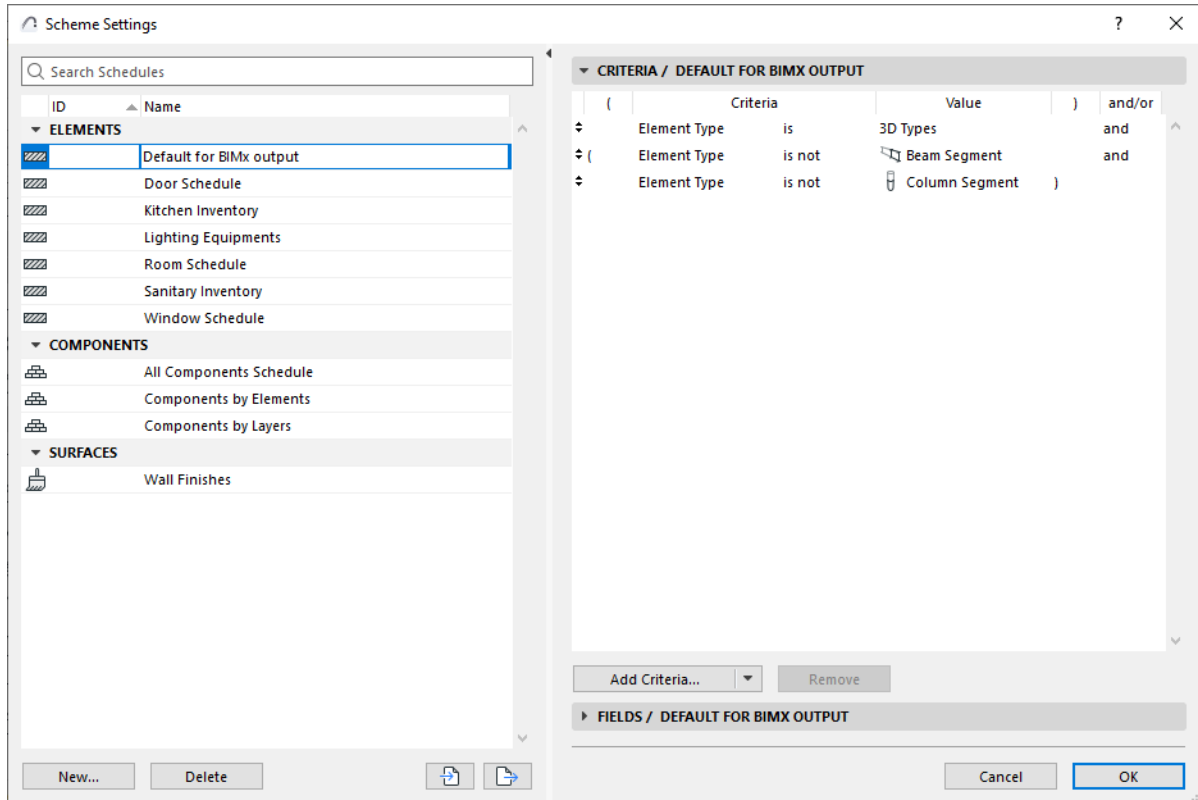
Select: Click this button to directly open that item's Master Window.

Create Clone: Click this button to create a copy of the highlighted Subtype with the same properties.

Scheme Settings Dialog Box (Interactive Schedule)

To access this dialog box, do one of the following:

- Click **Scheme Settings** from the opened Schedule window



- Use the Schedule's context menu in the Navigator
- In the Navigator Palette, select a Schedule, then click the "Settings" button at the bottom of the Navigator.
- Use **Document > Schedules > Scheme Settings**

For more information, see [Interactive Schedule](#).

See also [Reservation in Interactive Schedules \(Teamwork\)](#).

Topics in this Section:

[Schemes Panel](#)

[Criteria Panel](#)

[Complex Criteria Definition](#)

[Complex Criteria Example](#)

[Schemes Fields Panel](#)

[Additional Criteria and Fields: Library Part Parameters](#)

[Sort Schedule Fields](#)

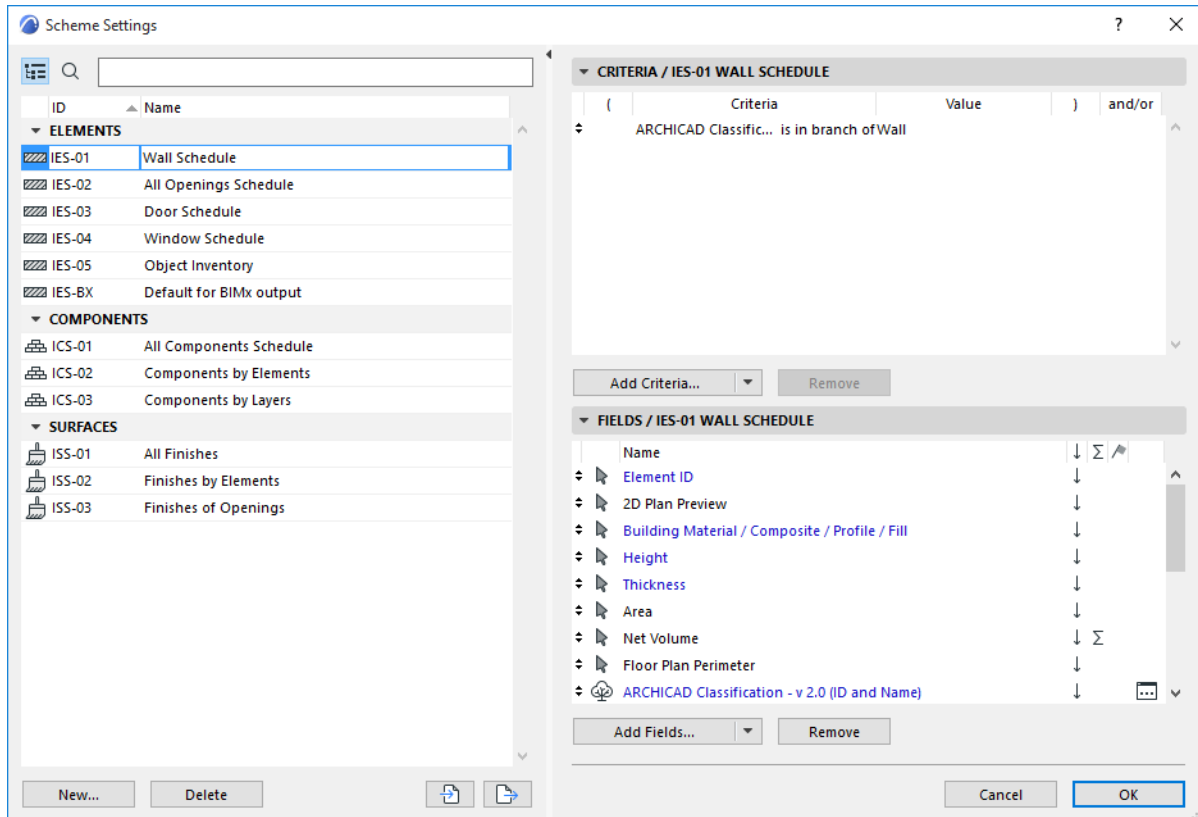
Schemes Panel

The left panel of Scheme Settings panel lists the available Schemes defined for your project.

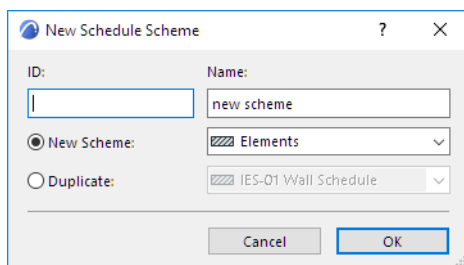
Click the panel header items (ID/Name) to sort them in ascending or descending order.

You can rename the Scheme and edit its ID directly in this list.

Edit the Schedule's Criteria and Fields in the panels at right.



- **New/Duplicate:** Click **New** to create a new Schedule Scheme.
In the appearing dialog box, choose whether to create a new Scheme, or to duplicate the current one. Choose a name and an optional ID number.
For a new Scheme, choose “Construction Elements,” “Components”, or “Surfaces”, depending on the type of schedule you are creating.

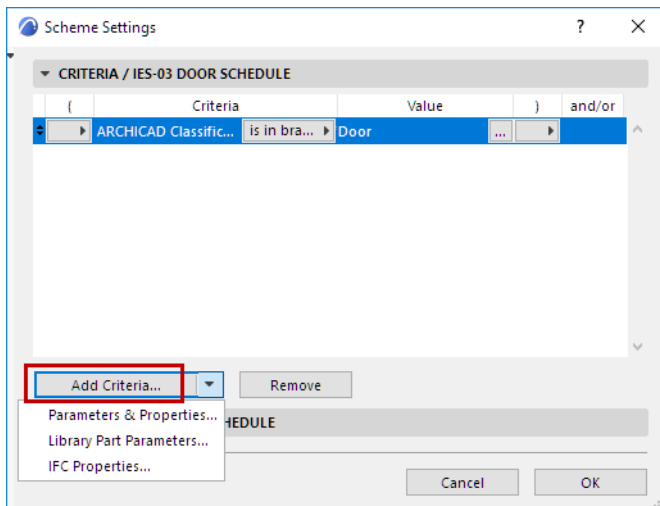


- **Delete:** Click to delete the selected Schedule Scheme.
- **Import:** Click to import a Scheme data file (XML format, created with Archicad 8 or above) or a Door or Window Schedule file (created with Archicad 7.0.)
- **Export:** Click to export the Scheme as an XML file that you will be able to import into another project.

Criteria Panel

Use this panel to define Element/Component/Surface Criteria for listing.

For a description, see [Define Element Criteria](#).



See also [Additional Criteria and Fields: Library Part Parameters and Define Element Criteria using IFC Data](#)

Remove: Click this button to delete the selected row. Inactive if no row is selected.

Click and drag the **up/down arrow** at the beginning of any row to move the row up or down within the Scheme.

Complex Criteria Definition

The **and/or** operator determines the relationship between adjacent criteria.

Keep in mind the meaning of “AND” and “OR” as logical operators.

- If you have a criterion with two statements joined by AND, the criterion will be fulfilled only if BOTH of the statements are true.
 - For example: Layer is “Interior - Partition” AND Element Type is Wall. Your schedule will list an element only if it is (1) on the given Layer, AND (2) it is a Wall.
- If you have a criterion with two statements joined by OR, the criterion will be fulfilled if one OR the other OR both of the statements are true.
 - For example: Element Type is Wall OR Slab OR Column. Your schedule will list any element which is EITHER a Wall, OR a Slab, OR a Column.

Thus, criteria that use AND are more restrictive, because both statements must be true.

Criteria that use OR are less restrictive, because it is enough for any *one* of the statements to be true.

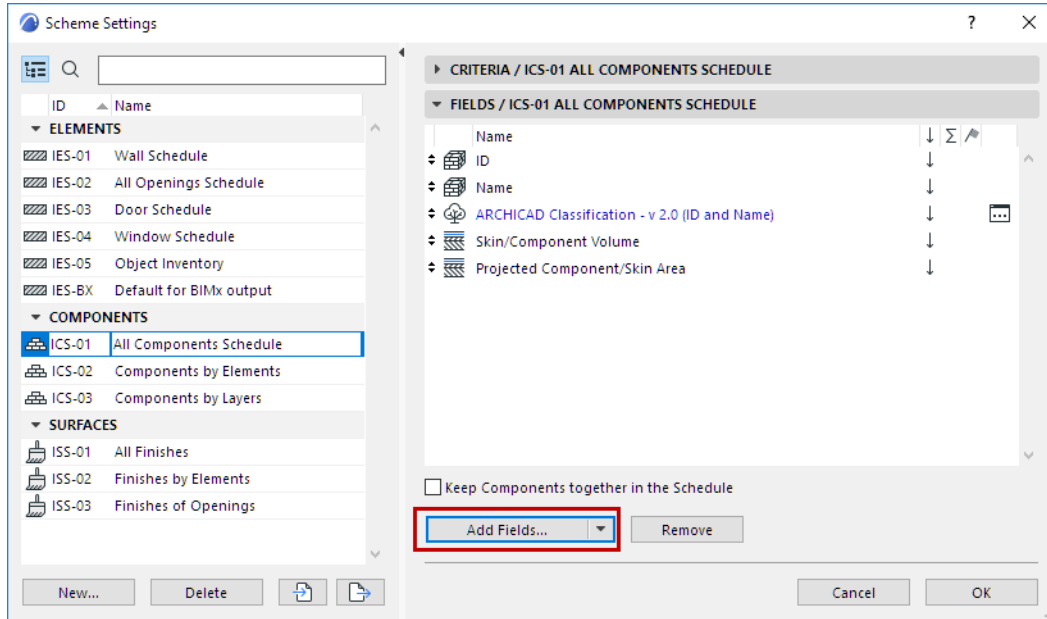
Note that if you want to list multiple Element Types, you must join them with an OR statement.

- For example: Element Type is Wall OR Slab. (If you join Element Types with AND, e.g. Element Type is Wall AND Slab, your schedule will return nothing, because an element cannot be both a Wall AND a Slab.)

This panel lists the parameters which will be displayed as fields of the schedule. Parameters that cannot be edited in the schedule are shown in black, while editable ones are shown in blue type.

- **Add Fields:** Click this button to access the list of available parameters and properties.

The list depends on the type of schedule, and may include General parameters; specific parameters related to Component and Surface lists; and Archicad Properties.



Choose a field (use the search field at the top to find it more quickly), then double-click to add it to the list. (Alternatively, click **Add** or press Enter.)

[See also Additional Criteria and Fields: Library Part Parameters and Define Element Criteria using IFC Data](#)

For more information on specific parameters used in element lists, component lists, and surface lists, see the following sections:

Element Listing Parameters

Component Data in Archicad

Surface Schedule

For information on using Archicad Properties, see:

Properties and Classification Systems

Keep Components together in the Schedule: This checkbox is available in the Scheme Settings of component-type lists.

- If you check the box, components are always grouped together, corresponding to the element they belong to. (For composite elements, these fill components are listed in the same order as in Composite Settings.)

Components by Elements	
Name	Building Material / Composite / Profile
Plaster	100 block insul. cav. plastered
Concrete Block	100 block insul. cav. plastered
Insulation	100 block insul. cav. plastered
Air Space	100 block insul. cav. plastered
Common Brick	100 block insul. cav. plastered
Plaster	Basement Wall
Structural Concrete	Basement Wall
Insulation	Basement Wall
Concrete Block	Basement Wall

- If you leave it unchecked, components will be listed according to the Scheme Settings (e.g. ordered by name rather than by element).

Components by Elements	
Name	Building Material / Composite / Profile
Air Space	100 block insul. cav. plastered
Common Brick	100 block insul. cav. plastered
Concrete Block	100 block insul. cav. plastered
Concrete Block	Basement Wall
Insulation	100 block insul. cav. plastered
Insulation	Basement Wall
Plaster	100 block insul. cav. plastered
Plaster	Basement Wall
Structural Concrete	Basement Wall

Remove: Click this button to delete the selected field from Scheme.

Once the new (or modified) scheme is defined, its name appears in the Navigator and you can use it like any other view.

Defining Schedule Fields for BIMx Info

You can set up one or more Interactive Schedule Schemes to define which Archicad element information should appear when navigating a BIMx Hyper-model in a BIMx application.

See [Step 3: Info Set at Publish BIMx Hyper-Model from Archicad.](#)

Archicad comes with a predefined Interactive Schedule Scheme use in BIMx, but you can define any number of additional schemes.

Display a URL in BIMx Element Info

This Scheme-defined element info can include a URL - that is, a hypertext link to a website (e.g. manufacturer information) or project-related data (e.g. a location in Google maps).

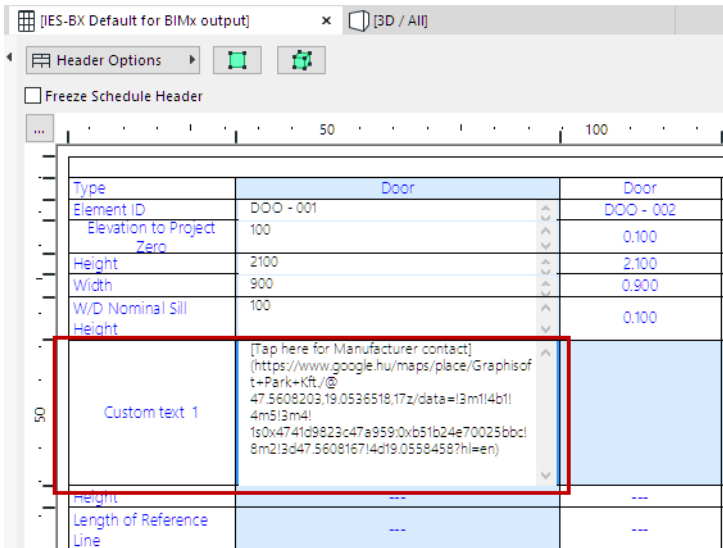
Here's the best way to do this:

1. In Scheme Setup, add a "Custom Text" field.
2. In the field, type what you want the BIMx user to see **in brackets**:

[Tap here for Manufacturer contact]

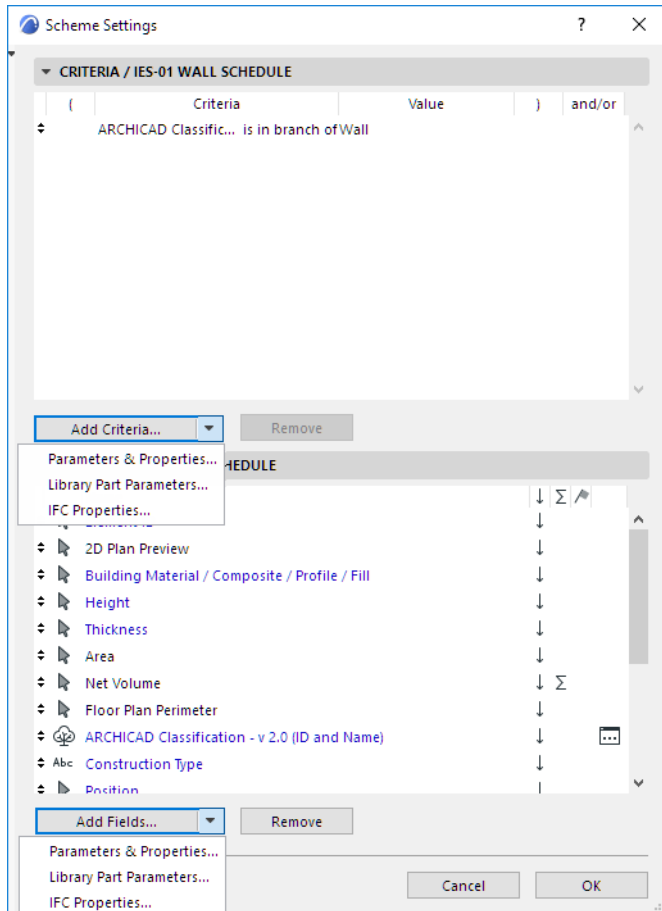
3. Immediately next to this - without leaving a space! - type the URL address **in parentheses**:

<https://www.google.hu/maps/place/Graphisoft+Park+Kft./@47.5608203,19.0536518,17z/data=!3m1!4b1!4m5!3m4!1s0x4741d9823c47a959:0xb51b24e70025bbc!8m2!3d47.5608167!4d19.0558458?hl=en>



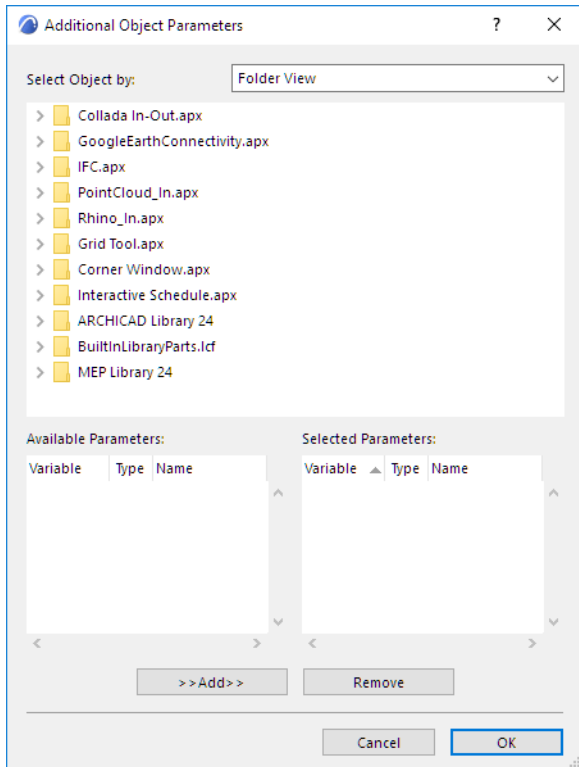
Additional Criteria and Fields: Library Part Parameters

In both the Criteria and Fields panels of Scheme Settings, use the black arrow at the right end of the **Add Criteria** or **Add Fields** button to access additional parameters and properties:



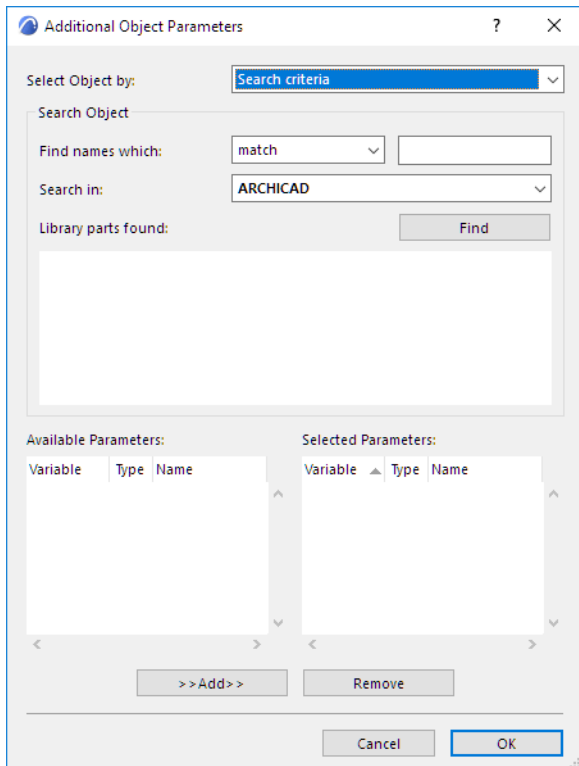
Library Part Parameters

This command makes the parameters of loaded library parts available for listing as criteria or as fields.



- **Select Object by:** Use the pop-up on the right to choose an option for searching for objects to select.
 - **Folder View:** lists all the loaded library folders that contain library parts.
 - **Folder View (Used Objects only):** lists the loaded library folders that contain the library parts used in the current project.
 - **Search Criteria:** displays a form that allows you to list the library parts whose name matches the chosen criteria.
 - **Favorite Objects:** shows the preview and the name of the library parts defined as Favorites.

- **Search Object:** If the Search Criteria option is active in the Select Object by pop-up field, this area allows you to search for library parts whose name matches some criteria.



- **Find names which:** With the pop-up control, choose a condition (match, contain, start with, end with) and enter a text string in the field on the right.
- **Search in:** Use this pop-up control to choose a loaded library to search. Click the Find button to display the results in the Library parts found box.
- **Find:** Click this button to perform a search based on the condition set with the Find names with and Search in controls.
- **Library parts found:** Displays the preview and names of the library parts that match the condition set with the Find names with and Search in controls.

Add: Click this button to add the available parameter selected in the box on the left to the list of selected parameters on the right.

Remove: Click this button to delete the selected field from the box on the right.

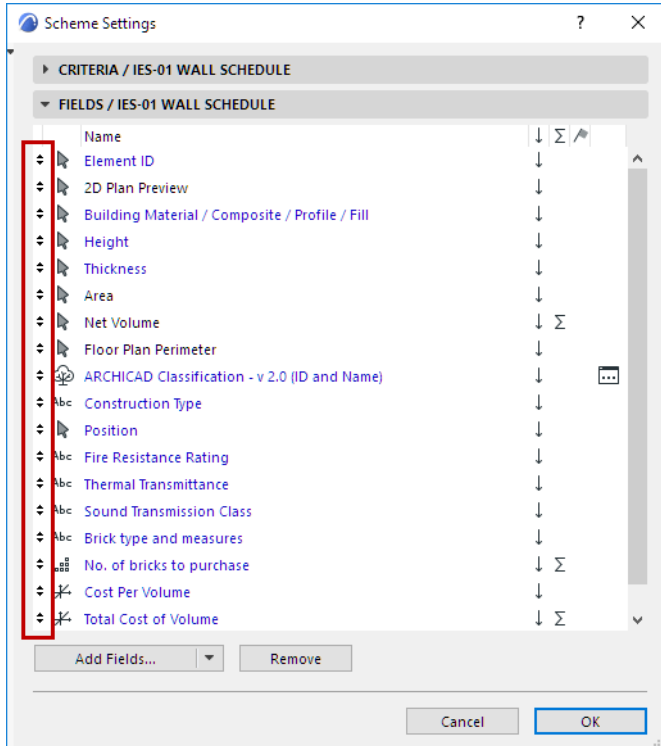
IFC Properties

To use IFC properties as criteria or fields for the schedule, click this command.

See [Define Element Criteria using IFC Data](#).

Sort Schedule Fields

To change the order in which fields are listed in the schedule, use the up/down arrows at the left edge of each row to move the field up or down.



See also [Schemes Fields Panel](#).

Sort By Value (Ascending/Descending)

For any field in Scheme Settings: Activate the up/down arrow to sort the items of this field in ascending/descending order; or leave this field blank, for “None”. “None” means that this field will not be taken into consideration when sorting.

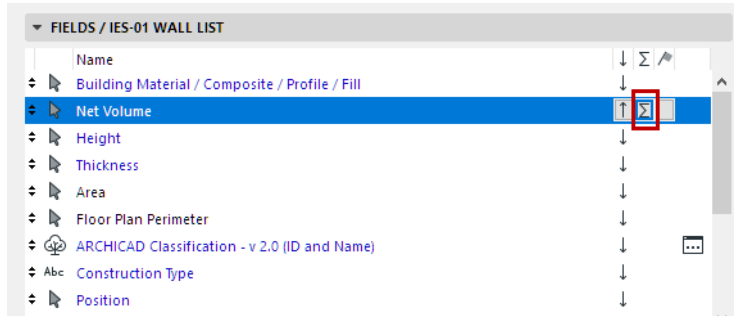
Note: If “Show Headline” is checked, and the first field’s sorting arrow is blank (“None”), then the sorting is automatically set to a descending order based on the first field (the source of the Headline)

In this example: the Volume field has activated the up arrow. The listed walls that are of the same structure type will be listed in order of their volume (top value the highest).

Wall Type	Net Volume [m3]
100 Block Insulated Cavity Plastered	6.27
100 Block Insulated Cavity Plastered	6.20
100 Block Insulated Cavity Plastered	6.20
100 Block Insulated Cavity Plastered	6.13
Generic Wall/Shell	6.65
Generic Wall/Shell	6.45
Generic Wall/Shell	6.45
Generic Wall/Shell	6.26

Display Sum

Display a “sum” to add a schedule cell that will contain the sum total for all items of this field.

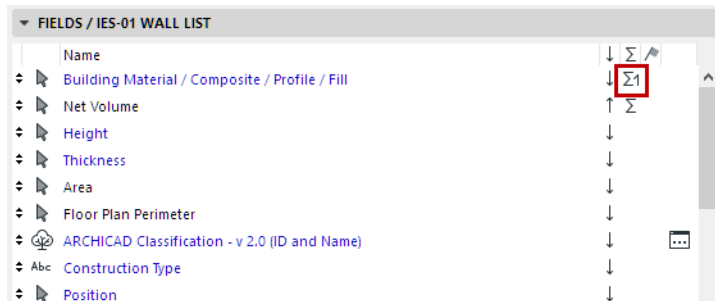


Wall Type	Net Volume [m3]
100 Block Insulated Cavity Plastered	6.27
100 Block Insulated Cavity Plastered	6.20
100 Block Insulated Cavity Plastered	6.20
100 Block Insulated Cavity Plastered	6.13
Generic Wall/Shell	6.65
Generic Wall/Shell	6.45
Generic Wall/Shell	6.45
Generic Wall/Shell	6.26
	50.61 m ³

Display Quantity

To show the quantity of items (rather than their sum), choose the “sum1” icon.

Here, we will have the schedule add a Quantity to the schedule’s Wall Type column.



Wall Type	Net Volume [m3]
100 Block Insulated Cavity Plastered	6.27
100 Block Insulated Cavity Plastered	6.20
100 Block Insulated Cavity Plastered	6.20
100 Block Insulated Cavity Plastered	6.13
Generic Wall/Shell	6.65
Generic Wall/Shell	6.45
Generic Wall/Shell	6.45
Generic Wall/Shell	6.26
8	50.61 m ³

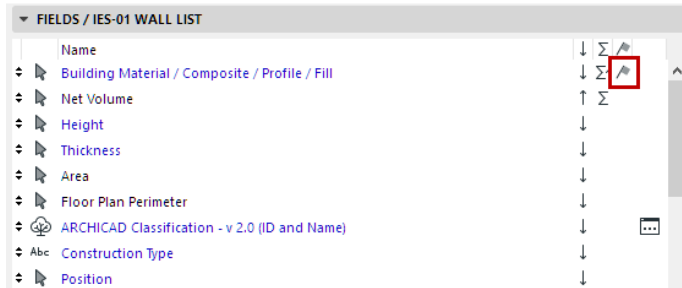
Display Subtotal

Display the flag icon next to the one schedule field to display its subtotal.

This field will display a separate sum and/or quantity for each group of identical elements in that field.

For example, the Building Material/Composite field already has a “Quantity” row, showing the total number of walls; to show a Quantity and Sum for each type of wall in that field, we have added a flag to that field in Scheme Settings.

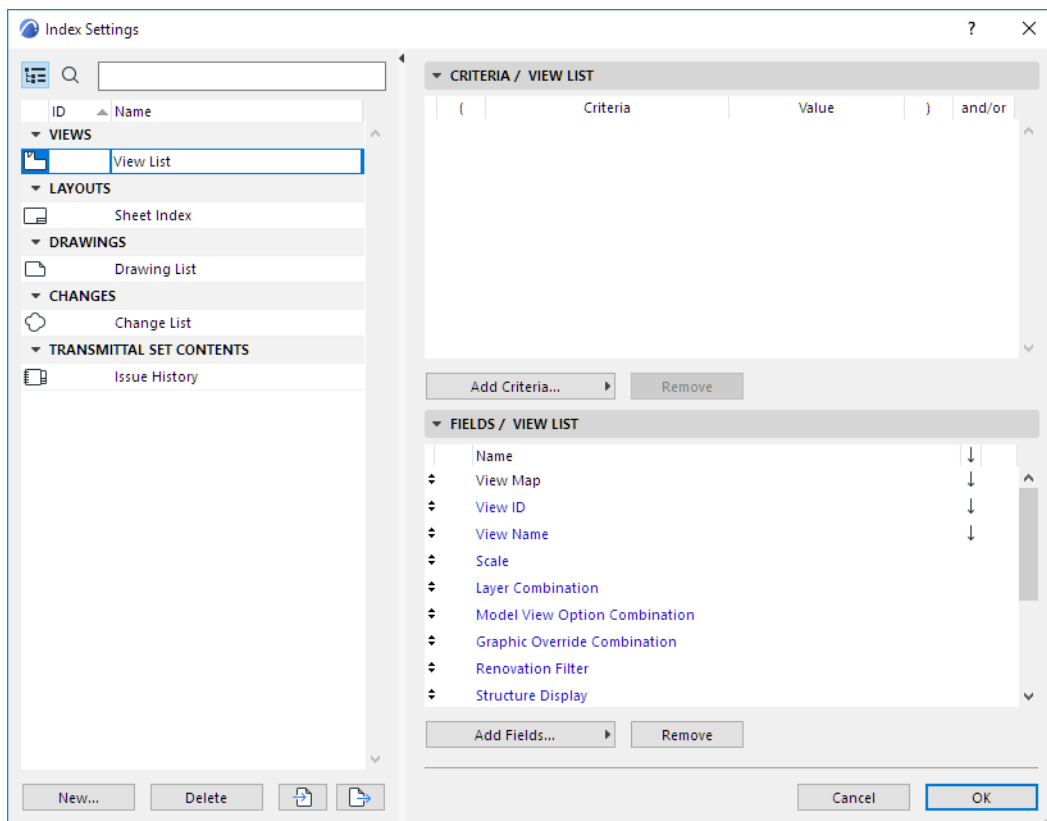
The result: Wall Types show a separate quantity for each separate wall type. Moreover, the Volume column also lists the corresponding volume sub-totals.



Wall Type	Net Volume [m3]
100 Block Insulated Cavity Plastered	6.27
100 Block Insulated Cavity Plastered	6.20
100 Block Insulated Cavity Plastered	6.20
100 Block Insulated Cavity Plastered	6.13
4	24.80 m ³
Generic Wall/Shell	6.65
Generic Wall/Shell	6.45
Generic Wall/Shell	6.45
Generic Wall/Shell	6.26
4	25.81 m ³
8	50.61 m ³

Index Settings Dialog Box

For general information, see [Project Indexes](#).



To access this dialog box, do one of the following:

- Choose **Document > Project Indexes > Project Index Schemes** command.
- Use the Project Index's context menu in the Navigator Project Map structure.
- Select Project Indexes in the Navigator Project Map structure and click "Settings" at the bottom of the Navigator panel.
- Select an Index in the Navigator View Map structure and use its context menu.

The left panel lists the project's Indexes, grouped by type. Select the Index you want to edit, or use the buttons at the bottom to create, duplicate, delete, import and export Index schemes.

The right side shows the Criteria and Fields of the selected Index.

- Use the Criteria Panel of Index Settings to define the items to be listed in the Index.
- Use the Fields Panel of Index Settings to define which parameters of the chosen criteria should be displayed in the Index.

The controls are identical to those of the Interactive Schedule Scheme Settings.

Related Topics:

[Schemes Panel](#)

[Criteria Panel](#)

[Schemes Fields Panel](#)

View Settings Dialog Box

Use the controls in the View Settings dialog box to adjust the settings of the currently selected view.

View Settings

Get Current Window's Settings Selected: 1, Editable: 1

IDENTIFICATION

ID: By Project Map 0.

Name: Custom GF bathroom

Source: 0. Ground Floor

GENERAL

Layer Combination: 02 Drafting

Scale: 1:100

Structure Display: Entire Model

Pen Set: 01 Color Conceptual

Model View Options: 03 Building Plans

Graphic Override: No Overrides

Renovation Filter: 00 Show All Elements

Note: Regardless of this view's scale, GDL objects will be represented according to the source viewpoint's scale.

DESIGN OPTIONS

Option Combination: Main Model Only

2D/3D DOCUMENTS

Floor Plan Cut Plane Settings...

Dimensioning: Plain Meter

Zooming: Zoomed Area

Ignore zoom and rotation when opening this view

3D ONLY

STRUCTURAL ANALYSIS

Structural Analytical Model: Disabled

Load Case: Load Case 1

Cancel OK

Open View Settings from any of the following:

- The tab context menu of the opened view
- The view item's context menu in the Navigator (or Navigator Palette/Organizer)
- The "Settings" button at the bottom of the Navigator View Map

Get Current Window's Settings: Click this button to overwrite the settings in this dialog box to correspond to the view currently in effect in the active (frontmost) window.

View Settings also opens whenever you use the **Save View** or **Save as View** commands.

For more information, see [Navigator View Map](#).

View Settings Identification Panel

- **ID:** Choose a method for assigning an ID to this view: None, Custom (enter desired ID in adjoining text field), or By Project Map.

“**By Project Map**” means that the view will inherit the ID from its corresponding Viewpoint in the Project Map.

- **Name:** Choose a method for assigning a name to this view: either Custom (enter desired name in adjoining text field), or By Project Map.

“**By Project Map**” means that the view will inherit its Name from its corresponding Viewpoint in the Project map.

- **Source:** This field provides feedback on the view’s source viewpoint, or the file path of its source file.

View Settings General Panel

- **Layer Combination:** Choose a Layer Combination to be applied to this view.

For more information, see [Layer Combinations](#).

- **Scale:** Choose a Scale to be applied to this view.

For more information, see [Scale](#).

- **Structure Display:** Choose a Partial Structure Display setting for this view.

For more information, see [Partial Structure Display](#).

If this pop-up is gray: This means that the original viewpoint is either Manual Update or Drawing, so you cannot edit the Partial Structure Display of this view. You can change the viewpoint to Auto Update, or else create a new view using the Partial Structure Display setting you need.

- **Pen Set:** Choose a Pen Set to be used with this view.

For more information, see [Pen Sets](#).

- **Model View Options:** Choose a Model View Option Combination to be applied to this view.

For more information, see [Model View Options](#).

- **Graphic Override:** Choose a Graphic Override Combination to be applied to this view.

For more information, see [Graphic Override Combinations](#).

- **Renovation Filter:** Choose a Renovation Filter to be applied to this view.

For more information, see [Renovation](#).

View Settings Design Options Panel

Choose a Design Option Combination for this view.

See [Working with Design Option Combinations](#)

View Settings 2D/3D Documents Panel

Floor Plan Cut Plane Settings

Click this button to access the Floor Plan Cut Plane Settings dialog box. Any adjustments to these settings will be applied to this view only, not to the project as a whole.

For more information, see [Floor Plan Cut Plane \(Global Setting\)](#).

Dimensions

Choose a Dimensioning standard from this pop-up menu. The standard you choose here will be applied to this view only, not to the project as a whole.

For more information, see [Dimensions Preferences](#).

Zooming

Choose a zoom to save with this view:

Note: A view's saved Zoom value includes the Pan and the Oriented View (rotation), if any.

- **Current Zoom:** The view will save the current zoom as part of its settings.

Note: Once a view is saved at "Current Zoom," that setting is henceforth called Zoomed Area in this pop-up list.

- **Fit in Window:** whenever you open this view, or whenever drawings from this view are updated or published, they will be displayed to include the project window's entire contents, subject to Layer status
- any custom-named Zoom you have saved earlier.

See [Save Zooms](#).

For more information, see [Zoom](#).

Ignore zoom when opening this view

If this box is checked, then whenever you open this view, it will be displayed using the zoom value of the last opened Archicad window (not the saved zoom of this view). However, the saved zoom level of this view will still be applied when you place this view as a Drawing, or publish it.

Note: A view's saved Zoom value includes the Oriented View (rotation), if any.

View Settings 3D Only Panel

Filter and Cut Elements in 3D...

Click to open the dialog box.

See [Filter and Cut Elements in 3D Dialog Box](#)

Generate in: Choose whether to generate the 3D view in the 3D window or in the PhotoRendering window.

3D Style: The settings for 3D Window display.

See [3D Styles](#).

Rendering Scene: Apply a saved Scene if you choose to always generate this 3D view as a PhotoRendering.

Size: View or edit the size of a rendered view.

Additional info is shown about:

- whether Render Safe Frame is applied

For more information, see [Select and Manage Rendering Scenes and Render Safe Frame](#).

- how the 3D-related and rendering-related settings (collectively "Image settings") are saved with the current view.

Redefine Image settings with current: Check this box if you want to overwrite the stored Image settings with the Image settings now in effect in the current (frontmost) window.

Note: Image settings are the following: 3D projection settings (including zooming and sunlight settings), filter elements in 3D, 3D Styles, 3D cutting planes, 3D cutaway and photo rendering settings.

If you check this box, the panel indicates whether the 3D view in the current (frontmost) window has been filtered or limited by a marquee, and whether Render Safe Area mode is on.

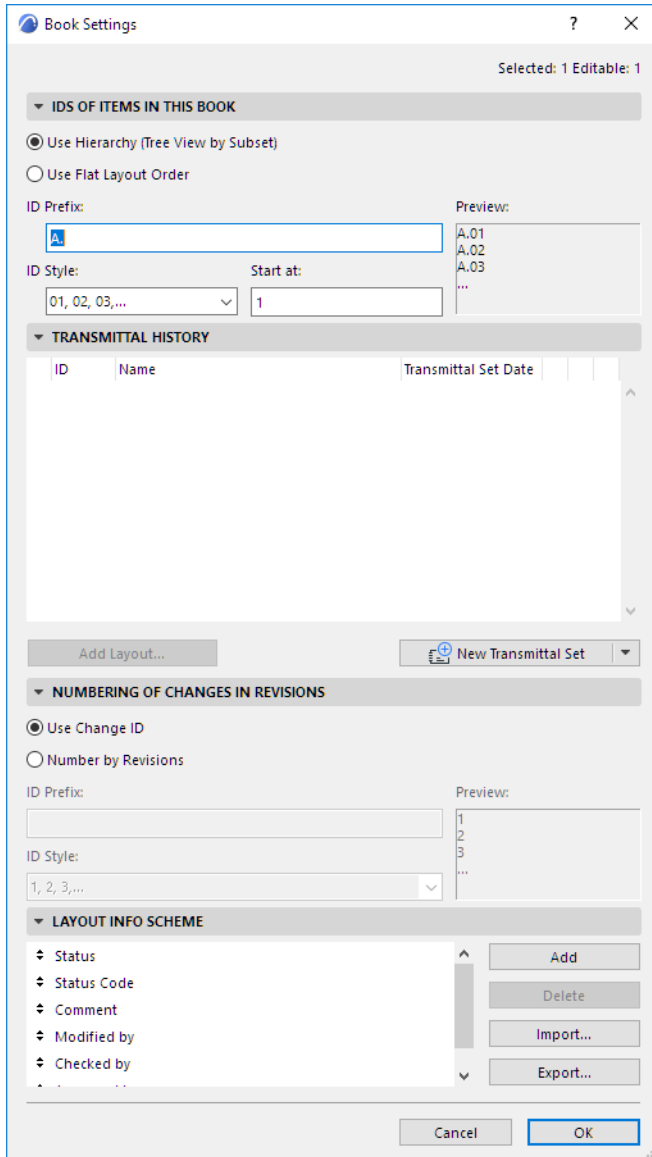
If you now click OK to close the View Settings dialog box, the 3D view will be redefined accordingly (i.e. it will only display the filtered or selected elements).

View Settings Structural Analysis Panel

- **Structural Analytical Model:** Enable or disable the Structural Analytical Model for this view.
- **Load Case:** Choose the visible Load Case for this view.

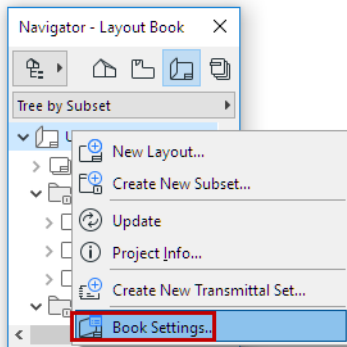
See [Structural Analytical Model](#).

Book Settings Dialog Box



To access this dialog box, do one of the following:

- Use the **Document > Layout Book > Book Settings** command
- Use the context menu of the Layout Book icon in the Navigator Layout Book structure, or the Settings button at the bottom of the Navigator Palette when the Layout Book icon is selected in the tree view.



- Click the Transmittal History button from the Change Manager

IDs of Items in this Book Panel

- The **Use Hierarchy** option assigns IDs that reflect the position of the Layouts within the tree view hierarchy, including their position in Subsets. In this system, the Subsets can also receive an ID, which shows up as part of the ID assigned to each of its Layouts.

Subset IDs are defined in Subset Settings.

For more information, see [Subset Settings Dialog Box](#).

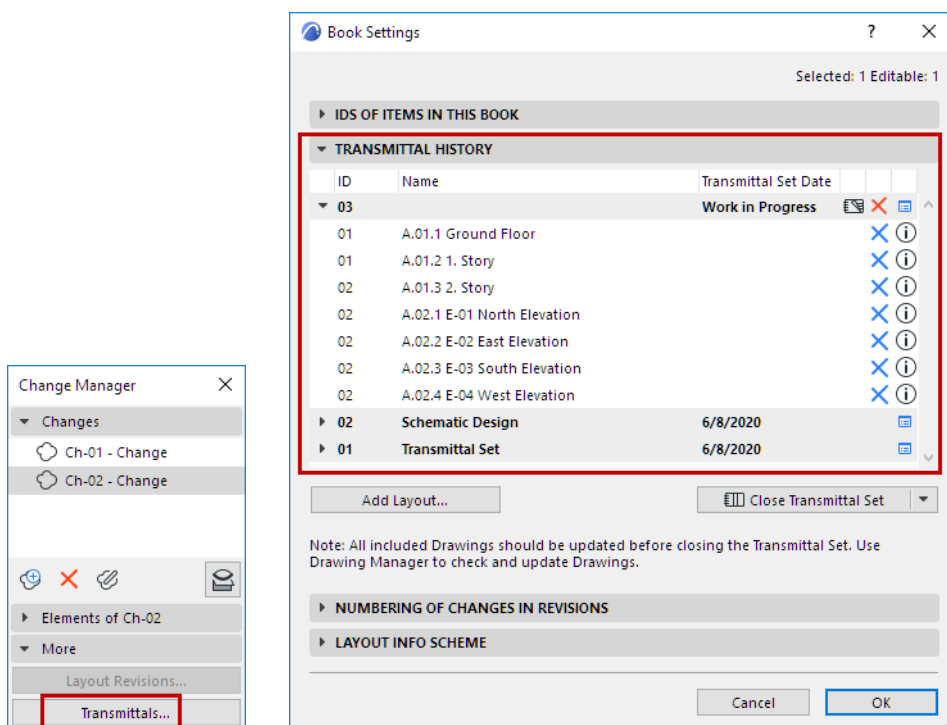
- The **Use Flat Layout Order** option assigns IDs to all the Layouts of the Book. The ID assignment is performed in the Navigator from top to bottom regardless of the Layouts' position in the hierarchy. In this case, Subsets will have no IDs. The Layout IDs will use the prefix and style you define here.

For more information, see [Layout and Drawing IDs](#).

Transmittal History Panel

Use this panel of Layout Book Settings to create and manage Transmittal Sets as part of Revision management.

Note: You can access this panel easily from the Change Manager Palette.



This panel lists all Transmittal Sets of the project, with the Layout Revisions that belong to each Transmittal Set.

For a selected Transmittal Set:

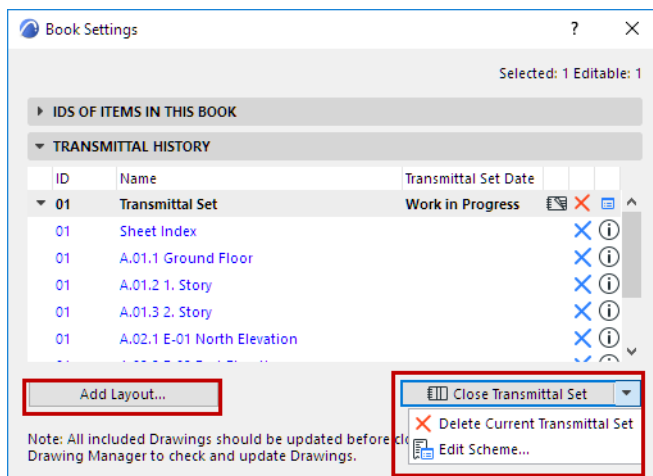
- click the icon to access its Transmittal Set Details
- click the red X to delete the Transmittal Set
- if the Transmittal Set is closed, its **Transmittal Set Date** is listed here. The Transmittal Set Date for an open Transmittal Set is shown as “Work in Progress”.

For a listed Layout Revision:

- click the Info button to view any Changes associated with that Revision
- if the Transmittal Set is still open: click the blue X to remove the Layout from the Transmittal Set

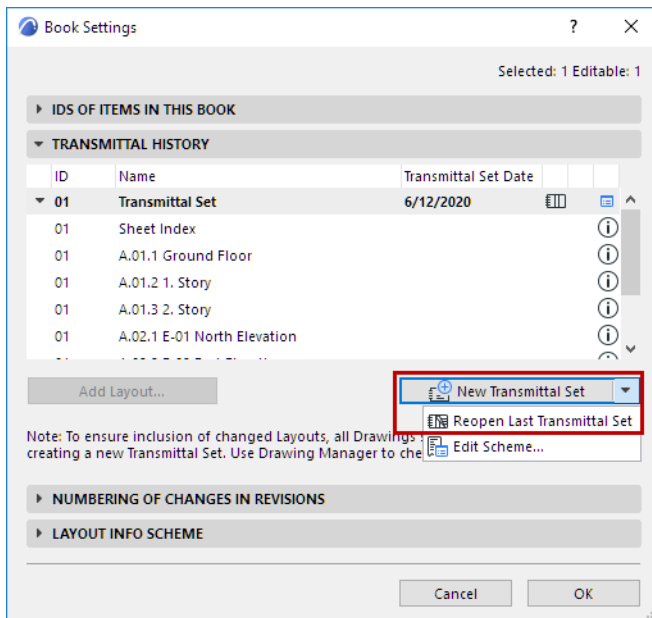
Only one Transmittal Set can be open at a time. When an Transmittal Set is open, the following commands are available on this panel:

- Close Transmittal Set
- Delete Transmittal Set



- Add Layout

When the current Transmittal Set has been closed, the panel looks like this:



Change Numbering in Revisions Panel

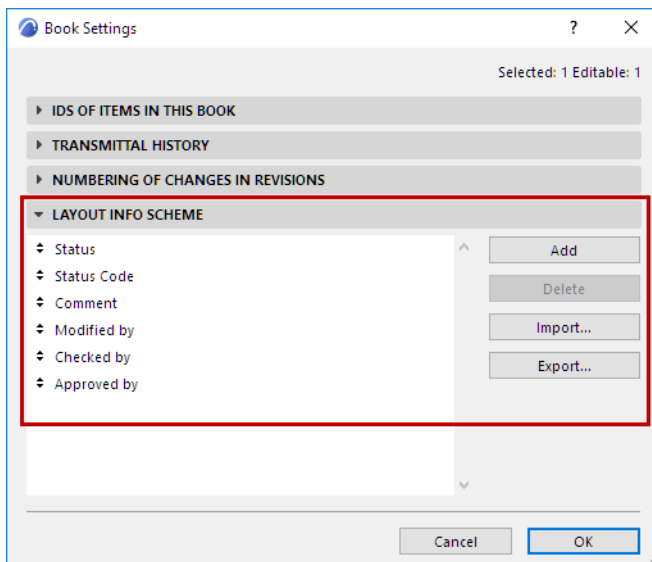
Set a Numbering logic for Revision Numbering for the current Transmittal Set:

- **Use Change ID (Default):** Changes should appear on Layouts using their own global ID
- **Number by Revisions:** Changes should appear on Layouts using the following ID: LayoutRevision ID + Suffix + numbering.

If you choose Number by Revisions, this will affect Change Numbering only where it appears in the Layout context: that is, in the Change Marker autotext; in the Revision History object on the Layout; and in the Layout Settings Revision History.

Layout Info Scheme Panel

Use this panel to customize or add new fields to be used for Layout-specific information.



For information on using these fields, see [Layout Info Panel \(Layout Settings\)](#).

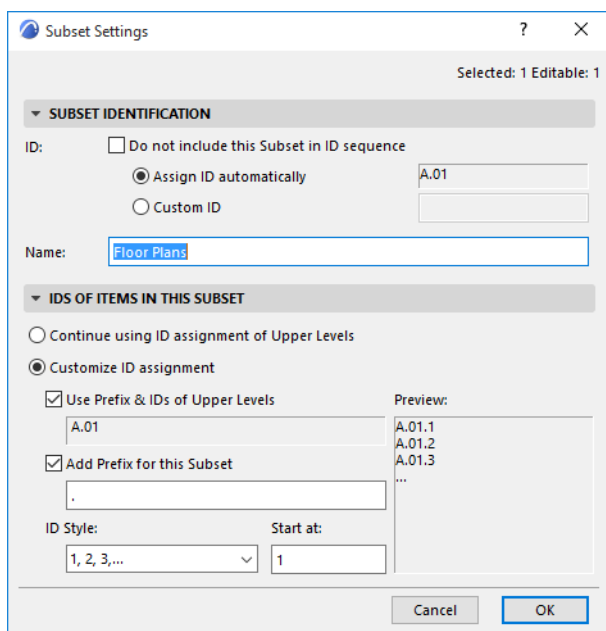
Subset Settings Dialog Box

Subset Settings provides a way to assign IDs to the Layouts contained in the selected Subset. The ID definitions in Subset Settings are applied if you have selected “Use Hierarchy” as the ID assignment method in Book Settings.

The Subset Settings dialog box can be accessed from the Navigator in one of three ways, provided that the Tree by Subsets view is displayed and the given Subset is selected:

- Click the **Settings** button at the bottom of the Navigator palette.
- Choose the **Subset Settings** command in the context menu opened by right-clicking (or Ctrl-clicking) the name of the Subset.
- Click the “Settings” icon at the middle of the Navigator palette.

For more information, see [Navigator](#).



Subset Identification Panel

Use the controls in this panel to assign an ID to the selected subset.

For more information, see [Layout and Drawing IDs](#).

- **Do not include this Subset in ID sequence:** Check this box to exclude the Subset from automatic ID assignment. (If you check this box, the following option for automatic ID assignment is grayed.)
- **Assign ID Automatically:** Choose this option to assign an ID automatically, based on the Prefix and Style defined in Book Settings.
- **Custom ID:** Choose this option to assign a Subset ID of your choice.

IDs of Items in this Subset Panel

These controls affect the IDs of Layouts contained in this Subset.

- **Continue Using ID Assignment of Upper Levels:** Check this option if you want Layouts within this Subset to be assigned IDs as if they were not within this Subset, but rather part of the level above. In this case you are only using the Subset as a logical grouping which has no effect on IDs.

- **Customize ID assignment:** Check this option if you want the Layouts inside the Subset to be assigned custom IDs:
 - Use Prefix & IDs of Upper Levels: Layouts in this subset will be assigned IDs based on the previous Layout in the Layout Book structure.
 - Add Prefix for this Subset:
Note: When IDs by letters exhausts the alphabet, the numbering continues as follows: y, z, aa, ab, ac. (The initial letter in double-letter numbering always consists of a single character regardless of the language being used.)

When moving a Subset within the hierarchy, IDs are automatically adjusted accordingly.

Layout Settings

The Layout Settings dialog box contains information about the selected Layout.

To access this dialog box, **select a Layout** - either by opening it in the Layout window, or by selecting it from the Navigator/Organizer. Then do one of the following:

- Choose **Document > Layout Book > Layout Settings**
- Use the Layout's context menu in the Navigator
- Click the "Settings" icon at the middle of the Navigator
- Click "Settings" at the bottom of the Navigator

Layout Identification and Format Panel

For more information, see [Layout and Drawing IDs](#).

- **Do not include this Layout in ID sequence:** Check to exclude the current layout from the automated layout ID assignment.
- **Use Automatic Book and Subset ID assignment:** Check to apply the automated layout book and subset ID assignment to the current Layout, as defined in Book Settings.

For more information, see [Book Settings Dialog Box](#).

- **Custom ID:** Check to apply a unique ID to the current layout.
- **Layout Name:** Enter a Name for the Layout.
- **Master Layout:** Select a Master Layout from the pop-up menu.

For more information, see [Master Layouts](#).

- **Size:** This field provides information on the horizontal and vertical size of the current layout (as defined in its Master Layout).

Note: Sizes are based on Working Units for Layout (Options > Project Preferences).

Drawing IDs on this Layout Panel

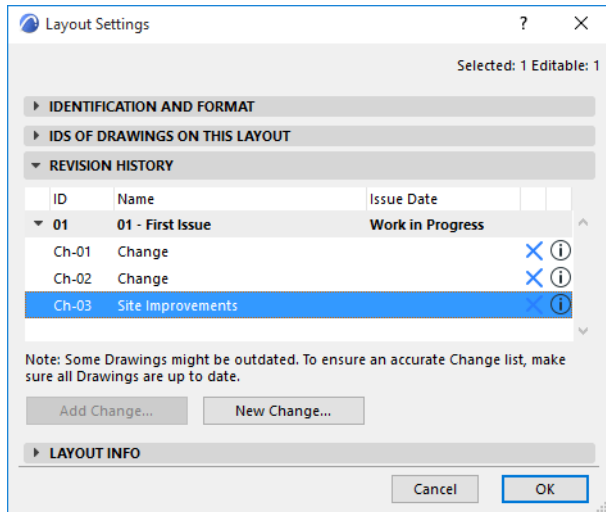
Settings in this panel will determine the IDs of all Drawings placed on this Layout, provided that the Drawing ID is set to "By Layout" in the Identification panel of its Drawing Settings dialog box.

For more information, see [Drawing Identification Panel](#).

- **Continue Drawing ID sequence from previous Layout:** Check this box to assign an ID by continuing the ID sequence from the previous layout. (This control is grayed if the selected Layout is the first one in the Layout Book.)
- **Drawing ID Prefix:** Enter item prefix characters for the drawing ID.
- **Drawing ID Style:** Choose any of the listed ID styles from this pop-up field.
- **Start at:** Enter either a number or a letter (depending on the selected ID style) with which to begin the ID sequence.
- **Preview:** This area shows a preview of the ID style you have selected.

Revision History Panel

The panel lists each Revision of this Layout, plus the Changes associated with each Revision and Transmittal Set.



Click **Add Change** to add an existing Change to this Layout.

[See also Add an Existing Change to a Layout.](#)

Click **New Change** to create a new Change in the project and add it to this Layout's current Revision.

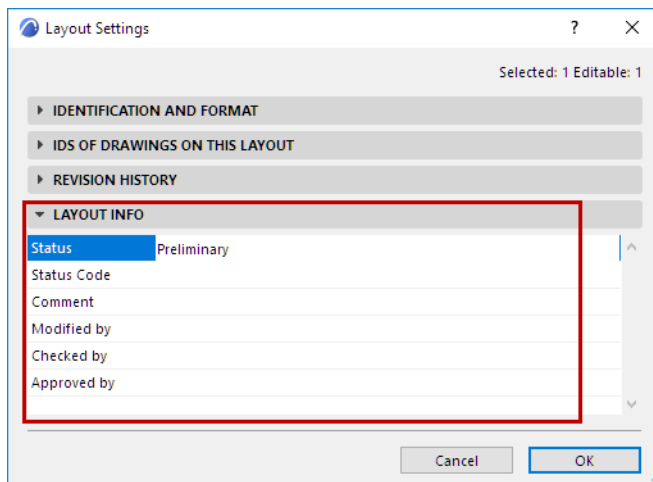
[See also Create New Change on a Layout Revision and Revision History Data on Layout and in Layout Settings.](#)

Layout Info Panel (Layout Settings)

Use these fields to define basic data values for the current Layout.

To customize these fields or add new ones, use the Layout Scheme Panel of Book Settings.

[See Book Settings Dialog Box.](#)



This can be useful if you need to differentiate these data for specific Layouts.

- For example, the project may already be in the construction phase, yet one Layout may still have a Preliminary status, while another Layout has the status of Approved by Contractor.

- Similarly, different parts of the documentation (e.g. Details vs. Floor Plans) may be the responsibility of different people, so you can use the Modified by/Checked by/Approved by fields to specify these names for any given Layout.

These data can then be displayed in the following parts of the Archicad interface:

- Autotexts placed on Layouts
- Names of Publisher Items
- Sheet Indexes
- Transmittal Set History Index
- Revision History object

Master Layout Settings

For general information, see [Master Layouts](#).

This dialog is available from

- Context menu of Layout or Master Layout
- In Navigator: Select Master Layout, then click Settings
- **Document > Layout Book > Master Layout Settings**

Name and Size

- **Name:** Enter a Name for the Master Layout in the text field.
- **Size:** Define the Master Layout size using the standard paper sizes listed in the drop-down list, or the Custom option. The two fields below give you feedback on the horizontal and vertical dimensions of the Layout.
Note: Sizes are based on Working Units for Layout (Options > Project Preferences).
- **Margins:** Enter values for the Master layout's four margins in the text fields.
- Choose **Portrait** or **Landscape** as the Master Layout's orientation.

Printable area: This area gives you feedback on the Master Layout's printable area (taking its Size and Margins into consideration).

Anchor Point: Click one of the nine points to use as an anchor for your master layout.

Display Master Layout

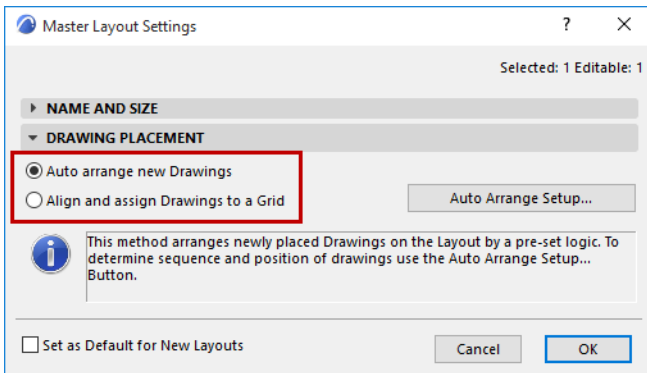
- **Above Layout:** Master Layout items will be placed on top of the Layout (in some cases, obscuring some of the Layout content).

- **Below Layout:** Master Layout items will be displayed behind the Layout. Contents of the Layout may obscure some Master Layout items.

Drawing Placement

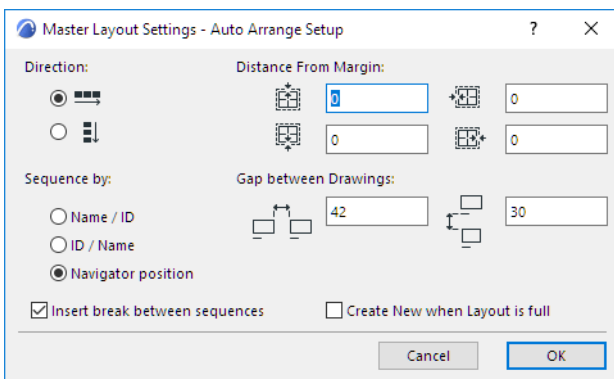
Choose one of the two logics for placing multiple drawings on the layout.

Note: The default logic is “Auto Arrange.” If you are placing only a single drawing on the Layout, it will be placed in the center of the Layout.



Auto Arrange New Drawings

If you choose this option, click **Auto Arrange Setup** to set the following:



Direction: Choose either horizontal or vertical to determine the direction in which subsequent drawings are placed on the Layout.

Distance from Margin: For each of the four sides of the Layout, define a margin of space.

Sequence by: When placing multiple drawings at once, the Sequence setting determines the order in which they appear on the Layout.

- **Name/ID:** Alphabetical by name; in case of identical names, drawings are placed in order of ID (numerical or alphabetical).
- **ID/Name:** In ID order; in case of identical IDs, drawings are placed according to their name (alphabetical order).
- **Navigator position:** Drawings are placed in the order (top to bottom) that they are listed in the Navigator View Map. (Floor Plans are an exception; they are placed according to their View Map order from the bottom up, e.g. Story 1, 2, 3...).

Insert break between sequences: “Sequence” depends on your choice in the **Sequence by** options above:

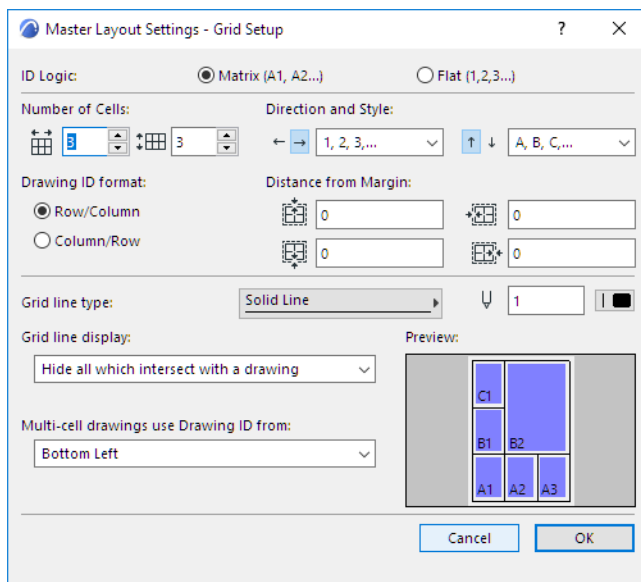
- if you are sequencing by Name/ID (or ID/Name): Check this box to start a new row/column every time the Name (or ID) changes.
- If you are sequencing by Navigator position: Check this box to start a new row/column for each new folder in the Navigator View Map. For example, Drawings from the Details folder will be placed in one row, from the Sections folder in the next, etc.

Gap Between Drawings: Define the length of the gaps between the rows and columns of placed drawings. (Length units here are Layout units, defined in **Options > Project Preferences > Working Units.**)

Create New when Layout is Full: Create a new Layout if the current layout is full.

Align and Assign Drawings to a Grid

If you choose this option, click **Grid Setup** to access the grid settings options. As you set these options, consult the **Preview** picture.



ID Logic: Assign an ID to each grid cell by a defined logic:

- **Matrix:** Each cell ID is based on its column and row number
- **Flat:** Cell IDs are numbered in sequence.

Number of Cells: Define number of columns and rows in your Grid.

Depending on which ID assignment logic you choose (Matrix or Flat), the subsequent controls will vary.

Matrix-Style ID Options

Direction and Style

- Choose how to assign IDs to grid columns/rows (left/right, and up/down)
- Choose an ID numbering style

Drawing ID Format: For Matrix-style IDs, define numbering format:

- Row, then Column, or
- Column, then Row

Flat-Style ID Options

Start From: Define one corner of the grid as the starting point for assigning cell IDs

Style: Choose an ID numbering style

Primary Direction: Define the direction (horizontal or vertical) in which to assign the cell IDs, starting from the corner you designated (Start From).

The rest of the controls in the panel apply to the Grid regardless of ID logic.

Grid Line Display Options

Note: These Grid line display settings also affect printed output.

- **Distance from Margin:** Define the distance of the Grid from the paper margin (top, bottom, left and right).
- Choose line type and pen color
- **Show/Hide All Grid Lines**
- **Hide all which intersect with a drawing:** Check this option to hide such grid lines.

Multi-Cell Drawings

Define which cell ID should be assigned as the drawing ID (You will choose one of the drawing's four corners; the ID of the cell which that corner occupies becomes the drawing ID.)

Set as Default for New Layouts

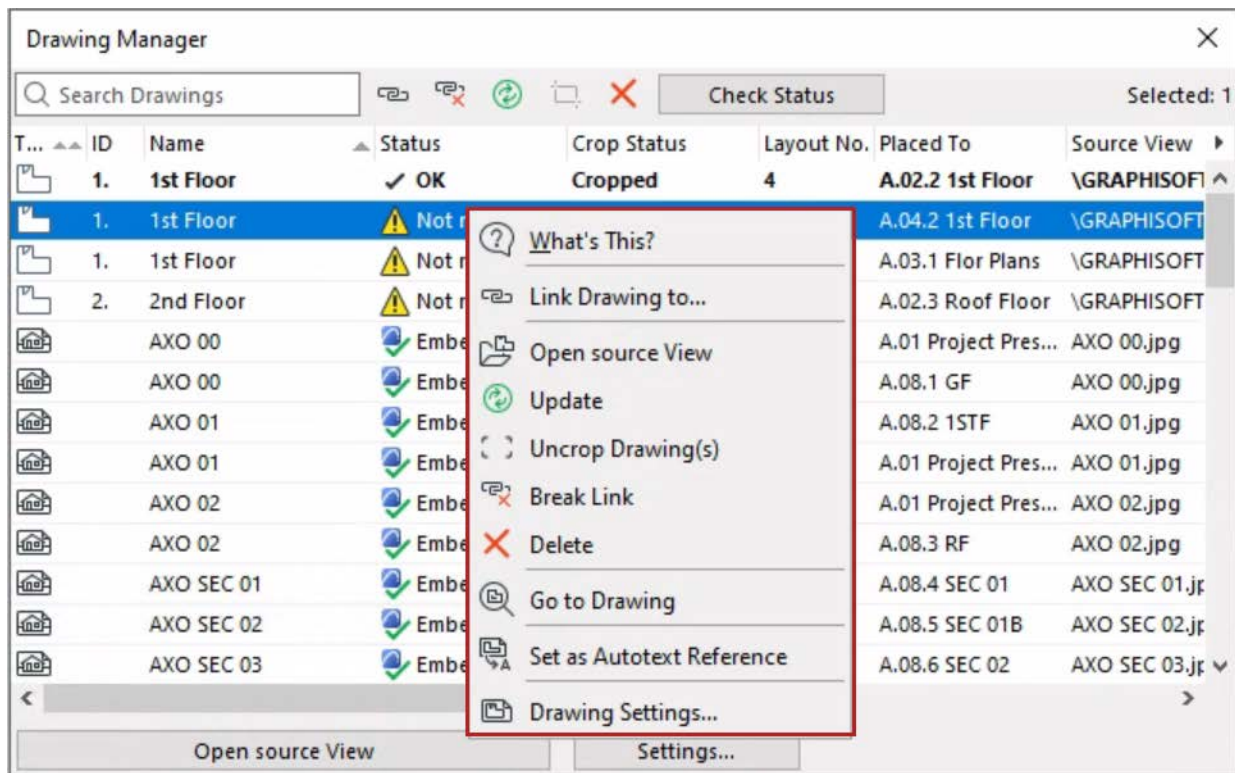
Check this box if you want this Master Layout to serve as the default Master Layout for new Layouts.

Drawing Manager

Drawing Manager lists all the Drawings you have placed in the Project (both Layout Book and Model views), including those referring to external projects, and external drawings such as PDF files.

Use the Drawing Manager commands to check the status of the placed drawings, modify their update type, reestablish, repair or change links.

For any selected drawing in the list, context menu commands are also available.



You cannot edit a selected drawing's parameters directly in the Drawing Manager. To edit drawing parameters, use Drawing Settings.

You can also check Drawings and access Drawing Manager through the Action Center. (**File > Info > Action Center**).

Topics in this Section:

[Open Drawing Manager](#)

[List of Drawings](#)

[Drawing Properties](#)

[Drawing Update Status](#)

[Drawing Manager Commands](#)

Related Topics:

[Action Center](#)

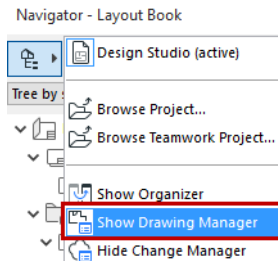
[Drawing Settings](#)

[Managing and Updating Placed Drawings](#)

Open Drawing Manager

Do one of the following:

- choose **File > External Content > Drawing Manager**
- click the Drawing Manager icon from the pop-up menu at the upper left corner of the Navigator/Organizer:

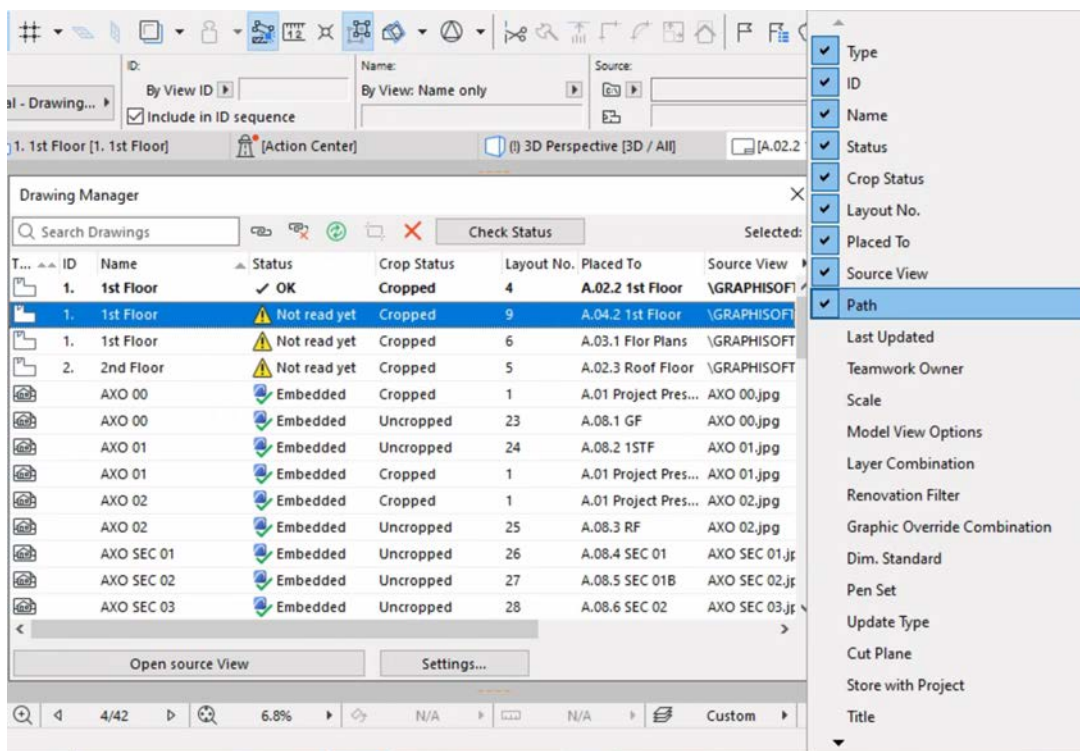


- choose **Window > Palettes > Drawing Manager**

List of Drawings

If the active window contains a Drawing, it is listed in the Drawing Manager in **bold**.

- Default info for each drawing: Type, ID, Name, Status, Crop Status, Layout (Placed to), Source View, Path
- Toggle columns on/off: right-click any column header to see list of available columns, or click the black arrow at the right end of the list columns



- Sort columns by clicking on the column header
- Secondary sorting: click a second column header to further sort within the hierarchy obtained by the first sorting
- Resize column widths graphically

Drawing Properties

- **Type:** Indicates the Drawing's source type (e.g. a Floor Plan view or a PDF file).
- **ID:** Shows Drawing ID.
- **Name:** Shows Drawing name.
- **Status:** [See *Drawing Update Status* below.](#)
- **Placed To:** Name of Layout or Model View on which Drawing is placed.
- **Source View:** Source view as displayed in Navigator Project Map. If the Drawing source is not an Archicad view, then this column displays the source file name and (in the case of a multi-page PDF document source) the page number from which the Drawing is created.
Note: If the Drawing source is a DWG paperspace, this field also indicates the viewport number.
- **Path:** Location of the drawing's source view (e.g. either Internal, or its file path).

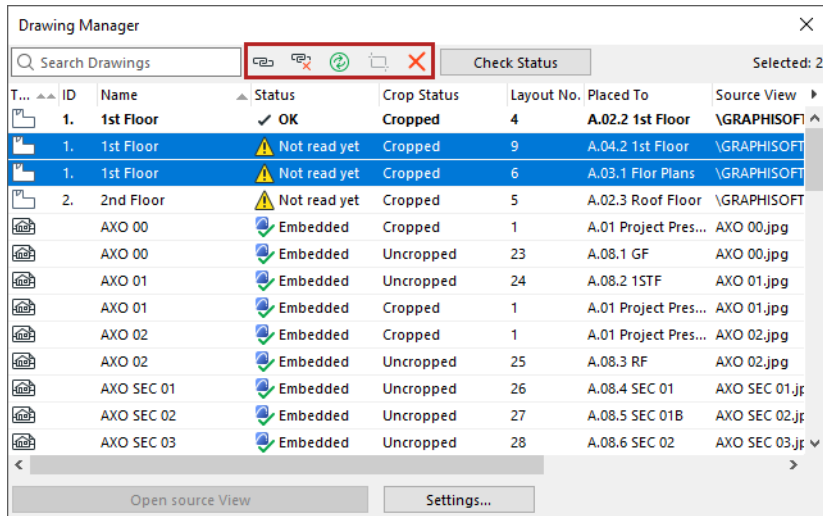
Drawing Update Status

For the selected Drawing(s):

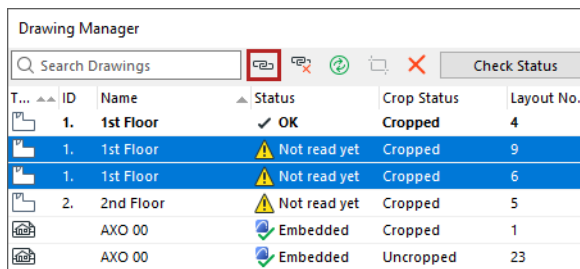
- **OK:** Drawing is updated.
- **Modified:** Source has been modified. Click the **Update** button to update the drawing.
- **Missing:** The source of the linked drawing cannot be found.
- **Not Accessible:** You do not have access to this source, because you are using either a Demo version of Archicad, or a non-Teamwork protection key; or you do not have Teamwork access rights to this source.
- **To be Updated:** Temporary status of a drawing included in the set of Drawings to be updated, but the update process has not affected this drawing yet.
- **Updating:** Temporary status of a drawing currently being updated.
- **Embedded.** This Drawing's link has been broken and cannot be updated. (The Drawing Manager displays its former source view for information purposes only.)
- **Needs Checking:** Archicad's automated background quick-check function cannot determine definitively whether the drawing needs an update or not. If this status appears, you can do one of the following:
 - Click the **Check Status** button to see whether the drawing is either OK or Modified. In the latter case, if the drawing's update method is Manual, you can decide whether to update it or not. (A Drawing set to Automatic update will be updated as soon as you activate its Layout or window.)
 - Click **Update**

Drawing Manager Commands

The commands (icons) at top left apply to the drawing(s) selected in the list. All applicable commands are also available from a selected Drawing's context menu.

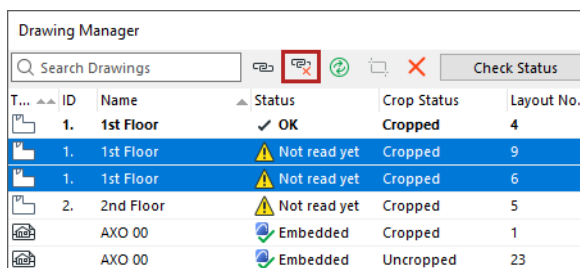


Link Drawing



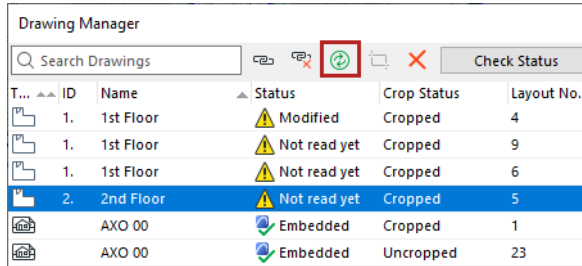
Links the selected drawing to either an external source or an internal view.

Break Drawing Link



Breaks the link of the selected drawing. The drawing is now embedded into the project. It cannot be updated, until you again link it to a source.

Update Drawing



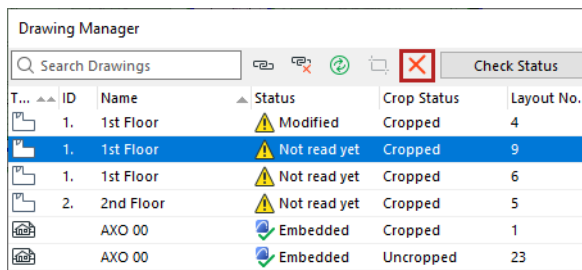
Click to update the drawing.

Crop Drawing to Frame

For a selected Drawing with a Manually resized Frame, which is not cropped: this command crops the unseen Drawing content.

[See *Drawing Frame*.](#)

Delete Drawing



Click to delete the selected drawing(s) after confirming the warning message.

Note: Deleting a drawing is undoable ONLY if the window containing the drawing is active at the time you delete it.

Check Status

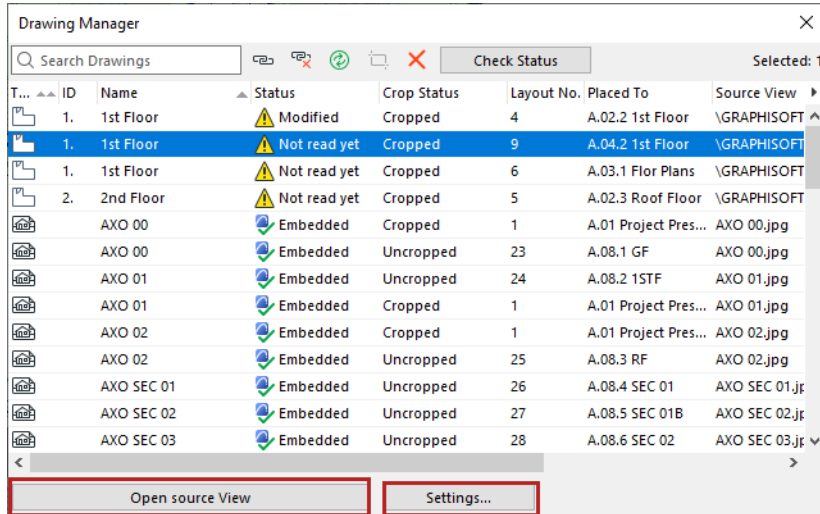
Check the update status of the selected drawing(s), if its status shows “Needs Checking”.

Open Source View

Open the source view of the selected drawing. (This button is grayed for Drawings having an external source.)

Settings

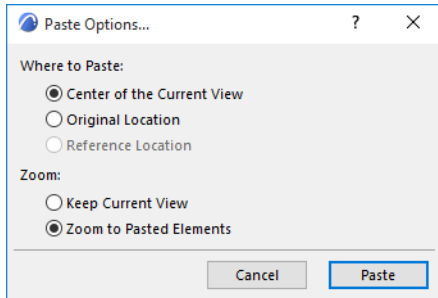
Open the selected drawing(s)' Settings dialog box.



Note: To assign parameters to multiple Drawings at the same time (such as a uniform Pen Set to all Drawings in the Layout Book), select the Drawings in Drawing Manager and click Settings. Any settings you modify will be applied to all the selected drawings (other Drawing Settings will remain as-is for each drawing).

Paste Options Dialog Box

This dialog box appears if you are pasting copied Archicad elements from the Clipboard, and the pasted elements cannot appear in their entirety in the Window at the current zoom level, or if it is possible to locate the pasted elements relative to either the Active view or a Reference view.



Where to Paste

Center of the Current View: Paste the Clipboard contents into the center of the current view with no regard for the pasted elements' source view origin.

Original Location: Paste the Clipboard contents into the current view at the same position as in the original source view (i.e., relative to the origin of the source view).

Reference Location: Paste the Clipboard contents into the current view relative to the origin of the Reference view.

This option is only available if a Reference view is shown; if the Reference view's origin is at a different location from the Active view's origin; and if that Reference view is of the same type (e.g. Floor Plan) as the contents of the Clipboard.

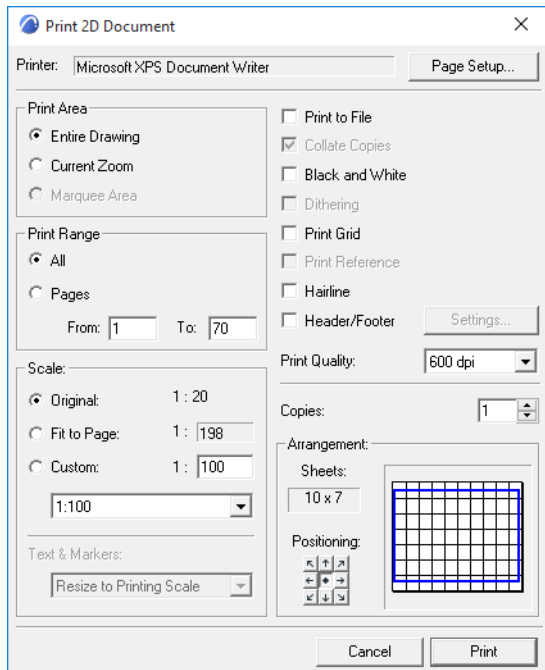
Zoom

Keep Current View: After the **Paste** command, the currently active window will remain at the same zoom level as before.

Zoom to Pasted Elements: After the **Paste** command, the current window will zoom in to the newly pasted elements.

Print 2D Document

This dialog box is opened with the **File > Print** command if a 2D window (including the 3D Document Window) is active, or from the Print Options command when printing from the Navigator Publisher.



Printer: This field displays the printer driver currently set up for your machine. To change the assigned printer, choose the Page Setup button at right.

Page Setup: Click this button to open the Page Setup dialog box, where you can choose a printer and set paper size and orientation.

Print Area: Choose which part of the active 2D window you wish to print.

- **Entire Drawing:** Choose this button to print the entire printable content of the window.
- **Current Zoom Only:** Choose this button to print only the visible content of the current window at its current zoom (including its rotated orientation, if any).
- **Marquee Area:** If you have placed a normal orthogonal Marquee in the active window, choose this button to print only the marquee area. (The option is available for other marquee shapes, but the output area will always be orthogonal.)

Print Range: Choose either **All** to print all pages, or choose a page range for multi-page output. See the Preview at the bottom right of this dialog box for how multi-page output will be arranged.

Scale: Choose a print scale for the output of the active window.

- **Original:** Choose this button to print at the original scale.
- **Fit to Page:** Choose this button to make the output fit the page. The scale will be adjusted accordingly.
- **Custom:** Choose this button to enter any other scale for the output. Either type in a custom scale, or choose a scale from the pop-up list below.

Text & Markers: Available only if you have chosen either *Fit to Page* or *Custom* as the Print Scale. Use this pop-up field to define the scale of text and markers in the output. These options also affect the output characteristics of other elements such as arrowheads, scale-independent dashed line types and fill patterns.

- **Resize to Printing Scale:** Outputs text in proportion to the size of other elements. This method is good for enlargement of a Project for a presentation to be seen from a distance.
- **Fix Size:** Outputs text at its exact size in the current paper units (inches or millimeters). To keep text at exactly the same size even when the printed size of the project changes, choose this option. This is acceptable for enlargements but can produce unwanted effects with reductions. Text may overprint other elements as they shrink.

Print to File: Check this box to save the output as a print file on your computer.

Collate Copies: Check this box if you are printing multiple copies, and would like each copy to be output in sequential order.

Black and White: Check this box for black and white output. With this option, colored elements will be output as follows:

- **Lines:** All colored lines will be black; white lines remain white.
- **Fill:** Foreground will be black; background will be white. Fills using RGB colors will be gray
- **Zones:** All will be displayed in grayscale cover fills.
- **Images:** All will be displayed in grayscale.
- **Trace Reference:** Will always be displayed in grayscale.

Dithering: Check this box to reduce the total number of colors present in the output while retaining visual fidelity. Dithering may be necessary due to a limited number of colors available on the display device. For example, if your fills turn out solid black, the Dithering effect may help.

Print Grid: Check this box if you would like the grid (construction or rotated) to be displayed on your printed output.

Print Reference: Check this box if the active window includes a Trace Reference, and if you would like to include its elements in the printed output.

Hairline: Check this box if you want to print all lines of the document using the thinnest line available on the printing device.

Header/Footer: Check this box if you wish to include a Header or Footer on the printed output. To define the Header/Footer content, click the Settings button at right.

For more information, see [Header/Footer Settings](#).

Print Quality: Enter a dpi (dots per square inch) setting to define print quality.

Copies: Enter the number of copies you wish to print.

Arrangement: This section helps you arrange the drawing on the output sheet(s). A preview window helps you get the desired outcome. In the preview window, the black square(s) represent the page(s) to be printed. The blue rectangle represents the drawing size. The Preview indicates the position of a Header/Footer if you have checked that option.

- **Sheets:** Shows the number of sheets of the paper size chosen in Page Setup.
- **Positioning:** Click these arrow-buttons to move the printed content around on the output sheets as shown in the Preview.

Print 3D Document

This dialog box appears if you issue the **Print** command while the 3D Window is active, using the Vectorial 3D Engine, or from the Print Options command when printing from the Navigator Publisher.

The controls are identical to those of the Print 2D Document dialog box, except certain 2D-related options (Text/Marker scale, grid, Trace Reference, etc.) are not available here.

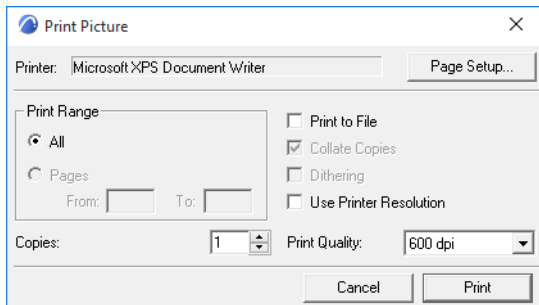
See [Print 2D Document](#).

If you issue the **Print** command while the 3D Window is active, using the Hardware acceleration Engine, then the Print Picture dialog box comes up.

See [Print Picture](#).

Print Picture

This dialog box appears if you issue the **Print** command while the 3D Window is active, using the Hardware acceleration Engine.



This printing function treats the entire contents of the 3D window as a single image. Consequently, the only available controls are:

Printer: This field displays the printer driver currently set up for your machine. To change the assigned printer, choose the Page Setup button at right.

Page Setup: Click this button to open the Page Setup dialog box, where you can choose a printer and set paper size and orientation.

Print Range: Choose either **All** to print all pages, or choose a page range for multi-page output.

Copies: Enter the number of copies of the image to print.

Print to File: Check this box to save the output as a print file on your computer.

Collate Copies: Check this box if you are printing multiple copies, and would like each copy to be output in sequential order.

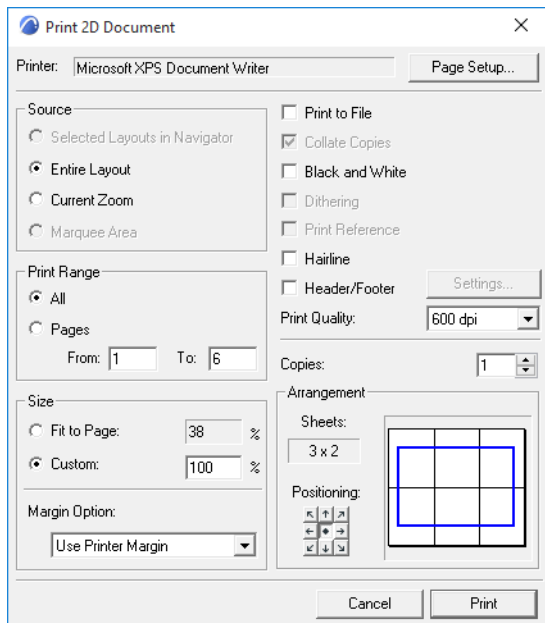
Dithering: Check this box to reduce the total number of colors present in the output while retaining visual fidelity. Dithering may be necessary due to a limited number of colors available on the display device. For example, if your fills turn out solid black, the Dithering effect may help.

Use Printer Resolution: Check this box to use the print resolution set up in the printer device.

Print Quality: Enter a dpi (dots per square inch) setting to define print quality.

Print Layout

This dialog box is opened with the **File > Print** command if Layout is active.



Printer: This field displays the printer driver currently set up for your machine. To change the assigned printer, choose the Page Setup button at right.

Page Setup: Click this button to open the Page Setup dialog box, where you can choose a printer and set paper size and orientation.

Source: Choose which layout(s) you wish to print.

- **Selected Layouts in Navigator:** Click this button to print only those layouts which are selected in the Navigator palette.

Note: This option is inactive if nothing is selected in the Navigator, or if several Layout Books (in the Navigator and Organizer) are open.
- **Entire Layout:** Print the entire currently active layout.
- **Current Zoom:** Print only the visible content of the active layout at its current zoom.
- **Marquee Area:** If you have placed a normal rectangular Marquee in the active layout, click this button to print only the marquee area. (The option is available for other marquee shapes, but the output area will always be rectangular.)

Print Range: Choose either **All** to print all pages, or choose a page range for multi-page output. See the Preview at the bottom right of this dialog box for how multi-page output will be arranged.

Size: Choose the layout output size.

- **Fit to Page:** Click this button to make the output fit the page. The size (in percentage) will be adjusted accordingly.
- **Custom:** Click this button to enter a size for the printed output, as a percentage of the paper size.

Margin Options: Use these controls to assign a margin to the printed output: either the printer margin or the layout margin.

Note: This option is available only if you have chosen **Selected Layouts in Navigator** or **Entire Layout** in the Source field.

Print to File: Check this box to save the output as a print file on your computer.

Collate Copies: Check this box if you are printing multiple copies, and would like each copy to be output in sequential order.

Black and White: Check this box for black and white output. With this option, colored elements will be output as follows:

- **Lines:** All colored lines will be black; white lines remain white.
- **Fill:** Foreground will be black; background will be white. Fills using RGB colors will be gray.
- **Zones:** All will be displayed in grayscale cover fills.
- **Images:** All will be displayed in grayscale.

Dithering: Check this box to reduce the total number of colors present in the output while retaining visual fidelity. Dithering may be necessary due to a limited number of colors available on the display device. For example, if your fills turn out solid black, the Dithering effect may help.

Print Reference: Check this box to include the Trace Reference in the Layout's printed output.

Hairline: Check this box if you want to print all lines of the document using the thinnest line available on the printing device.

Header/Footer: Check this box if you wish to include a Header or Footer on the printed output. To define the Header/Footer content, click the Settings button at right.

For more information, see [Header/Footer Settings](#).

Print Quality: Enter a dpi (dots per square inch) setting to define print quality.

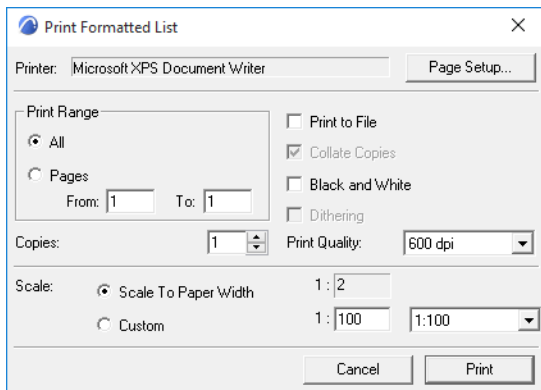
Copies: Enter the number of copies you wish to print.

Arrangement: This section helps you arrange the drawing on the output sheet(s). A preview window helps you get the desired outcome. In the preview window, the black square(s) represent the page(s) to be printed. The blue rectangle represents the drawing size. The Preview indicates the position of a Header/Footer if you have checked that option.

- **Sheets:** Shows the number of sheets of the paper size chosen in Page Setup.
- **Positioning:** Click these arrow-buttons to move the printed content around on the output sheets as shown in the Preview.

Print Interactive Schedule

This dialog box is opened with the **File > Print** command if a Schedule window is active.



Printer: This field displays the printer driver currently set up for your machine. To change the assigned printer, choose the Page Setup button at right.

Page Setup: Click this button to open the Page Setup dialog box, where you can choose a printer and set paper size and orientation.

Print Range: Choose either **All** to print all pages, or choose a page range for multi-page output.

Print to File: Check this box to save the output as a print file on your computer.

Collate Copies: Check this box if you are printing multiple copies, and would like each copy to be output in sequential order.

Black and White: Check this box for black and white output. With this option, colored elements will be output as follows:

- **Lines:** All colored lines will be black; white lines remain white.
- **Fill:** Foreground will be black; background will be white. Fills using RGB colors will be gray.
- **Zones:** All will be displayed in grayscale cover fills.
- **Images:** All will be displayed in grayscale.

Dithering: Check this box to reduce the total number of colors present in the output while retaining visual fidelity. Dithering may be necessary due to a limited number of colors available on the display device. For example, if your fills turn out solid black, the Dithering effect may help.

Print Quality: Enter a dpi (dots per square inch) setting to define print quality.

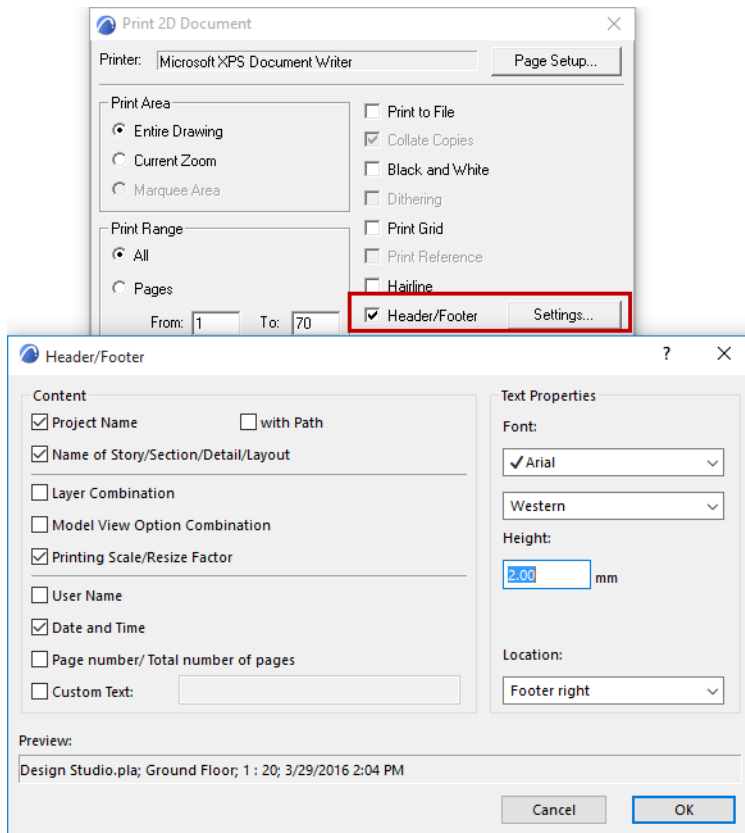
Copies: Enter the number of copies you wish to print.

Scale: Choose the layout output size.

- **Scale to Paper Width**
- **Custom:** Click this button to enter a scale for the printed output.
- **Every Zone/Page on New Sheet:** Check this box to split the output sheets by zones or pages.

Header/Footer Settings

Use the **Settings** button in **Print 2D Document**, **Print 3D** or **Print Layout** dialog boxes to open this dialog box. (Check the “Header/Footer” box first, to enable the Settings button.)



Content: In this section, check the items you would like to include in your header or footer. The available options are:

- Project name, optionally With Path
- Name of Story/Section/Detail/Layout
- Layer Combination (not available for Layouts)
- Model View Option Combination (not available for Layouts)
- Printing Scale (for 2D model windows)/Resize Factor (for Layouts)
- User Name
- Date and Time
- Page number/number of pages
- Custom Text. If you check this box, enter custom text in the field to the right.

Text Properties: Use the controls in this section to set the properties of text items in the header/footer, including Font Type, coding and font height3+.

Location: Use this pop-up field to choose between either a header (top of page) or a footer (bottom of page), and to set its horizontal location (left, center or right of the page.)

Preview: This field shows you the contents of your header or footer.

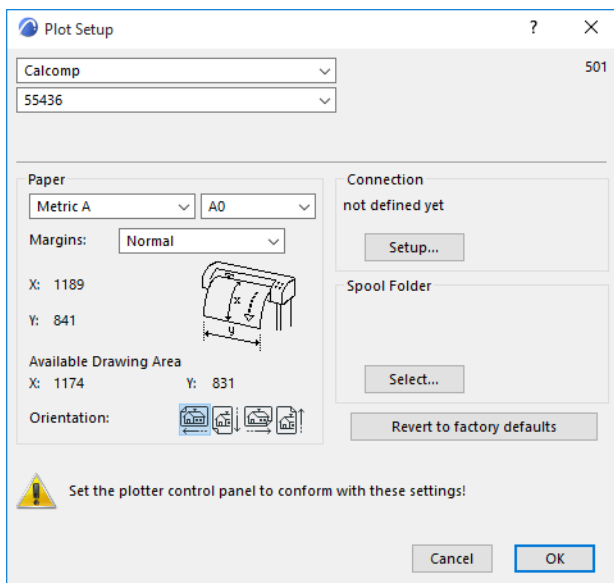
A symbolic preview of your header or footer is also viewable in the Print dialog.

When using a header or footer, please note:

- If you are working on a shared Teamwork Project, the User Name field changes to Team Member name.
- A header or footer can only be a single line.
- You cannot change the order of the fields.

Plot Setup

Note: The following descriptions are based on the Windows interface.



Use the two drop-down lists at the top of the dialog box to choose the plotter driver. The first lists manufacturers and the second the plotter driver for the given manufacturer. The contents of this list depends on the contents of the PlotWare folder.

- If the manufacturer of your plotter is listed but that particular model is missing, browse through the other models and select one with a similar paper size (e.g., E size). If there are several models with these criteria, experiment with all of them.
- If the manufacturer is not listed with any model, check the user manual of the plotter for compatibility and emulation modes. If the plotter accepts HPGL-2 and/or RTL data, there is a good chance of making it work. In this case, choose the HP DesignJet 755C model.

See also the list of supported models in the PlotWare folder/Read Me for Plotter Users.html file.

Paper: Use these controls to choose a paper size for your plotter. Invalid paper sizes are grayed in the menu and are not selectable.

If you choose **Custom** in the Paper area, the second pop-up list contains by default two items: Current paper and Modify list.

Choosing **Modify List** opens a dialog box in which you can define your own custom paper sizes.

Click **OK** when you have finished defining sizes manually. The new custom sizes will appear in the custom size pop-up. Custom paper sizes are saved with plotter settings.

Margins: Choose either “Normal” or “Extended” for the paper margins. The values in the X and Y fields below give you feedback on the result.

X/Y: These fields display the horizontal and vertical paper size. If you are using a “Custom” paper size, enter the desired paper size here.

Available Drawing Area: The X and Y values shown here indicate the size of the usable drawing area of the chosen paper.

Orientation: Use these controls to determine the location/direction of the output on the paper.

Connection

In the **Connection** area, specify the connection type you intend to use with your plotter.

To configure and use a plotter connected to one of your own computer's ports or to a network plotter, use the **Add Printer** tool of the Windows Printers Control Panel.

When first using a plotter with Archicad, you need to connect to it by clicking the **Setup** button in the Connection area. The **Select Plotter** dialog box appears.

You can choose one of the installed printer drivers from the pop-up menu. This driver will serve as a gateway to the plotter; Archicad will use the specific plotter driver selected on top of the dialog box for actually plotting data.

Spool Folder

Below the Connection options, you can define a folder for spooling files to be plotted either from your own computer or anywhere on the network in the background.

Plot 2D Document

Plotter: The name of the active plotter (chosen in Plot Setup) is shown at the top of the dialog box.

Plot Setup: Click this button to access the Plot Setup dialog box.

For more information, see [Plot Setup](#).

Destination: Choose a destination for the plotted output:

- **Plotter:** send the output to the designated plotter
- **File:** send the output to a plot (data) file
- **Spool Folder:** send the output into the spool folder specified in Plot Setup.

Scale: Choose a plot scale for the output of the active window.

- **Original:** Choose this button to plot at the original scale.
- **Fit to Page:** Choose this button to make the output fit the page. The scale will be adjusted accordingly.
- **Custom:** Choose this button to enter any other scale for the output. Either type in a custom scale, or choose a scale from the pop-up below.

Note: You cannot enter a custom value that will make the plotted image larger than the usable page size.

Text & Markers: *Available only if you have chosen either Fit to Page or Custom as the Scale.* Use this pop-up to define the scale of text and markers in the output. These options also affect the output characteristics of other elements such as arrowheads, scale-independent dashed line types and fill patterns.

- **Resize to Plotting Scale:** Outputs text in proportion to the size of other elements. This method is good for enlargement of a Project for a presentation to be seen from a distance.
- **Fix Size:** Outputs text at its exact size in the current paper units (inches or millimeters). To keep text at exactly the same size even when the printed size of the project changes, choose this option. This is acceptable for enlargements but can produce unwanted effects with reductions. Text may overprint other elements as they shrink.

Plot Grid: Check this box if you wish the output pages to display the Archicad grid.

Hairline: Check this box if you want to plot all lines of the document using the thinnest line available on the plotting device.

Plot with: Choose a color scheme for the plotted output (options may vary depending on the type of plotter used):

- Color
- Grayscale
- Black and White

Copies: Enter the number of copies to plot.

Arrangement: Use the preview window to view the arrangement of output on the plotted page.

Plot Layout

This dialog box is opened with the **File > Plot** command if Layout is active.

Plotter: The name of the active plotter (chosen in Plot Setup) is shown at the top of the dialog box.

Plot Setup: Click this button to access the Plot Setup dialog box.

For more information, see [Plot Setup](#).

Destination: Choose a destination for the plotted output:

- **Plotter:** send the output to the designated plotter
- **File:** send the output to a plot (data) file
- **Spool Folder:** send the output into the spool folder specified in Plot Setup.

Source

- **Selected Layouts in Navigator:** Click this button to plot only those layouts which are selected in the Navigator palette.

Note: This option is inactive if nothing is selected in the Navigator, or if several Layout Books (in the Navigator and Organizer) are open on screen, with several sets of selected Layouts.

- **Active Layout:** Plot the currently active layout.

Resize

- **Fit to Page:** Click this button to make the output fit the page. The size (in percentage) will be adjusted accordingly.
- **Custom:** Click this button to enter a size for the plotted output, as a percentage of the drawing's original size.

Margin Options: Use these controls to assign a margin to the plotted output: either the plotter margin or the layout margin.

Hairline: Check this box if you want to plot all lines of the document using the thinnest line available on the plotting device.

Plot with: Choose a color scheme for the plotted output (options may vary depending on the type of plotter used):

- Color
- Grayscale
- Black and White

Copies: Enter the number of copies to plot.

Arrangement: Use the preview window to view the arrangement of output on the plotted page.

For more information on plotting, see [Plot](#).

PhotoRendering Settings

For general information, see [PhotoRendering](#).

Topics in this section:

[Detailed Cineware Settings](#)

[Basic Rendering Engine](#)

[Sketch Rendering Engine](#)

[Using Redshift Renderer](#)

Basic Rendering Engine

Choose Basic Rendering Engine from the Engine drop-down menu in the PhotoRendering Settings dialog box (**Document > Creative Imaging > PhotoRendering Settings**).

Note: The speed of PhotoRendering is defined most of all by image size, while model complexity affects speed if shadows are used. The PhotoRendering effects define the final quality or character of your rendering.

Basic Rendering Engine Effects Panel

Use these controls to select the PhotoRendering effects you want to use:

Method

The **Method** setting determines how closely the program should adhere to its ultimate imaging capacity.

- **Best:** The program calculates the light attributes of each pixel. As the choices decrease in quality, Archicad calculates individual values for every second or third pixel only. The quality increase will affect curved surfaces more than others.
- Using multiple light sources or picture mapping may require the **Final** or **Best settings**.
- **Flat Shading:** Results in each shape being roughly approximated by flat surfaces only. Special effects like Transparency, Fog, Emission, Smooth Surfaces, Highlights and Light Decay are unavailable. The Flat Shading option is faster than all the other options in this dialog box, but slower than when the 3D Style uses Shaded mode.

Effects

Effects let you enable a variety of light and surface effects, some of which are initially set in the **Options > Element Attributes > Surfaces** dialog box.

For more information, see [Surfaces](#).

- **Highlights:** Check this to use specular reflections as **highlights**
Note: If Highlights is unchecked, or the Flat Shading method is used, specular reflections are ignored in the rendering.
- **Emission:** Check this if you have chosen, or created, light emitting Surfaces and want this quality to be displayed in the renderings.
- **Fog:** Check this box to apply a fog effect to the rendered image. Choose a color and intensity (low, middle, high) for the fog effect.
- **Transparency:** Transparent elements (such as glass) will be rendered as transparent. (If you leave this box unchecked, such elements will be shown as opaque.)
- **Smooth Surfaces:** Set the degree of smoothing, with up to one pixel accuracy. Curved surfaces, which are normally approximated by flat planes, are represented by more lifelike surfaces when this option is on.
- **Lamp Falloff:** If this is unchecked, the angle falloff and distance falloff values in the GDL Scripts of Lamps will be considered zero. This results in a sharp border between illuminated and dark areas. If it is checked, the values originally given will be used, resulting in a smooth transition between illuminated and dark areas.

Note: Lamps must be enabled at the Light Sources checkboxes for falloff to operate.

- **Textures:** Check this box to display bitmapped pictures on the corresponding surfaces.
It also controls whether the textures linked to surfaces in the Surfaces dialog box are displayed in PhotoRenderings.

Antialiasing

Controls the contour quality of your renderings. Jaggedness of the contour is gradually eliminated as you move from **Off** to **Best**.

You may want to use the Best option only for final documents, since processing time increases along with quality.

Light Sources

Select which **Light Sources** (Sun, Lamps or both) you want to use to illuminate the model during the PhotoRendering process.

The Lamps checkbox can only be effective if Lamp type library parts have been placed on the plan. They must be switched on in their Settings dialog box.

Shadow Casting

Define which of the available light sources you want to cast shadows in the PhotoRendered picture.

- Check the **High Accuracy** box if you notice incorrect results on your PhotoRendered picture, especially missing shadows of relatively small elements. This may occur if you zoom in on a small detail of an otherwise large project in the 3D Window and make a photorendering of that detail.
- If the **Use transparency** checkbox is checked, Archicad will create different shadows for surfaces with different transparency. The color of the transparent surface will also affect the color of the shadow in the final photorendering. This is also valid for surfaces with Alpha channel, where the Alpha channel is used to define the transparency of the surface.

Texture Antialiasing

Use the slider to set the desired image quality. As this procedure is very time consuming, use it only if you require a superior image quality for printing.

Basic Rendering Engine Background Panel

See [Cineware Basic Settings: Background Panel](#).

Basic Rendering Engine Brightness Panel

This panel of the dialog box gives you choices regarding exposure.

Use the dimmer switch to set the desired brightness from dark to bright.

You have three options for correcting possible Overexposures. (Overexposure occurs when the computational method results in a brighter color requirement than the brightest white on the monitor.)

- **Darken All Surfaces:** Archicad can make all surfaces darker than their computed color and make the brightest color white. This keeps the contrast between bright and dark parts of the image, but details in darker areas may be lost.
- **Darken Overexposed Surfaces:** You can choose to darken only the overexposed surfaces. This way, dark areas will not be extremely dark, but the contrast between shades in bright areas will be smaller.

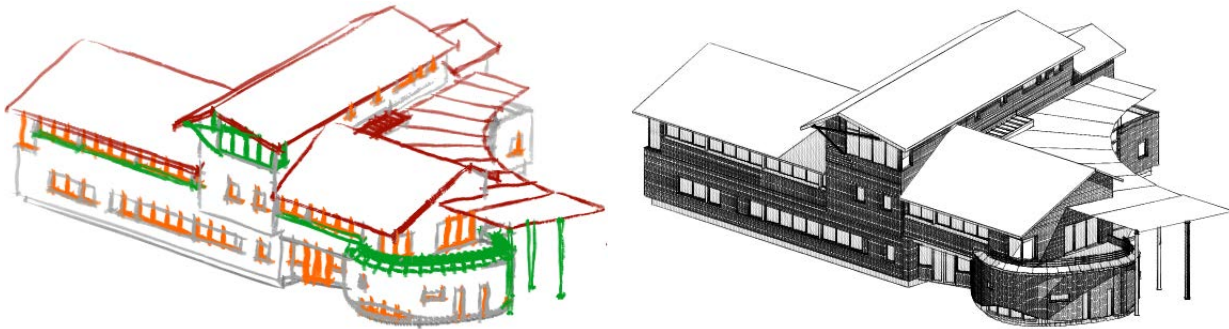
- **Paint Overexposed Surfaces with:** Select a color for these surfaces to show their place in the picture. This is an editing tool that helps you fine-tune the lighting of your renderings by changing elements of the model (e.g., relocating or dimming individual lamps) rather than applying one of the overall corrections described above.

Sketch Rendering Engine

Choose Sketch from the Engine drop-down menu in the PhotoRendering Settings dialog box (**Document > Creative Imaging > PhotoRendering Settings**).

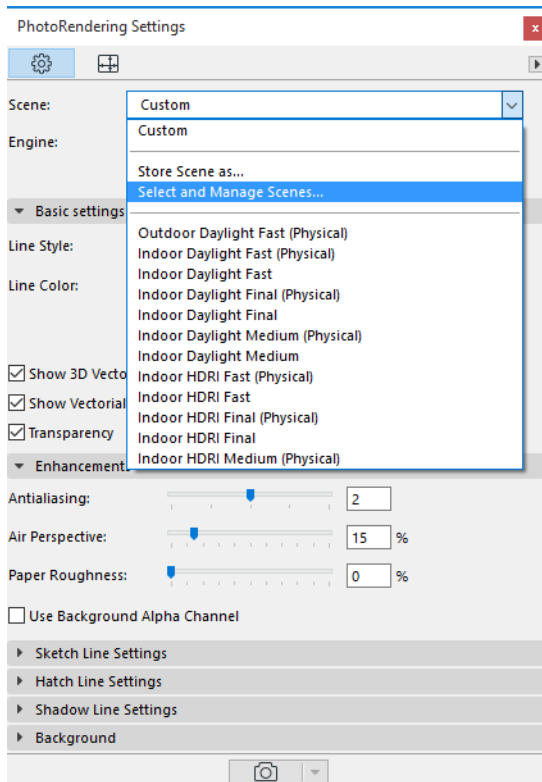
The Sketch engine produces non-photorealistic renderings that replicate sketched lines - best suited for draft views in the early project stages, using different free-hand effects. These effects include felt tip, pencil, graphite and marker. Files used for sketch rendering are placed in the Add-ons folder inside the Archicad folder. In the **SketchTextures** subfolder, you can find the predefined style file named SketchPreset.txt as well as the line texture files, each placed in a separate subfolder of **SketchTextures**.

Below, the same building rendered with two different Sketch scenes.



Sketch Basic Settings Panel

Scene: Click the drop-down to access sketch scenes.



The Sketch Rendering Engine provides a number of factory scenes with preset parameter values. If you override these factory settings, the scene will be renamed **Custom**.

Line Styles: Each linetype is represented by a dedicated folder with a set of bitmap images. These sets can be extended with similar scans of scripts using any graphic device. To get the best sketch effect, it is recommended to scan a large variety of line lengths, as the program will choose randomly among these styles when reproducing lines in the rendering.

Line color:

- **Custom:** Any preferred color can be chosen for monochrome images. The presets suggest traditional black, blue or tan colors.
- **Current element settings:** In this case, line color depends on the pen colors of individual model elements. Color images can be generated this way; however, it requires the careful adjustment of pen colors.
- **As in Line Style:** For the most natural effect, the color of the lines can be determined by the scans' original colors (graphite pencils, black art pen ink, sepia and peine gris aquarelles, and carmine.)
- If you check the **Show 3D Vectorial Hatching** box, the hatch lines (for example brick or tile contours) will be drawn in the rendered image. The properties of the hatch lines can be controlled by the settings of the Hatch Line Settings panel below.
- With **Show Vectorial Sun Shadows** active, the rendered image will include shaded areas representing sun shadow. The shade lines properties can be set on the Shadow Line Settings panel.

Sketch Enhancements Panel

Antialiasing: Smooths the lines by picking longer lines from the scans than necessary. In case of textured lines (e.g. brush, chalk), it is recommended to keep this value low (2 or less), as high values can slow down rendering.

Air perspective: Similarly to the Fog effect, fades lines further to the viewpoint. Although this effect is not normally done with manual sketching, a light air perspective to a rough wax sketch can be an attractive enhancement, and can be made to look especially natural with brush scans.

Paper roughness can simulate the effects of bumps on scans of real "rough paper". Normally, bumps are generated randomly by the rendering engine; however, for best results, it is advisable to use a background image with predefined alpha channel for the bumps (you can find some background images in the Archicad Library/Background Images/Alpha Paper Textures folder). Check the **Use background alpha channel** box if you use such a background image.

The sketch engine can detect these bumps and fade or erase the sketch lines at this area depending on the settings of the Paper Roughness slider. This setting has no effect if no background image has been scanned with the alpha channel.

Sketch Line Settings Panel

Thickness: Use a low value to replicate thinner devices (pencil, fine marker). Large values are appropriate for displaying the textures of brush or wax. However, you can experiment with different line settings and try getting interesting results which can be varied for a given line style. For best results, use higher values for Sketch Lines than for Hatch Lines and Shadow Lines.

Line distortion: Try various values with the chosen line style to achieve the desired effect. Generally, to replicate a pencil stretch you will use a lesser distortion than for brush, but you can experiment.

When **Line Overstretch** is set high, the lines will be longer than their counterparts in the Archicad 3D window, as if you had been drawing lines that do not connect properly and will cross each other. When the control is set low (below zero), the lines will stop short of reaching one another.

Sketch Hatch Line Settings Panel

The controls on this panel are to set the properties of the hatch lines of the rendered image. If the **Show 3D Vectorial Hatching** checkbox on the Basic Settings panel is not active, no hatch lines will be drawn and the choices you make here will have no effect whatsoever.

The **Thickness**, **Line overstretch** and **Line distortion** controls have the same effects as on the Sketch Line Settings panel.

When the **Clarify remote area** control is set high, the hatch lines in the background will be paler and thinner. This is useful and sometimes even necessary when a hatched surface is viewed from a reduced angle.

Without clarifying the remote area the hatch lines would be so dense that they would form a continuous dark area. These settings only take effect if you check the **Apply** box.

Sketch Shadow Line Settings Panel

The controls on this panel set the properties of the shadow shading lines of the rendered image. If the **Show Vectorial Sun Shadows** checkbox on the Basic Settings panel is not active, no shadow lines will be shown and the choices you make here will have no effect whatsoever.

- The **Thickness**, **Line overstretch** and **Line distortion** controls have the same effects as they do in the Sketch Line Settings panel.
- **Darkness** controls the opacity of the shadows. When it is set low, the shade is more transparent.
- **Spacing** controls the distance between the shadow lines. A high spacing value results in lighter shadows.

Note: The effects of **Thickness**, **Darkness** and **Spacing** may overlap. For example, if you set a high Thickness value when Spacing not very high, you will obtain a darker shadow, as if you had increased the Darkness value.

Sketch Background Panel

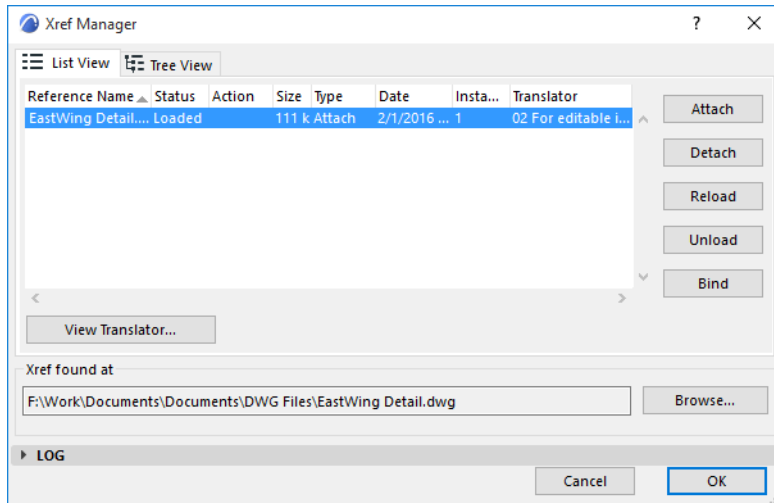
For a detailed description, see [Cineware Basic Settings: Background Panel](#).

XREF Management Dialog Box

To access this dialog box, go to **File > External Content > XREF Manager**.

Most XREF-related functions can be completed in this dialog box.

For an overview of this topic, see [Working with XREFs](#).



View Mode Buttons

The two buttons in the upper-left corner of the dialog box control how the list of attached XREF files is displayed.

- The left button will display all XREFs in the order they were attached.
- The right view mode button will display attached XREFs in a *hierarchical* tree structure. Since DWG files used as external references can also contain additional external references, nesting can occur. This nesting can be quite complex, reaching down several levels.

The hierarchical view mode lets you see which files refer to other external files, indicating what should load into the project when attaching an external reference file. If an attached file refers to other files, a + sign will appear on its left. Clicking the + sign reveals the names of the associated files.

XREF List: This list contains the information about the attached files. In the header, click the splitter bars to edit column width as needed. If you are in list mode you will see data about each file, including:

- **Reference name:** The name of the reference, usually the same as the name of the attached file.
- **Status:** The exact status of the file after each operation that affects it, such as Loaded, Detached, Reloaded, Unloaded or Bound.
- **Size:** The size of the referenced file.
- **Type:** The type of external referencing, either Attach or Overlay.
- **Date:** Last date the file was modified.
- **Insts (Instances):** The number of instances of a certain external reference file attached to the Project. As with library objects, you can attach an external reference several times to a Project file and in various locations (layers, different floors etc.).
- **Translator:** For each placed XREF, this column shows the translator used to place it, which will also be used to update it. If an instance has been placed multiple times with different translators, these instances are listed separately, by translator.

The file path below shows the location of the selected XREF file.

Attach

Click this button to display the **Attach XREF** dialog box.

For more information, see [Attach XREF](#).

For information on naming options for Attached XREFs, see: [Place Drawings into: in the section on DXF-DWG Translator Settings](#).

Detach

Click this button to detach the XREF from the drawing: closing the dialog box will remove it from the Project file. If you want to use it again you will have to attach it again. The Status column will reflect this “detached” state.

When the XREF is detached, a dialog lets you choose between retaining or removing the detached XREF's attributes, including layers, linetypes and fills.

Reload

When the DWG or DXF file used as an external reference has been modified, and you want to have the latest changes appear in your Archicad Project file, you can do so by clicking the **Reload** button. The program will load the whole file into Archicad again and display the latest state. Also, if you unloaded an external reference from the Project file, you can load it again with the **Reload** button. The status column of the dialog box will reflect this state.

Note: The reload process does not display an options dialog box to specify the input parameters. They are present in the database: the same data will be used as specified earlier with the **Attach** command.

Unload

Click this button to keep the selected external reference file attached to the project file, while unloading it from memory. The file's contents will not appear on the Floor Plan. As opposed to detaching, which removes the file from the project and the XREF Management file list, the **Unload** command keeps it attached. Clicking the **Reload** button will make it reappear. These changes are reflected in the Status column of the dialog box.

Note: After you unload an XREF, a text will appear at the insertion point of the XREF notifying you of this fact. For example, if you unload the XREF file “WALLS,” the message ‘Unloaded XREF “WALLS” will appear on the Floor Plan in place of the XREF.

Bind

Clicking the **Bind** button will merge the content of the external reference to the Project and delete the external reference to the file from the list.

Note: Binding will make all instances of the reference file part of the Project. It is not possible to bind only one instance of the XREF file.

When you bind an XREF, the layer names of the XREF will reflect the change, taking on following format:

```
'XREF_NAME $ NUMBER $ LAYER_NAME'.
```

For example, the PEN_5 layer of the XREF file with the name “WALLS” was converted to layer WALLS|PEN_5 while it was attached. When it is bound to the project file, its name is automatically changed to WALLS\$36\$PEN_5, with the number “36” being a number in the list of layers.

Browse

Click this button to open a directory dialog box where you can change the file that the selected XREF points to (**XREF Found at**). After you select another file and return to the **XREF Management** dialog box, you must click the **Reload** button for the change to take effect. After exiting the dialog box with the **OK** button, Archicad reloads the new file, which will then be associated with the given XREF name.

This feature is particularly useful when you have an XREF in your project whose original file was changed and saved under a new name.

For example, consider a Floor Plan that is attached to an interior design project with the name "Floor Plan0728.DWG." Later, as the Floor Plan changes, it is saved under the name "Floor Plan 0801.DWG." Clicking the **Browse** button and selecting this new file will associate the link with this latest version of the Floor Plan, enabling the Interior Designer to quickly update it and use it for further work.

Show Log

Click this button to expand the **XREF Management** dialog box downward to show a list of all steps performed in the current session of the **XREF Management** dialog box.

This log allows you to see what functions the program will execute once you click the **OK** button.

Click **Hide Log** to hide the log from view.

Translator Settings

Click this button to view the DXF-DWG Translator settings of any selected XREF from the list. The appearing dialog box is view-only.

For information on naming options for Attached XREFs, see: [Place Drawings into: in the section on DXF-DWG Translator Settings](#).

Attach XREF

This dialog box opens when:

- you use **File > External Content > Attach XREF**
- you click “Attach” from the XREF Management dialog box. (**File > External Content > XREF manager.**)

XREF Name

Specify the name of the file to be attached as an external reference.

- Use the **Browse** button to select from a file selection dialog box.
- Alternatively, select from the pop-up list of already attached files if you want to place another instance of the file into the Project.

Path

Shows the exact path to the selected file.

Reference Type

Choose a radio button to set how nested XREFs will be handled.

Note: Nesting occurs when you load an external reference to a file that has external references loaded to other files.

Attachment

All levels of nesting can be displayed in the Project file.

Overlay

Nested XREFs from the referenced file will not be visible in the Project.

The following table shows what happens when you attach/overlay DWG files that contain attached/overlaid XREFs.

	XREF visible (yes/no)	Nested visible (yes/no)
XREF attached, nested XREF within originally attached	YES	YES
XREF attached, nested XREF within originally overlaid	YES	NO
XREF overlaid, nested XREF within originally attached	YES	YES
XREF overlaid, nested XREF within originally overlaid	YES	NO

Examples:

1. A DWG file named BASE.DWG that has an attached XREF named COMMON.DWG is attached to the Archicad Project. Both files **will be** visible on the Floor Plan.
2. A DWG file named BASE.DWG that has an overlaid XREF named COMMON.DWG is attached to the Archicad Project. BASE.DWG will be visible on the Floor Plan but COMMON.DWG will not be visible.
3. A DWG file named BASE.DWG that has an attached XREF named COMMON.DWG is overlaid to the Archicad Project. Both files will be visible on the Floor Plan.

4. A DWG file named BASE.DWG that has an overlaid XREF named COMMON.DWG is overlaid to the Archicad Project. BASE.DWG will be visible on the Floor Plan but COMMON.DWG will not be visible.

Align Xref Origin and Rotate to Survey Point

Check this box to

- Place the Xref (using its own Drawing origin) at the Archicad Survey Point
- Rotate it to match the Survey Point orientation.

If you check this box, the Insertion Point, Rotation and Anchor Point fields below are grey.

[See also Survey Point.](#)

Insertion Point

Specify the location of the xref in the project:

- Enter the exact coordinates, or
- Check **Specify On-Screen** to locate the xref when placing it.

Scale

Either specify a scale factor in both X and Y directions, or check the **Specify On-Screen** box to define scale on the screen.

Note: If you specify **scale** factors, the XREF's enclosing box will be scaled to the appropriate size. If not, you will be able to stretch the enclosing rectangle just as if you were stretching an Archicad object. It is possible to input or specify negative or positive values for scale factors. By specifying negative values you can achieve mirroring effects.

Rotation

You can define an angle by which the whole placed XREF will be rotated around its insertion point, or check the **Specify On-Screen** box to define rotation manually, on the screen.

Note: If you specify a **rotation** angle, the XREF is rotated and then drawn. If not, you can rotate the enclosing rectangle until you input the value with a mouse click or coordinate input.

Anchor Point

Choose an option to define the point used to select/drag the Xref .

- **Drawing's own origin:** the Global Origin (0;0 coordinate) of the attached XREF.
- **Drawing's own anchor point:** the attached XREF drawing's own anchor point (if any)
- **Bounding box node:** Archicad calculates the size of the bounding box of the attached XREF. You choose any of the 9 significant points of the bounding box.

Place on Story

Use this drop-down list to choose the story onto which to place the XREF. Before the actual placement, Archicad will switch to that story.

Translator

This field displays the currently selected DXF/DWG Translator to be used during the conversion of the attached AutoCAD drawing. You may select any of the currently available Translators from the drop-down list. Clicking the **Settings** button will bring up the Translator Dialog where you can make modifications to the Translator Settings before attaching the XREF file.

[For more information, see DXF/DWG Translation Setup.](#)

Description

This field shows a short explanation and note about the currently selected Translator so you can decide if it fits your needs.

Attach

Click to attach the Xref.

Note: When you attach an external reference file to an Archicad Project, a layer will be created for it. The resulting layer names will have the following format:

'XREF_NAME | LAYER_NAME'.

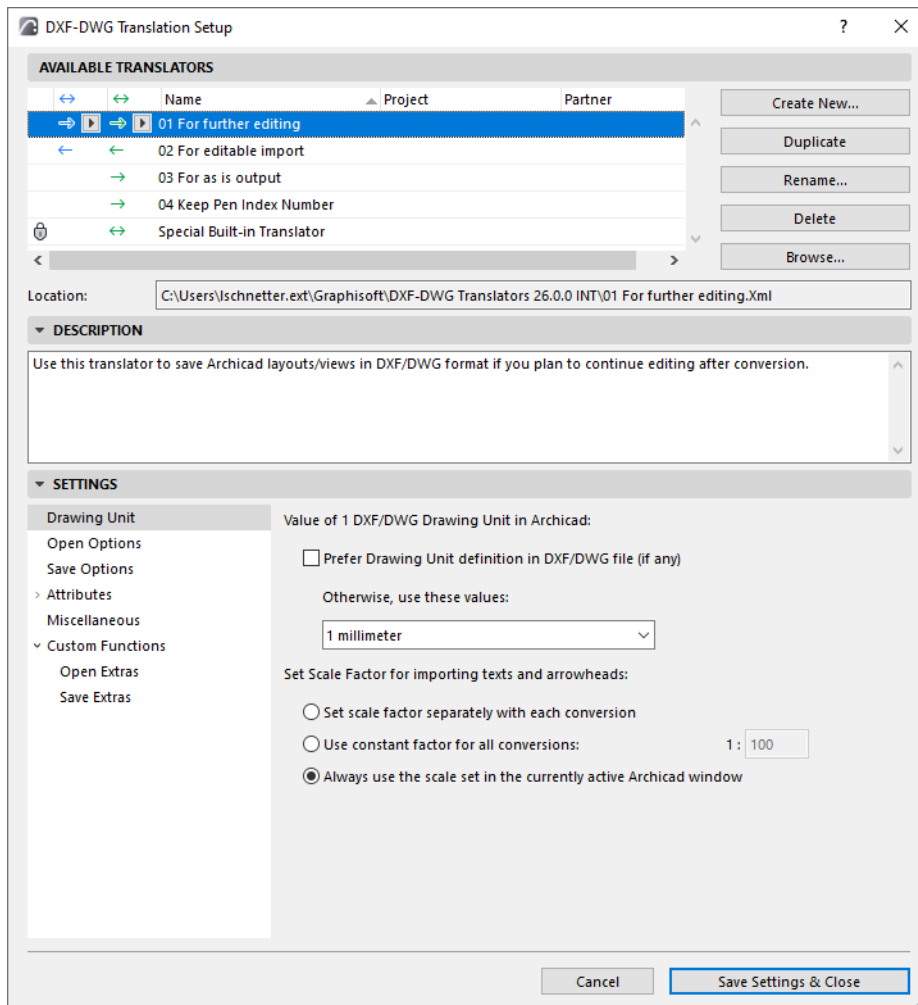
For example, the PEN_56 layer of a DWG file that was attached to the project with the XREF name 3D_STUFF will become the layer 3D_STUFF|PEN_56.

DXF/DWG Translation Setup

This dialog lists predefined and customized DXF-DWG translators. Use them to define how elements in Archicad will be interpreted when exported as DXF/DWG files, and vice versa: how the elements in DXF/DWG files will be interpreted when imported into Archicad.

DXF-DWG Translators are stored in the Defaults folder of the Archicad program.

Note for Teamwork users: Using common Translators ensures that all users have access to the same conversion options. It is recommended that the CAD Manager (or a person filling the same type of role in your office) manage the set of Translators and place them on a server. To use the translators, each user must add them to their list by using **Browse...** in the DXF/DWG Translator Setup dialog.



This dialog box is accessible:

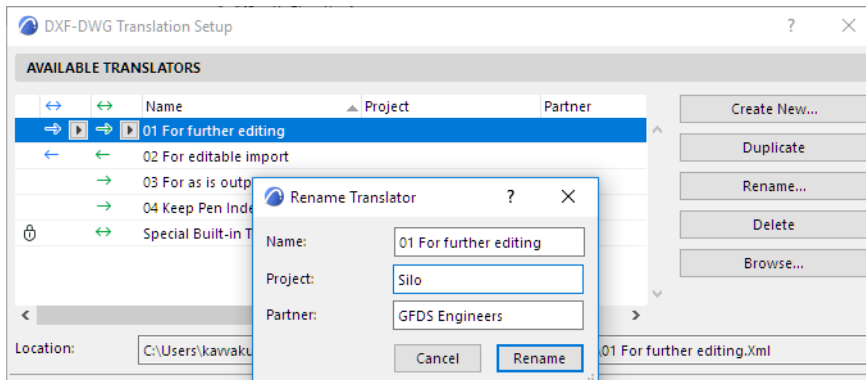
- When saving or opening DXF/DWG files: click the **Settings** button in the Save, Open or Merge dialogs, or the Attach XREF dialog box
- from **File > Interoperability > DXF-DWG > DXF-DWG Translation Setup**
- when opening a DXF/DWG file through the **File > Libraries and Objects > Open Objects** command,
- from **File > Libraries and Objects > Import Blocks from DXF/DWG**

Managing Translator Files

Translators are listed with the following data fields: Name, Project and Partner.

A locked translator cannot be edited, but you can create a duplicate, then edit the duplicate.

To edit these fields, double-click a list item to open the Rename Translator dialog, or click the Rename button.



Delete (available if the translator is not locked):

Click to remove unneeded translators:

- Delete the translator's XML. This option deletes all those translator settings permanently. Just remove the translator from the list in the dialog box. (To add it back to the list, click Browse.)

Create New: Using default settings, create a new Translator XML file and add it to the list. This XML file can be saved to any location, such as an office file server.

Duplicate: Duplicate the selected Translator. A new XML will be created in the same location as the selected Translator.

Remove (available if the translator is locked/unavailable/missing): This action will remove the translator from the list in the dialog box, but will *not* delete the translator XML file from your system. (To add the translator back to the list, click Browse.)

Browse: Click to access an existing Translator file.

Location: This field displays the path of the selected translator file and cannot be edited.

Note: If an XML file is renamed, moved or deleted, then it will be shown in italics in the Translator list and indicated as unavailable.

Why are my translators not editable?

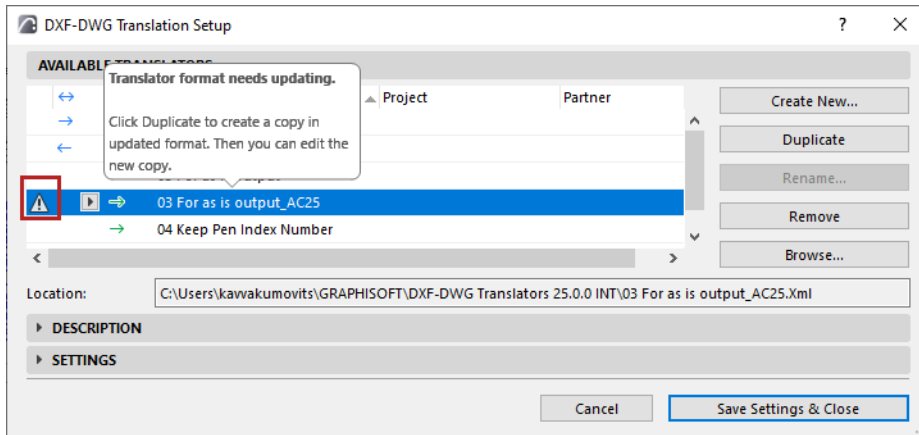
Translator is locked

A locked translator cannot be edited, but you can create a duplicate, then edit the duplicate.

Translator needs update

As of Archicad 26, the DXF-DWG translator file (xml format) uses an updated file format.

Translators imported from earlier-version Archicad projects are listed in the DXF/DWG Translator Setup, but they are outdated and therefore uneditable. They are marked with an icon and tooltip. Just create a duplicate - the new copy is now updated to the new format.



Translator's template file is inaccessible

If the outdated (and therefore uneditable) translator uses a template file that is inaccessible: it is marked with an icon and tooltip with instructions.

1. First open the translator in an earlier (pre-26) version of Archicad - now the translator settings are editable
2. Browse for the template file - now the template file and its settings are accessible
3. Reopen the translator in the current Archicad: now you can update the translator (create a duplicate), based on the proper template file.

DXF/DWG Description Panel

This is a multi-line edit box containing information about the selected translator. You can edit its contents with a single click.

DXF/DWG Settings Panel

The conversion rules for data exchange fall into six main groups:

- Drawing Unit
- Open Options
- Save Options
- Attributes
- Miscellaneous
- Custom functions

[Drawing Unit \(DXF/DWG Translation Setup\)](#)

[Open Options \(DXF/DWG Translation Setup\)](#)

[Save Options \(DXF/DWG Translation Setup\)](#)

[Attributes \(DXF/DWG Translation Setup\)](#)

[Layers \(DXF/DWG Translation Setup\)](#)

[Pens & Colors \(DXF/DWG Translation Setup\)](#)

[Line Types \(DXF/DWG Translation Setup\)](#)

[Fills \(DXF/DWG Translation Setup\)](#)

[Font Conversion \(DXF/DWG Translation Setup\)](#)

[Miscellaneous \(DXF/DWG Translation Setup\)](#)

[Custom Functions \(DXF/DWG Translation Setup\)](#)

Drawing Unit (DXF/DWG Translation Setup)

This is an option of the DXF/DWG Translation Setup dialog box

See [DXF/DWG Translation Setup](#).

Value of 1 DXF/DWG Drawing Unit in Archicad: As you export or import something in DXF or DWG, you must set the size and distance conversion factor to ensure that Archicad interprets the data correctly.

- **Prefer Drawing Unit definition in DXF/DWG file.** This checkbox means that Archicad will use the Drawing Unit definition as set in the original imported DXF-DWG file.
- If no such original definition is available, or if you want to redefine it, use the pop-up list to choose either a standard drawing unit (1 meter, 1 millimeter, 1 inch, 1 foot); or a custom unit (any number of millimeters or inches).

When opening a DXF/DWG file, you must define a scale at which you would like to see the correct ratios of elements drawn in AutoCAD. This particularly applies to arrowheads, symbol lines and symbol fills.

Set Scale Factor for converting texts and arrowheads (Choose one of the three options):

- **Set scale factor separately with each conversion:** In this case, a dialog box appears after clicking the Open button, letting you set the desired scale factor.
The scale factor effects Texts and Arrowheads of the imported AutoCAD drawing. Click OK to accept the scale factor.
- **Use constant factor for all conversions:** Set a scale factor for Archicad to use for these conversions. In this case the Scale Factor Dialog will not come up.
- **Always use the scale set in the currently active Archicad window:** The scale of the Archicad Floor Plan will be used for conversion. This is the same method which worked in Archicad versions up to 8.1.

Note: If you used the **Set scale factor separately with each conversion** option, the Scale Factor Dialog will also come up in case an AutoCAD drawing is dragged and dropped onto the Archicad Window background (drag-and-drop Open), and in case an AutoCAD drawing is merged into the Archicad Floor Plan, a Section/Elevation/IE Window or a Detail/Worksheet (drag-and-drop merge).

Open Options (DXF/DWG Translation Setup)

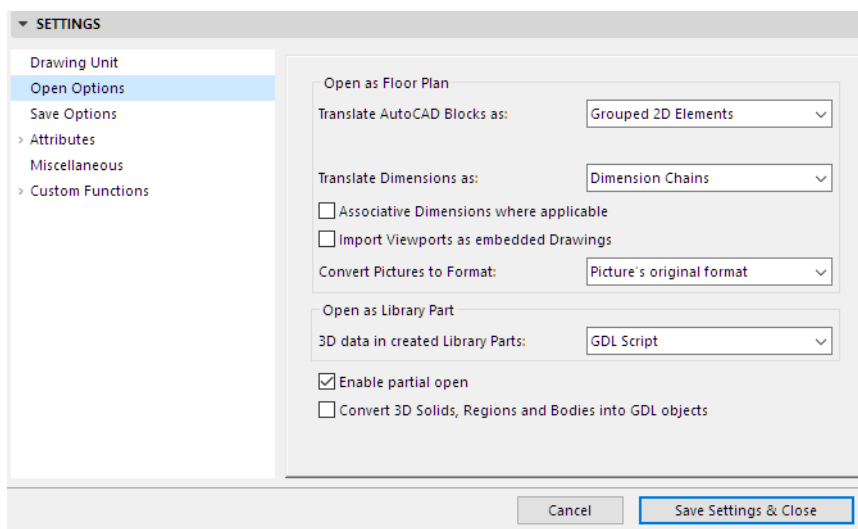
These are options in the DXF/DWG Translation Setup dialog box.

See [DXF/DWG Translation Setup](#).

The options vary depending on the performed operation:

- AutoCAD Block translation with the additional option of including 3D Data in the resulting Library Parts
- The conversion of Dimensions
- Conversion format of pictures contained in AutoCAD drawings
- 3D Conversion when AutoCAD drawing is opened as Library Part
- Filter, by Layer, the elements to be converted into Archicad.

For additional open options, see also [Custom Functions \(DXF/DWG Translation Setup\)](#).



Opening as Floor Plan:

- **Translate AutoCAD Blocks as:** 2D Elements
- Grouped 2D Elements
- Library Parts

When you choose **2D Elements** or **Grouped 2D Elements**, all the references of all the blocks get exploded into simple floor plan elements.

Translate Dimensions As:

- **Lines and Text:** you don't create a dimension out of the dimension entity; rather you use the drawing details stored in the file to get the same look (arrowheads, colors, text positions) as in AutoCAD.
- **Single Dimension Units:** every dimension unit in the AutoCAD file gets converted into an Archicad dimension unit.

Note: dimensions may have different visual features in Archicad and AutoCAD. Not all of them can be reproduced during conversion. Archicad has a fixed set of arrowheads, whereas in AutoCAD you can design your own set.

- **Dimension Chains:** In AutoCAD only separate 2-point dimensions are available (even though there is a method of creating sets of dimensions that look like dimension chains.)

Associative Dimensions where applicable: In former versions of AutoCAD, dimensions were not associative like in Archicad: glued to hotspots of other elements. If you check this option, the add-on will look for already imported elements to which to associate the processed dimension. It may be time-consuming. Therefore, the default behavior is to create hotspots on the element and associate the linear dimension to the hotspots.

Note 1: Radial dimensions always need something circular to be associated to.

Note 2: The most recent version of AutoCAD contains associative dimensions for certain elements, which cannot be converted.

Note 3: ADT has also introduced a kind of associative dimension, but it can be associated only to ADT objects.

Import Viewports as Embedded Drawings: Check this box to ensure that AutoCAD Viewpoints are imported as embedded Drawings after opening a DXF/DWG file in Archicad. If you uncheck the box, Viewports will be converted to Archicad views instead.

Convert Pictures to Format: Choose a file format for converting pictures: either retain the picture's original format, or choose another image format from the pop-up as a uniform format for all pictures.

Open as Library Parts

- **3D Data in created Library Parts:** Binary 3D: store the 3D information in binary format,
- GDL Script: store the 3D information in GDL script.

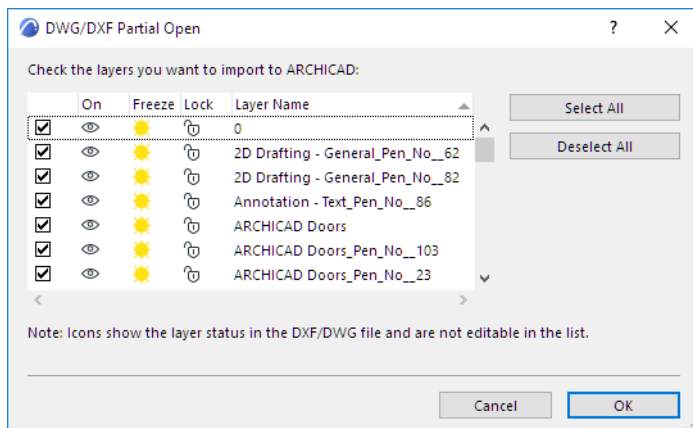
If you process 3D data, the Binary 3D form is recommended

Note: The **Open Object** command is the only way to get all the 3D data stored in a DXF/DWG file!

Enable partial open

Check this box and save it as part of the Translator to filter, by Layer, the elements of the DWG file to be opened/merged or XREF'd in Archicad.

In this case, when you use **File > Open or Merge on a DWG/DXF file**, or when you Attach an XREF file, the **DWG/DXF Partial Open** dialog box appears, listing the source file's Layer set and the Layers' status.



Uncheck the Layers which you do *not* wish to include in the Archicad project.

This function lets you restrict the DWG conversion to just those elements that you really need, thereby increasing conversion speed and reducing the size of the resulting Archicad project. When attaching XREFs, those Layers which you do not check here will *not* be added to the Layer attributes of your Archicad project.

See also [Attributes of XREF Files](#).

Convert 3D Solids, Regions and Bodies into GDL objects

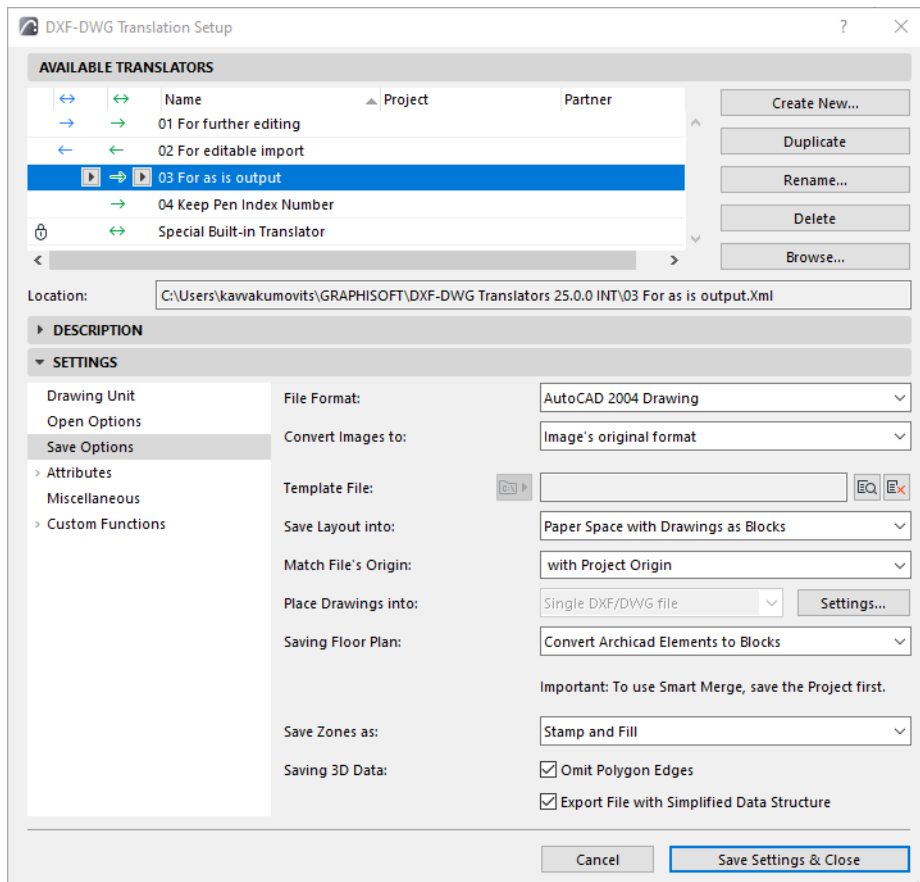
Check this box to convert the listed items into GDL objects upon opening the file in Archicad.

Save Options (DXF/DWG Translation Setup)

These are options in the DXF/DWG Translation Setup dialog box.

See [DXF/DWG Translation Setup](#).

For additional save options, see also [Custom Functions \(DXF/DWG Translation Setup\)](#).



File Format

Define which version of AutoCAD you are saving your files to.

Convert Images to

Choose a format for images saved to AutoCAD.

Template File

Some settings in AutoCAD cannot be matched to anything in Archicad, and vice-versa. For example, in Archicad, Layers do not have color and line type; in AutoCAD they do.

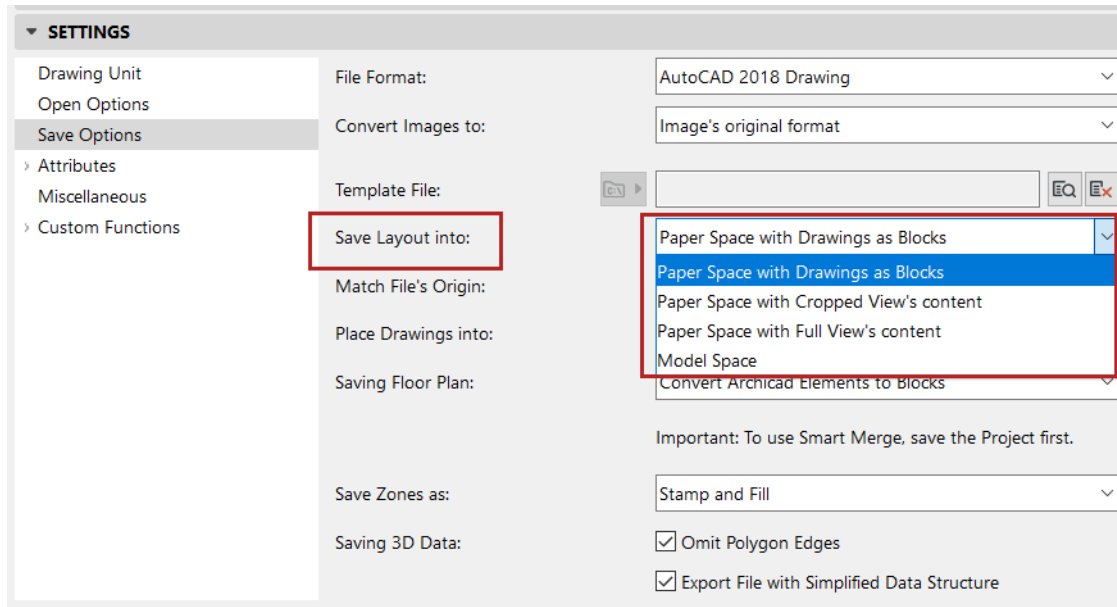
You can convert such settings based on a DXF/DWG file template, which you can choose here: browse for and select the template file.

If the selected template file becomes inaccessible, it is listed in italic type, and marked in the translator list with a warning icon. Browse for the file and make sure it is accessible.

Note: Use a template if you want to map Archicad attributes (e.g. Layers, Line types, Fills, Pens) manually. Otherwise, you can set attributes using the Translator controls. See [Attributes \(DXF/DWG Translation Setup\)](#).

Save Layout Into

Controls how the Layout is saved into the Spaces of a DXF/DWG file.



Paper Space with Drawings as Blocks

Use this option to get a perfect reproduction of the Archicad Layout appearance.

- The Layout is placed in a single Paper Space, with each Drawing converted into an AutoCAD Block.
- Drawings are cropped: only the visible part of the Archicad Layout is visible in the Paper Space.
- Nested blocks are shown in a flat list.
- The Model Space of the resulting AutoCAD drawing is empty.

Model Space

Use this option if you use automatic tools to validate DWGs, which can only see the Model Space.

- Drawings on the Archicad Layout are converted into AutoCAD Blocks in the Model Space.
- The Drawings are cropped: only the visible parts of Archicad Layouts are visible in the Model Space
- A viewport element is created in the default Paper Space.

Note on “Merge to one DXF file”:

If you are saving DWGs using the Publisher, you have the option to “Merge to one DWG” file. This option *does not work if you are saving to Model Space*. See [DXF Format: Merge to one DXF file](#).

Paper Space with Full View's content

- Drawing contents are saved to the Model Space
- Viewports are created in the Paper Space which points to those Drawing contents
- Even if a Drawing was cropped in Archicad, the entire source view is exported
- To control how these Drawings are saved, use this setting: [Place Drawings into:](#)

Note: If the exported Layout contains a Drawing based on a Floor Plan view: make sure the Drawing is updated before you save to DWG. That way, any Library Parts, Zones and Doors/Windows on the Floor Plan can be converted to intelligent AutoCAD Blocks.

Paper Space with Cropped View's content

Use this option to reduce the size of the DWG, and to hide content as needed (e.g. for privacy concerns)

- Just the visible parts of Drawings are exported
- If any part of an element is visible on the Layout, the entire element is exported
- Otherwise, this option works like “Paper Space with Full View's Content”

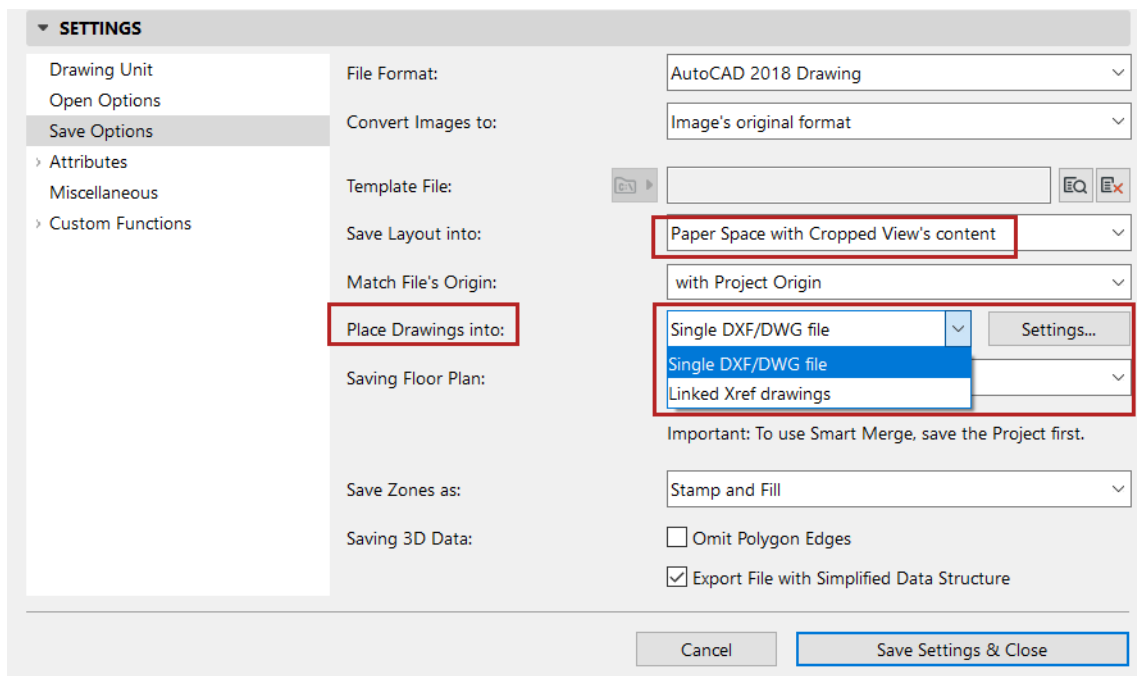
Match File's Origin

Choose either Project Origin or Survey Point.

The Survey Point option ensures that the Model Space content is correctly oriented in AutoCAD based on the coordinates set in Archicad.

Place Drawings into:

If you are saving Layouts to Paper Space (either cropped or full view): use this control to choose how to save the Drawings:



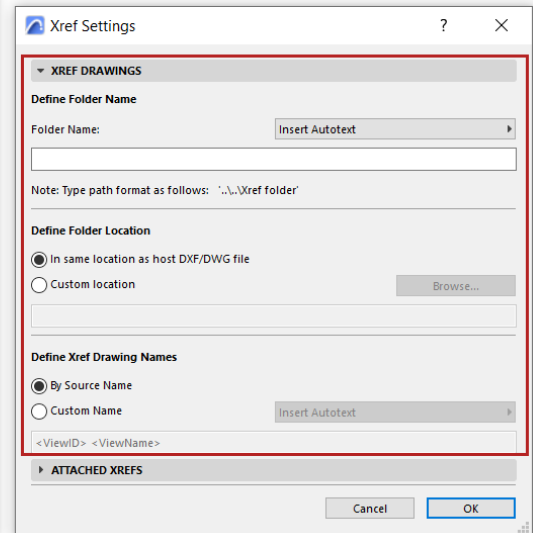
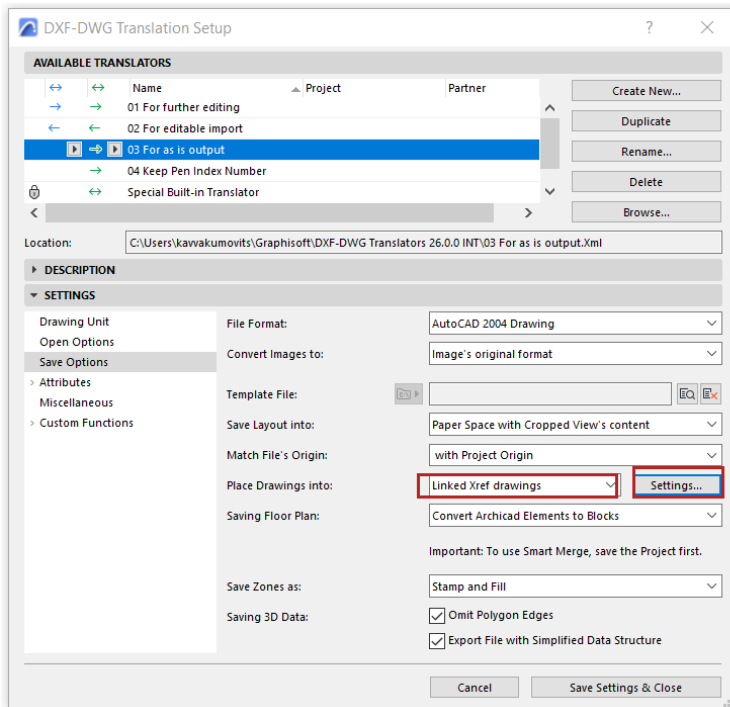
Single DXF/DWG file

- All drawings are saved into a single DXF/DWG file
- Multiple drawings are placed side by side in the Model Space
- A single Paper Space shows the cropped view content in viewports, plus any additional drawing elements, as they looked on the Archicad Layout

Linked XREF drawings

- Each drawing is saved in a separate, external DXF/DWG file
- Multiple drawings are placed side by side in the Model Space, and each is linked to its source file
- In the Paper Space, viewports recreate the Archicad Layout's appearance.

To manage the names and location of these linked drawings: click **Settings**, and define your preferences in the XREF Drawings panel.

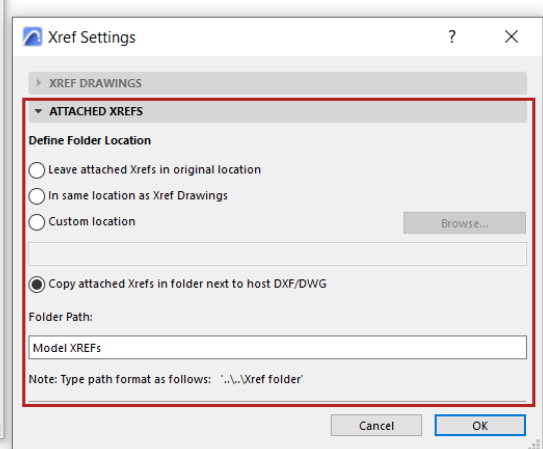
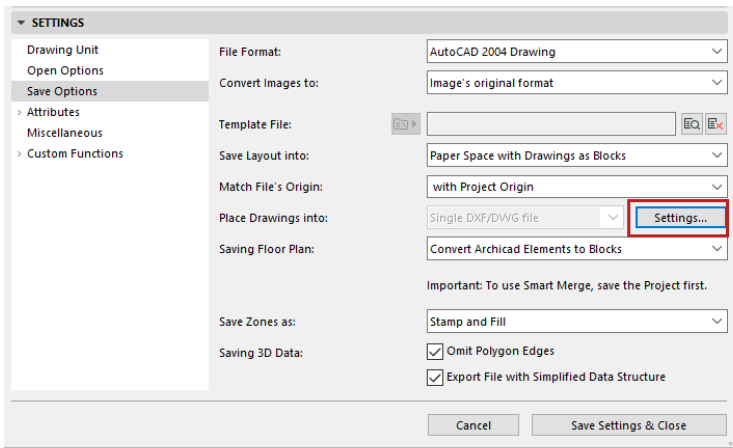


Manage attached DWG files

Your Archicad Layout may contain dwg files that you have attached as XREFs. These are read-only drawings linked to an outside source.

When you save the Layout to DWG, such attached drawings maintain their link to their outside source.

However, if needed, you can redefine this source location, for example by saving all such linked drawings to a new folder. Click **Settings**, and define your preferences in the **Attached XREFs** panel.



Saving Floor Plan

The following options apply when saving Floor Plans (on views or Layouts):

- **Convert Archicad Elements to Blocks:** Each element will be converted to 2D drawing primitives and combined in a block, without any attributes or properties or metadata.
- **Convert only Objects to Blocks:** Only GDL Objects are combined into blocks. All others will be 2D drawing elements.

- **Explode Archicad Elements:** All elements are exploded into separate 2D primitives. Use this if you are exporting DXF files from Archicad to a program which cannot process blocks.

Prepare File for Smart Merge: Use this if you plan to merge a DXF/DWG file as part of a round-trip workflow.

For more information on Smart Merge, see [Round-Trip Conversion \(Smart Merge\)](#).

Saving Zones as

Save Zones as **Stamp only**, or **Stamp and Fill**.

Saving 3D Data

The following options are available only when saving 3D Data:

- **Omit Polygon Edges:** The color of polygon edges in the 3D window of Archicad is usually different from the color of surfaces. If you want to reproduce this look in AutoCAD, leave this checkbox empty.
- **Export File with Simplified Data Structure:** The effect of this option is similar to the “Explode Complex Archicad Elements” option – all the data goes into the entity section, instead of blocks.

Attributes (DXF/DWG Translation Setup)

Use these pages to convert AutoCAD attributes to Archicad attributes and vice versa, at Open and Save.

There are five groups dedicated to different attributes: **Layers, Pens and Colors, Line Types, Fills, and Font Conversion**.

Each group (except the Font conversion) consists of a “Method” page (where you define options for handling attributes) and one or more conversion mapping tables.

On any Conversion page:

- Click **New** or **Add** to create a new entry
- Click **Edit** or double-click on an existing to modify the entry
- Select one or more items and click **Delete** to remove them from the dictionary

Note on Attribute Names:

- Archicad supports attribute names of essentially any number of unicode characters.
- AutoCAD 2000 and later versions allow almost any character attribute names. Names can contain up to 255 characters. (Character lengths are halved for Japanese, Chinese and Korean versions).

Note on Colors and Fonts:

- While Archicad pens can be set to any colors you like, AutoCAD versions have set pen colors that cannot be changed. In AutoCAD 2004 and later versions, it is possible to define further pen colors by specifying RGB values for them. These pens will have a pen index number of 256 or higher in these programs.
- Font names are stored differently in DXF/DWG files and in Archicad. They are converted based on the Font conversion mapping table. (On a PC, Archicad supports all available fonts including those installed by AutoCAD.)
- Symbol line types are stored differently in Archicad and in AutoCAD. Due to these differences, some of Archicad’s symbol line types won’t look the same in AutoCAD. There are two solutions to this problem:
 - Redesign the problematic line type until the disturbing differences disappear.
 - Set up a template file containing an acceptable substitution for it.

The following sections describe the method and conversion options for each of the five attribute groups.

[Layers \(DXF/DWG Translation Setup\)](#)

[Pens & Colors \(DXF/DWG Translation Setup\)](#)

[Line Types \(DXF/DWG Translation Setup\)](#)

[Fills \(DXF/DWG Translation Setup\)](#)

[Font Conversion \(DXF/DWG Translation Setup\)](#)

Layers (DXF/DWG Translation Setup)

These are options in the DXF/DWG Translation Setup dialog box.

See [DXF/DWG Translation Setup](#).

Methods

The options on the Methods tab are relevant for Save operations.

Create Layers According to

- **Archicad Layers:** Create Layers in the DXF/DWG file based on the Layers used in Archicad. The Layers will carry their hidden and locked status with them.

Note: Pen-based Layer names conversion is not available with this option, Custom Layer names take precedence. When you choose this option, the Layer name conversion dictionary will be used to replace some of the Layer names.

- **Do not export empty Layers:** This option is available only if “Create Layers according to” is set to Archicad Layers. (If this box is checked, then only layers that contain elements will be exported.)
- **Pen Numbers:** This option places elements having the same color in a common Layer. The Custom tags to Pen Numbers area has a Prefix field and a Postfix field. You can fill out these to specify the format of the created Layer names. The resulting Layer name will follow the following format: <Prefix>Pen No.<Postfix>. For example, if Prefix='AA_' and Postfix='_PEN', then all elements drawn using Pen 23 will be converted to Layer 'AA_23_PEN' in the resulting AutoCAD drawing.

Note 1: When you choose this option, the Pen-based Layer names dictionary will be used to replace some of the Layer names.

Note 2: Custom Layer names are not available with this option.

- **Element types:** Places common element types into a corresponding Layer, such as a WALLS Layer.
Note: There is no conversion dictionary associated with this Layer creation method.
- **Layer or declared Pen Number:** Creates AutoCAD Layers according to Archicad Layers, but definitions in Pen-based Layer names conversion dictionary take precedence over Archicad Layer conversion. Within Archicad Layer conversion, Custom Layers take precedence over normal Layer conversions.
- **Layer extended by Pen Number:** Creates AutoCAD Layer according to Archicad Layers, but out of one Archicad Layer it may create several Layers in the resulting AutoCAD drawing, because it will place elements located on the same Layer but using different Pens to different Layers in AutoCAD. The names of Layers will be created by the following pattern: <ArchicadLayer>_<Prefix>Pen No.<Postfix>.

Note 1: When you choose this option, the Pen-based Layer names dictionary will be used to replace some of the <Prefix>Pen No.<Postfix> portion of the Layer names.

Save Elements on

- **All Layers:** Save elements on hidden Layers and visible ones.

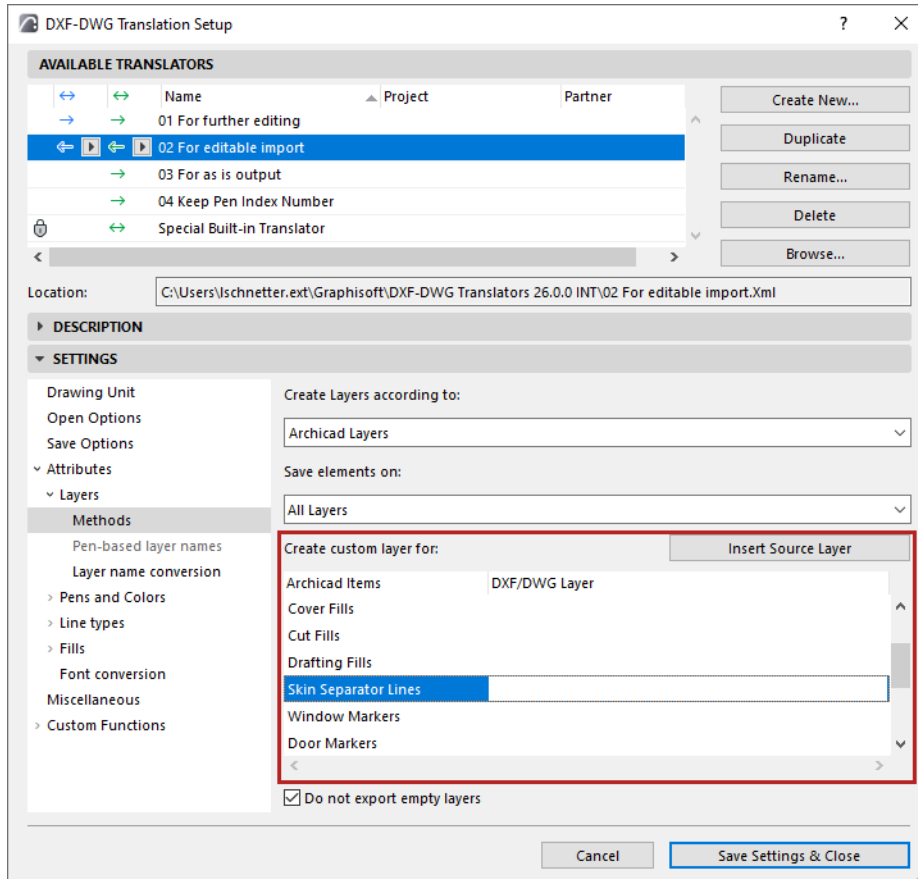
Visible Layers only: Elements on hidden Layers are omitted.

Note: The **Visible Layers Only** option works for Drawings only if you save the Drawing file separately. If you save a Layout as a DXF/DWG file, all Layers (both visible and hidden) of that Layout's Drawings will be saved.

Custom Layers

These fields allow you to assign Windows, Doors, Drafting Fills, Cover Fills, Cut Fills and Skin Separator Lines of Walls to separate Layers, instead of remaining on the same Layer as the construction element they belong to.

See also [Fill Categories](#) and [Line Categories](#).



- **Windows:** If you enter a Layer name into the 'Windows' edit field, all the Windows will be part of that Layer instead of the Layer of the Wall containing those Windows. The same rules apply to **Doors**.
- **Drafting Fills** will go to the Layer specified in the 'Drafting Fills' edit field.
- **Cover Fills:** Fills belonging to Slabs, Roofs, Meshes and Zones and Fills having the 'Cover Fill' category will go to the Layer specified in the 'Cover Fills' edit field.
- **Cut Fills** showing cut surfaces or sections of construction elements and Fills having the 'Cut Fill' category will go to the Layer specified in the 'Cut Fills' edit field.
- **Skin Separator** Lines of cut composite structures, plus Lines having the 'Skin Separator Line' category will go to the Layer specified in the 'Skin Separator Lines' edit field.
- **Contours** of cut construction elements and composite structures ('Cut Line' category) and Lines having the 'Drafting Line' category will remain on the Layer they were in.
- In case of a **Column:** if there is no veneer, the contour of the Column belongs to the 'Cut Line' category. If there is a veneer, the contour of the Column belongs to the 'Cut Line' category, and the lines between the core and the veneer belong to the 'Skin Separator Line' category and will be saved accordingly.
- In case of **Fills:** the contour of a Fill belonging to the 'Cut Fill' category will belong to the 'Cut Line' category. Contours of other Fill categories will belong to the 'Drafting Line' category.

- Window Markers
- Door Markers
- **Zone Background** (any visible Zone cover fills)
- Zone Outline
- **Zone Stamp** (geometry of Zone stamp)
- **Zone Texts** (textual content of Zone stamp)

The Custom Layer names you define will work even if they do not exist in the Archicad project.

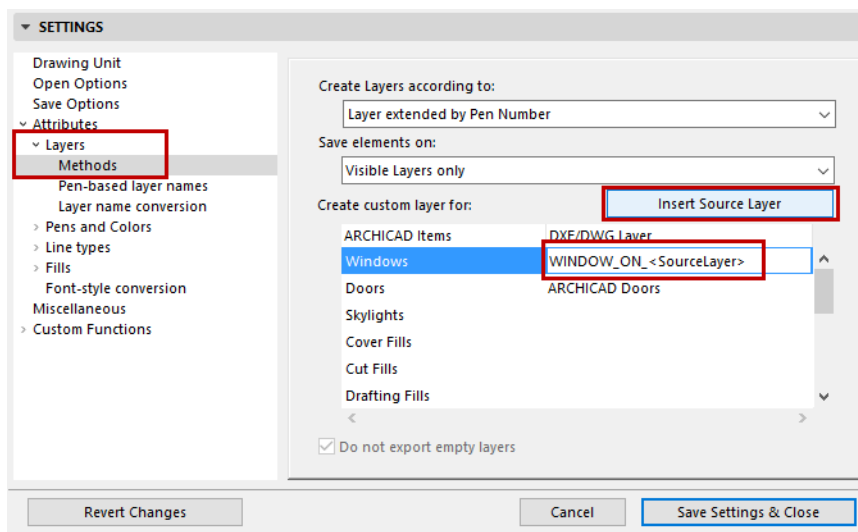
If you want your Custom Layer name to include the source Layer name of the wall containing the element: click **Insert Source Layer**.

Example:

The Wall containing Windows is on the Layer: EXTERIOR WALLS

If you enter in the **Windows** DXF/DWG Layer field the following: WINDOW_ON_

and then click **Insert Source Layer**, the Windows field shows: WINDOW_ON_<Source Layer>



Then the resulting Layer name will be: WINDOW_ON_EXTERIOR_WALLS

Pen-based Layer Names

If you save a DXF/DWG file with the “Create Layers according to” option set to Pen Numbers, the export file will contain a Layer with the name PEN_<n> where <n> is a pencolor index. Options to state the actual color name (i.e. PEN_BLUE) can be set here.

Pen-based Layer names conversion is very similar to Layer name conversion – the only difference is that you have a standard Archicad pen selection control instead of the Layer name pop-up. You can write only valid pen numbers in the Archicad pen field.

Layer name Conversion

Use custom tag to indicate renovation status: Check this box to append a tag corresponding to the renovation status to Layer names. This way, all elements on a particular Layer in Archicad will be assigned to one of three different DWG Layer names, depending on each element’s renovation status.

To create a dictionary item in the Layer-name conversion dictionary, click on the “New” button and assign the Layers in the dialog box.

Note: A problem may arise when you convert both “Exterior Walls” and “Interior Walls” to “WALLS” in AutoCAD. When converting wall Layers back to Archicad, the dictionary will not distinguish between the two types of Archicad wall Layers.

Pick a valid Archicad Layer from the pop-up under the edit field in the Layer conversion dialog. Press OK to add the item to the dictionary. Then assign a corresponding Layer on the DXF/DWG side.

To modify an existing item, just double-click on it, modify it, and write it back to the dictionary.

To delete one or more items from the dictionary, select them and click **Delete**.

Pens & Colors (DXF/DWG Translation Setup)

These are options in the DXF/DWG Translation Setup dialog box.

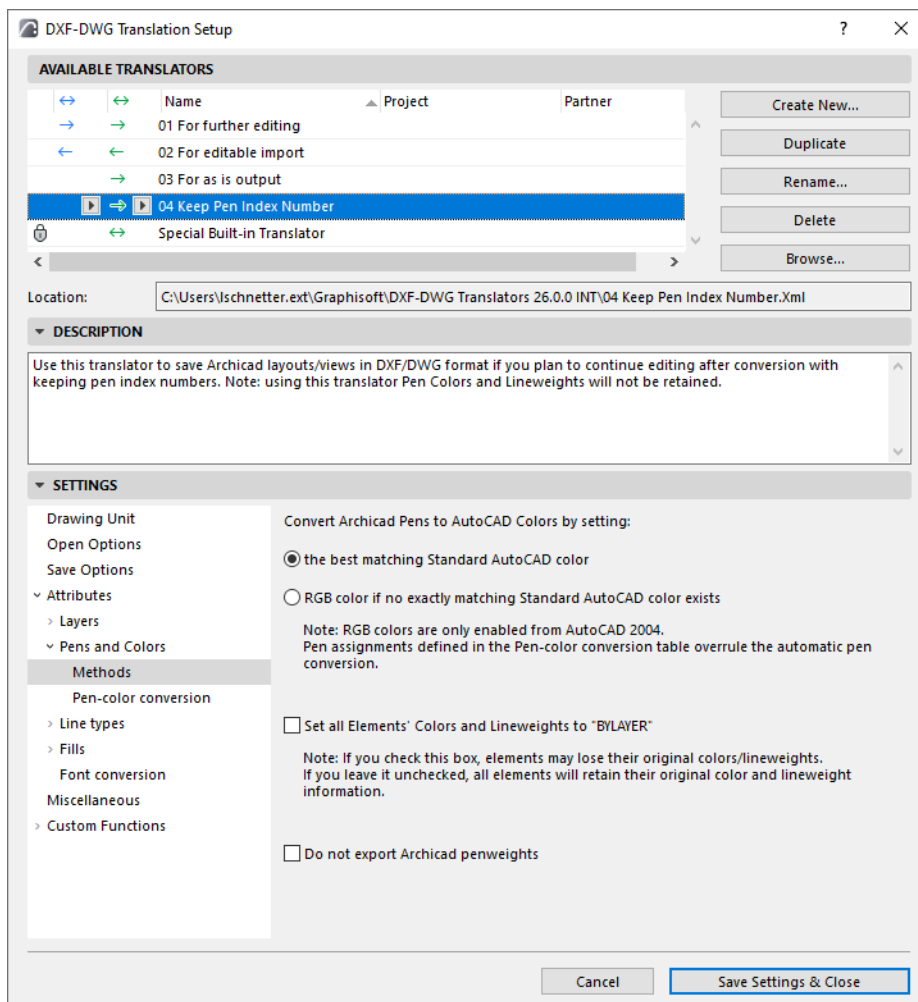
See [DXF/DWG Translation Setup](#).

Archicad Pen Colors are configurable, while AutoCAD's pen number association to colors cannot be modified. You can convert Archicad Pens to AutoCAD Colors by searching for the best color match, or set up a conversion dictionary.

To convert by pen index numbers, use the default translator entitled "Keep Pen Index Number." (Choose this option from the pop-up translator list at the top of the DXF-DWG Translation Setup dialog box.)

- If Background fill pen in Archicad is set to window background (-1), it will be converted to an AutoCAD Wipeout.
- A Patch object's background fill color is assigned the window background pen color (-1): as a result, the patch will be displayed correctly when the background color changes. When the Patch is converted to AutoCAD, its background is transformed to an AutoCAD wipeout (WPOut) without a frame, and will be displayed correctly.

Note: If you are using an AutoCAD version preceding AutoCAD 2004, Wipeout is available as part of "Express Tools." Wipeout is integrated into AutoCAD 2004.



Pen Methods

Open

- **Open:** When you open a DXF/DWG file, the pen table of Archicad is set to the AutoCAD colors. The pen numbers will be the same as the AutoCAD color indexes in the DXF/DWG file.

Other Import: With any other DXF/DWG import operation, in which pens should not be altered (Merge, Attach XREF, Open Object, etc.), every AutoCAD color will be mapped to the Archicad pen with the most similar color (RGB) components.

Note: All AutoCAD pens with a pen index number of 256 or higher will have their pen index changed to 255 in Archicad.

Save

The following two options are available when saving to AutoCAD 2004 or later versions.

- **Best matching Standard AutoCAD Color:** With a Save operation, every Archicad pen will be mapped to the AutoCAD color with the most similar color (RGB) components.
- **Note:** A round-trip will not necessarily result in the same colors as the original. For example, if Archicad pen 25 finds AutoCAD color 236 the closest match, it is possible that AutoCAD color 236 will find Archicad pen 32 the closest match instead of 25. **RGB Color:** If no match exists, it will save the RGB value of the Archicad Pen and assign to it a pen index number of 256 or higher in the AutoCAD drawing.

Set All Elements' Colors and Lineweights to BYLAYER

- For Save: Check this to override the Pen-Color conversion preferences above.
- Every saved entity will have the color BYLAYER. When the AutoCAD user changes the color attached to a Layer, everything on that Layer will change its color accordingly.

Do not export Archicad penweights

- For Save: With this option, penweights of Archicad Pens will not be exported to the resulting AutoCAD drawing.

The following table shows how Archicad penweights are converted to AutoCAD penweights when this checkbox is not checked and penweights are exported.

Conversion of Pen weights from ArchiCAD to AutoCAD					
ArchiCAD (mm)	AutoCAD (mm)	ArchiCAD (mm)	AutoCAD (mm)	ArchiCAD (mm)	AutoCAD (mm)
0	0	-0.30	0.30	-0.90	0.90
-0.05	0.05	-0.35	0.35	-1.00	1.00
-0.09	0.09	-0.40	0.40	-1.06	1.06
-0.13	0.13	-0.50	0.50	-1.20	1.20
-0.15	0.15	-0.53	0.53	-1.40	1.40
-0.18	0.18	-0.60	0.60	-1.58	1.58
-0.20	0.20	-0.70	0.70	-2.00	2.00
-0.25	0.25	-0.80	0.80	2.00-100	2.11

Pen-Color Conversion

Check the box to enable settings for conversion between Archicad Pens and AutoCAD Colors .

These settings take precedence over the automatic conversion methods set on the Methods page.

Line Types (DXF/DWG Translation Setup)

These are options in the DXF/DWG Translation Setup dialog box.

See [DXF/DWG Translation Setup](#).

When saving files, you can either set up a line type conversion table or choose to convert all Archicad line types to a single type in AutoCAD named BYLAYER.

Note: symbol line types are stored differently in Archicad and in AutoCAD. Due to these differences, some of Archicad's symbol line types will not look the same in AutoCAD.

Methods

Set All Elements' Line Types to BYLAYER: This checkbox is available only during Save. It overrides the effect of the Linetype-linetype conversion dictionary.

When this check-box is checked, every saved entity will have line type BYLAYER. When the AutoCAD user changes the line type attached to a Layer, everything on that Layer will change its line type accordingly.

Keep Existing Line Types on Open: If you check this checkbox, Archicad will keep line types currently defined and also import line types defined in the imported DXF/DWF file. In this case, if the DXF/DWG file contains line types with the same name as line types existing in the currently open Archicad Project, the line types of the DXF/DWF file will not be imported.

If this checkbox is not checked, all line types from the DXF/DWG file will be imported, and none of the existing Archicad line types will be kept.

LTScale value in output field: Set the LTScale to the desired value.

The LTScale variable used in DWG files are correctly interpreted when opening or merging DWG files in Archicad. New line types will be created as needed, regardless of the length of the original line pattern.

Linetype-Linetype Conversion

This conversion dictionary works exactly the same way as the Layer-Layer conversion. The only condition restricting its use is the option "Set all Elements' Line Types to BYLAYER."

You can pick the requested line type name from the pop-up list. Its name will appear in the **Archicad line type** field. In the AutoCAD line type field, type the name of the line type you would like this **Archicad line type** to be converted to in the resulting DXF/DWG drawing. Or vice-versa: from this line type, Archicad will create a line type specified in the **Archicad line type** field when opening DXF/DWG files.

Fills (DXF/DWG Translation Setup)

Archicad Fill patterns can be saved as solid Fills. You can also export Fill backgrounds as additional polygons (as this feature is not supported by AutoCAD).

Methods

In this popup menu you can choose how Archicad Fills will be exported to AutoCAD.

- **Export all Fills “as is”:** This option will recreate Archicad Fills in AutoCAD.
- **Export all Fills as “SOLID”:** This option will convert all Fill patterns to solid Fills.
- **Export all Symbol Fills as “SOLID”:** This option will convert all symbol Fill patterns to solid Fills.

Export Fill Backgrounds as Additional Solid Hatches: Archicad Fill patterns have a separate background Fill; AutoCAD Fills, in contrast, are all transparent between the pattern lines. If you want the exported file to look like the original Archicad Fill, check this option.

Note: Export Fills as “SOLID” disables this option.

The results of this choice:

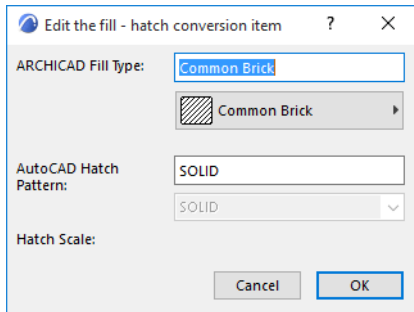
- **Background Fill, Non-Transparent Background:** When a background Fill with a non-transparent color is exported, it will be exported as a solid Fill whose color is that of the background (plus the border, if it exists).
- **Background Fill, Transparent Background:** no Fill will be exported (only the border, if it exists).
- **Foreground Fill:** Exported as-is; background has no effect.
- **Vectorial Fill, Non-Transparent Background:** a foreground Fill with the background color is exported first; then, a pattern Fill with the foreground color is exported as a Hatch entity (and the border if it exists). If the Hatch lines reach a certain level of density, then it becomes a solid.
- **Vectorial Fill, Transparent Background:** Only the pattern Fill with the foreground color is exported (and of course the border if necessary).
- **Symbol fill:** Has no AutoCAD equivalent; the symbol pattern is generated into a Block. If it has a background in Archicad, this background will become a Hatch in AutoCAD.
- **Gradient:** Exported as solid.
- **Image:** Exported as solid.

Explode Gradient Fills: Gradient Fills in your project will be exploded into separate components. If your Gradient Fill had a transparent background, it will also be a separate component.

Use the Fill Conversion Table: Check this box to convert Fills to hatches using the Fill Conversion table. Click on the Fill-Hatch Conversion table to define correspondences.

Fill-Hatch Conversion Table

Click **New** to define each new correspondence in the **Edit Fill-Hatch Conversion** dialog box.



This requires that you have selected an AutoCAD template file in the Save Options page of the DXF-DWG Translation Dialog Box, and the template file includes at least one placed instance of the desired hatch.

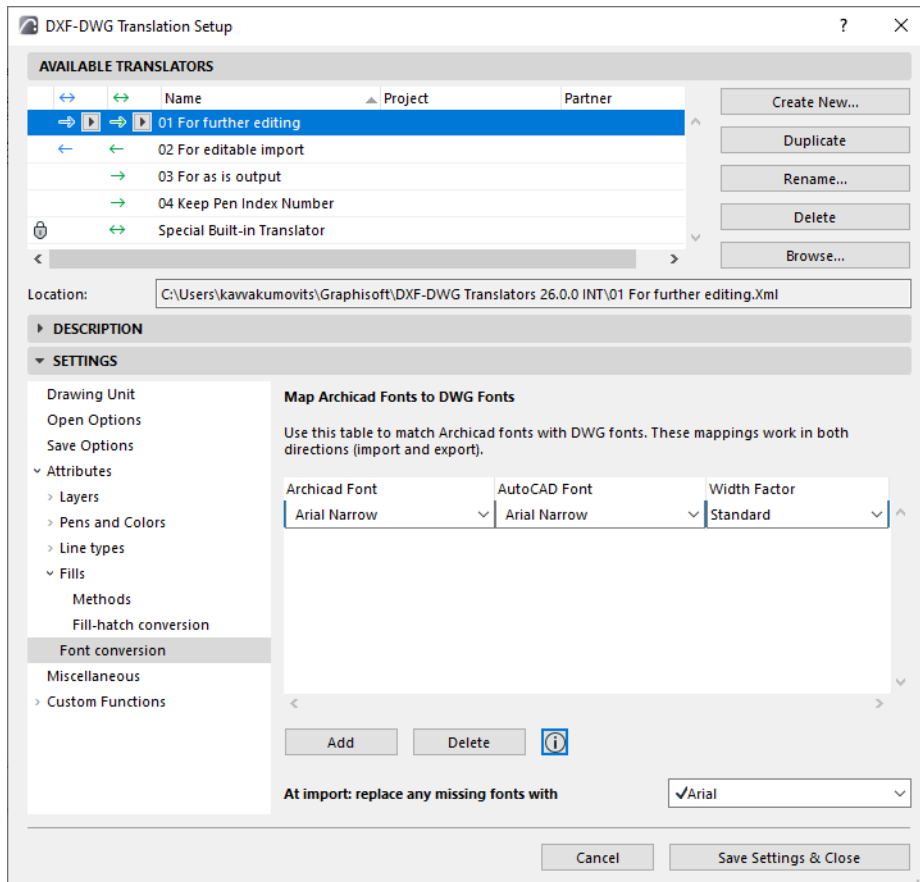
Choose from among the available hatch patterns using the drop-down menu below AutoCAD Hatch Pattern.

Enter a **Hatch Scale** in which to display the converted Fill.

Font Conversion (DXF/DWG Translation Setup)

These are options in the DXF/DWG Translation Setup dialog box.

See [DXF/DWG Translation Setup](#).



Use this panel to map the conversion of Archicad fonts to AutoCAD fonts, and vice versa.

1. Click **Add** to create a mapping rule.
2. Use the font pop-ups to match Archicad with AutoCAD fonts.
3. The Width Factor field affects exported text.
 - Optionally, click this field to enter a factor to increase or decrease the font width in AutoCAD. (The default “Standard” is = 1.00, that is, no change in font width.)

Replace missing fonts at import

Use this field to enter a backup font to use at import, in case any incoming fonts are not yet mapped.

Miscellaneous (DXF/DWG Translation Setup)

These are options in the DXF/DWG Translation Setup dialog box.

See [DXF/DWG Translation Setup](#).

These settings concern character conversion including the use of national characters not in the English alphabet, font substitution rules for missing specifications and binary DXF.

Allow national characters in Layer, linetype, block names: If the recipient AutoCAD works in the same national environment as your Archicad, you can check this checkbox.

Write binary DXF: This compressed version of DXF takes up about half the space of the text DXF format.

Note: If you really want the data to be compact, it is suggested that you write to DWG format.

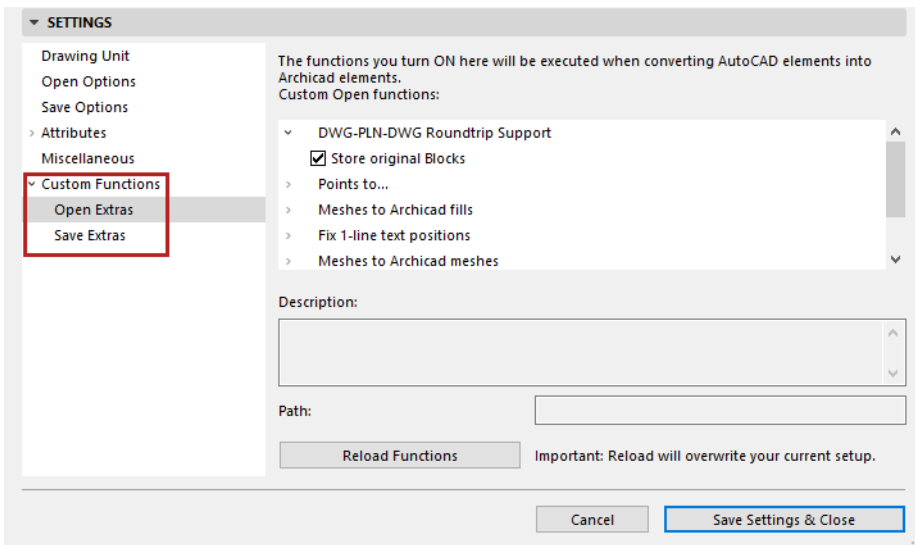
Convert Splines into Polylines: Check this box to convert Splines into Polylines in Archicad. Archicad will approximate the AutoCAD Spline using a series of connected straight line segments (that is, a Polyline). This may be useful for fine-tuning the Archicad Spline.

If you leave this box unchecked, AutoCAD Splines are converted into Archicad Splines.

Replace Missing Renovation Status with: Choose a Renovation status in Archicad for imported elements that do not have an assigned renovation status.

Custom Functions (DXF/DWG Translation Setup)

These are options in the DXF/DWG Translation Setup dialog box. The available options will vary depending on the language version of Archicad you are using.



Custom Functions

Contains add-ons for both **Open Extras** and **Save Extras** operations. Several functions are loaded by default. Turn them on or off using the checkboxes.

Drag and drop to change the order in which they are executed.

If you add or delete any functions, press the **Reload Functions** button to refresh the list.

Store original Blocks

Optimizes the roundtrip of DWG files to and from Archicad. Original blocks will be stored, so that unmodified DWG blocks, once reexported from Archicad, will remain intact.

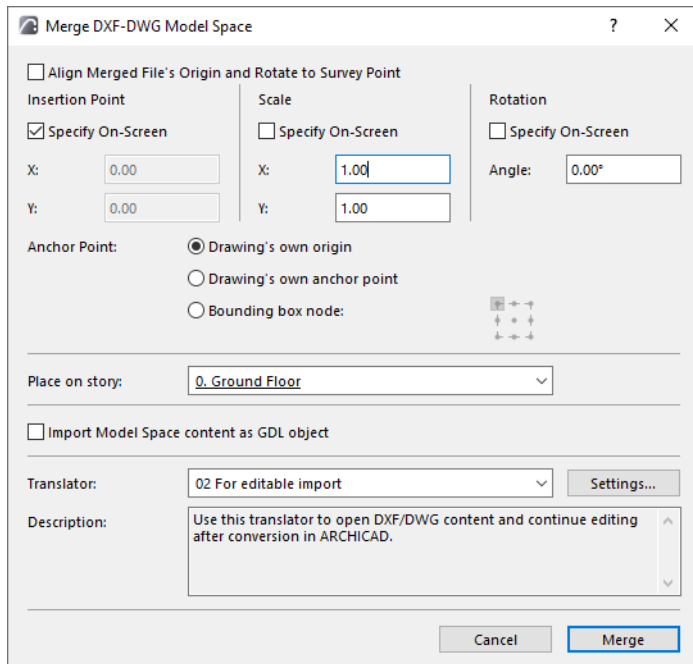
Leaders to Labels

This function works only if the Leader's content is in Text or Mtext, and if the Leader was created as a simple Leader. Multileaders and non-text content are not supported.

Merge DXF-DWG Model Space Dialog Box

For a description of the merge process, see [Merge DXF-DWG](#).

Use these controls define where and how you want the DXF-DWG file to be placed on the Archicad Floor Plan.



- **Align Merged File's Origin and Rotate to Survey Point:** check this to insert the merged model at the position and with the rotation of the Archicad Survey Point.

The next controls let you define the insertion, scale and rotation of the placed drawing.

- **Insertion point:** (not available if you checked Align to Survey Point)
 - Enter X/Y coordinates for the lower left corner of the bounding box or Drawing origin. Or check **Specify On-Screen** to define the insertion point by clicking on the Floor Plan.
- **Scale:**
 - Specify a scale factor in X/ Y directions. Negative values will mirror the drawing.
 - Or check **Specify On-Screen** to define scale on the screen.
- **Rotation:** (not available if you checked Align to Survey Point)
 - Define rotation angle for the placed drawing around its insertion point.
 - Or check **Specify On-Screen** box rotate manually on screen.

Anchor Point: Defines the point of the bounding box used to place the drawing.

- **Drawing's own origin:** the Global Origin (0;0 coordinate) of the attached drawing will be located at the coordinates specified by you on screen or in this Dialog.
- **Drawing's own anchor point:** the attached drawing's own anchor point (if any) will be located at the coordinates specified.
- **Bounding box node:** Choose any of the 9 bounding box points. The drawing will be placed using that point.

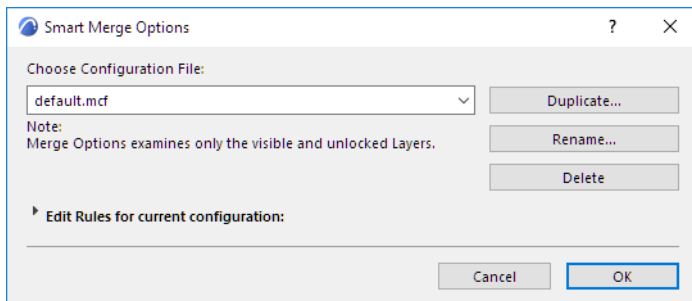
Place on Story: Use this drop-down list to choose the story onto which to place the drawing.

Import Model Space content as GDL Object: Check this box to import the model content as a single GDL Object.

Translator: Choose a DXF/DWG Translator to convert the attached AutoCAD drawing . Click Settings to access and change Translator Settings if needed.

Smart Merge Options

To access this dialog box, use **File > Interoperability > DXF-DWG > Smart Merge Options**.



Use these controls to configure the Smart Merge process for DXF/DWG files.

For more information, see [Round-Trip Conversion \(Smart Merge\)](#).

You can choose an existing configuration, or create a new one. The configuration files (mcf) must be located in the folder of the Add-on.

You can also duplicate, rename, or delete these configuration files with the appropriate buttons. You can save your configurations for your different needs into different files. When you merge a DXF/DWG file, Archicad first looks for the mcf file defined in the #MERGECONFIG section of the DXF-DWG Config.xml file. If this section is not used, then Archicad looks for the dxf.mcf configuration file and the default.mcf file, successively. If none of these exist, select another mcf file at **File > Interoperability > DXF-DWG > Smart Merge Options** dialog box.

A configuration set tells you what will happen when you merge a DXF/DWG drawing into the plan.

The Merge process affects only visible and unlocked elements, so if you want to avoid changing any elements, you should hide or lock them.

You can study or edit the details of the configuration set, by clicking the small arrow next to **Edit rules for current configuration text**. The rules in the list are editable. The maximum number of rules is 16. A rule is made of a condition part and one or more actions. If an element coming from the DXF/DWG drawing meets a condition of a rule, the actions of that rule will be executed on it.

Those elements that do not meet any of the conditions in the list are ignored.

Add or Remove Rules

The **New Rule** button adds a new but empty rule to the end of the list. The condition and the actions for this rule will be undefined. A rule becomes valid if the condition is fully defined and at least one action is set.

You can remove a rule entirely by selecting its line and clicking the **Delete Rule** button. (Separator lines are not selectable.)

Editing a Rule

If you want to edit a condition, you have to click that line in the list, then choose another type of condition in the displayed pop-up window. To edit an action, click the row with the action you want to change, and select another *enabled* action from the pop-up. If you want to delete an action from a rule, select it and click the **Delete Action** button. A rule must have at least one action, thus you cannot delete the last one. The **Add Action** button inserts an action after the selected one. At first it will not be defined, so you must set a valid value for it.

You have to read the actions sequentially, because there are cases when the execution order is irrelevant (e.g. you get different results if you first duplicate a wall in its original position and then move the duplicated original item or if you first move the original item and then duplicate it in the new position). Usually the actions are applied to the elements on the Archicad plan, but not always. The operations following a duplicate element action are applied to the replica. The actions following the Outline from Merged file action are applied to the “Outlined drawing” stored in the DXF/DWG file.

For more information on “Outlined drawing,” read the next section: [More Information on Smart Merge](#).

More Information on Smart Merge

To understand how Merge works, let’s have a look at what happens in detail:

In Archicad:

- You may add new elements. Even if you create them by copying existing elements, they are still new, as they get new unique IDs.
- You may change elements created earlier. That is, you may drag them, stretch them, or even just alter a surface parameter. When you insert or change a window or door in a wall, it means the wall will change, because its modification date will change. This means that when merging the DWG/DXF file back into Archicad, your only option will be to display the “Outlined drawing” next to the original Archicad wall model element.
- You may delete anything.

What the AutoCAD-user consultant can do with your DWG file:

- He/she can add new elements.
- He/she can move, rotate or mirror items you exported. He can even stretch them, but it is not recommended: the Merge function processes only size-preserving transformations.
- He/she can delete these elements and he can duplicate them.

But how do we know that an item of the drawing (Autodesk terminology calls it entity) is something we exported or the other user created. How do we know what he/she did to items of the former category?

Autodesk provides means to add custom data (called extended entity data) to each entity.

This allows us to save Archicad-specific information along with the image of each of the elements we export. These elements can be simple, so they can be represented by one AutoCAD entity (e.g. a circle); or complex (e.g. a wall that is drawn as at least 2 lines and 1 hatch).

The complex elements have to be joined together in blocks (something like Library Parts in Archicad) in order to handle them as units.

Merge needs the original, unchanged element present in the plan in order to incorporate the changes the AutoCAD user made on it, so only minimal information will be stored:

- The unique (Archicad) ID and type of the element
- The three coordinates that AutoCAD (or the other program) transforms if the entity is moved, rotated or stretched. (If the other program is not AutoCAD, it must be done similarly)
- The AutoCAD ID of the Layer of the element, so that Archicad can detect if the AutoCAD user moved the element to another Layer
- The AutoCAD ID of the originally exported element to separate the original from the duplicates; this is what happens in the Merge process to the different elements
- Elements created on the Floor Plan after exporting the DWG: no action

Entities added by the AutoCAD user (the duplicates of the Archicad-generated elements not included) are added to the plan.

Elements exported to the drawing (which may or may not have been modified by the other program): note that nothing prevents you and the AutoCAD user from simultaneously modifying/deleting these elements. The situation can be described by a matrix like this:

	1	2	3
A	1/A	...	
B			3/B
C	...		
D			

where:

1: the Archicad user leaves the element unchanged

2: the Archicad user modifies the element

3: the Archicad user deletes the element

A: the AutoCAD user leaves the element unchanged

B: the AutoCAD user modifies the element

C: the AutoCAD user copies the element

D: the AutoCAD user deletes the element

- the case 1/A is simple enough: no one modified the element so it remains unchanged;
- 1/B: the original state of the element can be retrieved from the plan so there is no problem applying the changes;
- 1/C: like 1/B just applied to the copy of the original element;
- 1/D: the element gets deleted.

But if you have modified (2) or deleted (3) an element, the results may be very different. If you moved a wall the AutoCAD user left in place (2/A) but mounted tubes on it: now the tubes will appear in the middle of nowhere.

Or both of you moved an element (2/B): your move has been completed, but you do not know how to apply his movement because the original position is not preserved in the plan.

The only way to deal with this problem is this: the AutoCAD drawing contains the image of the element reflecting the intended changes. This image (the "Outline") can be drawn directly in the Floor Plan so you can see what the AutoCAD user meant to do. You can then decide what to do manually.

Note: If an element was scaled in AutoCAD, the Merge Add-on will outline the drawing, and scaled elements will appear in Archicad as single lines, arcs and Fills.

- 3/D : both of you meant to delete the element. Since you have already done it, you have nothing to do.

Please bear in mind that Merge is configurable; the above mentioned solutions are just suggestions.

Many other possibilities exist:

- You can move the original element to another Layer.
- You can move the element coming from the AutoCAD drawing to another Layer.

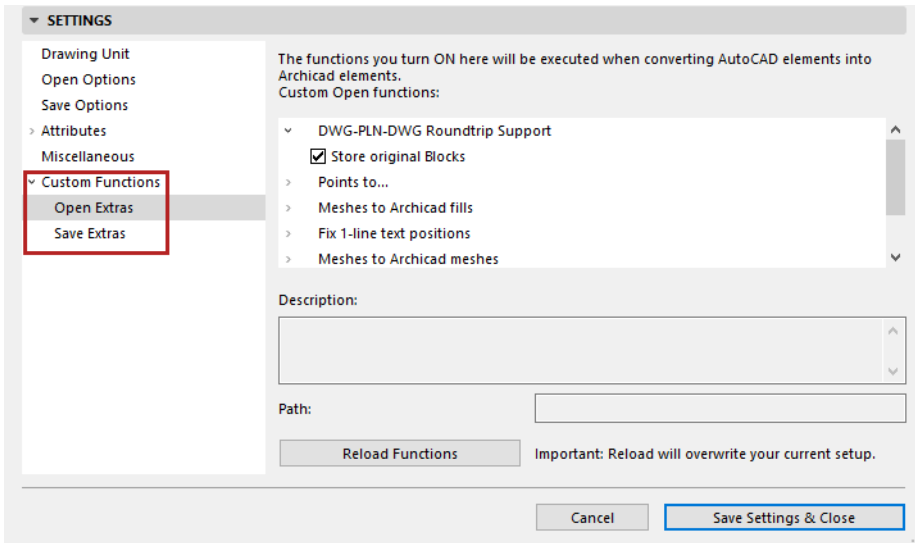
- You can ask Archicad to request confirmation at any point of an action sequence. This means that Archicad will zoom to the element in question so you can examine the changes and decide on Merge Actions.

The available actions are:

- Put Element on <layer> Layer: this action puts the Floor Plan element to the Layer specified in the DXF/DWG file
- Put Element on “Deleted” Layer
- Put Element on “Edited” Layer
- Put Element on “Copied” Layer: these actions put the element on a special Layer created by the Add-on. This is useful when you would like to separate elements edited by the AutoCAD user.
- Duplicate Element: creates a copy of the element
- Transform Element: executes the transformations applied to the element in the DXF/DWG file (dragging, rotating, mirroring or even changing Layers if needed). Note that the last change is only carried out if the “Put Element on Layer” action is also active. Size changing operations are ignored.
- Delete Element: removes the element from the plan
- Outline from Merged File: this action should be set when you want to put the “Outlined drawing” of the element on the plan, or when no other action can be chosen. This is the case when the original element was changed or deleted in Archicad. This means the element will be generated from the AutoCAD entities as a set of 2D Archicad elements.

Custom Functions (DXF/DWG Translation Setup)

These are options in the DXF/DWG Translation Setup dialog box. The available options will vary depending on the language version of Archicad you are using.



Custom Functions contains add-ons for both **Open Extras** and **Save Extras** operations. Several functions are loaded by default. Turn them on or off using the checkboxes.

Drag and drop to change the order in which they are executed.

If you add or delete any functions, press the **Reload Functions** button to refresh the list.

Store original Blocks

Optimizes the roundtrip of DWG files to and from Archicad. Original blocks will be stored, so that unmodified DWG blocks, once reexported from Archicad, will remain intact; no unnecessary new blocks will be created.

Leaders to Labels

This function works only if the Leader's content is in Text or Mtext, and if the Leader was created as a simple Leader. Multileaders and non-text content are not supported.

Merge DXF-DWG Model Space Dialog Box

For a description of the merge process, see [Merge DXF-DWG](#).

Use these controls define where and how you want the DXF-DWG file to be placed on the Archicad floor plan.

- **Align Merged File's Origin and Rotate to Survey Point:** check this to insert the merged model at the position and with the rotation of the Archicad Survey Point.

The next controls let you define the insertion, scale and rotation of the placed drawing.

- **Insertion point:** (not available if you checked Align to Survey Point)
 - Enter X/Y coordinates for the lower left corner of the bounding box or Drawing origin. Or check **Specify On-Screen** to define the insertion point by clicking on the Floor Plan.
- **Scale:**
 - Specify a scale factor in X/ Y directions. Negative values will mirror the drawing.
 - Or check **Specify On-Screen** to define scale on the screen.
- **Rotation:** (not available if you checked Align to Survey Point)
 - Define rotation angle for the placed drawing around its insertion point. Or check **Specify On-Screen** box rotate manually on screen.

Anchor Point: Defines the point of the bounding box used to place the drawing.

- **Drawing's own origin:** the Global Origin (0;0 coordinate) of the attached drawing will be located at the coordinates specified by you on screen or in this Dialog.
- **Drawing's own anchor point:** the attached drawing's own anchor point (if any) will be located at the coordinates specified.
- **Bounding box node:** Choose any of the 9 bounding box points. The drawing will be placed using that point.

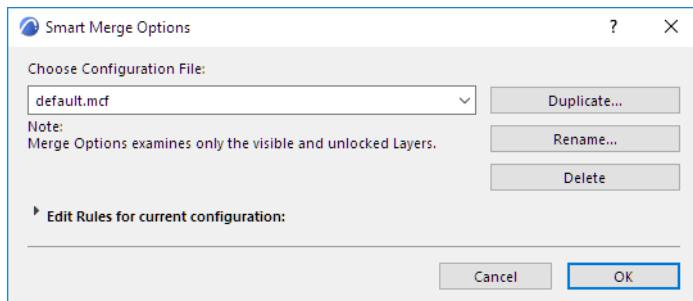
Place on Story: Use this drop-down list to choose the story onto which to place the drawing.

Import Model Space content as GDL Object: Check this box to import the model content as a single GDL Object.

Translator: Choose a DXF/DWG Translator to convert the attached AutoCAD drawing (the default translator, “For editable import”, is usually appropriate). Click Settings to access and change Translator Settings if needed.

Smart Merge Options

To access this dialog box, use **File > Interoperability > DXF-DWG > Smart Merge Options**.



Use these controls to configure the Smart Merge process for DXF/DWG files.

For more information, see [Round-Trip Conversion \(Smart Merge\)](#).

You can choose an existing configuration, or create a new one. The configuration files (.mcf) must be located in the folder of the Add-on.

You can also duplicate, rename, or delete these configuration files with the appropriate buttons. You can save your configurations for your different needs into different files. When you merge a DXF/DWG file, Archicad first looks for the .mcf file defined in the #MERGECONFIG section of the DXF-DWG Config.xml file. If this section is not used, then Archicad looks for the dxf.mcf configuration file and the default.mcf file, successively. If none of these exist, select another .mcf file at **File > Interoperability > DXF-DWG > Smart Merge Options** dialog box.

A configuration set tells you what will happen when you merge a DXF/DWG drawing into the plan.

The Merge process affects only visible and unlocked elements, so if you want to avoid changing any elements you should hide or lock them.

You can study or edit the details of the configuration set, by clicking the small arrow next to **Edit rules for current configuration text**. The rules in the list are editable; the maximum number of rules is 16. A rule is made of a condition part and one or more actions. If an element coming from the DXF/DWG drawing meets a condition of a rule, the actions of that rule will be executed on it.

Those elements that do not meet any of the conditions in the list are ignored.

Add or Remove Rules

The **New Rule** button adds a new but empty rule to the end of the list. The condition and the actions for this rule will be undefined. A rule becomes valid if the condition is fully defined and at least one action is set.

You can remove a rule entirely by selecting its line and clicking the **Delete Rule** button. (Separator lines are not selectable.)

Editing a Rule

If you want to edit a condition, you have to click that line in the list, then choose another type of condition in the displayed pop-up window. To edit an action, click the row with the action you want to change, and select another *enabled* action from the pop-up. If you want to delete an action from a rule, select it and click the **Delete Action** button. A rule must have at least one action, thus you cannot delete the last one. The **Add Action** button inserts an action after the selected one. At first it will not be defined, so you must set a valid value for it.

You have to read the actions sequentially, because there are cases when the execution order is irrelevant (e.g. you get different results if you first duplicate a wall in its original position and then move the duplicated original item or if you first move the original item and then duplicate it in the new position). Usually the actions are applied to the elements on the Archicad plan, but not always. The operations following a duplicate element action are applied to the replica. The actions following the Outline from Merged file action are applied to the “Outlined drawing” stored in the DXF/DWG file.

For more information on “Outlined drawing,” read the next section: [More Information on Smart Merge](#).

More Information on Smart Merge

To understand how Merge works, let’s have a look at what happens in detail:

In Archicad:

- You may add new elements. Even if you create them by copying existing elements, they are still new, as they get new unique IDs.
- You may change elements created earlier. That is, you may drag them, stretch them, or even just alter a surface parameter. When you insert or change a window or door in a wall, it means the wall will change, because its modification date will change. This means that when merging the DWG/DXF file back into Archicad, your only option will be to display the “Outlined drawing” next to the original Archicad wall model element.
- You may delete anything.

What the AutoCAD-user consultant can do with your DWG file:

- He/she can add new elements.
- He/she can move, rotate or mirror items you exported. He can even stretch them, but it is not recommended: the Merge function processes only size-preserving transformations.
- He/she can delete these elements as well and he can duplicate them.

But how do we know that an item of the drawing (Autodesk terminology calls it entity) is something we exported or the other user created; and how do we know what he/she did to items of the former category?

Autodesk provides means to add custom data (called extended entity data) to each entity.

This allows us to save Archicad-specific information along with the image of each of the elements we export.

These elements can be simple, that is, something that can be represented by one AutoCAD entity (e.g. a circle); or complex (e.g. a wall that is drawn as at least 2 lines and 1 hatch).

The complex elements have to be joined together in blocks (something like Library Parts in Archicad) in order to handle them as units.

Merge needs the original, unchanged element present in the plan in order to incorporate the changes the AutoCAD user made on it. This means that only minimal information will be stored:

- The unique (Archicad) ID and type of the element
- The three coordinates that AutoCAD (or the other program) transforms if the entity is moved, rotated or stretched; if the other program is not AutoCAD, it must be done similarly
- The AutoCAD ID of the layer of the element, so that Archicad can detect if the AutoCAD user moved the element to another layer
- The AutoCAD ID of the originally exported element to separate the original from the duplicates; this is what happens in the Merge process to the different elements
- Elements created on the Floor Plan after exporting the DWG: no action

Entities added by the AutoCAD user (the duplicates of the Archicad-generated elements not included): you will most likely want to keep them, so they are added to the plan.

As to elements exported to the drawing (which may or may not have been modified by the other program): the problem here is that nothing prevents you and the AutoCAD user from simultaneously modifying/deleting these elements. The situation can be described by a matrix like this:

	1	2	3
A	1/A	...	
B			3/B
C	...		
D			

where:

1: the Archicad user leaves the element unchanged

2: the Archicad user modifies the element

3: the Archicad user deletes the element

A: the AutoCAD user leaves the element unchanged

B: the AutoCAD user modifies the element

C: the AutoCAD user copies the element

D: the AutoCAD user deletes the element

- the case 1/A is simple enough: no one modified the element so it remains unchanged;
- 1/B: the original state of the element can be retrieved from the plan so there is no problem applying the changes;
- 1/C: like 1/B just applied to the copy of the original element;
- 1/D: the element gets deleted.

But if you have modified (2) or deleted (3) an element, the results may be very different. If you moved a wall the AutoCAD user left in place (2/A) but mounted tubes on it: now the tubes will appear in the middle of nowhere.

Or both of you moved an element (2/B): your move has been completed, but you do not know how to apply his movement because the original position is not preserved in the plan.

The only way to deal with this problem is this: the AutoCAD drawing contains the image of the element reflecting the intended changes. This image (the "Outline") can be drawn directly in the Floor Plan so at least you can see what the AutoCAD user meant to do. You can then decide what to do manually.

Note: If an element was scaled in AutoCAD, the Merge Add-on will outline the drawing, and scaled elements will appear in Archicad as single lines, arcs and fills.

- 3/D is the only simple thing: both of you meant to delete the element; since you have already done it, you have nothing to do.

Please bear in mind that Merge is configurable; the above mentioned solutions are just suggestions.

Many other possibilities exist:

- You can move the original element to another layer.
- You can move the element coming from the AutoCAD drawing to another layer.

- You can ask Archicad to request confirmation at any point of an action sequence. This means that Archicad will zoom to the element in question so you can examine the changes and decide on Merge Actions.

The available actions are:

- Put Element on <layer> layer: this action puts the Floor Plan element to the layer specified in the DXF/DWG file
- Put Element on “Deleted” layer
- Put Element on “Edited” layer
- Put Element on “Copied” layer: these actions put the element on a special layer created by the Add-on. This is useful when you would like to separate elements edited by the AutoCAD user.
- Duplicate Element: creates a copy of the element
- Transform Element: executes the transformations applied to the element in the DXF/DWG file (dragging, rotating, mirroring or even changing layers if needed). Note that the last change is only carried out if the “Put Element on layer” action is also active. Size changing operations are ignored.
- Delete Element: removes the element from the plan

Outline from Merged File: this action should be set when you want to put the “Outlined drawing” of the element on the plan, or when no other action can be chosen. This is the case when the original element was changed or deleted in Archicad. This means the element will be generated from the AutoCAD entities as a set of 2D Archicad elements.

Surfaces (Cineware)

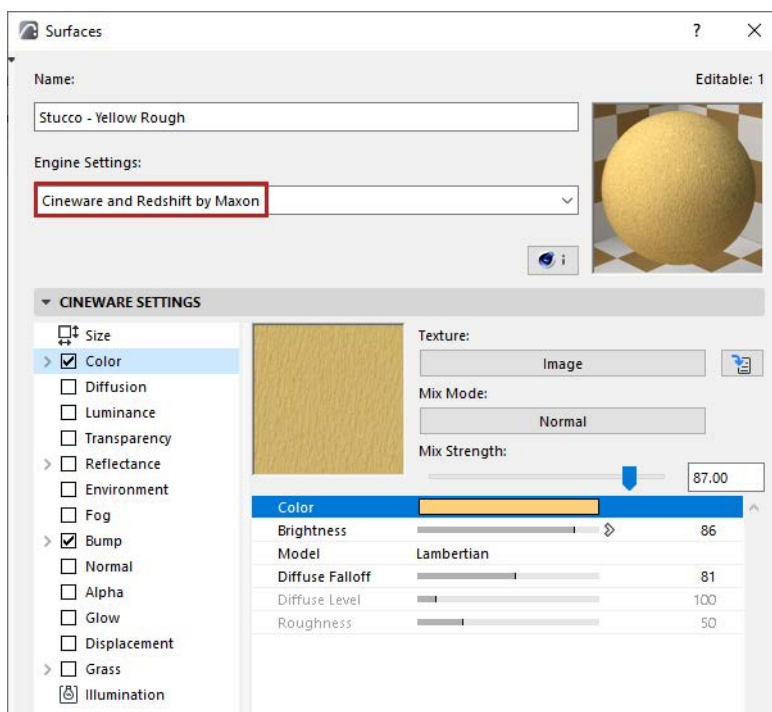
Open the Surfaces dialog:

- Options > Element Attributes > Surfaces
- Attributes toolbar (Windows > Toolbars > Attributes)

From Engine Settings, choose Cineware and Redshift. This means that the surface settings here are optimized to work with the Cineware-Redshift PhotoRendering engine.

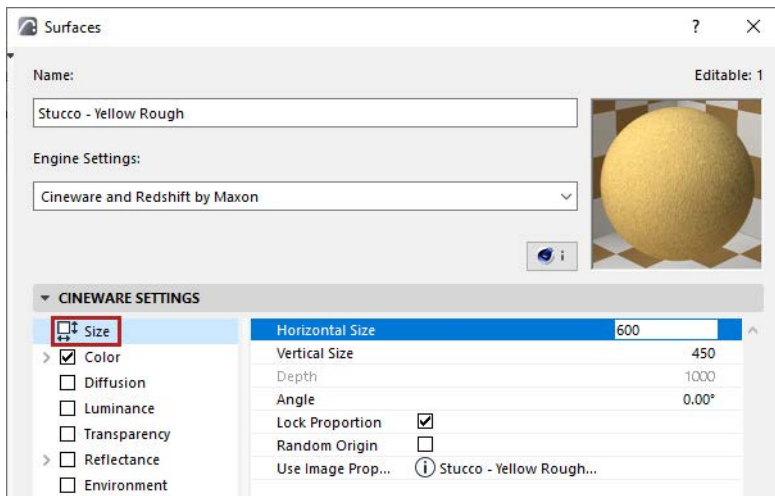
Important: All surfaces are specific to the engine for which they are set up. If you change a surface parameter for one engine, those modifications will not take effect for the same surface parameter if you switch to another engine.

Note: The Redshift renderer (GPU based) from Maxon, based on Maxon's R23 engine, is included in Archicad as a Technology Preview, with limitations. See [Using Redshift Renderer](#).



Choose a surface from the pop-up chooser.

For general information, see [Using the attribute dialogs](#).



Size settings for surface unit

Define how the surface unit will be scaled on the object to which it is applied.

- Vary the size of the surface unit with **Horizontal Size** and **Vertical Size** parameters.
- Adjust the angle of the surface unit (e.g. if it uses a pattern)
- Click **Lock Proportion** to keep these parameters relatively constant.
- Enable **Random Origin** to begin the surface pattern at a random point, rather than starting it at the project origin.

Related Topics

[Cineware Surface Channels](#)

[Textures \(Cineware Surfaces\)](#)

[Special Settings for Image Textures \(Cineware Surfaces\)](#)

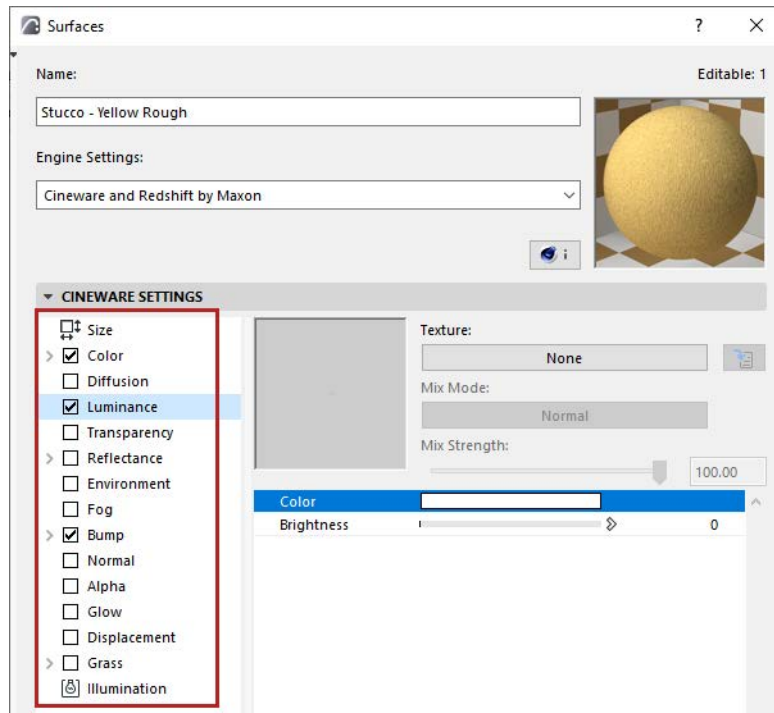
[Shaders List \(Cineware Surfaces\)](#)

[Using Layers to Combine Shaders \(Cineware Surfaces\)](#)

[Watch the Video](#)

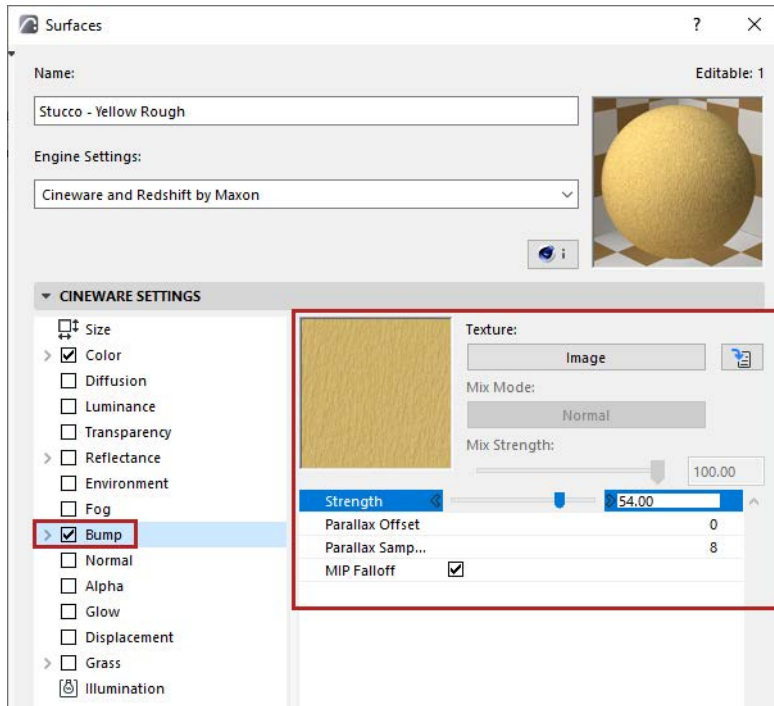
Cineware Surface Channels

Surfaces for the Cineware engine consist of a set of channels, listed on the left side of the dialog box. Each channel represents one surface property (e.g. Luminance, Fog, Environment, Normal, Alpha, Glow, Transparency, Specular, Displacement).



Many surfaces can be simulated using just a few channels. So don't be intimidated by the numerous channels and their wide range of parameters. Generally speaking, all everyday surfaces can be created using just a handful of these channels.

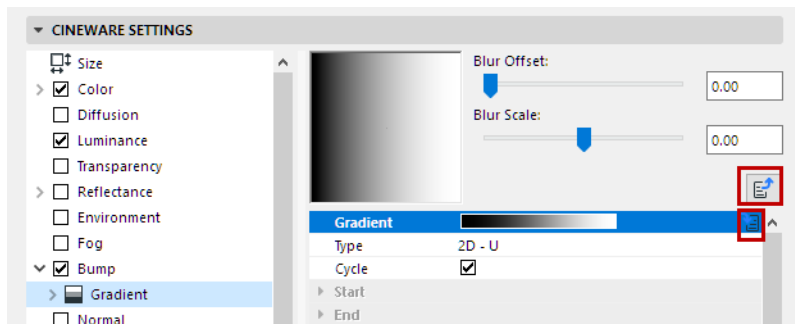
For any selected channel in the list, its settings and a preview window are seen on the right.



Navigation in Surface Channels

Surface Channels and their associated textures/shaders often include several levels of parameters.

- Click the green down-arrow icon to open the list of associated parameters.
- Use the blue “Step Out” button to return to your previous view.



Color (Cineware Surface Channel)

Diffusion (Cineware Surface Channel)

Irregularities in surface color (works by brightening and darkening the color channel)

Luminance (Cineware Surface Channel)

Transparency (Cineware Surface Channel)

Transparency (including refraction index)

Reflectance (Cineware Surface Channel)

Ability to reflect other objects. Also controls Specular and Color effects.

Environment (Cineware Surface Channel)

Environment reflection (simulates reflection)

Fog (Cineware Surface Channel)

Fog effect

Bump (Cineware Surface Channel)

Virtual bumps on a surface

Normal (Cineware Surface Channel)

Virtual surface irregularity that is lit realistically

Alpha (Cineware Surface Channel)

Localized texture invisibility

Glow (Cineware Surface Channel)

Glow Halo around an object

Displacement (Cineware Surface Channel)

Realistic bumps on a surface

Grass (Cineware Surface Channel)

Create realistic-looking grass as a surface

Illumination (Cineware Surface Channel)

Use the Illumination channel in Surfaces to enable or disable the generation and reception of Global Illumination (GI) to and from this surface, and to control the strength of the caustics effects.

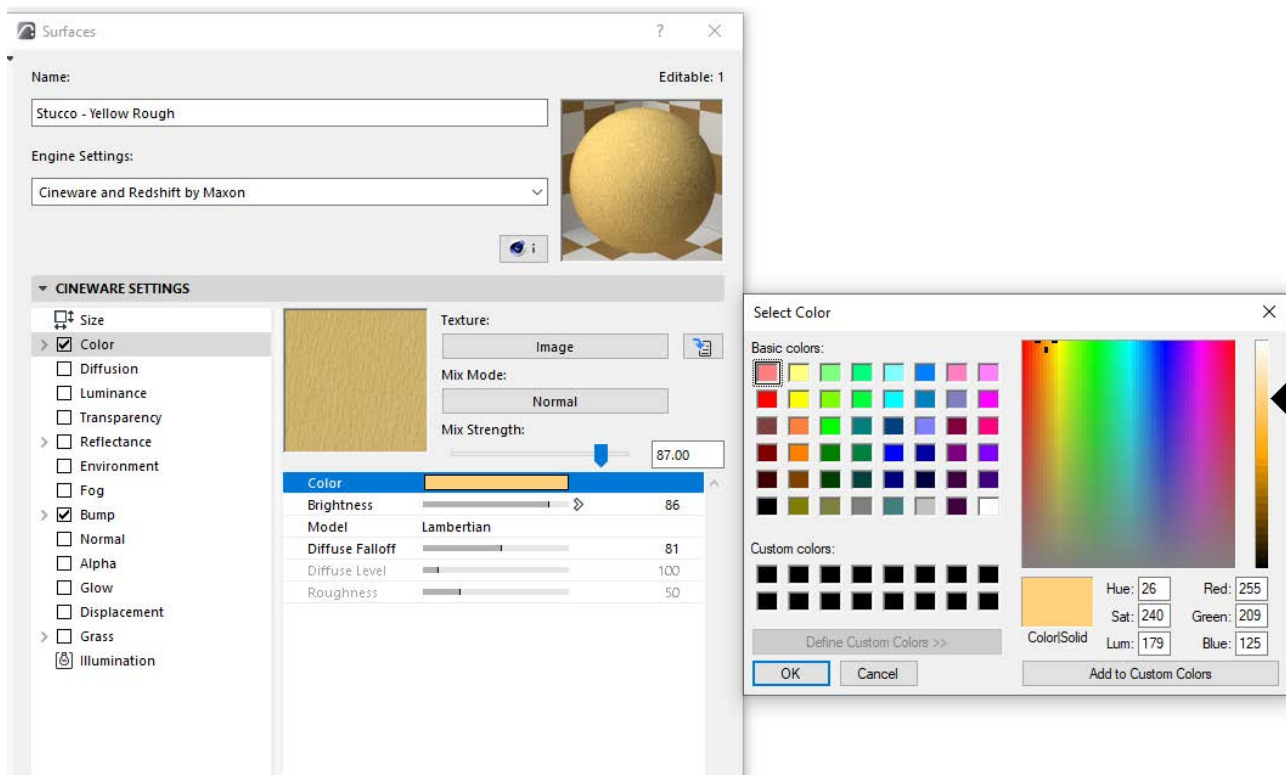
Color (Cineware Surface Channel)

The settings on this page define the basic color of the surface.

Defining Color and Brightness Values

Many surface channels contain (apparently identical) color selection options. Once a color has been defined it can be brightened or darkened using the Brightness slider.

Click the Color swatch to access the Color picker.

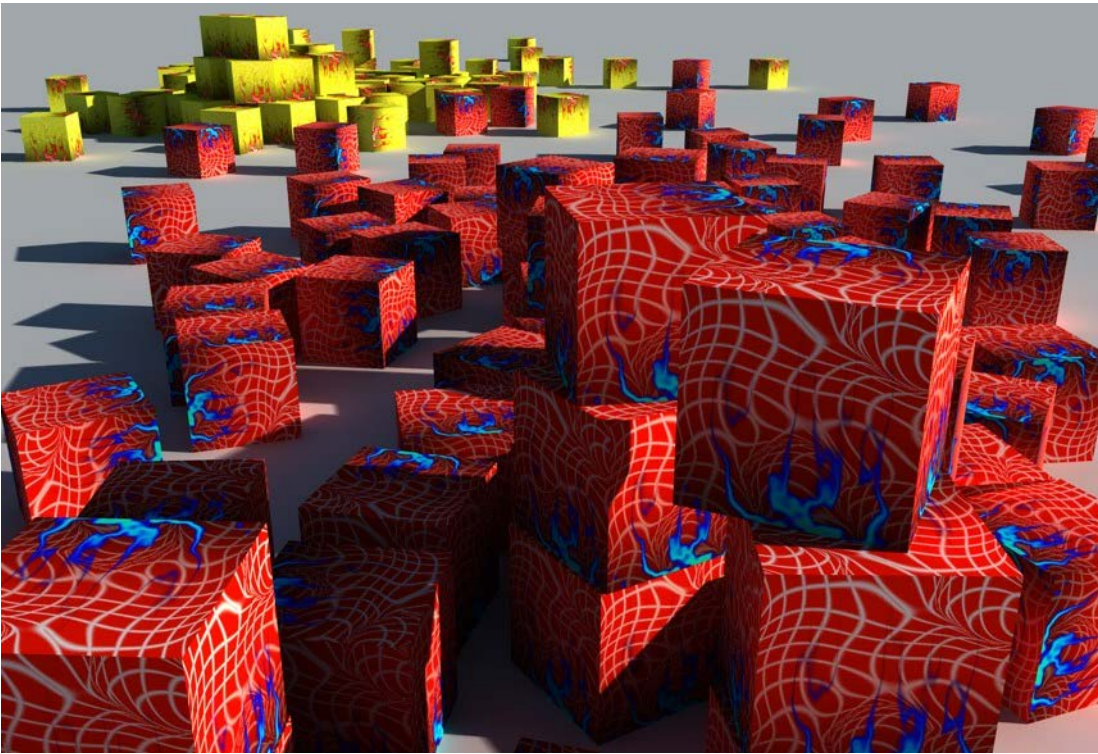


If you want to create a more complicated surface, such as a checkered pattern that uses several colors, you can load a texture. The texture is layered above the color.

[See *Textures \(Cineware Surfaces\)*.](#)

If you want to see the color only and not the texture, set the **Mix Strength** value to 0%.

[See *Mix Mode and Mix Strength \(Cineware Surfaces\)*.](#)



Model

Choose the illumination model for the surface: Lambertian or Oren-Nayar.

- For a shiny surface needing accurate highlights, choose Lambertian.
- For rough surfaces — like cloth, sand, concrete and plaster — choose Oren-Nayar.

Note: The Blinn model is now located in the Reflectance channel's *Type* menu as Specular-Blinn (Legacy).

Diffuse Falloff

This value defines the falloff of brightness between the specular highlight (surface Normals point in the direction of the light source) and the regions in which the surface Normals have an angle of 90° to the light source, i.e., they do not reflect light from the object surface. The default value of 0% reflects a normal falloff behavior. The larger the value, the more uniform the brightness will be, up to the point of producing a comic-like look (uniform colors). If negative values are used, the 90° angle mentioned above will be reduced accordingly, which will produce correspondingly smaller regions from which light will be reflected.

Diffuse Level

Available only with Oren-Nayar shading.

Use this parameter to alter the general diffuse brightness that is reflected for the Oren-Nayar model (very similar effect to the above-described Brightness setting).

The default of 100% means normal diffuse reflection. Enter a lower value to reduce the amount of reflection.

Roughness

Available only with Oren-Nayar shading.

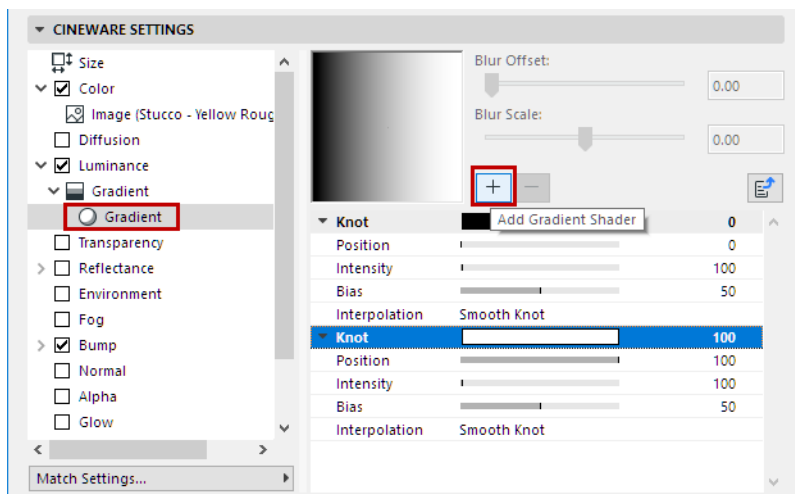
Larger values will produce a correspondingly more diffuse light dispersion; the surface will become correspondingly darker and duller.

Smaller values will produce a light dispersion increasingly similar to the Lambert model.

Adjusting Gradient Parameters

Every time you adjust a color gradient, you can play with the **Knot** parameters, used to set the color and position of the gradient.

- Add as many knots as needed by clicking the Plus icon (Add Gradient Shader) at the top of the list.
- For each knot, you can set its color, intensity, position, interpolation, and bias (the horizontal distance that the curve is displaced between any two knots).



- If you select an existing knot and click Add Gradient Shader, it will duplicate the selected knot. Knots are automatically ordered from 0 to 100 based on their Position value.

Brightness

Once a color has been defined it can be brightened or darkened using the Brightness control. Click the arrow at the right of the slider to increase the value past 100%: this increases brightness dramatically. This can be useful if the brightness of a Luminance channel, for example, is used in conjunction with Global Illumination to illuminate a scene, or if the shadows generated by a weak light source should be strengthened using the surface's Color channel.

Note: Depending on the value of the Glass/Mirror Optimization parameter (Cineware Detailed Settings > Global Illumination > Details), GI will ignore all surfaces whose transparency or reflection Brightness value exceeds this optimization value.

See [Glass/Mirror Optimization](#).

Texture

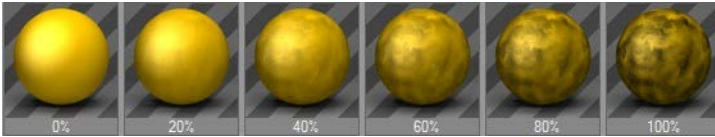
Here a texture or a 2D shader can be defined. If you load a texture or a 2D shader, it is placed on a layer above the color (i.e. the texture is placed on top of the color).

See [Textures \(Cineware Surfaces\)](#).

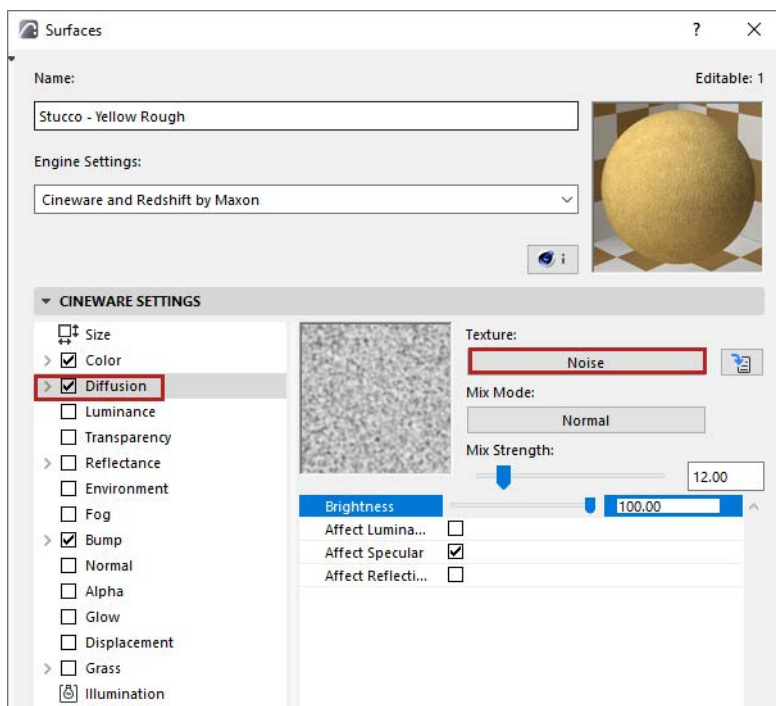
Diffusion (Cineware Surface Channel)

Use this channel to create irregularities in surface color (it works by brightening and darkening the color channel).

Just about any surface can use a little variation to make it look more natural. Objects in the real world almost always show some type of wear and tear, signs of usage or dirt. Adding such characteristics makes a 3D surface look more natural and realistic.



To create such an effect, a texture (image or shader) can be loaded into the Diffusion channel.



Use the Brightness slider to adjust the brightness of the channel's color. The brightness functions somewhat like a multiplier.

Affect Luminance

Enable the Affect Luminance option if you want the diffusion map to affect the luminance property. The darker a pixel in the diffusion map, the darker the corresponding region of the luminance. This helps you add irregularities to the luminance to achieve a more natural look.

Affect Specular

If the Affect Specular option is enabled, the diffusion map is applied to the specular property as well. This will reduce the surface's specular values where the diffusion map is dark. This option is enabled by default since it adds to the realism considerably.

Affect Reflection

Enable Affect Reflection if you want to apply the diffusion map to the reflection and environment properties for a more natural look. The darker a pixel in the diffusion map, the darker the corresponding region of the reflection.

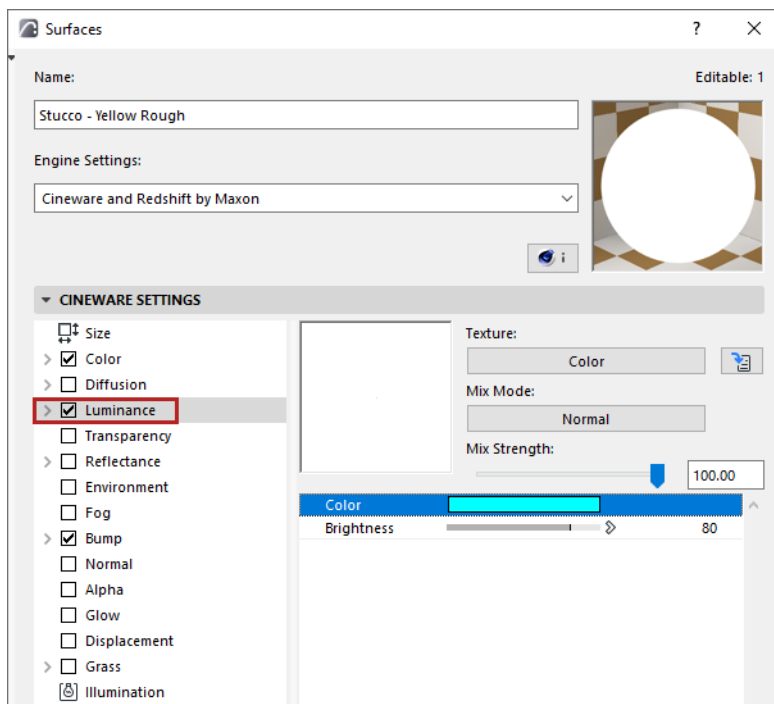
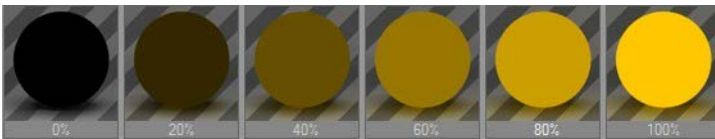
Luminance (Cineware Surface Channel)

Use this channel to add a light-independent, luminescent color to your surface.

A luminescent object can be seen even when there are no lights in the scene. It is self-illuminated.

If you are using an image as your texture, load a file to use as a luminance map. The brighter a pixel in the luminance map, the more luminescent the corresponding region of the surface. If you have chosen a Luminance Color and loaded a texture (luminance map) as well, the color will be added at 100% strength to the texture. If you want to see the result without the chosen color, set Mix Strength to 0%.

Luminescent surfaces are used to help simulate objects that seem to be self-illuminated in the real world, such as the windows of an office block at night, neon writing or a TV screen.



Color and Brightness

See [Defining Color and Brightness Values](#).

Texture

See [Textures \(Cineware Surfaces\)](#).

Mix Mode/Mix Strength

See [Mix Mode and Mix Strength \(Cineware Surfaces\)](#).

Transparency (Cineware Surface Channel)

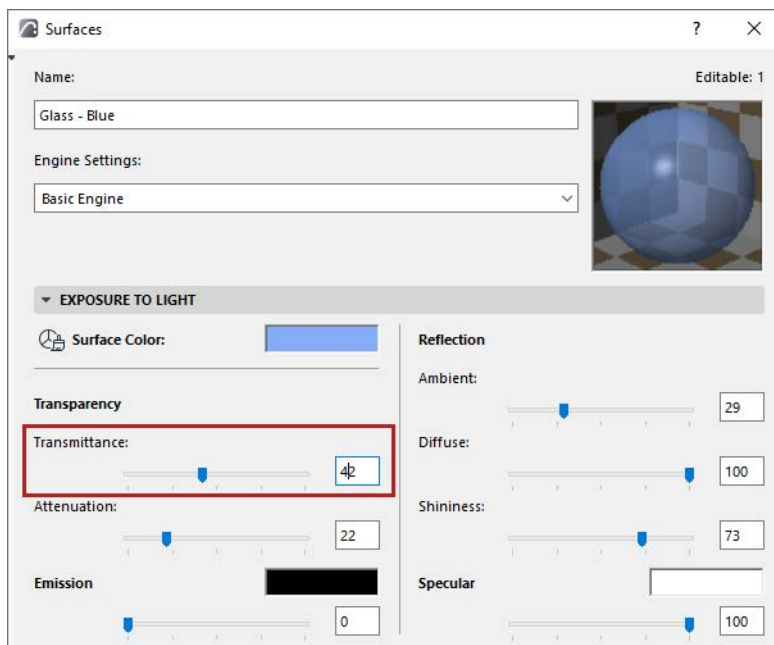
Here you can define the transparency, including a refraction index.

Use the Color and/or Brightness parameters to set a transparency. Using a 50% gray is similar to using white and 50% brightness.

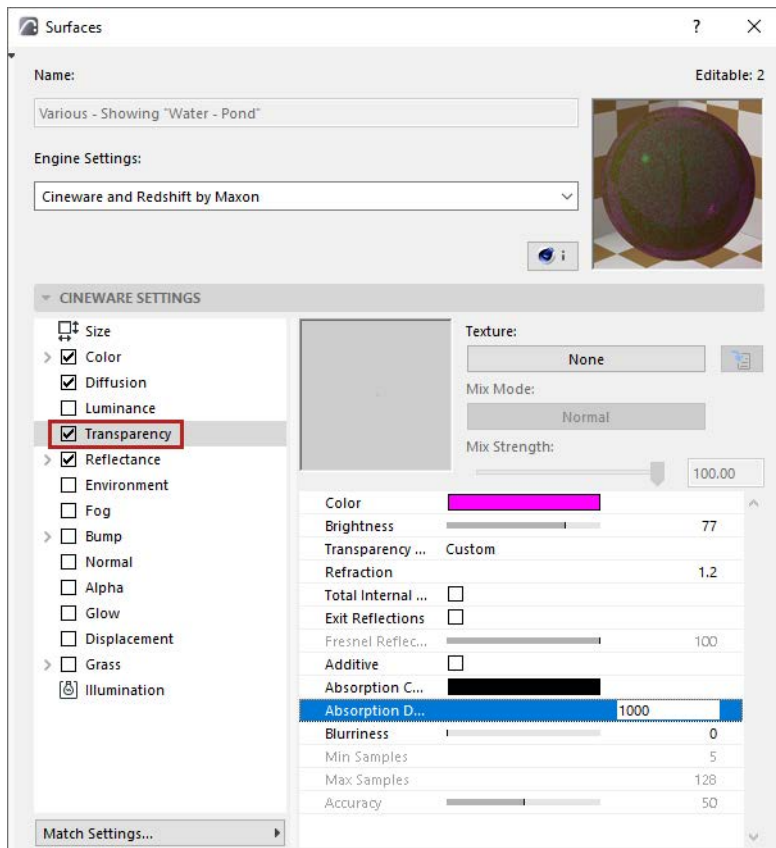
Color percentage + Transparency percentage = 100%

Surface Color	Transparency Percentage	Result
White surface	zero transparency	Totally White (100% white)
White surface	50% transparency	Gray (50% White)
White surface	100% transparency	No color

If the surface has a color (in the Color channel), the color is automatically reduced with increasing transparency.



Tip: Much better results can be achieved if Absorption is used ([see Absorption, below](#)).



You can load a texture as a transparency map. The brighter a pixel in the transparency map, the more transparent the corresponding region of the surface. A transparency texture is similar to a photographic slide: Red parts of the slide allow only red light to pass through; white parts allow all light through. With black, no light can pass through the slide.

Color and Brightness

See [Defining Color and Brightness Values](#).

Transparency Refraction Preset

Choose a refraction preset that automatically applies a Refraction value. These are the same presets that can be found in the Reflectance channel's Fresnel function.

[Watch the Video](#)

Refraction

When not using a Preset, you can set an exact Refraction value.

Useful refraction values

- Vacuum 1,00
- Air 1,00
- Obsidian 1,480 - 1,510
- Onyx 1,486 - 1,658
- Acrylic glass 1,491
- Benzene 1,501
- Crown glass 1,510

Jasper 1,540

Agate 1,544 - 1,553

Amethyst 1,544 - 1,553

Amber 1,550

Quartz 1,550

Sugar 1,560

Diamond 2,417 - 2,419

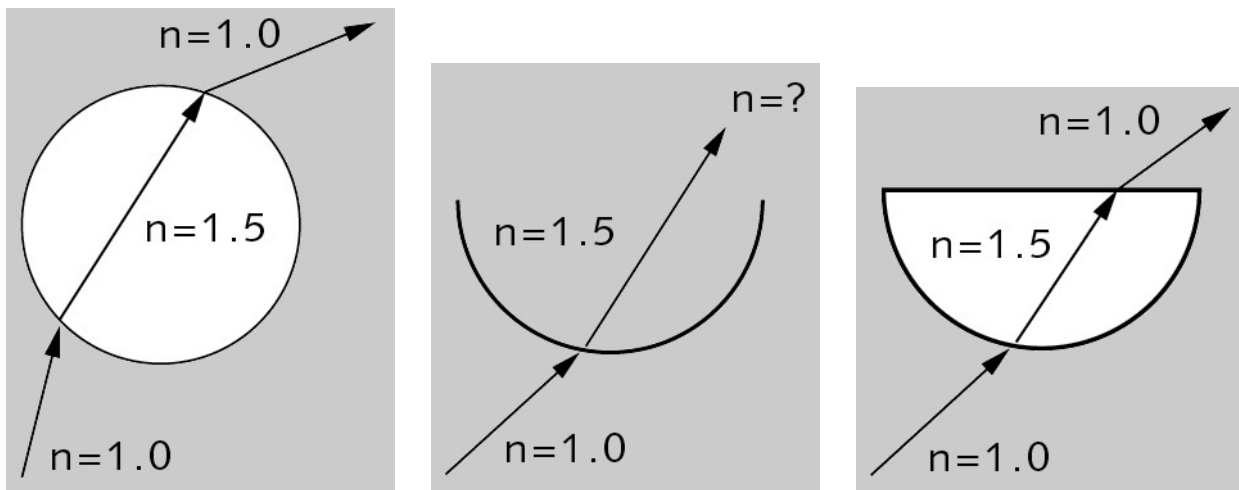
Glass noodle 3,999

When a ray hits a closed surface (leftmost image, below) with transparency and refraction, the ray is bent to simulate refraction. The bent ray is considered to be inside the object. When the ray reaches another surface of the object (the exit side), it is bent back as in real life.

However, if the object is open, the ray may not hit a second surface of the object (center image, below). Thus the ray may not be bent back and the refraction effect may be inaccurate.

For this reason, ensure that all transparent surfaces in the scene are closed (as in the right-side image below).

If you are using refraction with windows - with the refraction parameter other than 1 - the glazing must be closed (that is, you must use an even number of panes). (In Archicad, to improve 3D generation speed, glass panes are modeled as surfaces that have no thickness.)



Total Internal Reflection

Enable the Total Internal Reflection option to activate the **Fresnel Reflectivity** effect (see below). This will automatically reduce the transparency and increase reflectivity when the surface is viewed at an angle. This is a common and physically correct characteristic of glass and should therefore not be disabled for glass.

Note that the reflective properties (e.g. Roughness) can be fine-tuned in the Reflectance channel's *Transparency* setting.

See [*Transparency*](#).

Exit Reflections

As soon as a Reflection meets a surface after being refracted through glass one of two slightly different reflections can be calculated - ones that result when they enter glass and those that exit the glass. A single

reflection is often visually the most appealing one (even if it is in fact false). To achieve this effect, disable the **Exit Reflections** option.



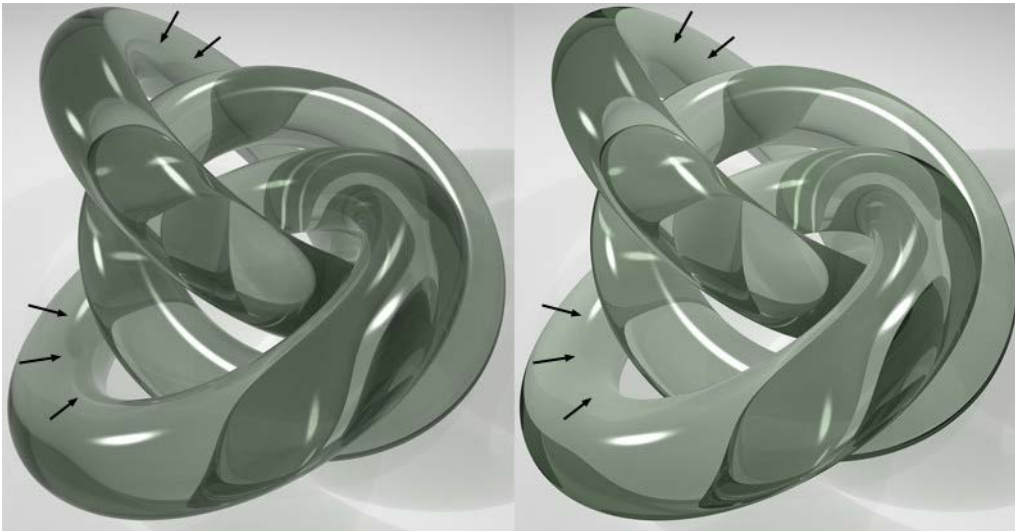
At top, Exit Reflections enabled; at bottom, disabled.

Fresnel Reflectivity

The Fresnel Reflectivity controls the degree to which the transparency and reflection values are dependent on the viewing angle. If the Fresnel Reflectivity value is larger than 0%, the viewing angle — the angle between the camera and the surface — will be taken into account.

If you observe a real pane of glass with your eyes parallel to the pane (i.e. with a 90 degree viewing angle), you will notice that the pane barely reflects, almost all light passes through. However, look at the pane from a narrow viewing angle and you will see that it reflects much more of its surroundings. The transparency and reflection values are dependent on the viewing angle.

The Fresnel option simulates this phenomenon for you.



Fresnel Reflectivity set to a high value (left) and low value (right).

For example, provided Fresnel Reflectivity is set to 100%, if you set transparency with RGB values of 80%, 80%, 80%, the surface is 80% transparent and 0% reflective when the viewing angle is 90 degrees. With a very low viewing angle, the surface is approximately 0% transparent and 80% reflective. If you have entered a reflection value in addition to transparency, the reflection value is added to the angle-dependent reflection. If the Fresnel Reflectivity value is larger than 0% the transparency and reflection values are used as they are irrespective of the viewing angle.

Additive

Usually, if the surface has a color, the color is automatically reduced with increasing transparency. This can help to ensure a realistic effect. However, if you don't want this automatic behavior to take place, enable the Additive option.

Generally, more realistic rendering results if this option is *disabled*.

Absorption

When light enters a transparent medium, it weakens and changes color to one degree or another (glass is very rarely completely colorless). The thicker the surface, the weaker the light becomes as it passes through (glass is not completely transparent). These effects can be emulated using the following two parameters. In the following, Color refers to the Color parameter at the top of the Transparency channel.

Tip: Note that Absorption works best with closed volumes. Otherwise faulty calculations can result if there are holes in the volumes.

Absorption Color

The Absorption Color is the color that is multiplied by the Color when a "ray of light" has traveled the distance defined in the Absorption Distance parameter.

Thus, the Transparency channel has two colors:

- **Color:** The color that appears in a very thin object (e.g. a window pane or a thin film of water)
- **Absorption color:** The color a massive object assumes.

When creating glass you will almost always set the Color to white.

Absorption Distance

Use this setting to define the distance rays of light have to travel before Absorption Color replaces Color. The lower the value the more intense the Absorption Color will be, even for thin glass. A value of 0 will turn Absorption off completely.

Blurriness

You can blur the transparency using the Blurriness, Min Samples, Max Samples and Accuracy settings. 0% means no blur. Increase the value to increase the strength of blur.

Min Samples/Max Samples

Note: Has an effect only in Standard rendering mode.

These parameters will only be available if Blurriness is set to a value higher than 0%.

The program uses samples to create the blur effect and, rather than take samples at even distances, it concentrates the samples where they are most needed. Max Samples defines the number of samples used for the most important areas. Min Samples defines the number of samples used for the least important areas. Increasing Min Samples and/or Max Samples gives a higher quality blur, but a longer render time.

Accuracy

Note: Has an effect only in Standard rendering mode.

This parameter will only be available if Blurriness is set to a value higher than 0%.

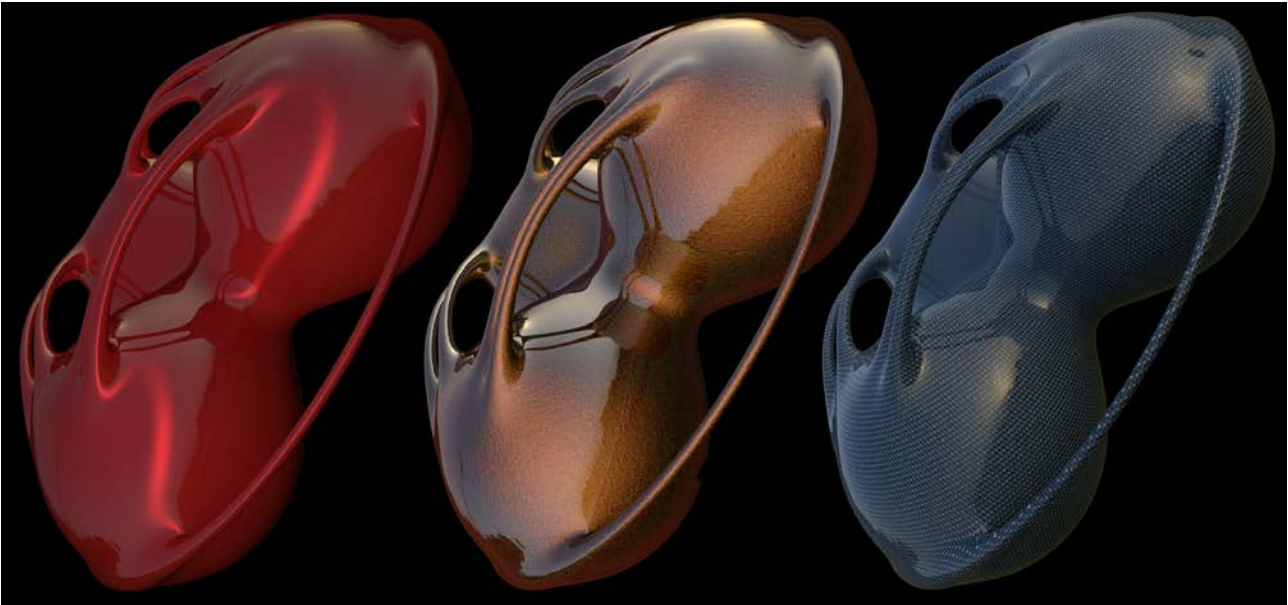
Increasing the Accuracy value gives a more accurate blur, but a longer render time also.

Affects the accuracy of the blur effect by controlling the number of samples per shading point. With a value of 100%, the maximum number of samples defined under Max Samples will be used in critical areas. Lower the value to reduce the number of samples used.

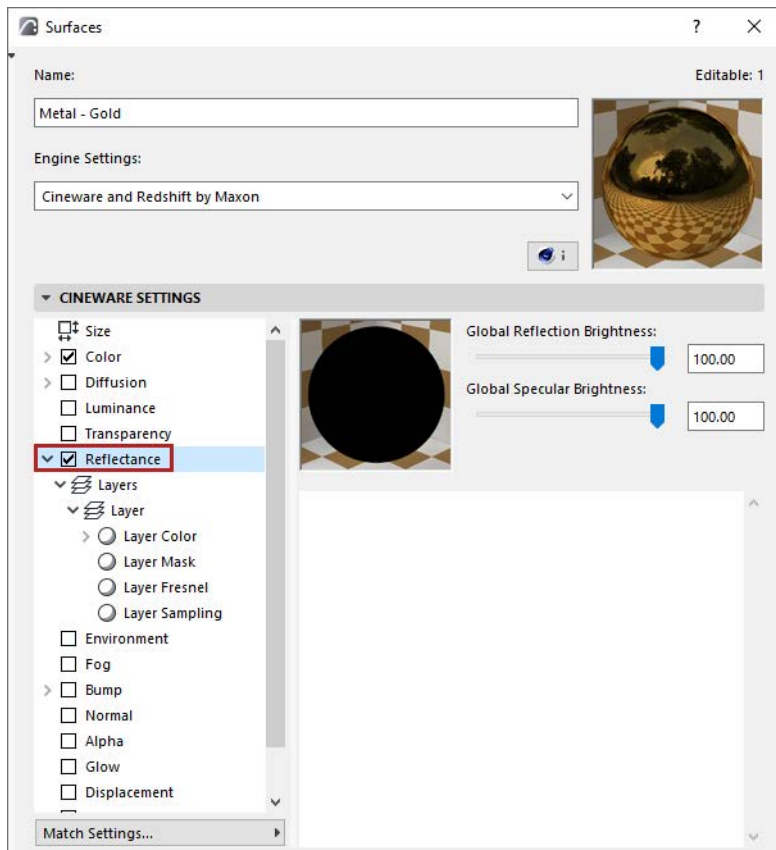
Reflectance (Cineware Surface Channel)

Use the Reflectance page to set a surface's ability to reflect.

The Reflectance channel combines the Reflection and Specular channels of earlier versions. Reflectance can be stacked and organized in layers to create effects such as metallic car lacquer.



Use up to 16 Reflectance Layers, plus the *Transparency* Layer. Each layer can have its own Alpha, Bump or Normals channel.



Topics in this section:

[Global Reflection Brightness](#)

[Global Specular Brightness](#)

[Layer Color](#)

[Layer Mask](#)

[Layer Anisotropy](#)

[Layer Cloth](#)

[Layer Fresnel](#)

[Layer Sampling](#)

[*Transparency*](#)

[Watch the Video](#)

Global Reflection Brightness

Compared to the Reflection Strength setting, which is defined *per layer*, the **Global Reflection Brightness** setting regulates the strength of the overall reflection. For example, if you create a complex combination of reflections, each with its own reflective strength and the overall reflection is too intense when rendered, this slider can be used to tone the reflection down without having to adjust individual reflection settings.

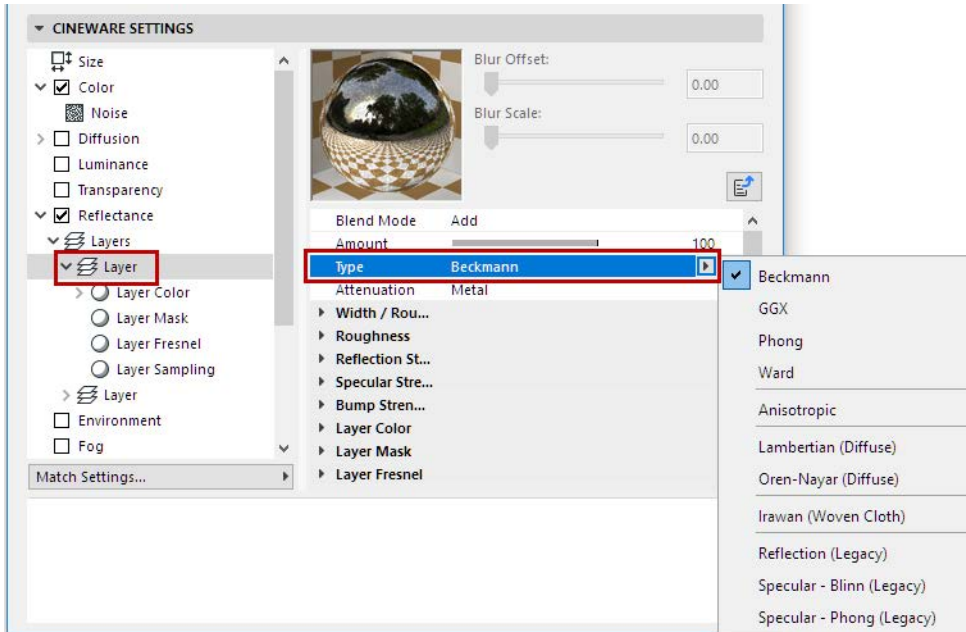
Global Specular Brightness

Use Global Specular Brightness to adjust or fine-tune a surface's overall specular brightness.

Layers

Lists the reflective layers of the Reflectance surface. Each layer's effect can be combined with another layer using one of the two available mix modes.

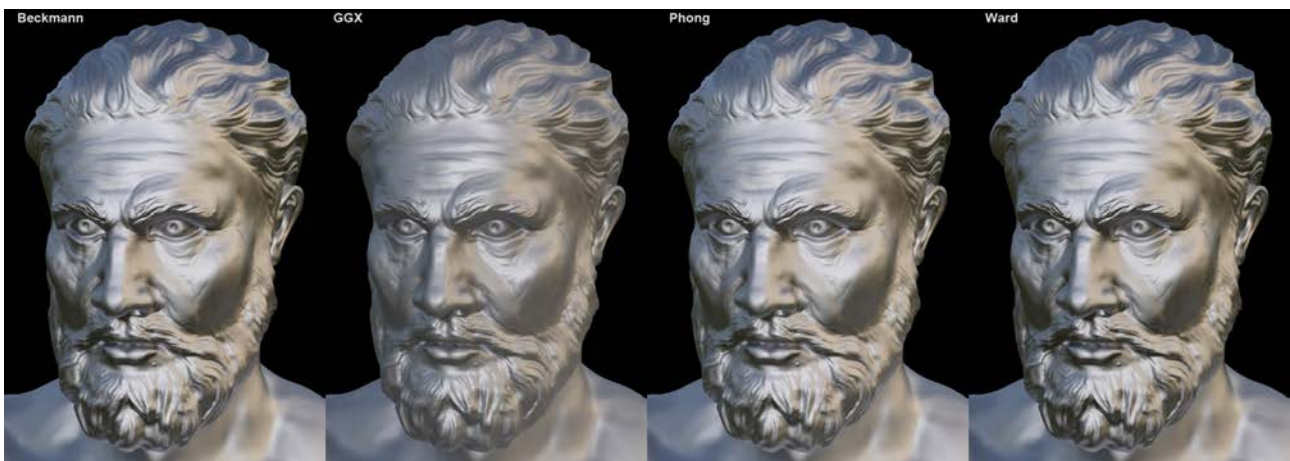
Type



Beckmann, GGX, Phong, Ward

These types only differ in the manner with which they uniformly weaken reflections from the ideal reflection angle (= angle of incidence).

The four most important types shown on a real-world reflective model (© Eric Smit) with 60% Roughness:



- Beckmann is a physically correct and fast type and should therefore be the preferred method for normal use.
- GGX produces the greatest dispersion and works best for simulating metal surfaces: a bright specular highlight with diminishing brightness.
- Ward is best suited for soft surfaces such as rubber or skin

Other Types

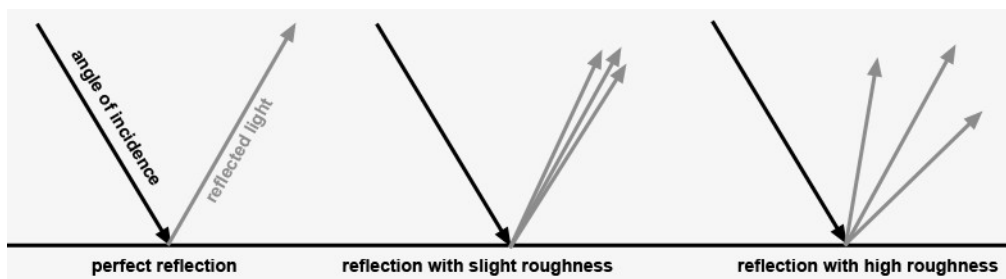
- **Anisotropic** bends the reflected bundle of rays in certain directions, which produces a distortion of the reflection, as with brushed or scratched metal, for example.

See [Layer Anisotropy](#).

- **Reflection (Legacy), Specular - Blinn, Specular - Phong (Legacy):** These three are only available for reasons of compatibility if an older file is loaded. It is recommended that a different mode be used that works physically correctly. If Specular Blinn/Phong (Legacy) is selected, specular highlights can be adjusted freely without having to take roughness into consideration.
- **Lambertian (Diffuse), Oren-Nayar (Diffuse):** These modes are diffuse models (i.e. “perfect” matte reflections). They produce a result similar to the surface channel Color (which doesn’t reflect). These channels should be used with caution (they can’t be cached by GI’s Irradiance Cache) and are basically only available for reasons of compatibility. The Color channel should be used instead, because it renders faster.
- **Irawan (Woven Cloth)** is a type of special anisotropy that contains several cloth patterns internally that can be used to create realistic-looking cloth surfaces.

See [Layer Cloth](#).

Dispersion of light is a product of matte reflections. These types have no effect on surfaces with perfect reflections (Roughness set to 0%).



The following illustrates the Type settings, at various degrees of Roughness.

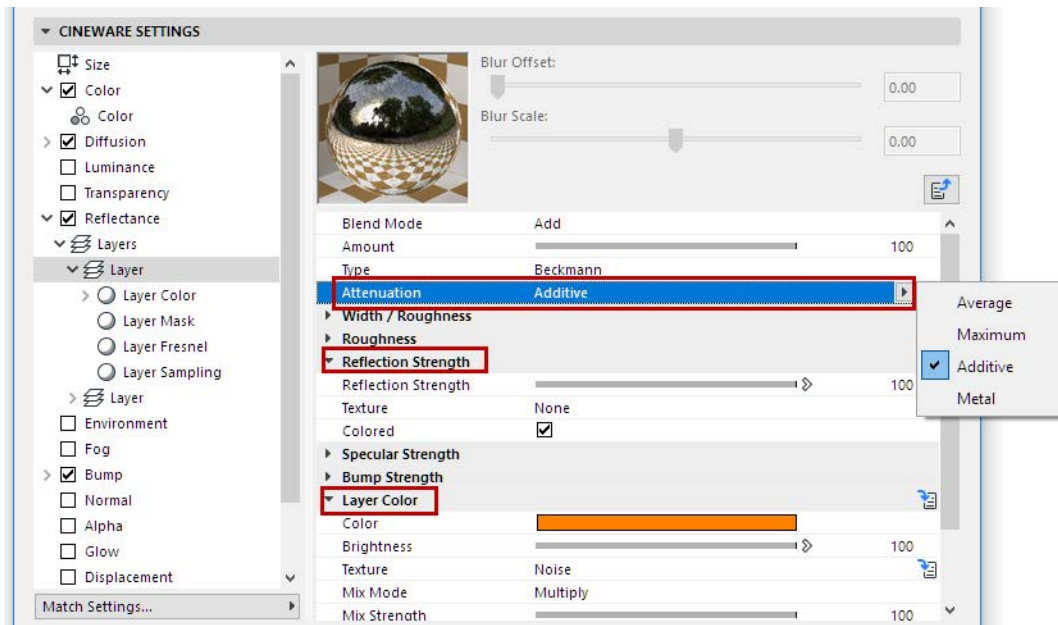


The large, reflective object reflects a HDRI sky, the depictions inset each show the specular highlights from two light sources. Both the Diffuse and the Irawan options produce a result very different from the rest because they do not allow Roughness to be defined separately (exception: Oren-Nayar).

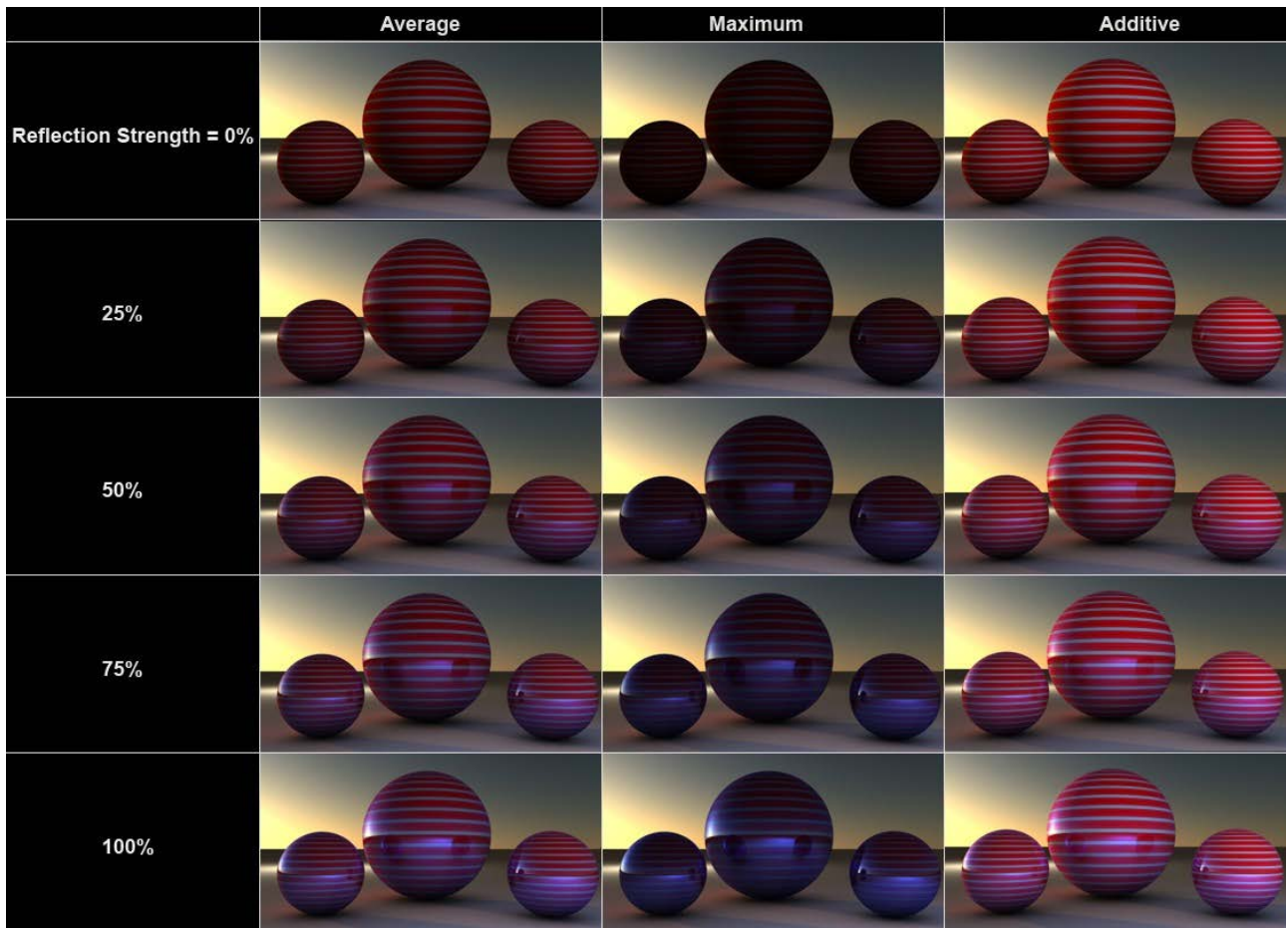
Attenuation

These options define how the **Color channel** should be mixed with the **Layer Color** with various **Reflection Strength** settings.

Note: If the Color channel is disabled, these settings will have no effect!

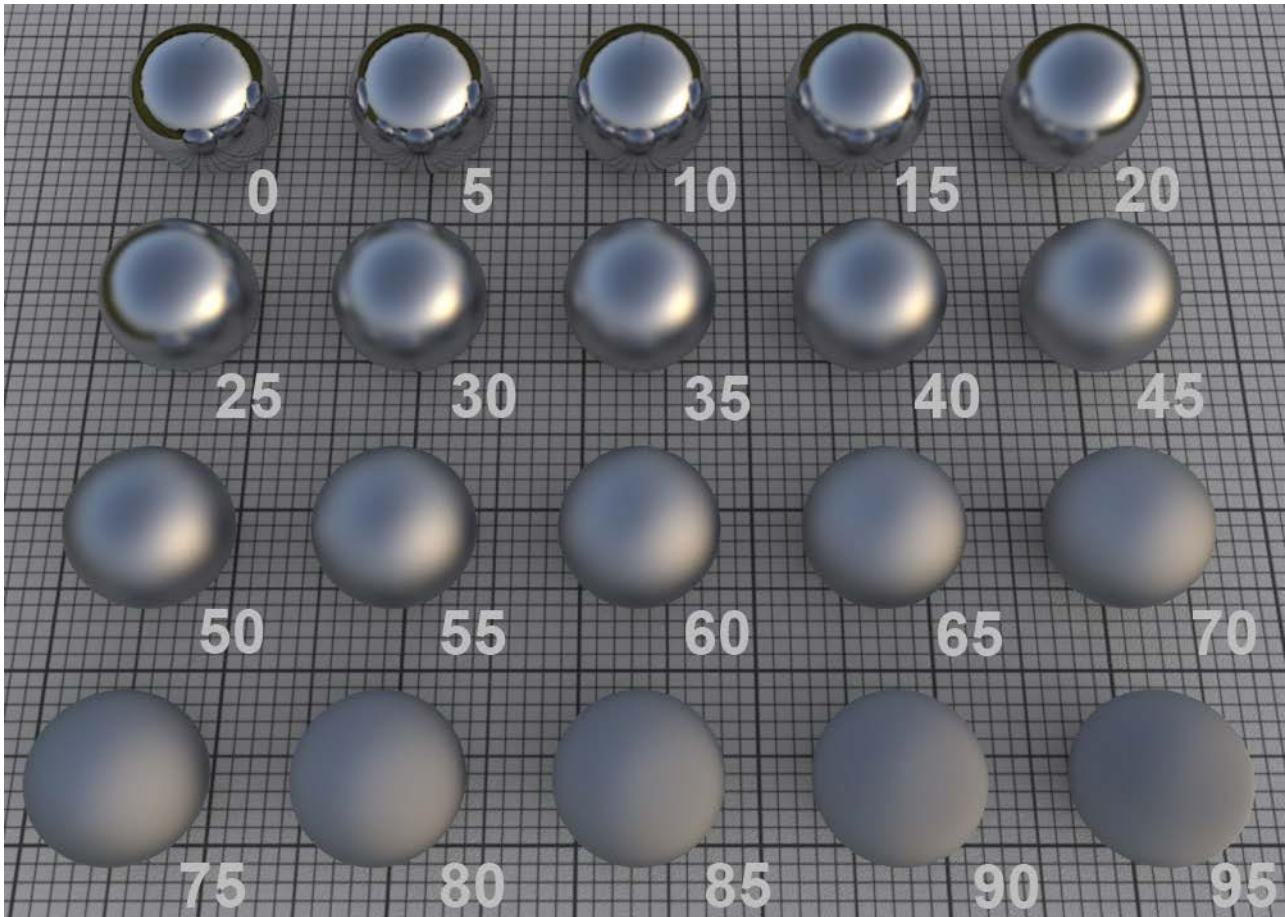


- **Average:** Both colors will be averaged. This mode produces the most realistic-looking results. (If no color is defined in the **Layer Color** setting, then the effect is the same as the next mode, **Maximum**).
- **Maximum:** This mode is best suited for creating colored reflections: The Color channel's effect will be reduced and the Layer Color will dominate.
- **Additive:** Both colors will be added. (In the real world, the Color channel's effect should weaken as the surface's reflectivity increases, which is something that the Additive mode ignores completely. To achieve a physically correct result, the Color channel must be darkened manually.)
- **Metal:** Only available for reasons of compatibility if an older Project is loaded (in older versions of Cineware, this mode was found in the Specular channel's **Mode** setting; it used the color defined in the Color channel to color the specular highlight).



The sphere surface has a red-and-white striped Color channel and a Reflectance channel with Layer Color set to blue

Roughness



Roughness values in increments of 5

In the real world, surfaces are made up of an innumerable number of microscopic reflective facets, each with a different orientation, which, when viewed from a distance, produce a specular highlight or reflection.

A perfectly polished (reflective) surface is made up of facets with the same orientation. The rougher a surface is, the more random the orientation of these micro facets is, which produces a correspondingly more spread out and duller specular highlight.

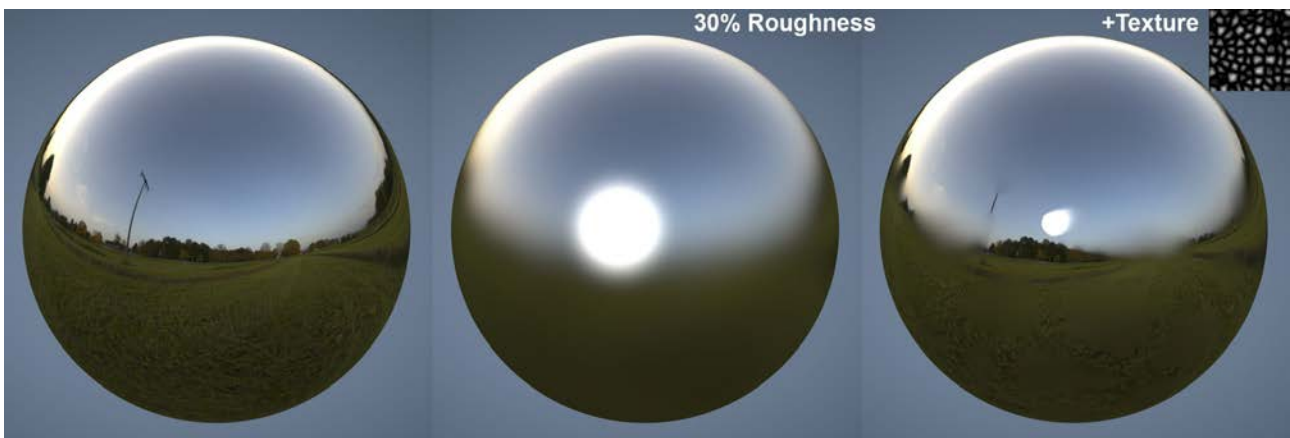
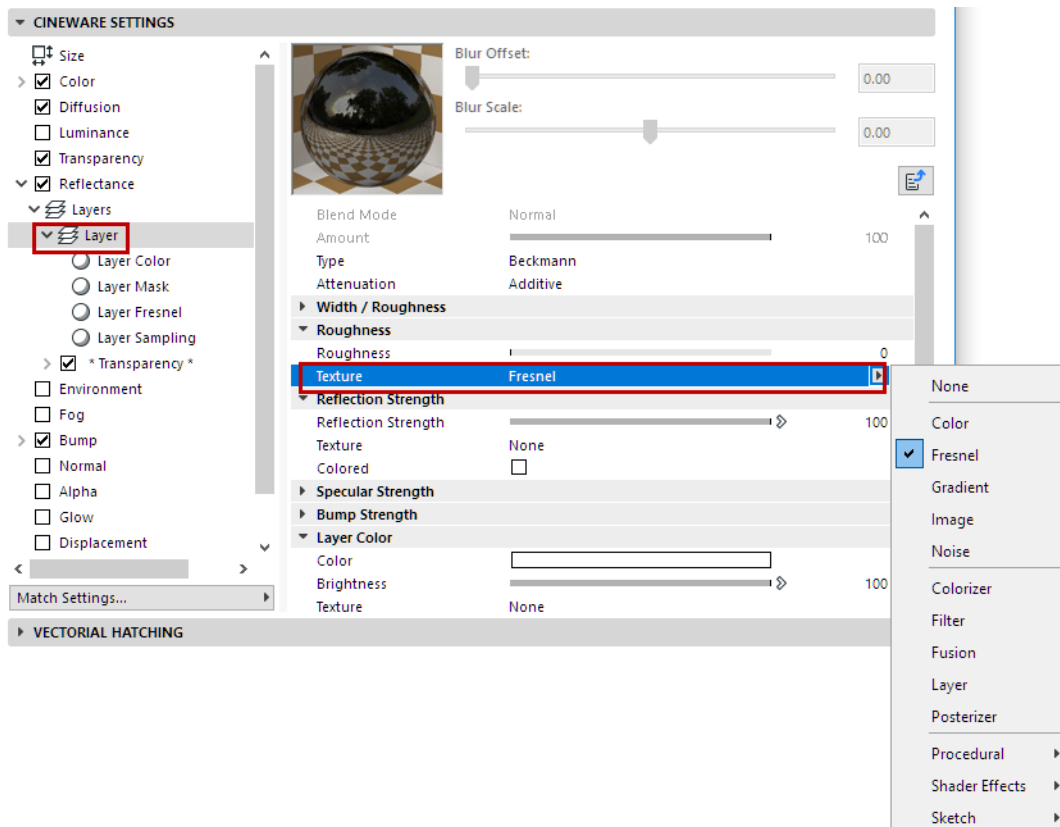
A roughness of 100% is referred to as a Lambertian surface. This type of surface reflects light in a circular dispersion in all directions, which constitutes a perfect diffuse surface. This is exactly what is defined in the surface channel Color: a diffuse surface's color without reflective properties. You can even ignore the Color channel completely and use the layer color to control the diffuse surface color (which, however, takes longer to render).

Note that the render time will increase as roughness increases, because reflections from a correspondingly wider range of reflections must be included.

Note: A **Roughness** value of greater than 0% is required for the calculation of specular highlights and anisotropy.

Texture

The **Texture** setting can be used to apply the Roughness setting (described above) to an entire texture.



Matte reflections can be controlled using textures

[See Textures \(Cineware Surfaces\).](#)

The grayscale values of the texture loaded here control the setting as follows: white represents the setting's full value and black = 0. In-between gray values have a corresponding effect.

Reflection Strength

This setting defines the strength with which the surface should reflect. The Attenuation setting above defines how this setting should work in conjunction with the surface channel Color.

Generally speaking, the intensity of the surface's color subsides as reflection strength increases (which occurs for all modes with the exception of Additive). This effect is also known as energy conservation.

If Reflection Strength is set to 0%, the surface will not reflect at all; a value of 100% will produce the maximum amount of reflection. Every surface in the real world reflects to some degree, even if it is very minor. Keep this in mind when creating highly realistic textures.

Note: The layers' combined reflective strength can be adjusted together using the Global Reflection Brightness slider.

Colored

If this option is enabled, the Color defined in the Color channel will be used as the reflection color (rather than the Color setting in the Layers Color sub-menu).

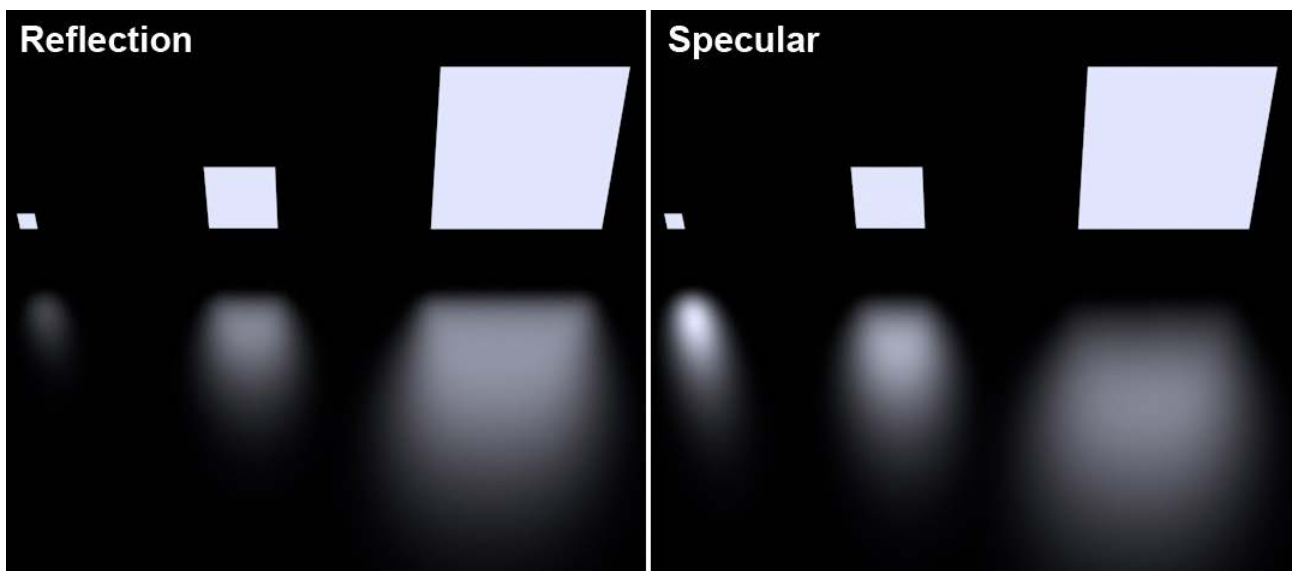
Specular Strength

This value defines the strength of the specular highlight. The use of specular highlights is a trick to add simple reflections to surfaces without noticeably increasing render time.

In the real world, specular highlights are nothing more than light sources reflected in objects' surfaces. If you want to create a photo-realistic scene, set this value to 0 and use the Reflection Strength setting (above) instead.

Note on Specular Highlights: The size and appearance of specular highlights depends on Type and Roughness (must be set to a value greater than 0% for most modes), and in some cases the anisotropic settings. This is different from earlier versions of Cineware, which let you make physically incorrect adjustments to the specular highlight's height and width. This can still be done, if necessary, if Type is set to Specular Blinn/Phong (Legacy). See [Specular highlight settings](#), below.

Each light source in a project generates specular highlights. Note that specular highlights are only generated by real Archicad light sources - as well as by Polygon (area) lights (however, these differ in size and other aspects from real, reflected Area lights):



Area lights reflected on the floor

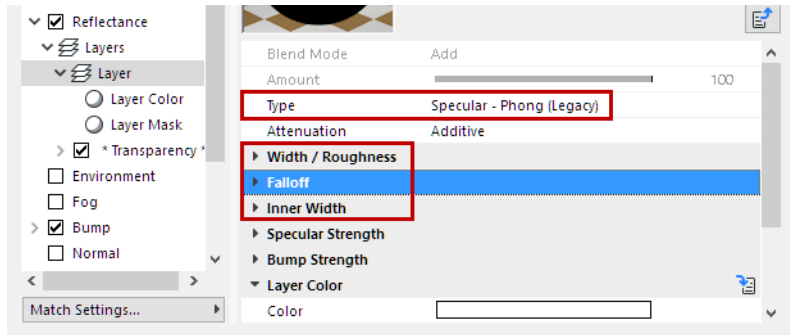
Note how the small Area light is rendered far too large, making it look unrealistic.

For example images of the effect of different Type and Roughness settings, see [Type](#).

Tip: Note that the combined specular highlight strength of multiple reflective layers can be adjusted together using the **Global Specular Brightness** slider in the **Reflectance** tab.

Specular highlight settings

The following 3 settings apply only to *Specular Blinn/Phong (Legacy)*:



- **Width/Roughness:** Defines the width of the specular highlight. Larger widths should be used for matte surfaces (for less specular strength). Polished and shiny surfaces should have very small widths (for greater specular strength).
- **Falloff:** Defines the curve of the specular highlight - anything from pinpoint to bell-shaped or rectangular shapes can be created.
- **Inner Width:** Defines the specular highlight's inner region, which is not affected by abatement of brightness.

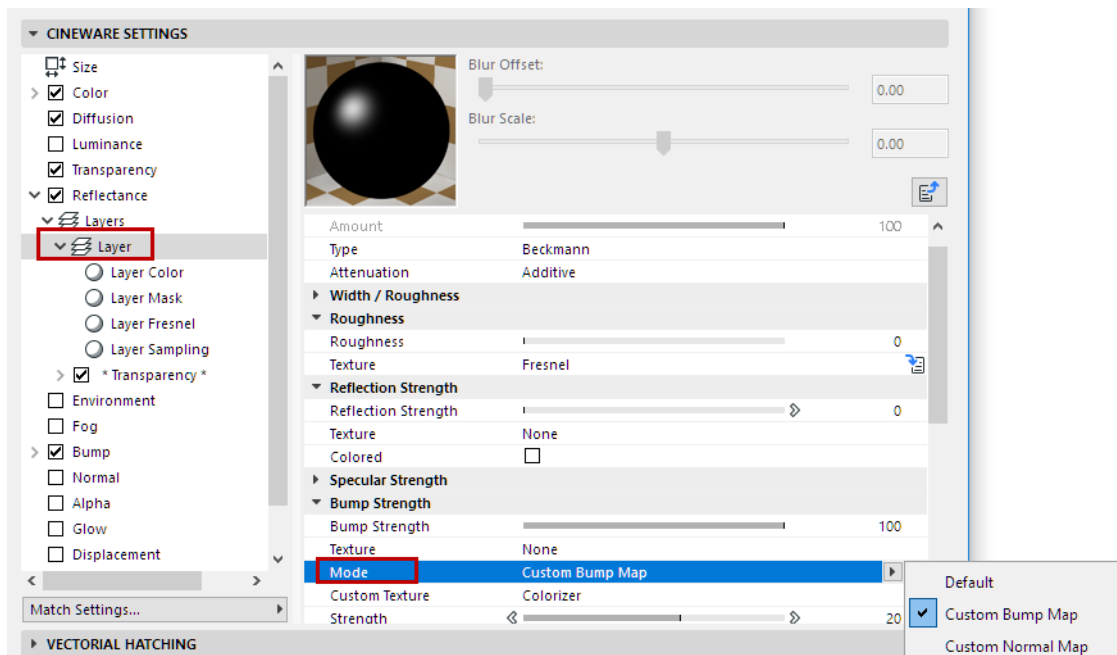
Bump Strength

Each reflection layer can have its own, autonomous Bump or Normal channel (see Mode, below). They function exactly like the surface channels of the same name.

Use this slider to adjust the strength of the bump or normal effect. If set to 0%, the effect will be disabled; 100% is the maximum effect.

Mode

This setting defines the type of Bump or Normal channel that should be used.



- Default: No specific reflection layer channels.
- **Custom Bump Map:** Select this to load a custom bump map into the **Custom Texture** field. You can also load a normal map (using the next option) for better results.
- **Custom Normal Map:** elect this to load a custom normals map into the **Custom Texture** field. This can also be a bump map in a Normalizer shader.

A very common use for layer-specific Bump channels is for glitter in metallic car lacquer. The reflective particles are simulated using a very small Noise Texture.



Metallic glitter and scratches were created using separate Noise shaders in both reflection layers

Strength

Use this value to define the strength of the bump/normal mapping.

For details, see [Strength \(Normal Channel\)](#) and [Strength \(Bump Channel\)](#).

MIP Falloff

Used with Custom Bump Map. Enabling this option will reduce the bump mapping the farther away it is from the camera (see also MIP Falloff).

Method

Used with Custom Normal Map. Use this setting to define the coordinate system for the Normals map. Details can be found under Method.

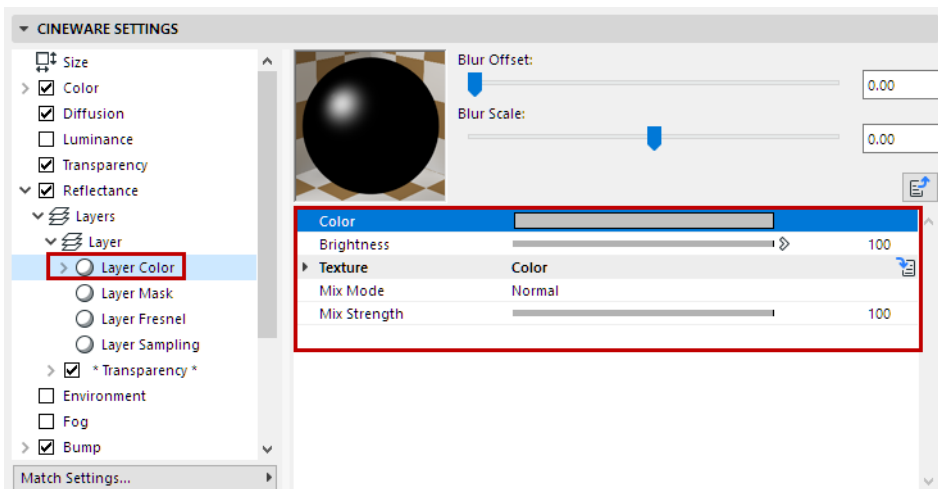
Use these settings to define which axis should have which color, with a Custom Normal Map:

- Flip X (Red)
- Flip Y (Green)
- Flip Z (Blue)
- Swap Y & Z (Y up)

See also the [Swap Y & Z \(Y up\)](#) description under the Normal Channel.

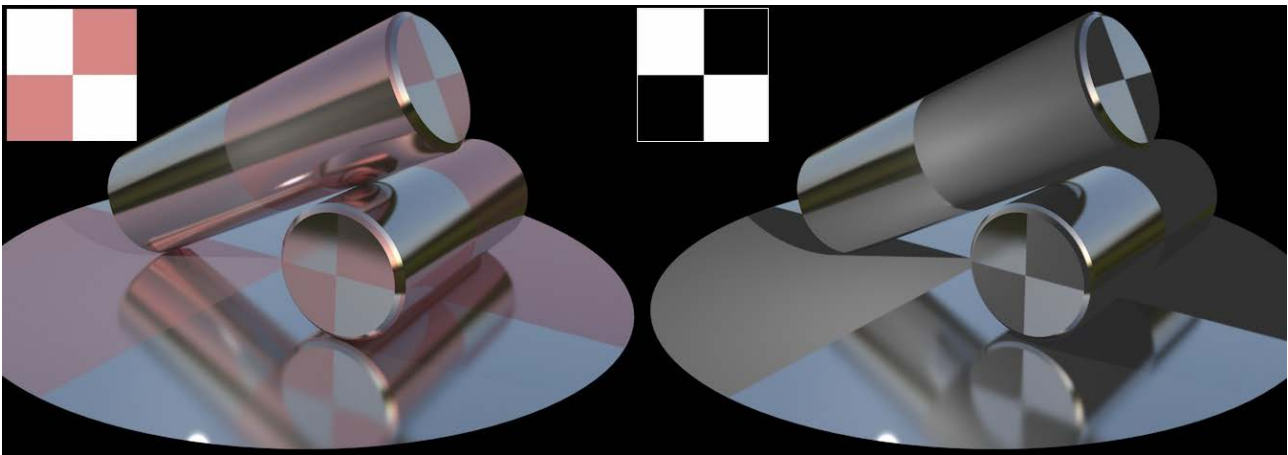
Layer Color

This menu's settings serve as a type of color filter for the reflections.



The layer color and the color from the surface's Color channel will be added. This method lets you use a black-and-white texture to define where reflections should take place (white = yes; black = no).

This is also the color you should use if you want to create colored metal surfaces.



The textures from the Layer Color menu. White produces total reflection, black shows the Color channel's color.

- **Color:** Defines the layer filter color. The default color is white, which allow reflections without having to modify the color.
- Brightness

[See *Defining Color and Brightness Values.*](#)

- Texture

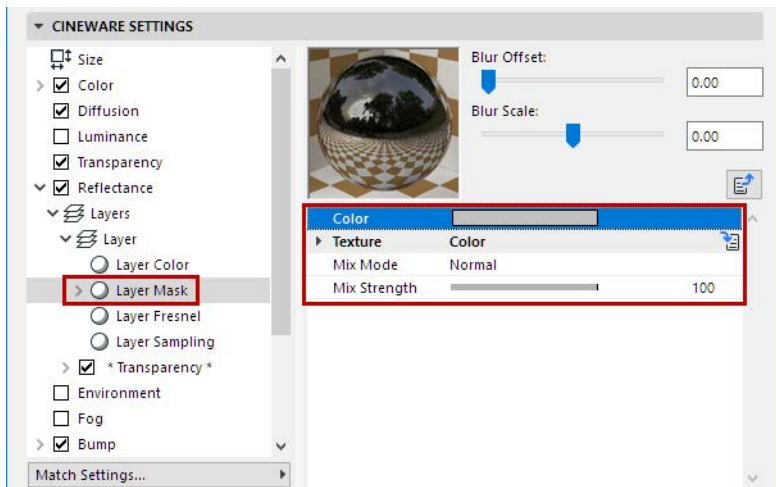
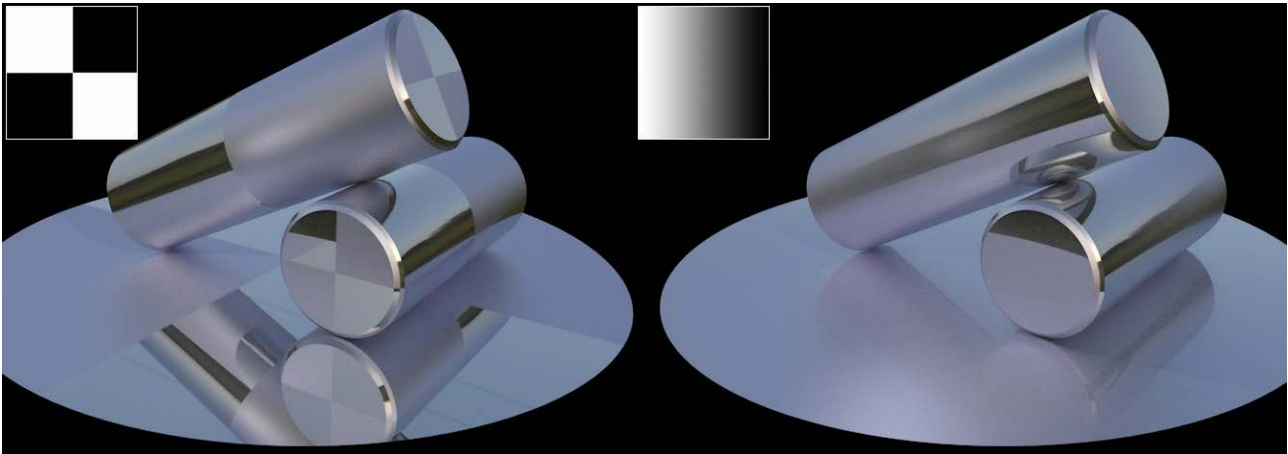
[See *Textures \(Cineware Surfaces\).*](#)

- Mix Mode
- Mix Strength

[See *Mix Mode and Mix Strength \(Cineware Surfaces\).*](#)

Layer Mask

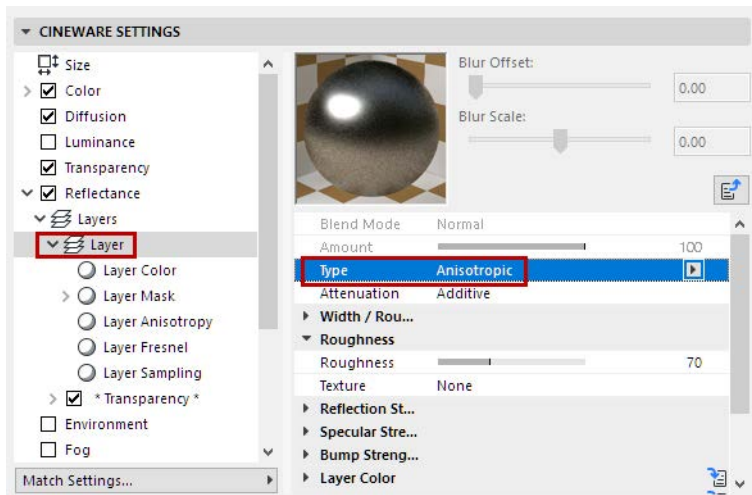
This menu's settings can be used to create an alpha channel for a layer that defines the visibility of the layer beneath it. White will not hide anything, black hides all, grayscale values hide accordingly.



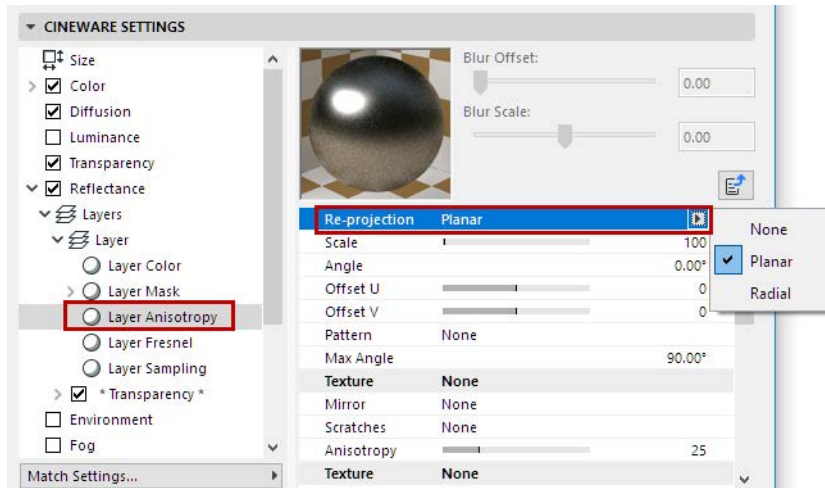
Color: This setting is used to define the color of the alpha. The default color is white, which allows reflections without having to modify the color.

Layer Anisotropy

To create an anisotropic effect, set Type to Anisotropic and Roughness to a value greater than 0%. Also make sure that there is enough present in the environment to reflect. HDRI textures on a Sky object are well-suited for this.

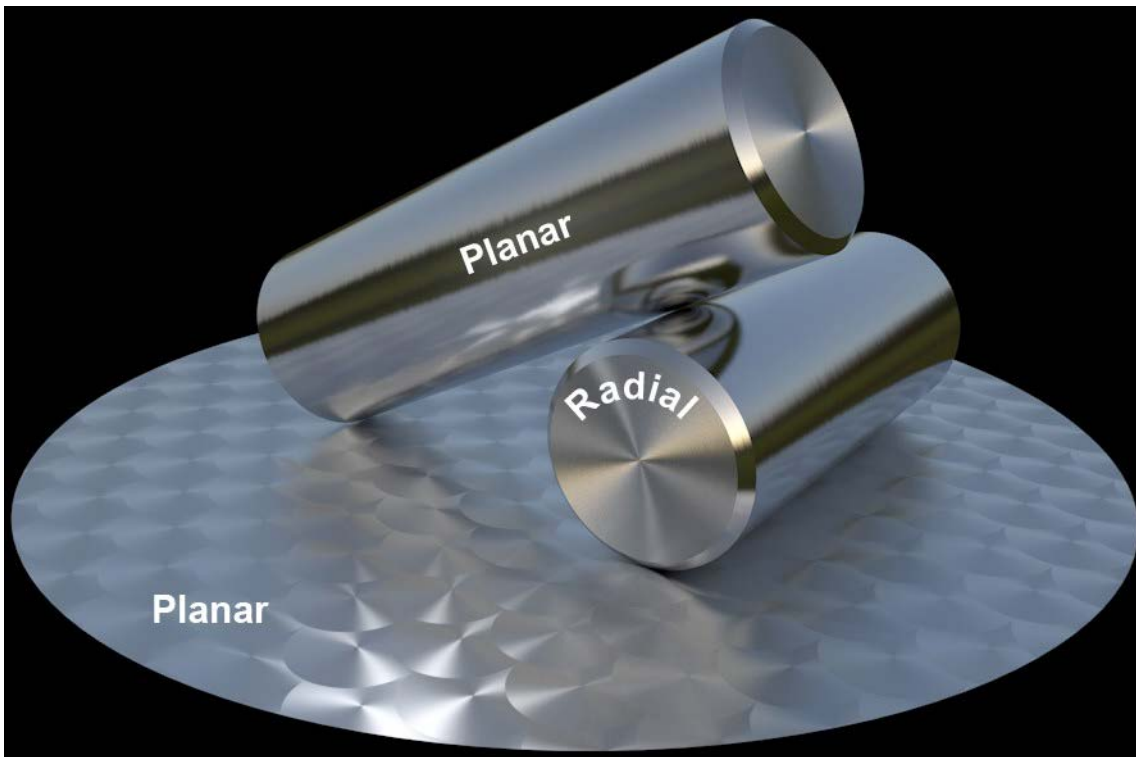


Re-projection



The scratches' vector tangents, which define the scratch direction internally, can be projected onto an object in one of three ways:

- None: Vectors will be projected onto the object without modification
- Planar: Vectors will be scaled, rotated and moved
- Radial: Vectors will be curved circularly, e.g., to simulate the look of the disc in the image below. An image can be loaded into the Pattern field, Offset values can be increased, and Count can be varied to create all kinds of interesting spiral/symmetrical effects but the effect in the example below can only be created without using the Pattern setting.



Different surfaces for the cylinder heads and sides (Planar and Radial).

Scale

Defines the size of the Pattern or Scratches selected.

Angle

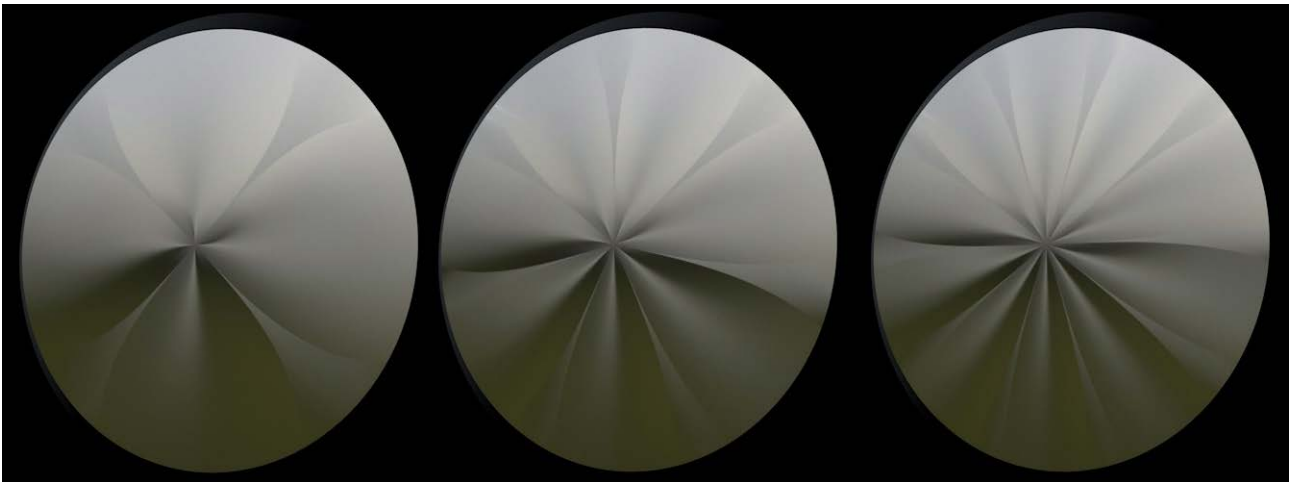
Used to rotate the anisotropic microscopic scratches (and, if defined, the parallel ridges). The rotational axis is defined by the Offset U/V values.

Offset U, Offset V

Used to move the Pattern/Scratches in the U and V directions, respectively. For the effect on the disc in the image above, both values were set to 50%, which means that the rotational axis lies at the center of the circular scratches.

Count

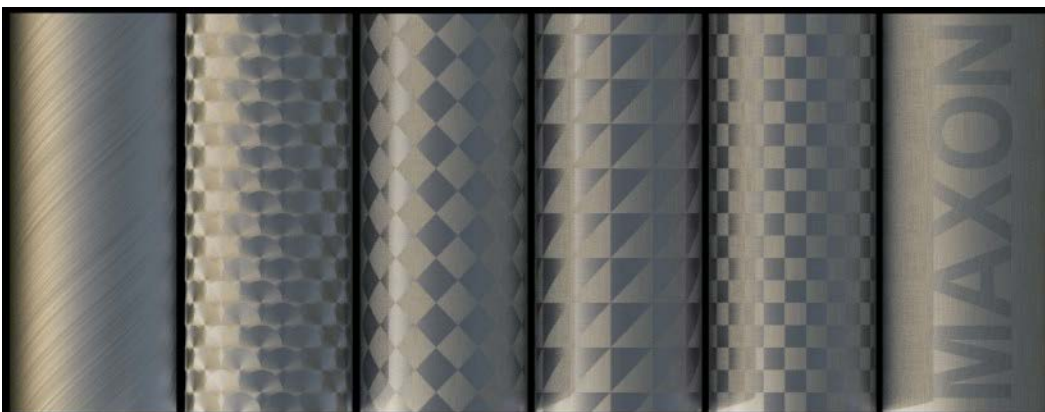
When applying this setting, a Pattern should be selected first. The Count value then defines the number of “threads” that converge at the center defined by the Offset U/V settings.



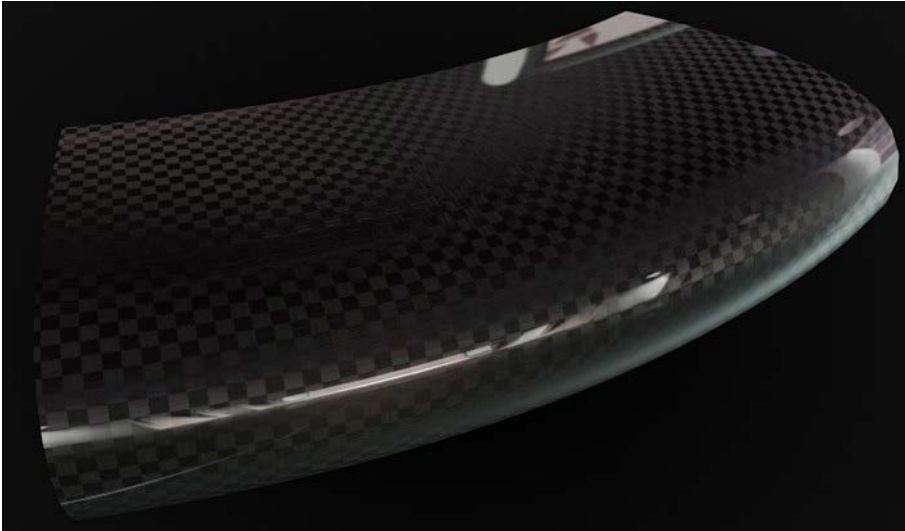
Pattern

This setting can be used to create common anisotropic surfaces such as carbon, brushed metal, etc. In addition, textures can be used to create numerous other patterns.

- None: Linear scratches whose orientation will remain constant will be created.
- Circular, Box, Diagonal, Lattice, Custom, from left to right:



Lattice and Box are well-suited for creating carbon fiber surfaces:

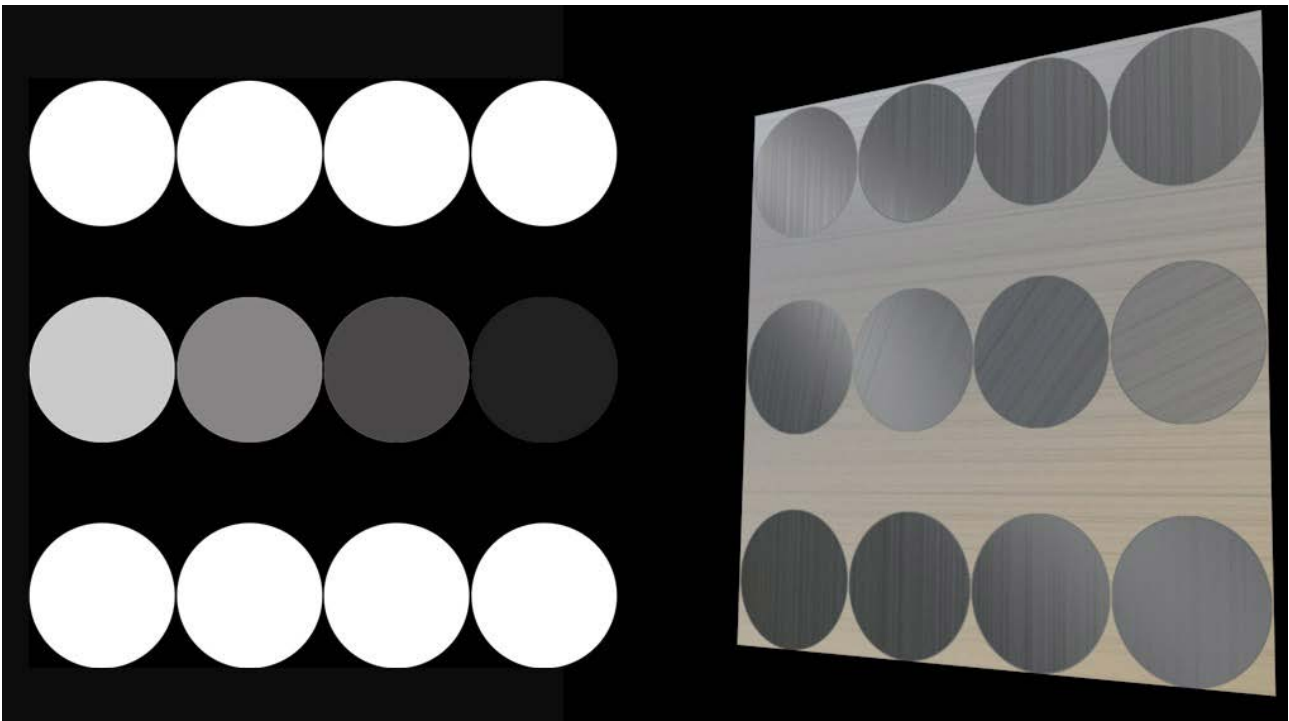


*The fitting pattern can be used to create a carbon fiber surface.
Here, an additional reflective layer was added to the anisotropy layer*

Texture

Instead of using a default pattern, you can also use a custom pattern. To do so, create a black texture with a white pattern on it.

Grayscale values will also be evaluated, whereby Max Angle will represent the scratches' angle at white regions. Note that you will not see the difference if a scratch is rotated 180°.



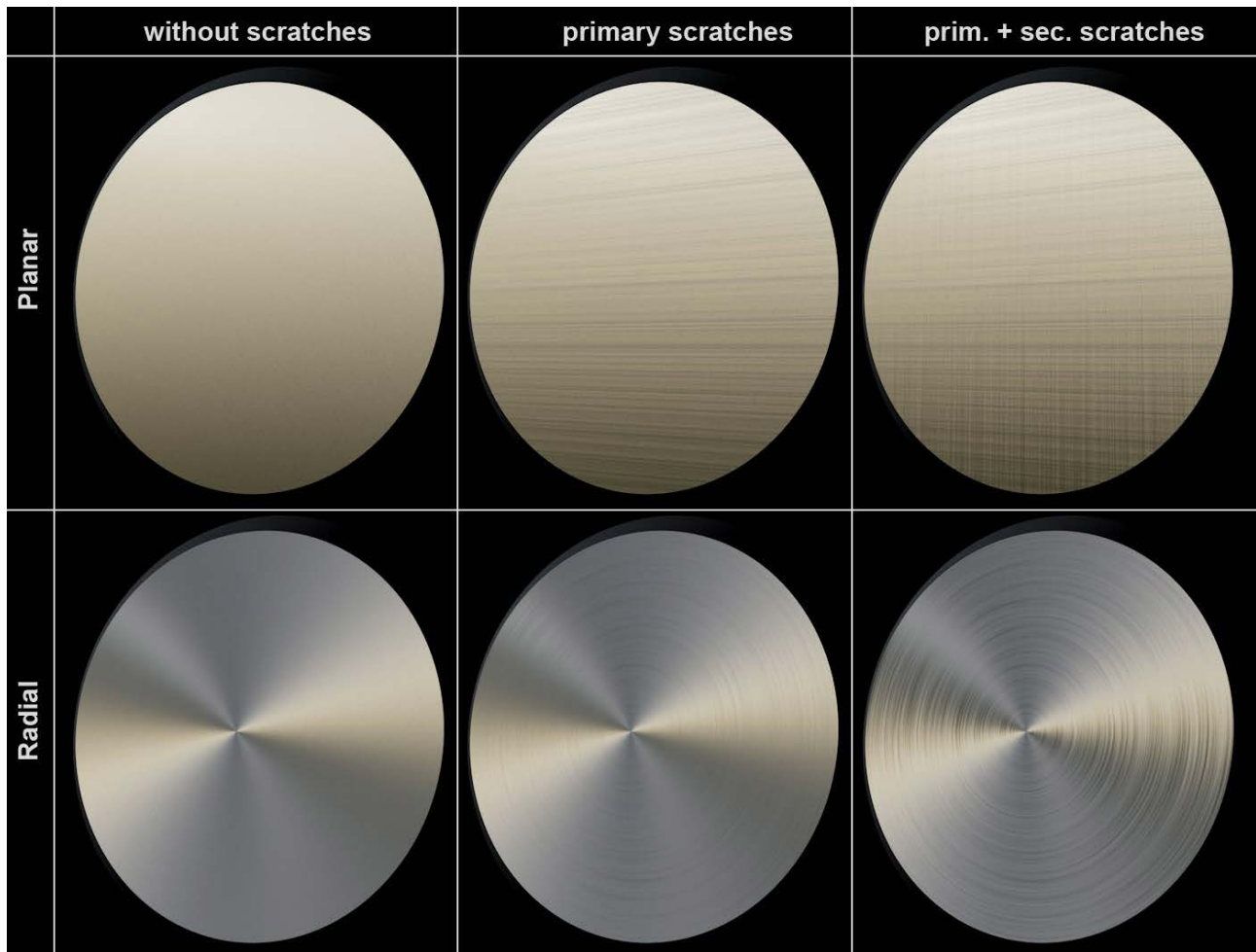
The texture on the left produces the scratch pattern on the right (Max Angle set to 90°).

Mirror

This option lets you mirror the Pattern in the U and or V directions (with the exception of Custom patterns).

Scratches

Besides anisotropic microscopic scratches that can uniformly deform specular highlights/reflections, additional, even larger scratches (ridges) can also be defined. There are primary ridges that run parallel to the microscopic scratches and secondary ridges (these make less sense in conjunction with Re-projection Radial) that run in a perpendicular direction. Both ridge types can be applied simultaneously, as shown at the top right of the image below.



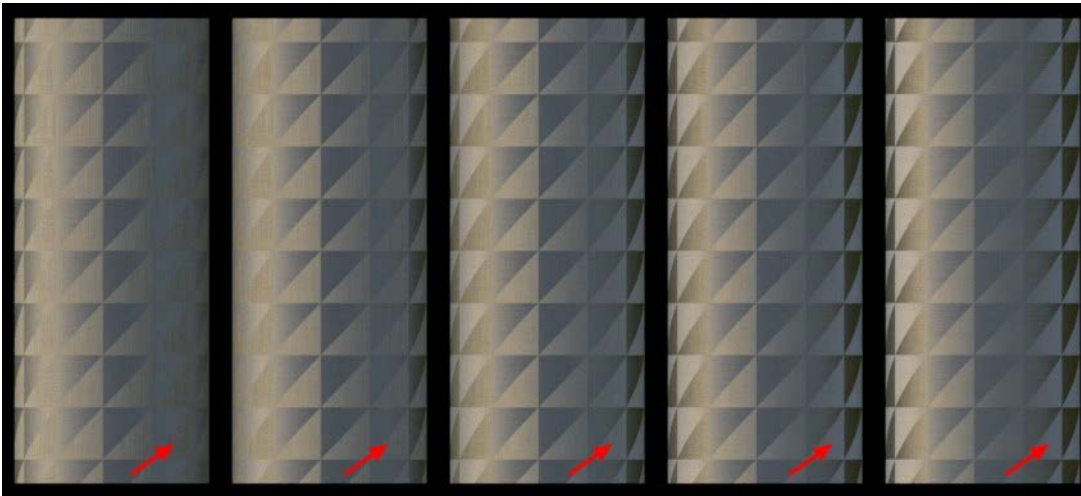
Primary and secondary scratches in Planar and Radial modes.

Anisotropy

Use this value to define the overall strength of the anisotropy effect. A value of 0% will disable the effect.

Orientation

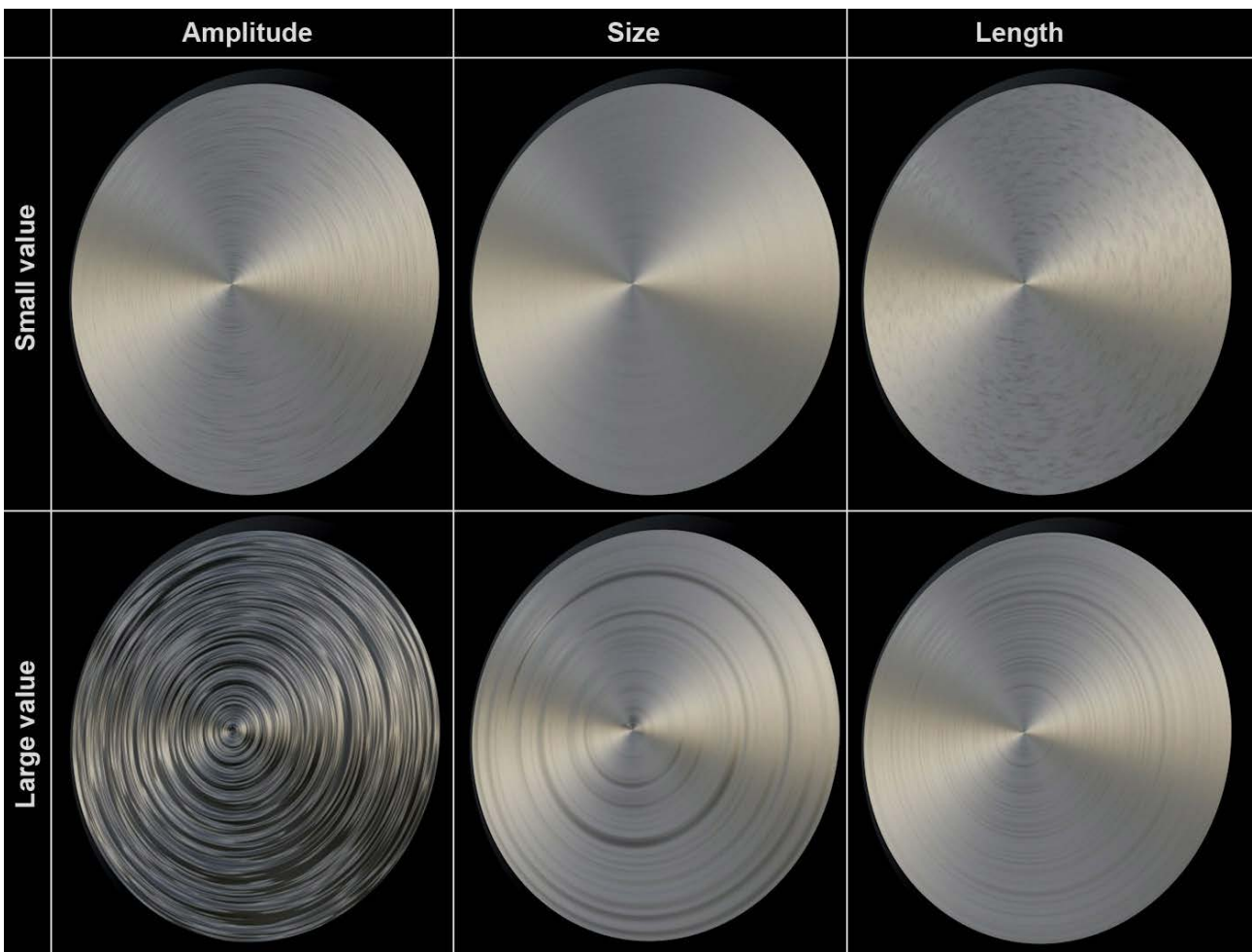
This setting regulates a property that does not actually exist in the real world: it rotates the vector tangent scratches mentioned above. Its effect is to change the reflective behavior. This makes it possible to fine-tune the anisotropy effect without having to change the position of the light source.



Orientation values. Note the changes at the marked locations.

Primary Amplitude, Secondary Amplitude

These values are used to define the ridge depth. Note that you can also enter values in excess of 100% to create very deep ridges.



Small and large values for amplitude, scale and length.

Primary Scale, Secondary Scale

Used to define the ridge width.

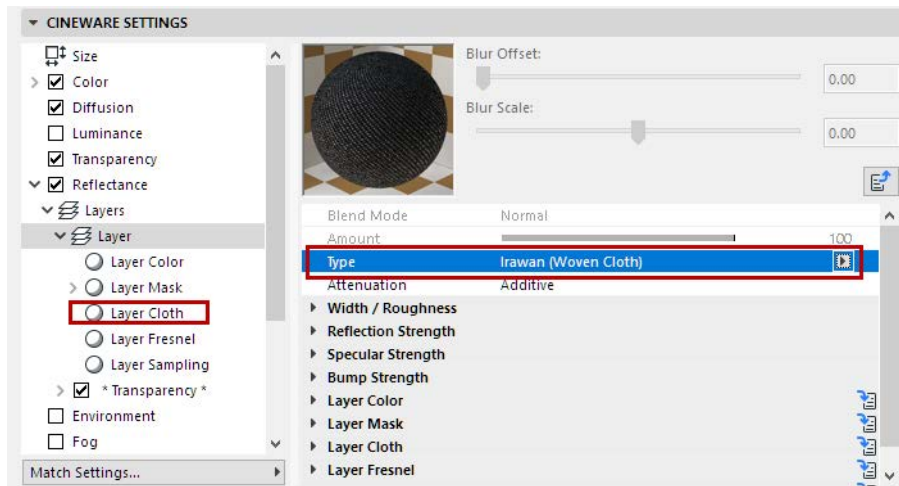
Primary Length, Secondary Length

Used to define the ridge length. Very high values lead to concentric (radial) or long ridges with a uniform depth (Planar), very small values lead to very short ridges.

Primary Attenuation, Secondary Attenuation

Layer Cloth

Cineware has its own cloth mode - Irawan (Woven Cloth) - which can be selected in the Type menu.



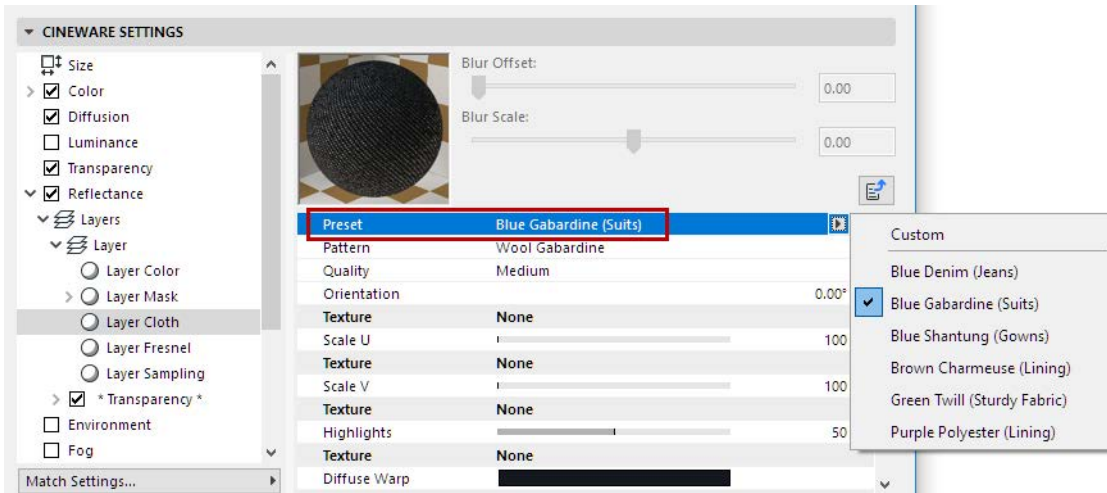
Cloth is most often made of yarn (and this in turn from fibers) that are woven to create a certain pattern. They are made up of structured surfaces that produce characteristic anisotropic specular highlights/reflections.



The Color surface channel can be disabled in conjunction with this mode.

Preset

A selection of presets with pre-defined values for each respective weave pattern.



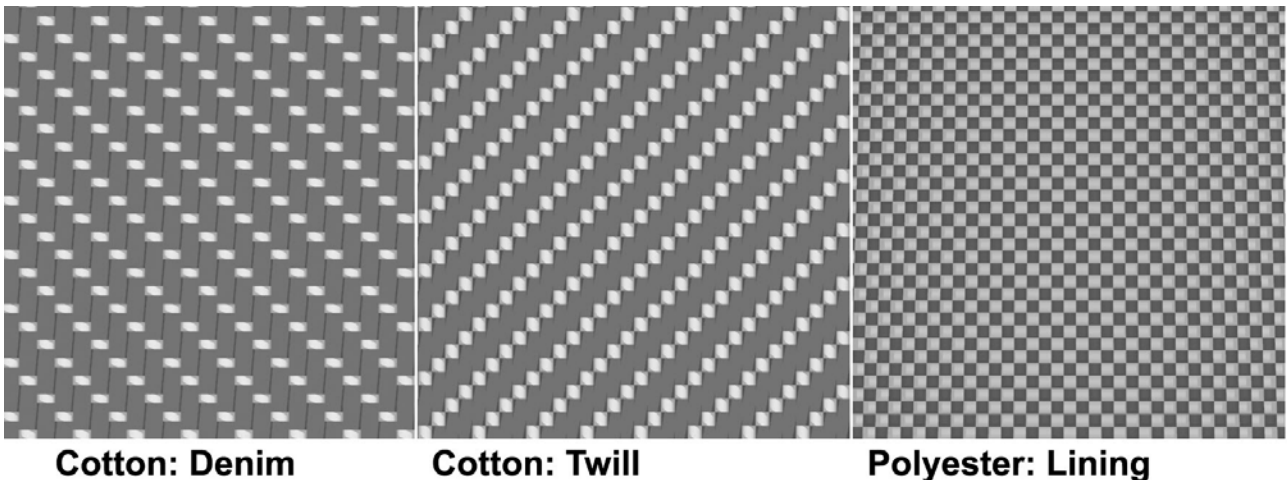
The following presets are available:

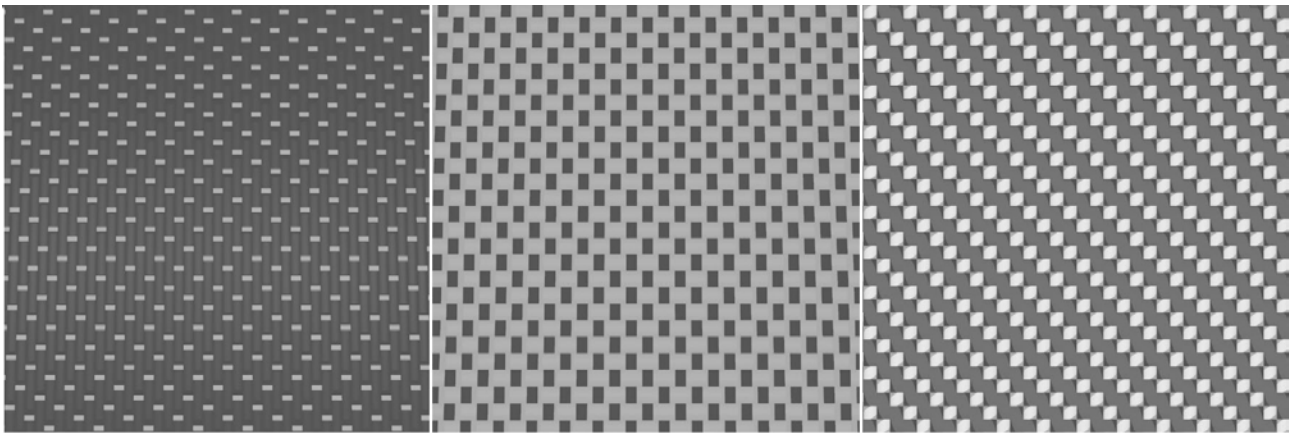


If a preset's settings are changed, it will automatically switch to Custom mode.

Cloth Pattern

Woven cloth is made up of various warp and weft threads that are woven perpendicular to each other. When viewed close up, the Cineware weave patterns look like this:



**Silk: Charmeuse****Silk: Shantung****Wool: Combed Yarn**

When viewed from a distance, these structures blend to form a characteristic cloth, which will appear in varied form depending on the angle from which it is viewed. In reality, warp and weft threads are made up of yarns having different reflective properties. For example, Polyester lining is made of very shiny elements that lie perpendicular to each other and have strong reflective properties.

Quality

Internally, the reflective behavior of the weave pattern is treated like a procedural texture with a strong noise property (analogous to the Noise shader, this would reflect a small scale value). When rendered, this texture will be evaluated using various small samples. The three quality levels available here are used to adjust the sample count.

Notes:

- The effect this has on render quality is minor, but the effect on render time is huge, unless you use the Low option, designed for creating test renderings.
- The better the antialiasing settings, the less the Quality setting will affect render quality.
- The higher the quality, the sharper the cloth structure will be and the more the Moiré pattern is reduced.

Orientation

Use this setting to rotate the cloth structure (rotational axis U/V=0/0). For example, if a pant leg's pattern lies perpendicular to the leg, this setting can be used to correct this state. Depending on the weave pattern, even small modifications (e.g., 10°) can have a major effect on the rendered result.

Scale U/Scale V

Use these settings to vary the cloth structure in U and V directions. Normally, the scale values should be the same in both directions to correctly reproduce the cloth's structure.

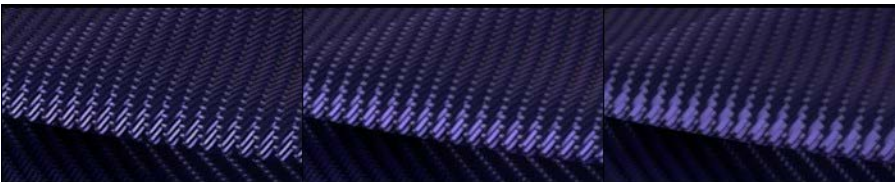
Note that very large values will produce correspondingly unrealistic-looking results.



Increasing Scale values from left to right (polyester lining)

Highlights

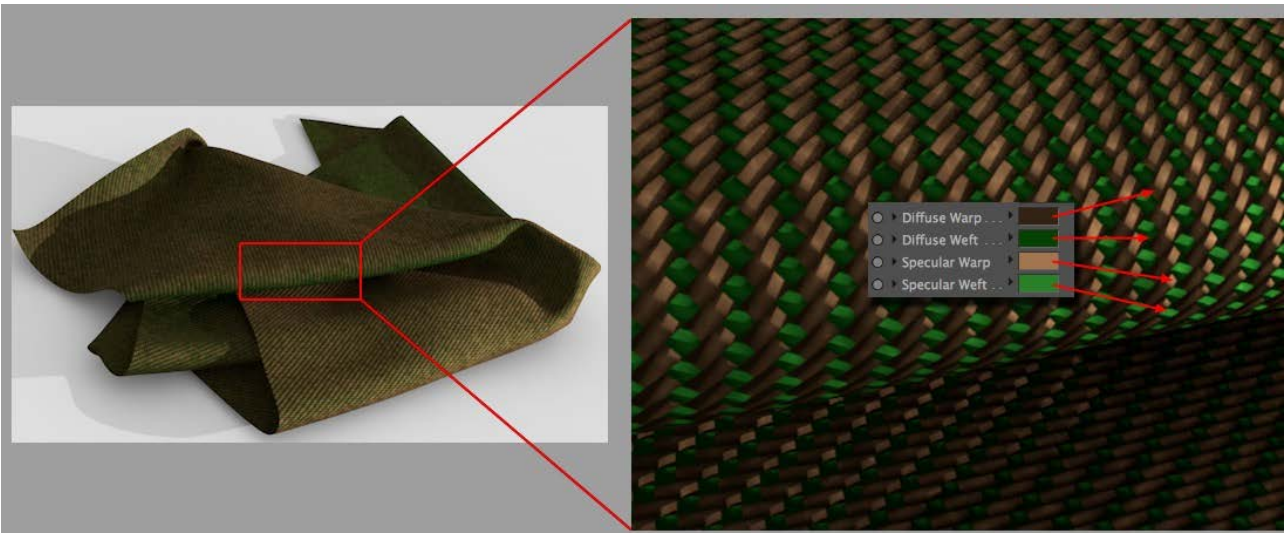
The Highlights setting defines the 'width' of the specular highlight/reflection. Smaller values produce sharp, bright highlights on individual threads; larger values will soften the highlights accordingly (they will become correspondingly darker and more expansive).



Increasing Highlights values from left to right

Diffuse Warp/Diffuse Weft; Specular Warp/Specular Weft

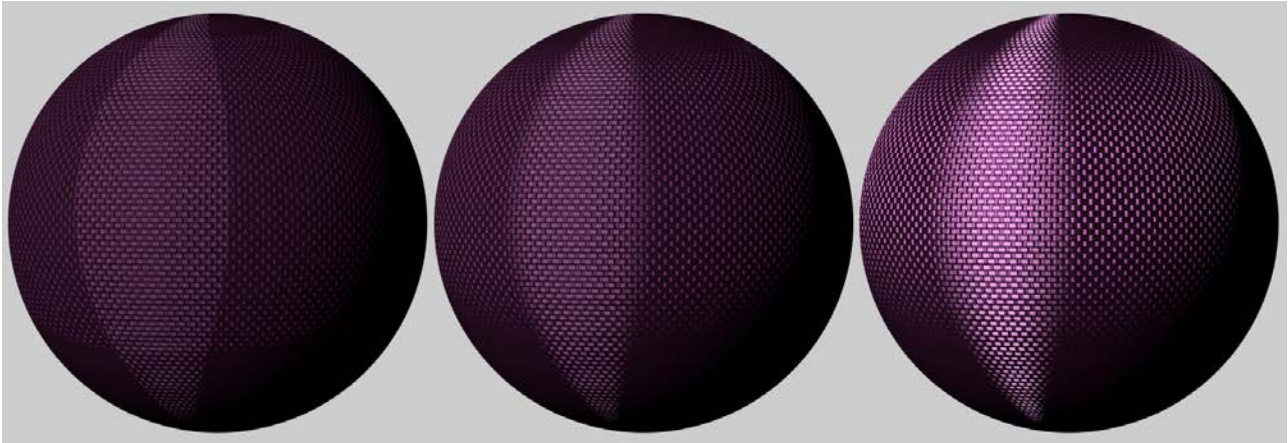
These color settings can be used to define separate diffuse and specular colors for the warp and weft threads (see Cloth Pattern).



Make sure that the specular color is always brighter than the diffuse color and that they have a similar tone. A shader or bitmap can be loaded into the Texture field to define the color. Note that the texture will be multiplied with the respective color. If you only want to use the texture to define the color, then set the color for that setting to white.

Smoothing

This setting is for use with the Polyester and Silk Pattern options only. Generally speaking, the anisotropic specular highlight becomes more blurred - but brighter - with increasing values.



Increasing Smoothing values from left to right

Scattering - Uniform

This setting defines the overall specular/reflection strength. The larger the value, the brighter/more reflective the cloth will be.

Scattering - Forward

This setting can also be used to adjust specular/highlight strength - but larger values will produce correspondingly smaller effects. More importantly, this setting primarily affects the bright regions of the yarn's specular highlight, while the darker regions will barely be affected.

Noise Strength

Defines the strength of the noise applied to the yarn's color (the thread color will be brightened and darkened in other regions).

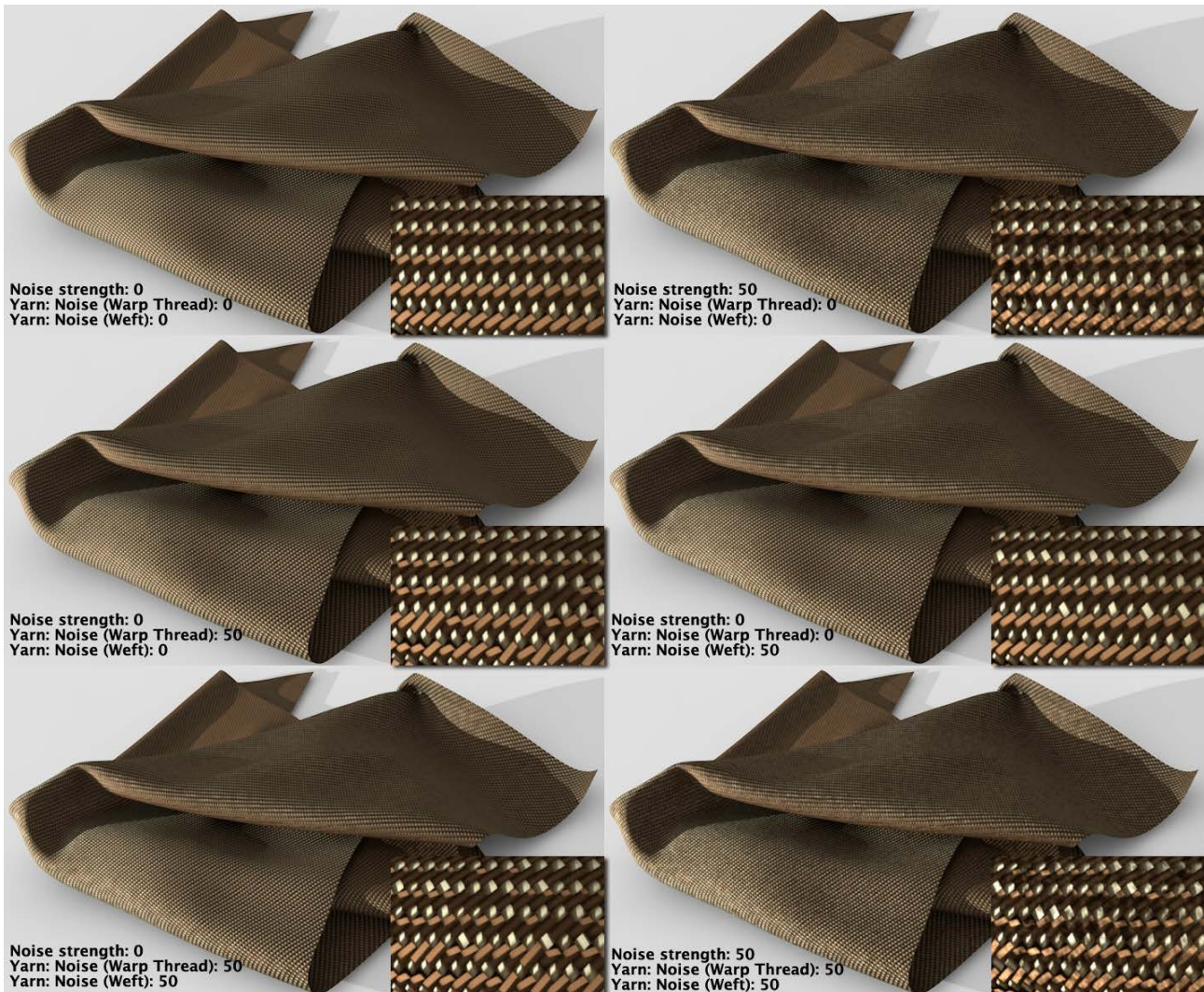
Noise Scale

Defines the scale of the noise applied to the yarn's color. Note that very large values can be used, which will produce a correspondingly spotty cloth.

Various Noise Effects

Since just about every cloth has irregularities in its threads and its weave, Cineware offers three Noise methods that can be used to simulate such irregularities in cloth:

- Yarn color using Noise Strength
- Shape/orientation of the warp thread's specular highlight using Yarn Noise (Warp)
- Shape/orientation of the weft thread's specular highlight using Yarn Noise (Weft)



Various Noise effects (insets: greatly enlarged view)

Yarn Noise (Warp)/Yarn Noise (Weft)

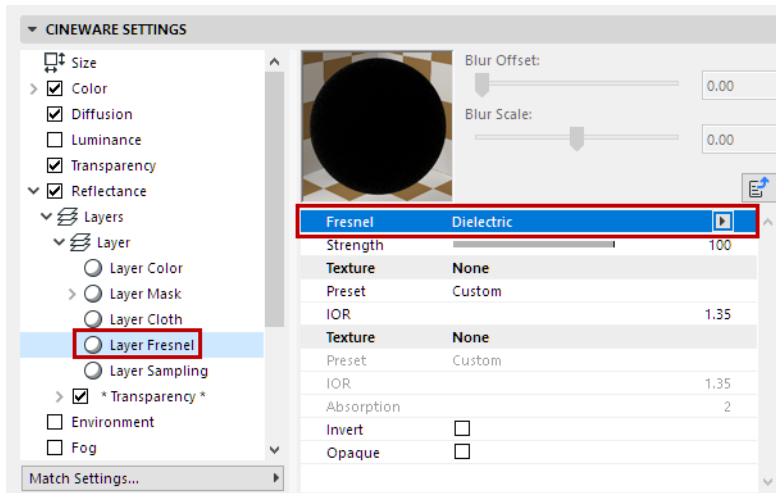
Define the strength with which the internal noise affects the shape and size of the specular highlight (see also image above). The larger the value, the greater the deviation from the parallel thread position and the more the look of the cloth will change - e.g., from a silky surface to a rough canvas-type surface.

Yarn Noise Scale

Defines the scale of the noise effect. The smaller the value, the faster the deviation from the optimal shape. Larger values will slow deviation correspondingly, exactly like scaling a Noise shader.

Layer Fresnel

These settings define the degree to which the reflection will increase as the surface normals increase their angle to the surface, which is referred to as Fresnel reflection.



Make sure that the reflectance strength is greater than 0. Otherwise, no Fresnel effect will appear.

Fresnel

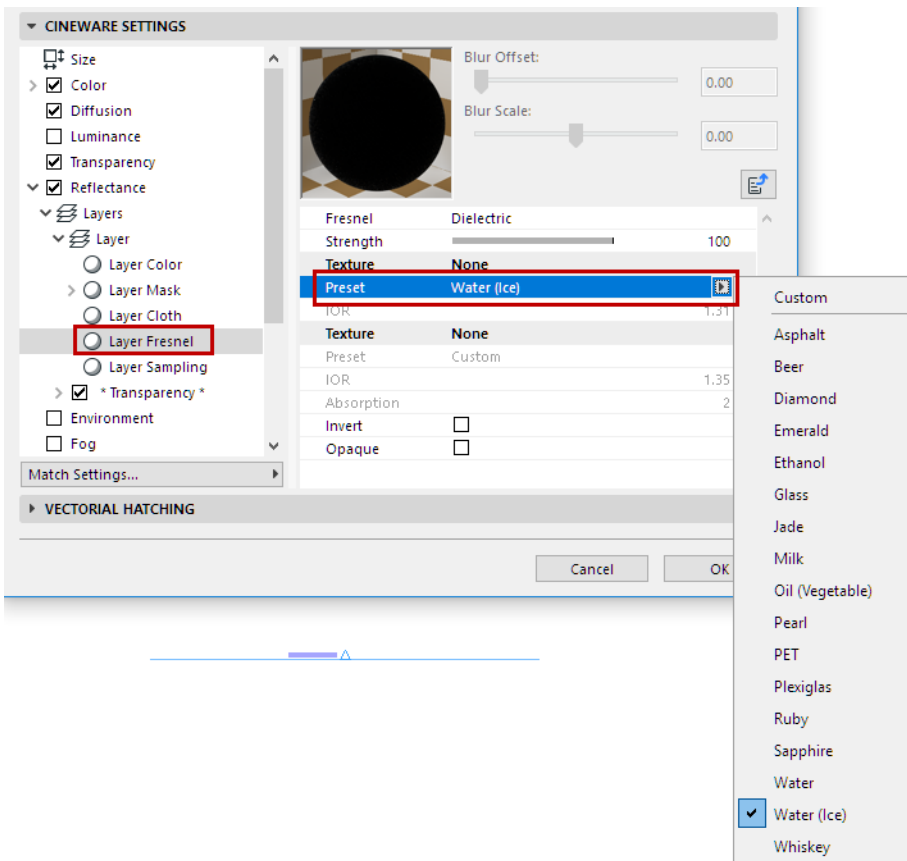
Select from:

- **Dielectric** for transparent surfaces such as glass, water, clear coat lacquer, etc.
- **Conductor** for opaque, reflective surfaces such as metal, minerals, etc.

Make sure that the Color channel is disabled for more realistic-looking results. Metal surfaces use reflections almost exclusively.

Preset

A series of presets (primarily for metals) is integrated that are based on real-world values. These cannot be modified individually using the Fresnel settings described below.



Different surfaces will be made available for selection depending on the Fresnel mode. selected. Select Custom if you want to modify the settings yourself.

Note that the Conductor presets will affect the internal color system (the Layer Color should be 100% white!).



A selection of **Conductor** presets

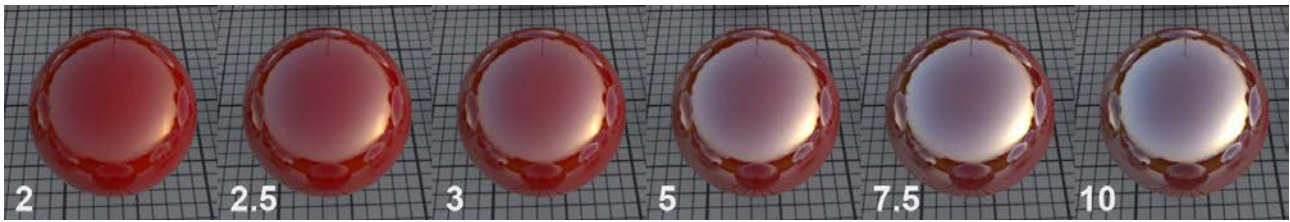
Strength

The Strength slider can be used to seamlessly move from reflection (0%) to full Fresnel reflection (100%).

IOR

The IOR is mentioned mainly in conjunction with refractive light effects, but it is also a measure for the angle of incidence for a surface’s light reflections (regardless if it’s transparent or opaque). As you can see in the images above, low values cause the surface to reflect almost exclusively in regions at which the camera’s angle of view is extremely low. Larger values cause the reflection to move correspondingly more to the frontal regions.





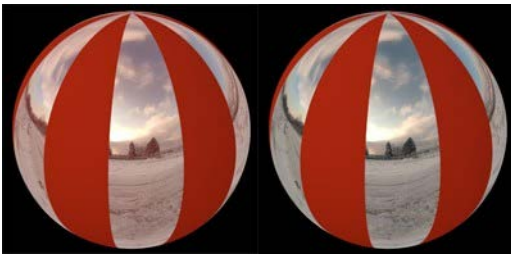
The Conductor mode behaves similarly but the reflections are generally much more pronounced. (However, the increase in reflection that corresponds to the increased IOR is not linear, because it is also affected by the Absorption value; it is recommended that you use the preset values to achieve a higher degree of realism)

Invert

Enable this option to invert the Fresnel reflection. Example (on a sphere): the maximum reflection that is normally at the sphere's center will be switched to its edges.

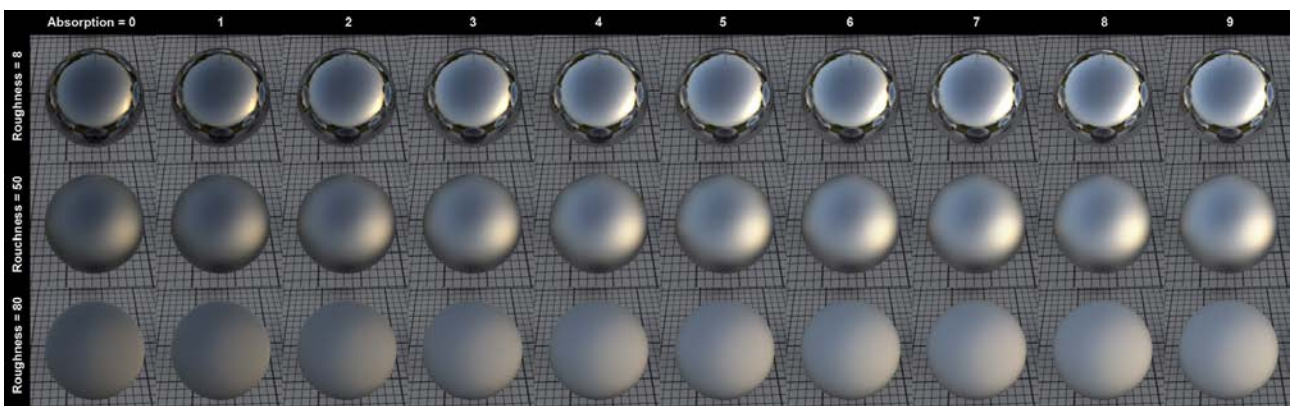
Opaque

In some cases you might want to prevent the Fresnel effect of neighboring surfaces from affecting a given object. In this image, a reflective surface with an alpha map lies on top of a red, diffuse layer. The Opaque option is disabled at the left: note how the red color from the underlying layer shines through while it is blocked on the right side of the image where the Opaque option is enabled.



Absorption

This setting is only available for the Conductor mode. Increasing values will produce correspondingly stronger overall reflections. This setting can be used to fine-tune the degree of reflection.



The Absorption effect in conjunction with Roughness (IOR = 2.97)

Layer Sampling

Sampling Subdivisions

This setting is designed for use with the Standard Renderer only (not the Physical Renderer). It can be used to define the quality of matte reflections.

Larger values will slow rendering and increase quality correspondingly.

Clamp Secondary

A reflected HDR image/sky can produce extremely bright points (fireflies) on a third object - i.e., a reflection of a reflection. To prevent the sampling settings from being increased dramatically, this setting can be used to limit the intensity of this reflection. The higher the value, the more the intensity of the reflection will be *restricted*.

Cutoff

This is a threshold value that defines the degree to which weak reflections will be calculated, if at all. Let's say you have a project with numerous reflective objects. An object at the rear of the image that is reflected in a small object near the front can be omitted because its role in the overall image is so small. The smaller this value is, the more reflections will be ignored.

The image below is an example of how this setting can be applied (the very bright HDR image helps make the effect more pronounced). A sphere with a Fresnel reflection reflects a HDRI sky. The Cutoff value on the left is too large; the corrected value is on the right.



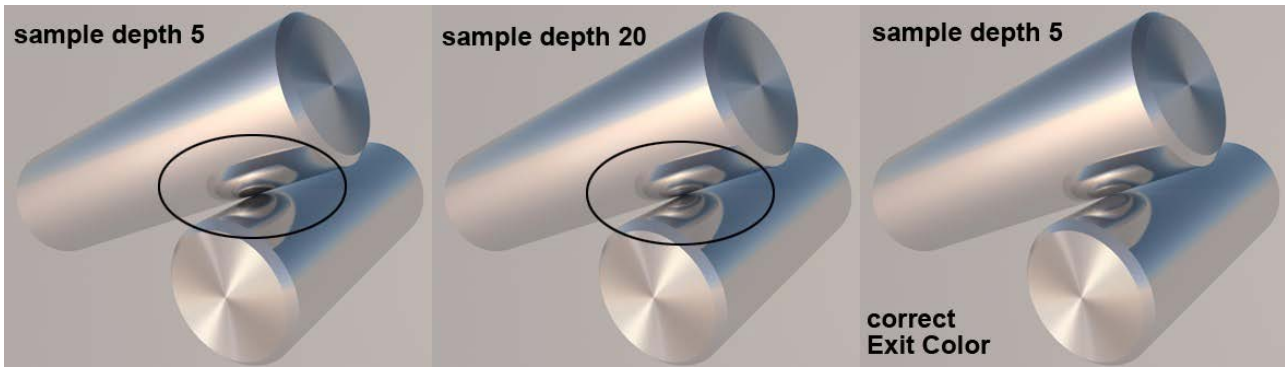
If the effect at the left takes place, then the Cutoff value needs to be reduced

Exit Color

The **Reflection Depth** PhotoRendering setting (Options/General Options) defines the number of reflections that should take place. For example, if you have two mirrors facing each other, this setting can be used to limit the number of reflections that take place. Otherwise an endless number of reflections would be rendered with a more-or-less endless amount of render time.

[See *Reflection Depth*.](#)

The **Exit Color** setting defines the color the reflection should assume in the end. In previous versions of Cineware, this was black by default. However, as you can see in the image below, this produced unwanted black spots. This can be avoided if a fitting color is defined for the last reflection, as seen below.



A fitting exit color is used to create a realistic-looking result without increasing the diffuse depth

Distance Dim/Distance/Falloff



Left: Distance Dim disabled, Right: Distance Dim enabled

Distance Dim can be used to omit objects from a surface's reflection depending on their distance from the surface.

- If **Distance Dim** is disabled, all objects will be reflected.
- If enabled, the **Distance** value defines the distance at which an object will no longer be reflected.
- The reflection falloff up to the Distance value is defined using the **Falloff** value. A value of 0 produces a linear falloff; other values produce the effects shown in the example above.

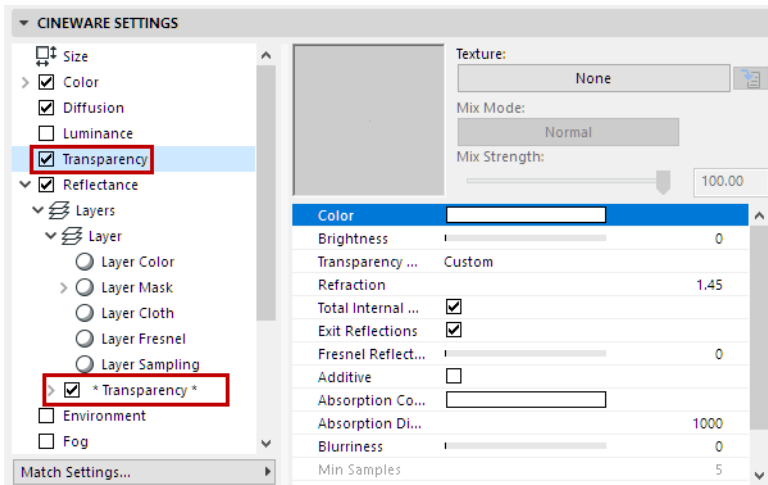
The reflection of the Sky object was disabled because it is considered to be an endless distance away.

Distant Color

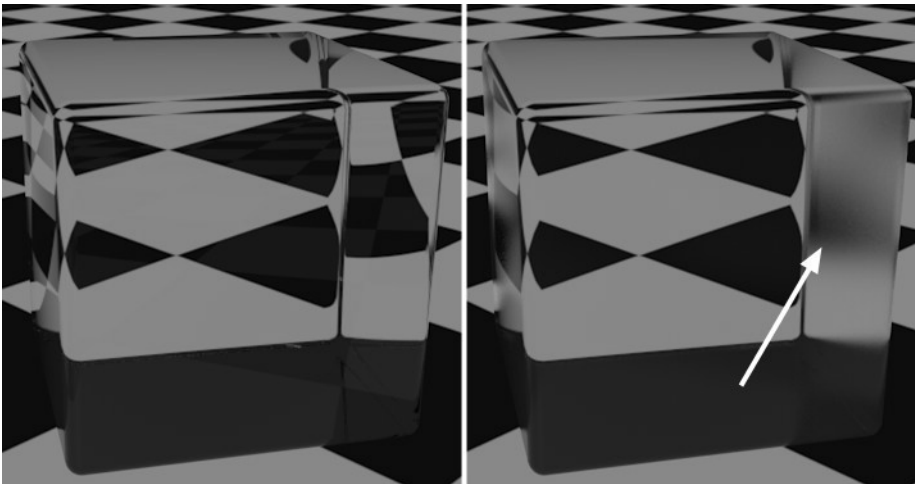
For this setting, imagine a sphere with a Distance radius around the reflective object. This sphere's color is reflected when the reflection samples are longer than the Distance value defined. Normally, black would be the correct color in this case because it would prevent anything from being reflected. Other colors can be used depending on the intended result.

Transparency

The * Transparency * layer will be added when the Transparency channel is enabled.



This page contains reflection settings that let you control the Total Inner Reflection's reflective properties.



*The Total Inner Reflection is also part of the Reflectance channel;
the Roughness was increased on the right*

Normally, these settings do not have to be modified. If you're not trying to achieve a very unusual effect, these settings can be left as they are.

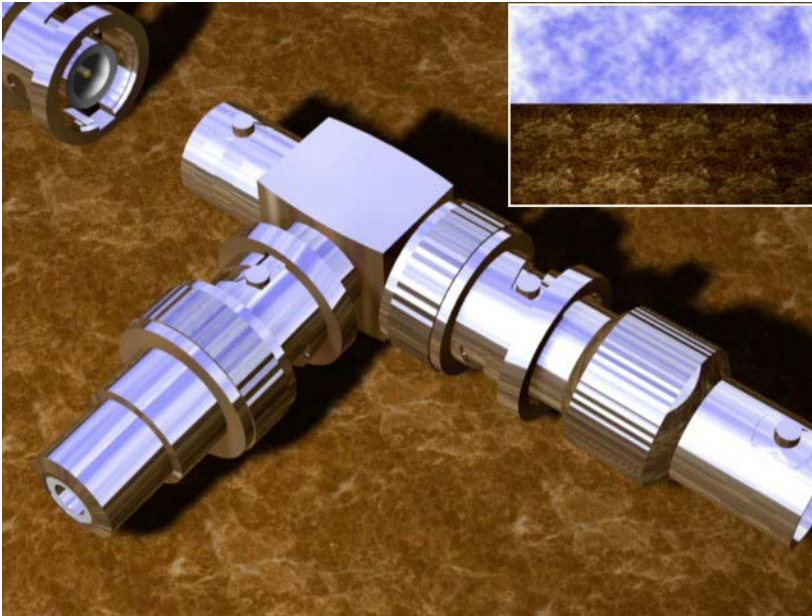
Note: The Total Inner Reflection must be enabled in the Transparency channel and Refraction should be set to 0.

See [Total Internal Reflection](#).

Environment (Cineware Surface Channel)

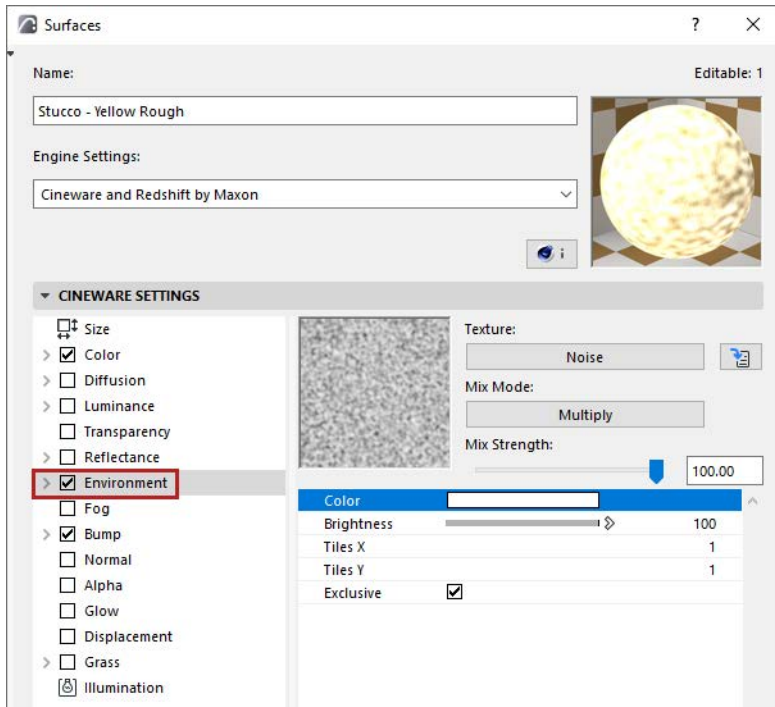
The Environment page uses a texture to simulate reflection. Here, in contrast to the other pages, the Color and Texture are multiplied by the Mix mode instead of being added.

So why would you want to use the environment property instead of Reflectance? One reason is that your scene may not have enough objects to produce good results when reflected. Another reason is that the Environment property renders more quickly than the Reflectance property.



Reflections using the environment property only.

Note: If you use the Environment channel in conjunction with the Reflectance channel, make sure that the Reflectance channel's Additive option is enabled. Otherwise the Environment channel will not be visible.



[Watch the Video](#)

Color and Brightness

[See Defining Color and Brightness Values.](#)

Texture

[See Textures \(Cineware Surfaces\).](#)

Mix Mode and Mix Strength

For the Environment channel, the default Mix Mode is Multiply.

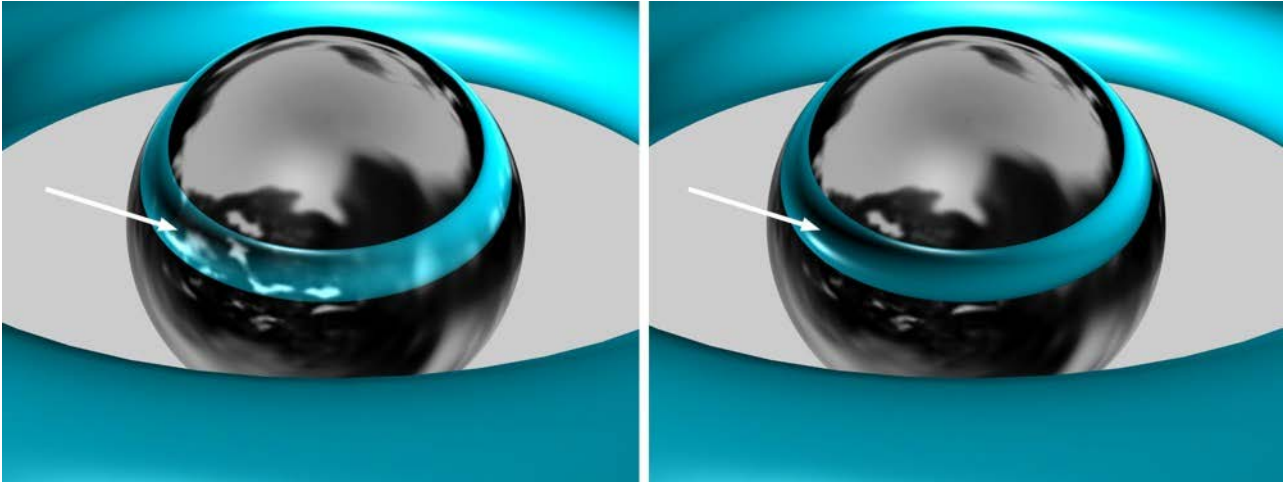
[See Mix Mode and Mix Strength \(Cineware Surfaces\).](#)

Tiles X/Tiles Y

The environment property is independent of the projection type of the surface. The environment is always placed spherically around the object. Use Tiles X and Tiles Y to set the number of tiles in the X and Y directions.

Exclusive

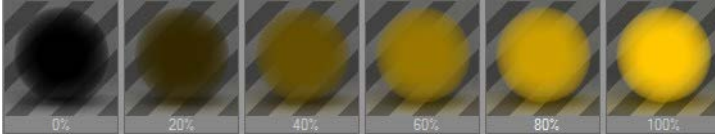
If this option is enabled, the environment reflections will only appear in places where there are no real reflections produced by the Reflectance channel.



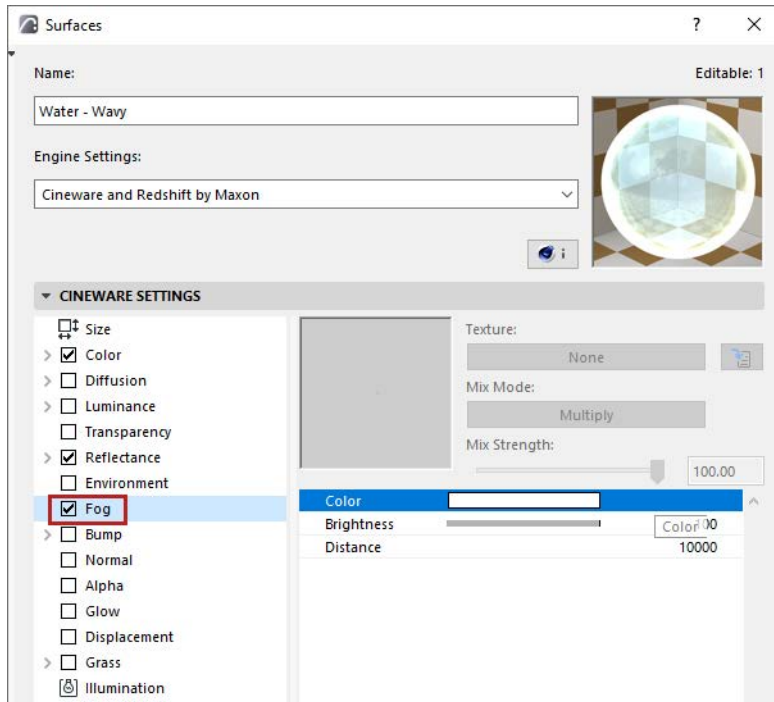
Left: Exclusive disabled, right: Exclusive enabled

Fog (Cineware Surface Channel)

These parameters enable you to simulate fog or gas clouds. Objects with such surfaces are translucent, but weaken the light that shines through them according to their density.



Use fog surfaces with closed objects only. Fog is a volume effect applied to the inside of an object.



Color and Brightness

See [Defining Color and Brightness Values](#).

See also the [discussion of Fog color below](#).

Distance

Light rays weaken as they pass through the fog. You can control this weakening with Distance. The larger this value, the thinner the fog. Distance defines how far a light ray can pass through the fog.

You can color the fog and this also affects its visibility. The further you look into the fog, the less the objects are visible and the more the **fog color** becomes visible. The fog color therefore also depends on the Distance value.

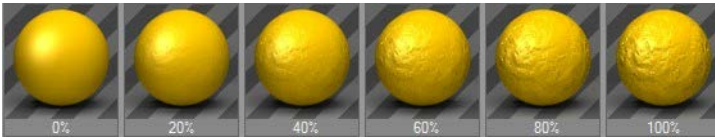
If you choose, say, a Distance of 500 units, a light ray of originally 100% intensity will have an intensity of 50% after a distance of 250 units and is extinguished completely after a further 250 units. The shorter the distance, the thicker the fog appears. In addition to this effect, after 250 units, half the fog color is added to the light and after 500 units, the full fog color is added.

Bump (Cineware Surface Channel)

The settings on this page enable you to simulate bumps.

Note: In contrast, the Displacement channel uses “real” bumps. ([See *Displacement \(Cineware Surface Channel\)*](#).)

A surface has a uniform brightness. However, if you use a bump map for the same surface, the program interprets the brightness values of the picture as height values for the surface.

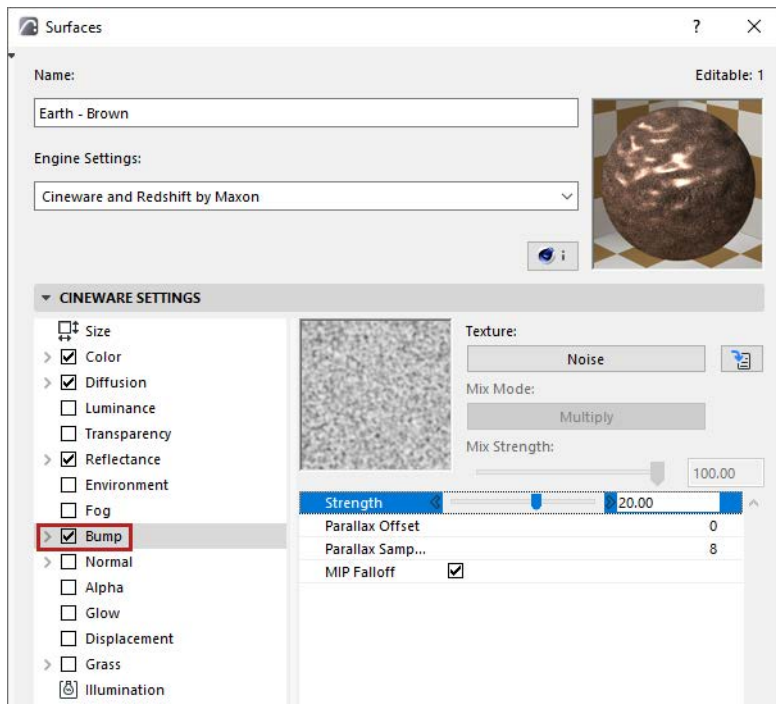


These height values are converted into a profile, whose height affects the inclination of the normal vectors. Although the surface is actually smooth, through the change in the normal vectors an apparently three-dimensional surface with a bump-like structure is created at render time.



You must use a texture with this channel. It is only from the grayscales in this image that the bump map (a height or relief map) is calculated.

[See *Textures \(Cineware Surfaces\)*](#).



Strength

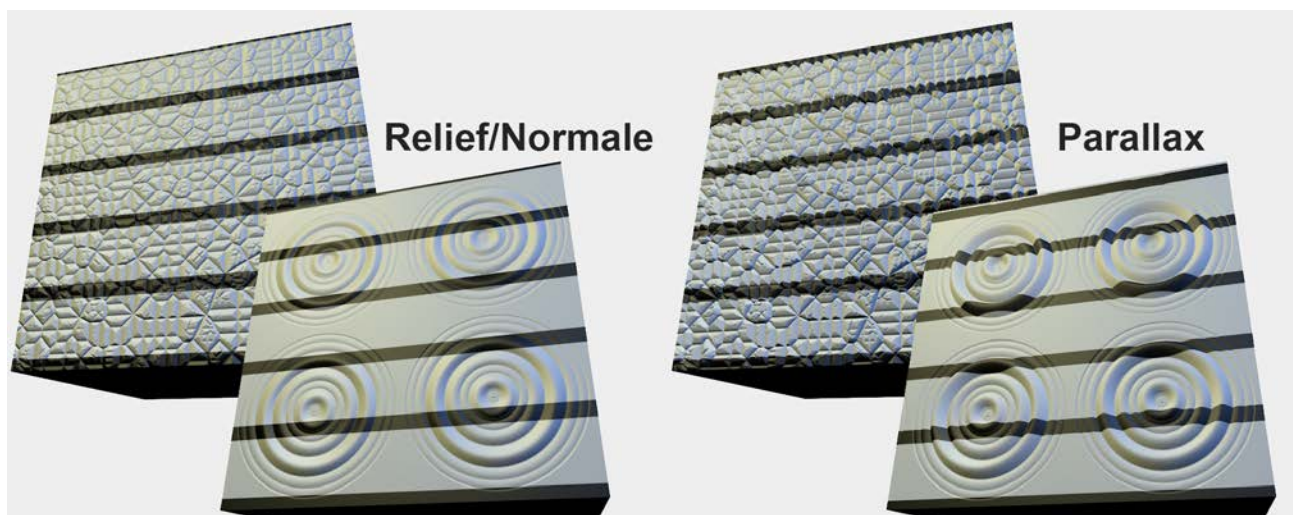
The strength of the bumps. The higher the value, the rougher the surface. If you choose negative values, this reverses the effect of the bump map — bright pixels then cause the surface to indent while darker pixels elevate the height of the surface.

You can enter values much higher than 100%. High values are especially useful when using MIP or SAT mapping, because these interpolation types tend to flatten the surface slightly.

[See *Special Settings for Image Textures \(Cineware Surfaces\)*.](#)

Parallax Settings

Parallax mapping provides approximately the same quality as subpolygon displacements but without having to subdivide the object internally. This reduces render times dramatically. Internally, Parallax mapping “bends” the UV coordinates to feign elevation on a surface. The color texture is distorted in a manner that makes its perspective distortion appear to be correct (e.g., the black stripes at the right the image below).



Imagine Strength and Parallax Offset as separate sliders: Strength defines the shading based on the bump effect and Parallax Offset calculates the texture distortion. The best results can be achieved when both settings are defined, i.e., bump and Parallax mapping are implemented simultaneously.

Parallax Offset

Use this setting to define the strength of the effect, i.e., the degree with which the UV coordinates should be moved to feign an elevated structure. Make sure to adjust the Strength setting (it most often has to be increased; it cannot be equal to 0) to produce convincing results.

Negative values can also be used to reverse the effect: raised regions will be sunken and vice-versa.

If the value is set to 0, no Parallax Mapping will take place. Bump mapping will take place instead.

Parallax Samples

Normally, these values do not have to be modified. However, if Parallax Offset were set very high, artifacts can appear. To offset this, increase the Parallax Samples value.

Higher values will result in correspondingly longer render times.

[Watch the Video](#)

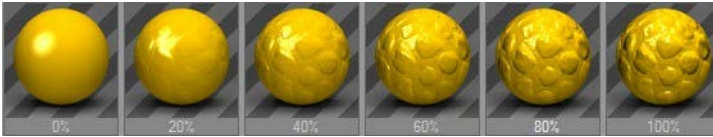
MIP Falloff

Enable MIP Falloff to enhance the MIP/SAT mapping effect for bump maps. This will reduce the strength of the bump map with increasing distance from the camera.

Normal (Cineware Surface Channel)

This surface channel produces a virtual surface irregularity that is lit realistically.

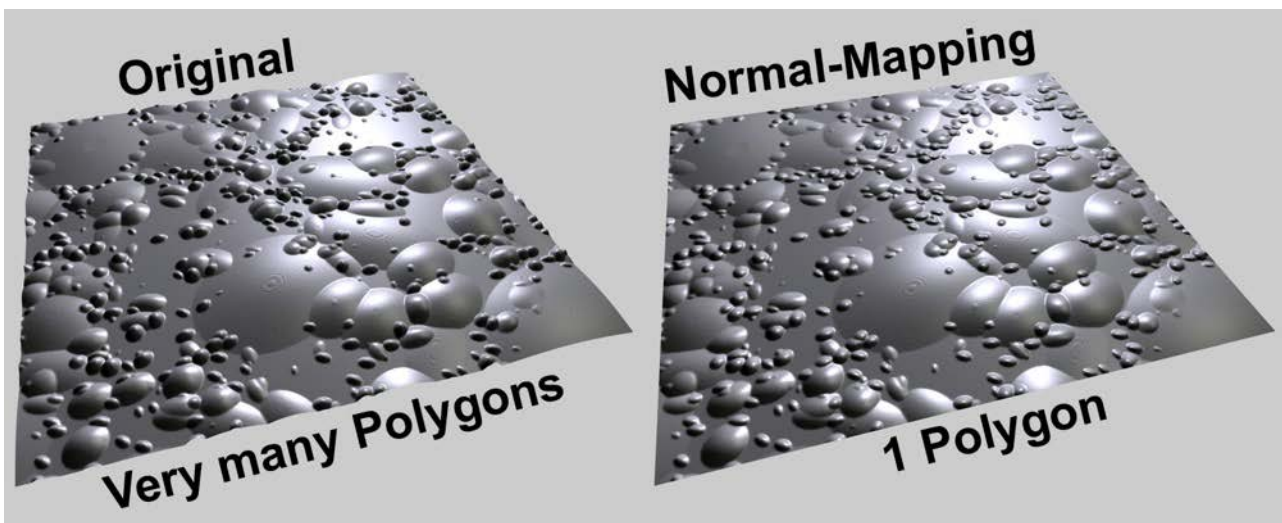
This entails technique called Normal Mapping (as opposed to Bump mapping). Normal mapping makes it possible to give a low-poly object (an object with a low level of detail) a detailed, seemingly structured surface, resulting in low render times.



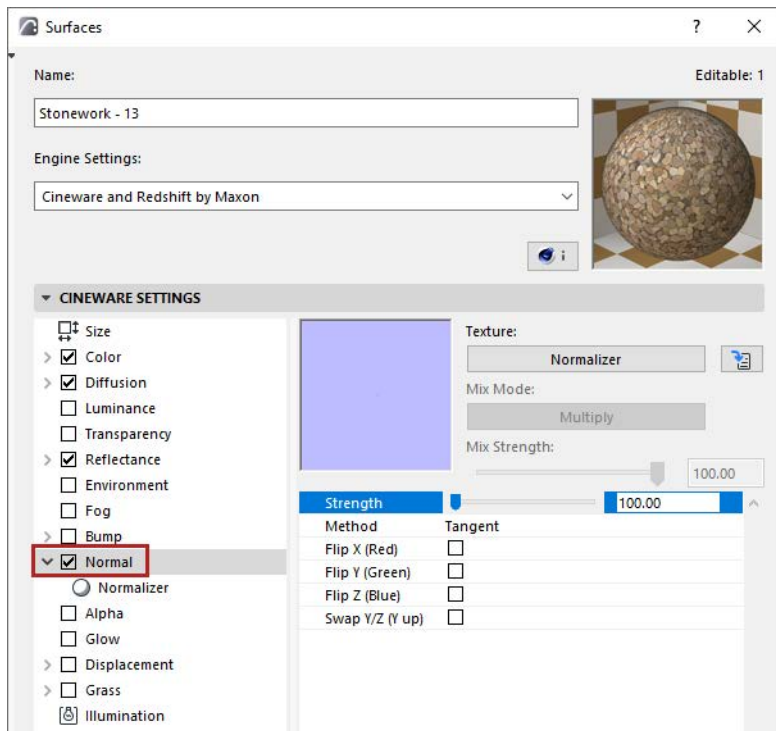
- **Bump mapping** uses a grayscale bitmap to generate height data.
- **Normal mapping** uses an RGB texture with coded Normal orientation (the Normals can be oriented in various directions).

You can create Normal textures with the help of the Normalizer shader (which can load normal color textures, usually bump textures).

The rendered result will very much resemble the original high-poly object.



*Left, the original object rendered with sub-polygon displacement;
right, the result using a Normal texture*



Strength

Regulates the strength of the Normal mapping. Do not use very high values because the effect will be increasingly unrealistic.

Basically, the higher the value used, the more perpendicular the view to the surface must be to keep the look of the surface realistic.

Method

There are several methods with which Normal maps can be defined:

- **Tangent:** The most common method. The Normal orientation is defined with respect to the underlying surface. Such textures can be identified by their mostly light blue or light green coloring; these are the areas in which the Normals are oriented perpendicularly to the surface.
- **Object:** Uses the object's coordinate system to code Normal orientation. Such textures emit, for the most part, in all colors of the rainbow.
- **World:** Uses the world coordinate system to code Normal orientation. These textures emit in all colors of the rainbow as well.

When selecting a method, you must know which method was used to create the Normal texture. This same method should be defined here.

Note: The Normalizer shader uses Tangents.

Flip X (Red)/Flip Y (Green)/Flip Z (Blue)

Swap Y & Z (Y up)

Since no standard method of creation of Normal textures exists, each application pretty much does it as it wishes: Sometimes the color component for the Y-orientation is green, sometimes blue. In order to assure maximum compatibility, these settings let you switch all color components.

The following rule of thumb applies to the most common method, Tangent:

- If the Normal texture is mostly light green, activate Swap Y & Z (Y up).
- If the Normal texture is mostly light blue, deactivate Swap Y & Z (Y up).

Alpha (Cineware Surface Channel)

An alpha channel enables you to achieve localized texture invisibility. The alpha channel uses an image to mask out areas of the surface, allowing any background to show through. This is useful for faking detail in 3D. The idea is to define areas of the surface that effectively become non-existent so that any underlying surfaces or objects show through.

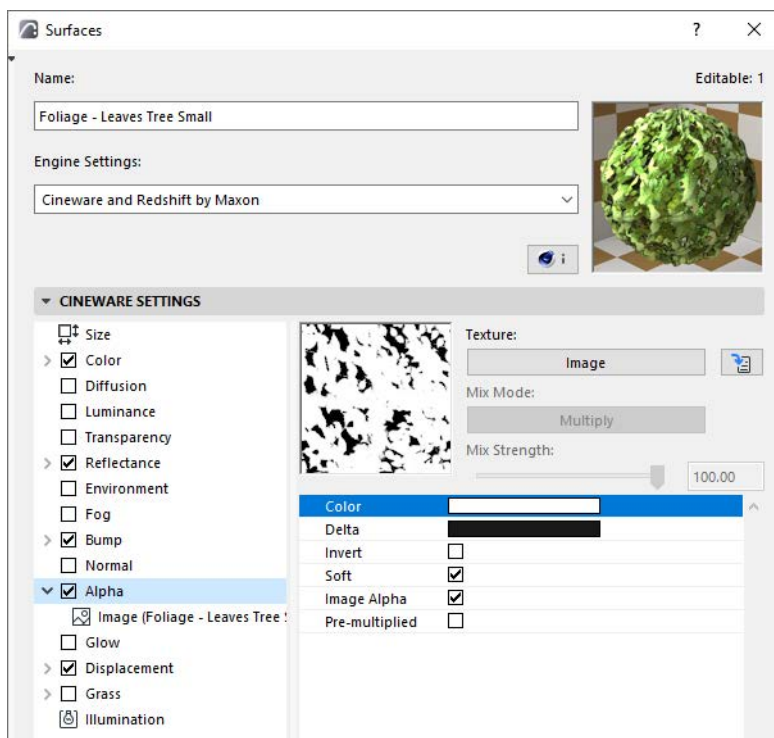
Note: When rendering bitmap images, make sure to use the Physical Renderer.



If you notice black areas when rendering objects that have multiple alpha surfaces, increase the Ray Depth value on the Options page of the render settings.

[See Ray Depth.](#)

If you apply a surface having an Alpha channel to an object that has no underlying surface, the object will be non-existent at the points where the alpha would allow the underlying surface to show through.



Alpha Channel and Antialiasing Filter

Depending on the Filter Antialias settings, the Alpha channel can end up being smoothed as well.

[See Filter.](#)

An Alpha texture can also contain a grayscale gradient that can be used for blending.

Color

You can use clip mapping to mask particular areas with a **Color** value. Using this method, you pick the color from the selected image that you wish to become transparent. The downside is that often the areas you wish to become transparent contain several colors and the resulting alpha can leave a seam around the object. However, by setting the **Delta** color (see below), you can define by how much the color may deviate; this can help to reduce the seam.

Delta

Clipping is often attempted with texture images that are antialiased; this produces a bright border around an object caused by the antialiasing of colors between the main texture and the alpha color; you can remove this border by adjusting the deviation (delta) color.

Use alpha channels built in to the texture to mask areas softly and more accurately.

[See Image Alpha below.](#)

Invert

Enabling the Invert option will invert the effect of the Alpha channel. All black areas of the alpha image will remain visible and the white areas will be masked.

Soft

This enables you to fade textures and surfaces from one to another, which gives you even more ways of creating realistic-looking objects. With Soft enabled (the default setting), the Color and Delta settings lose their meaning. The texture map is now used to decide which ranges should be faded. A white texture pixel in the image means that here the surface is to be 100% opaque. If the texture pixel is black, the underlying surface shines through 100%.

Image Alpha

With Image Alpha enabled, you can use any existing alpha channel of the loaded image. Alpha channels of the TIF, TGA, PICT, PNG, and Photoshop PSD. If no alpha channel is present in the image, the Image Alpha option will be ignored. Use the Invert function to invert the alpha channel without having to rework the texture in your image editor.

Pre-multiplied

Enable this option if you are using a texture with a pre-multiplied alpha channel. Certain graphics applications only produce this type of alpha channel.

Glow (Cineware Surface Channel)

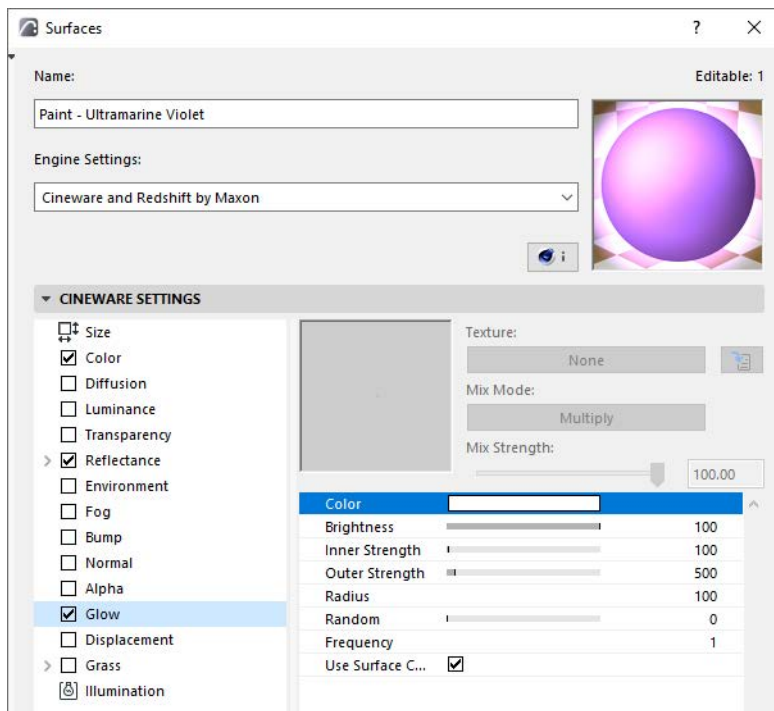
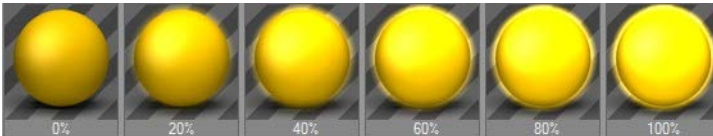
This page enables you to create a soft glow.

Glow is a post effect. It will take place only if you also turn on Object Glow in the Rendering Settings dialog.

[See *Object Glow*.](#)

Glows cannot be seen through transparent objects, nor in reflections. The glow you specify here does not act as a light; other parts of the scene will not be lit by the glow, nor will any shadows be cast. Glows are restricted to a maximum image resolution of 4,000 x 4,000 pixels.

[Watch the Video](#)

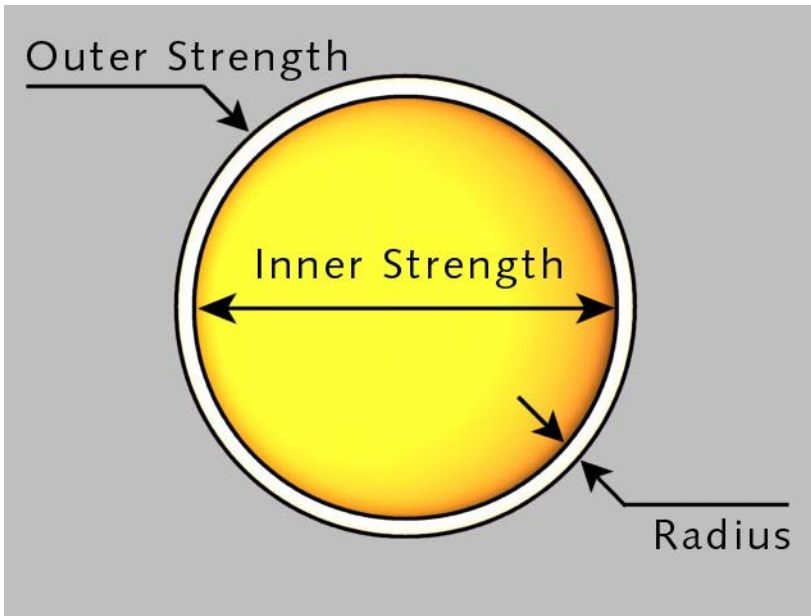


Color and Brightness

[See *Defining Color and Brightness Values*.](#)

Inner Strength/Outer Strength

Inner Strength specifies the intensity of the glow over the surface; Outer Strength is the intensity of the glow at the edges.



Radius

The Radius determines how far the glow (either the inner or the outer) extends from the surface. This value is rendered relative to the distance of the object from the camera. The further away the object is, the smaller the glow, and vice versa.

Random

If a random percentage is defined, the intensity of the glow in each animation frame is increased and decreased in a random pattern, as follows:

0%: No change

100%: Maximum change

Frequency

Frequency specifies how often the glow radius changes. The amplitude of the change is given by the Random value.

1Hz: The glow reaches a new random value after 1 second

25Hz: The glow has a new value for each frame (for 25 fps), which causes a flicker

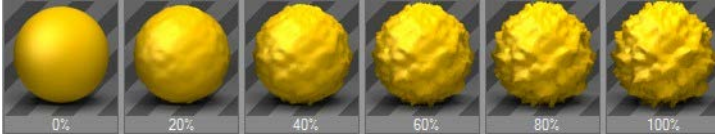
Use Surface Color

If Use Surface Color is enabled, the glow is calculated on the basis of the surface color rather than the color specified here.

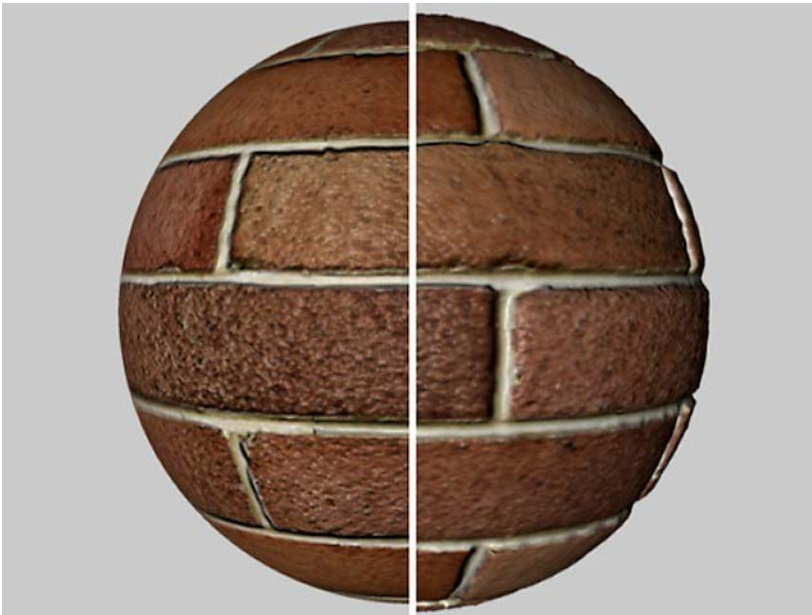
If the option is disabled, the object and glow colors are mixed. For example, green objects will appear yellowish under a red glow.

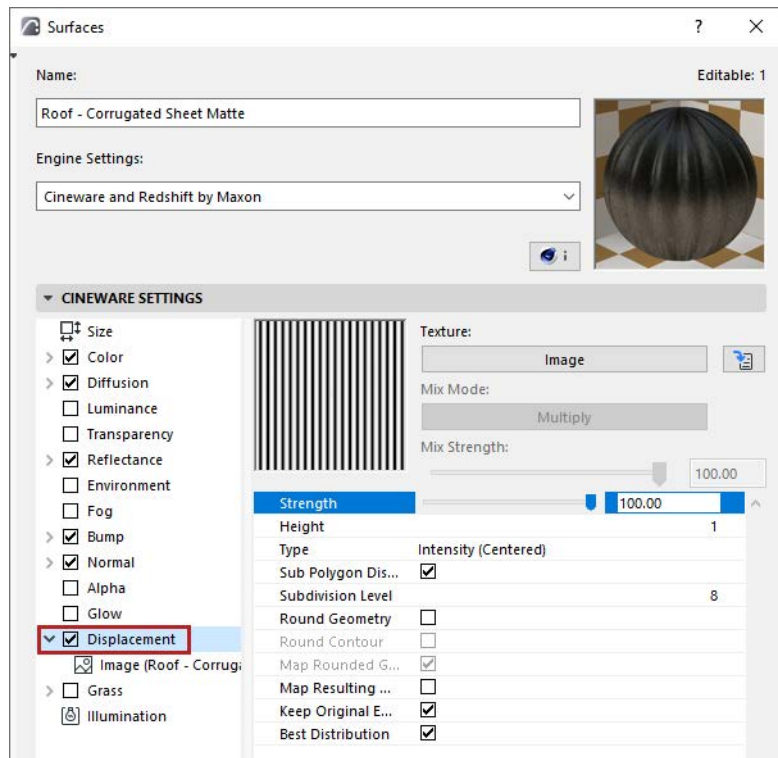
Displacement (Cineware Surface Channel)

Displacement is similar to Bump, the difference being that here the object is actually (not just apparently) deformed. This difference is best seen at object edges.



In this object: the left half of the sphere uses bump mapping, while the right half uses displacement mapping. ([See also *Bump \(Cineware Surface Channel\)*.](#))





Strength

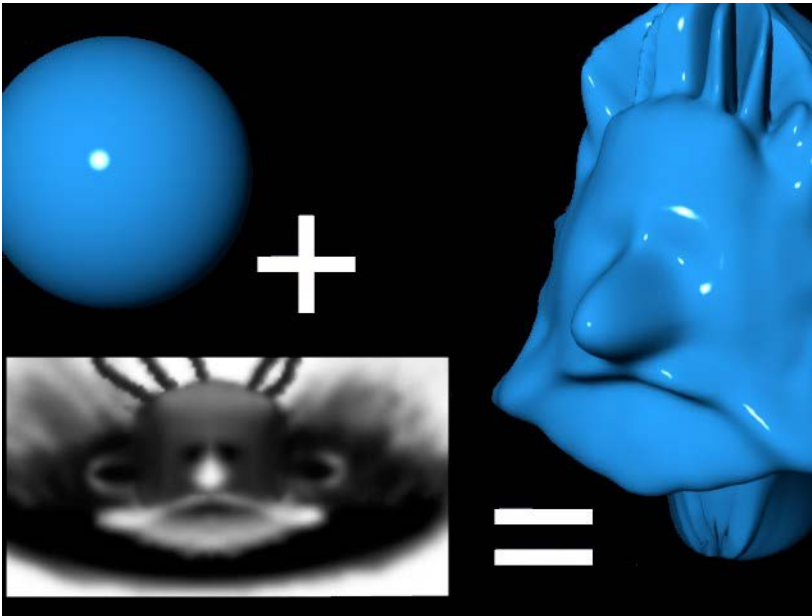
This control allows you to adjust the maximum displacement defined by the Height setting (the Strength and Height values will be multiplied together to control the maximum displacement).

Height

Defines the height of the displacement, which can be modified by Strength value.

Type

- **Intensity:** The displacement takes place in the positive direction. Black parts in the displacement map produce no displacement, white parts produce maximum displacement.



- **Intensity (Centered):** The displacement can take place in both positive and negative directions. A gray value of 50% results in no displacement. White produces maximum positive displacement, while black produces maximum negative displacement.
- **Red / Green:** The displacement can take place in the negative or positive direction according to the red and green values in the texture. Green values raise the displacement, red values lower the displacement. A black color results in no displacement, it is purely the red and green components that control the displacement. A pure green (RGB: 0,255,0) and a pure red (RGB 255,0,0) lead to maximum displacement in the positive and negative directions respectively.
- **RGB (XYZ Local)/RGB (XYZ World)**

These modes control the displacement spatially according to the texture's RGB components. Depending on the mode selected, different coordinate systems will be used for deformation.

The components define the following directions:

- Red: X
- Green: Y
- Blue: Z

Texture

Here an image texture or a 2D shader can be defined.

[See Textures \(Cineware Surfaces\).](#)

Sub-Polygon Displacement

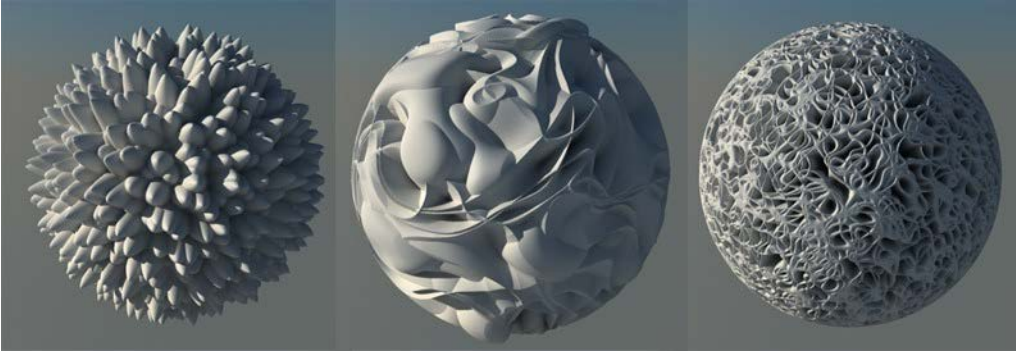
Check to activate Sub-Polygon Displacement (SPD).

In principle, SPD is similar to displacement: an object is deformed during rendering based on a texture's grayscale palette or, in some modes, color palette. The trick lies in the object's own internal, adjustable and relatively high subdivision, which allows for very detailed structures without having to actually subdivide the object permanently. In many cases, it is impossible to achieve the same degree of detail through modeling because too much memory would be required.

SPD adds to the render time, since rendering real geometry using sufficient memory is, with few exceptions, always faster than rendering virtual polygons.

SPD offers the following advantages:

- Small details can be rendered without having to permanently subdivide the object; permanent subdivisions can lead to problems such as file size that is too large or even impossible to manage.
- Details can be implemented faster using 2D textures instead of modeling.
- The quality of the rendered image is much better than using a texture map in the bump channel.



Noise shaders shown with SPD

If the option is disabled, normal displacement will be used (only existing object points will be affected).

Subdivision Level

This is where you determine the SPD subdivision. This goes for the entire object to which the surface has been applied. It can be well worth the trouble to delete the object's hidden sides in order to reduce the number of surfaces that must be subdivided.

As is often the case, higher values produce better results but add to the render time.

Note that you may need to change this value for each face of an object (polygon): thus, you might use duplicated surfaces with different SPD parameters for different faces.

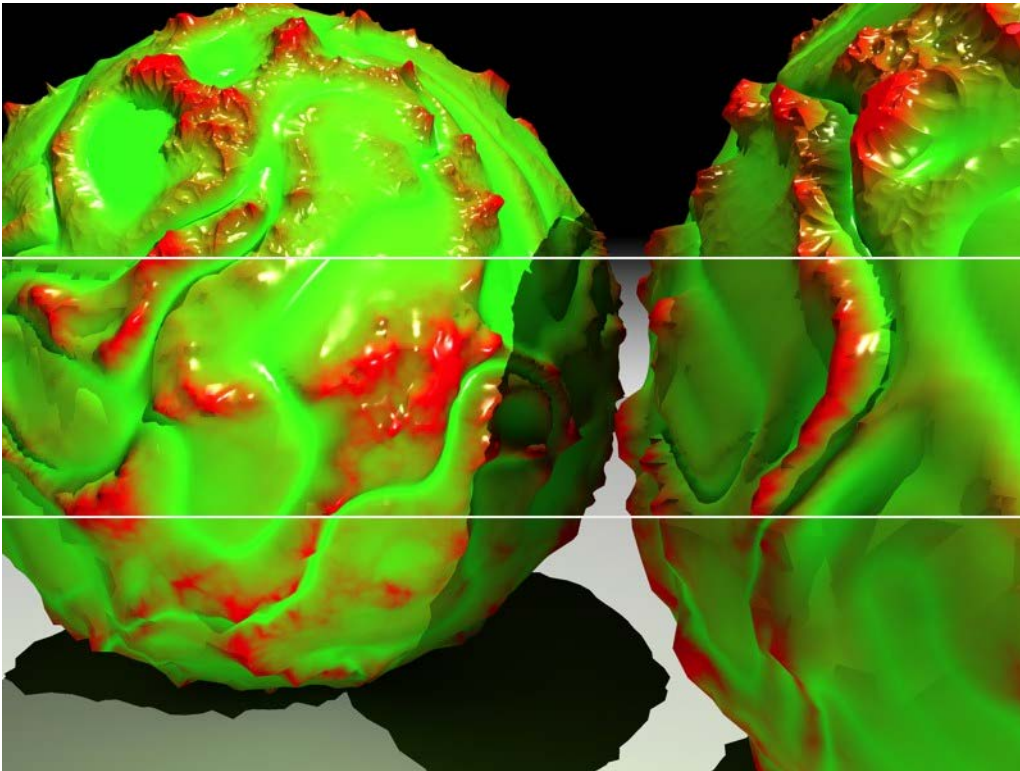
A 1m by 1m wall face needs less SPD value than a wall of 1m by 100 m.

For each existing polygon, the following polygon count will be calculated internally:

- Triangle: $(2 \text{ to the power of Subdivision Level}) * (2 \text{ to the power of Subdivision Level}) / 2$
- Quadrangle: $(2 \text{ to the power of Subdivision Level}) * (2 \text{ to the power of Subdivision Level})$
- Cube: $6 * 256 * 256 = 393,216$ polygons.
- Plane: $400 * 256 * 256 = 26,214,400$ polygons.

Note:

- If several displacement surfaces are assigned to a given object, the maximum values will always be used.
- Example: An object has 2 different displacement surfaces on 2 selections. The first displacement surface has a subdivision level of 4 and the second 6. The entire object has a subdivision level of 6. Even though the entire object is subdivided, the displacement will only be applied to the defined selections.

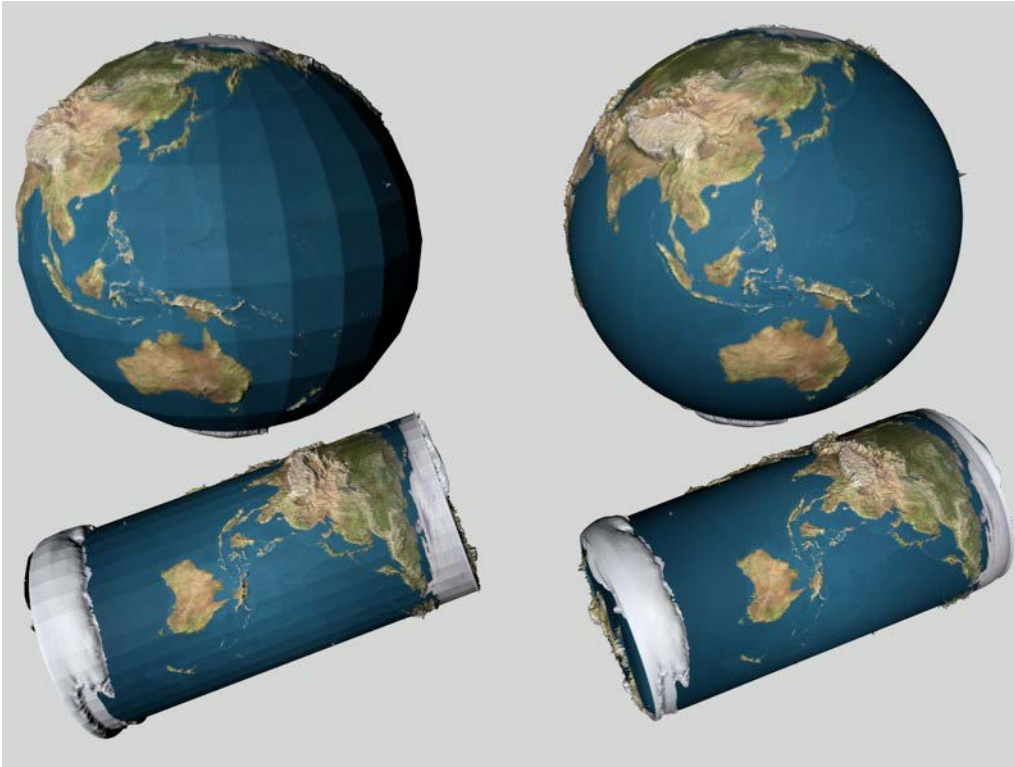


Two sphere primitives with a subdivision of 2, 4 and 8 (from bottom to top)

Round Geometry

Since SPD cannot calculate a normal Phong shading when it is applied, a special algorithm is used (if you enable this option) to ensure that the object is rounded before the SPD is rendered.

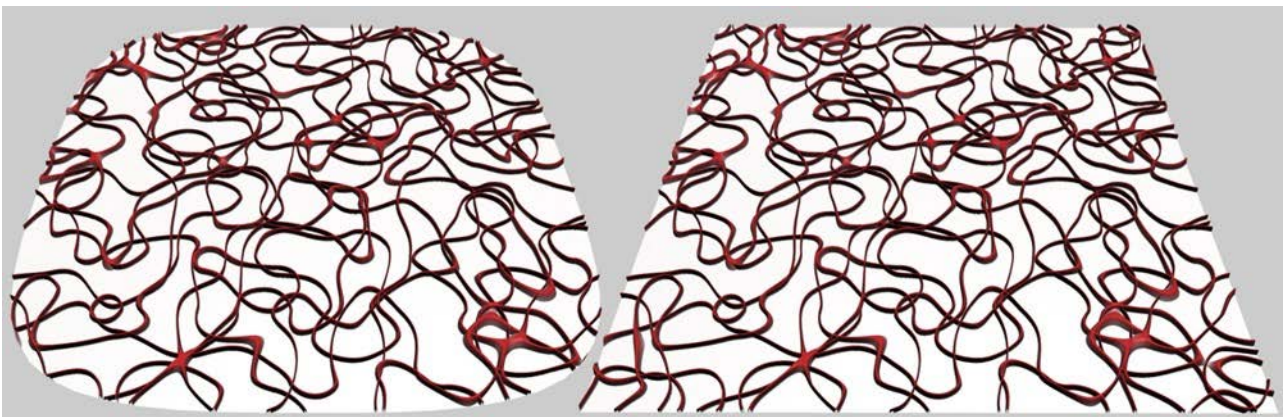
This is specially designed to smooth surfaces that appear faceted after SPD rendering. Contours (polygon edges that have no adjoining polygons) remain unaffected if Round Contours is disabled.



Since this can also cause unexpected results, you can also disable the option.

Round Contour

If this option is enabled, contours (polygons that have at least one adjoining polygon missing) will also be rounded. Disable the option for objects whose contours would be adversely affected by the smoothing.



Round Contours enabled (left) and disabled (right)

Map Rounded Geometry

This option determines whether the rounded geometry should be used to define the texture coordinates. In most cases this will lead to more intuitive results. It can also lead to fewer artifacts.

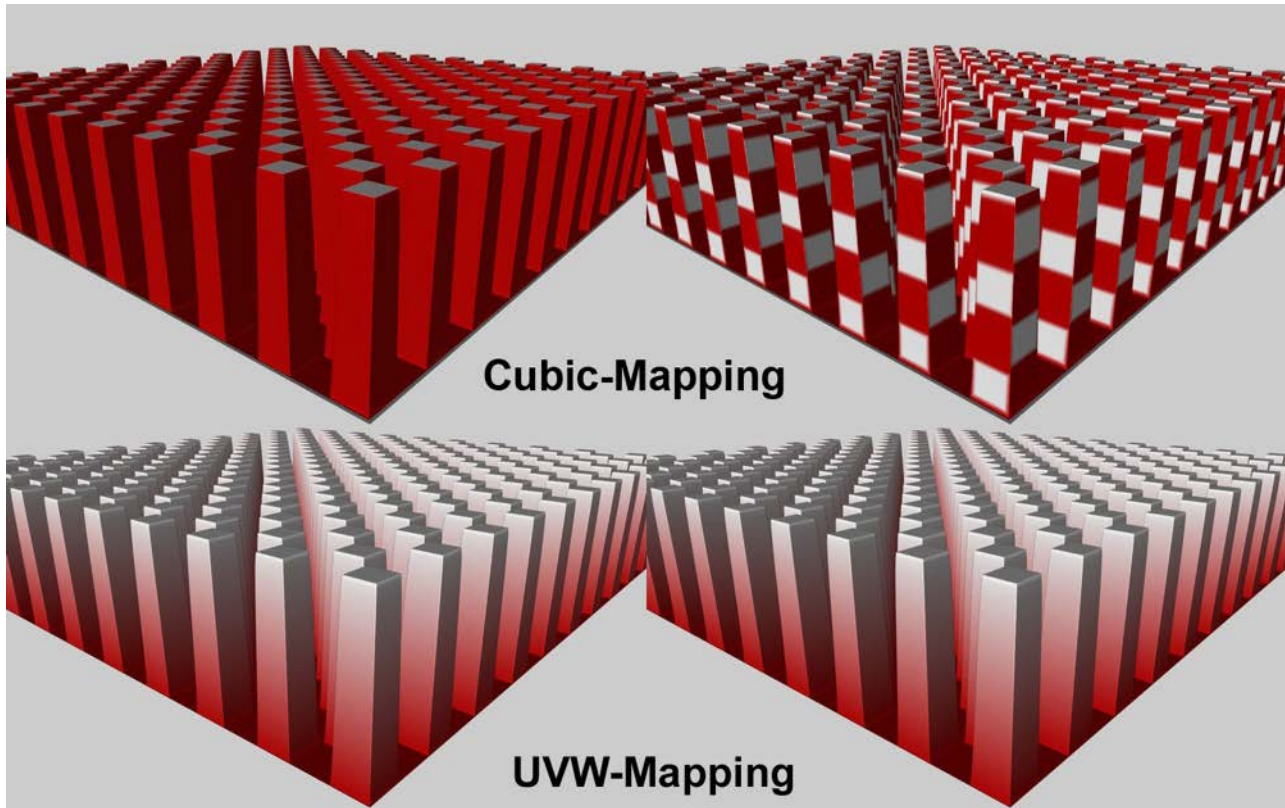
Because the calculation can take up to 10% longer and in some cases the projection must be onto the non-rounded geometry, you have the option to disable this feature.

Map Resulting Geometry

Use this setting to define how the texture should be projected:

- Setting is active: Texture is projected *before* applying the Sub-Polygon Displacement.
- Setting is inactive: Texture is projected *after* applying the Sub-Polygon Displacement.

When mapping a cube you can clearly see how the colors are projected, depending on whether or not Map Resulting Geometry has been activated. In contrast, no difference can be seen when using UVW Mapping since SPDs do not change any UV coordinates.



Various projection effects:

Left: Map Resulting Geometry not active, right: Map Resulting Geometry active

3D-based shaders (e.g. noise, wood, rust, etc.) behave differently. Since they do not require UV coordinates, they can also be used to texture the edges of extreme displacements.

Keep Original Edges

If this option is enabled, hard Phong edges will remain hard. If the option is disabled, the edges will be rounded according to the Subdivision Level.

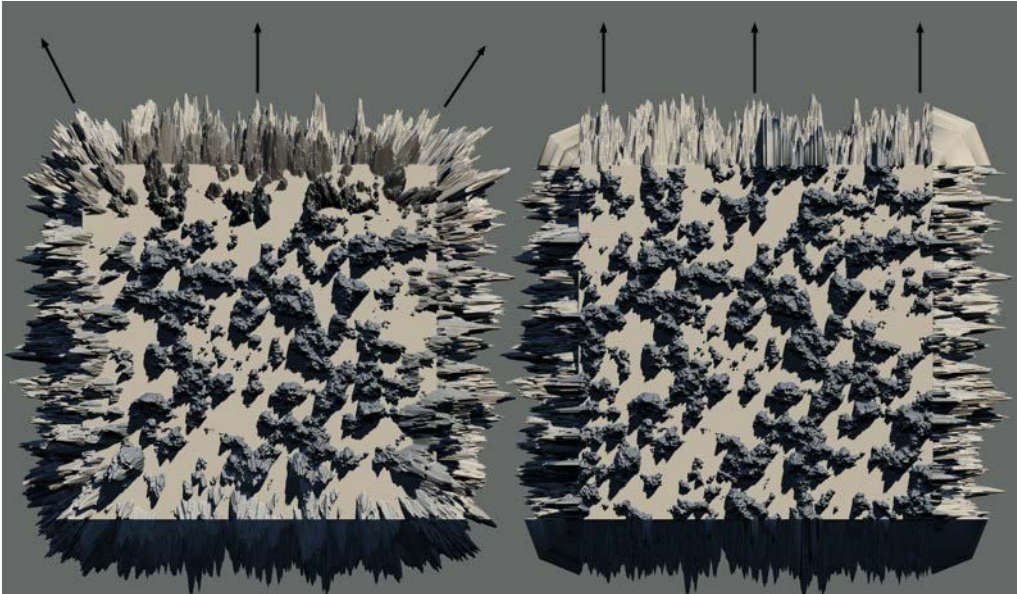
The effects of this option mostly occur only when the Round Geometry option is disabled.

Best Distribution

The enabled Best Distribution option changes the direction of the displacements towards the Phong edges. The closer a displacement is to the edge, the closer it is brought to this edge and the closer it follows the virtually rounded Phong normal.

In most cases, this setting will ensure a soft, uninterrupted transition of the SPD over phong edges.

If Best Distribution is deactivated (which only makes sense if one of the intensity modes is set in the Type setting), each displacement will point up perpendicularly. This can lead to inflated displacements at the edges. This is a result of displacements that overlap edges and basically stand at a right angle to each other: the space between them will be filled. Activate this option if modeling cities that are viewed from a greater distance or, for example, castle stones.

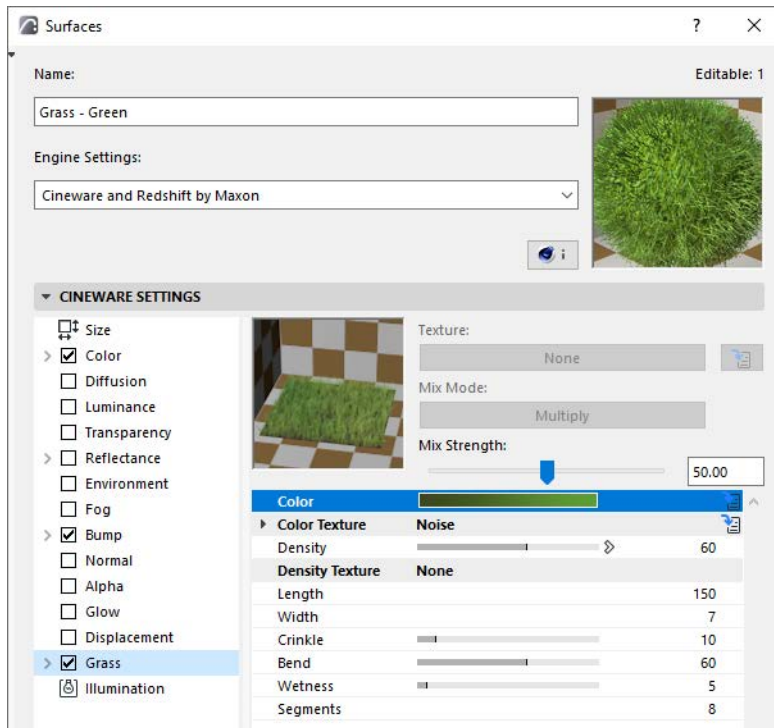


Best Distribution enabled (left) and disabled (right)

Grass (Cineware Surface Channel)

Use this surface channel to quickly and easily create realistic-looking grass.

Note: Use the parameters in Render Settings to activate and define grass rendering for the scene as a whole. See [Grass \(PhotoRendering Option\)](#).



[Watch the Video](#)

Color

This color gradient defines the color from the root (left) to the tip (right) of the grass blade.

See [Defining Color and Brightness Values and Adjusting Gradient Parameters](#).

Color Texture

You can also control the color of grass using a texture. The texture is assigned to the grass using UVW mapping. The blades of grass will assume the color of the texture. The Mix Strength value can be used to seamlessly mix this effect with the gradient's colors.

Density

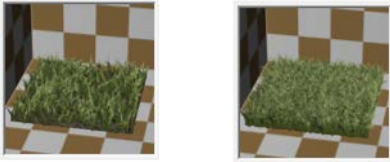
Defines the number of blades of grass that should be dispersed across a given surface. Note that increasing the Density value will lengthen render times accordingly.

To be more exact: a value of 100% will generate 1 blade of grass per 5 surface units (on a surface that is 10 cm square, this would be approximately $10 * 10 / 5 = 20$ blades of grass).

Note: Because of the effect on render time, keep the density value low. To simulate a denser effect, try using thicker and longer blades instead of increasing the density value.

Density Texture

A grayscale texture can be loaded here, which will define the grass density according to the texture's grayscale values. The effect is a more realistic, blotchier appearance. The density will range from white regions having the most density to black regions having none.



Length/Width

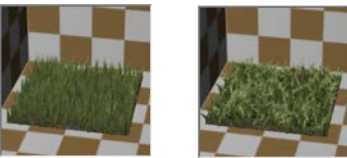
Defines the maximum length (the default value generates a relatively short blade) and width (at the root of the tapered grass blade) of the individual blades of grass.

Crinkle

This parameter works similar to the Bend parameter (see below) and varies the shape of the individual blades of grass to make them look more realistic. The larger the value, the more blades will randomly crinkle multiple times in different directions.

Bend

The Bend parameter causes the blades of grass to bend. The higher the value, the more the blades of grass will randomly bend in one (!) direction (in contrast to the Crinkle parameter that randomly crinkles the blades multiple times).



Wetness

Use this setting to define the intensity of the specular light. The more intense the specular light, the wetter the grass appears to be.

Segments

Use this setting to set the number of segments per grass blade (from 1 to 20). 1 segment means the blade is entirely straight. The default segment number is 8.

To speed up rendering, set this number lower. This is worthwhile if you have large grassy areas in the distance where more precise rendering is unimportant.

Tips and Tricks for Using Grass

- If you want to let grass grow on a plane with a green surface assigned to it, the Density setting, which can greatly influence render speed, can be reduced without affecting the overall look very much.
- Grass that is thicker and longer than real-world grass helps simulate fuller growth without having to increase the Density value too much.
- If the grass surface is very far from the camera you should consider using a texture instead of grass (for much faster rendering).

- Use a 3D Density gradient within the camera space in order to let 3D grass gradually fade into a simple grass texture with distance. This is the best way to reduce the number of unnecessary 3D grass blades.

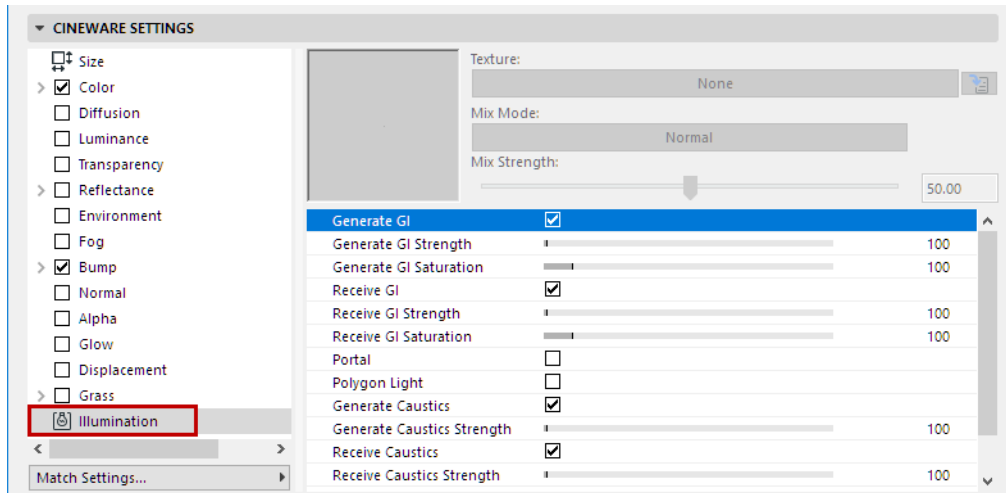
Illumination (Cineware Surface Channel)

Use the Illumination channel in Surfaces to enable or disable the generation and reception of Global Illumination (GI) to and from this surface, and to control the strength of the caustics.

If, for example, you have a project with many transparent objects it would be very laborious to calculate caustics for each object. In this case, you can exclude particular surfaces from caustics calculation.

This page also defines the surface's Global Illumination model setting (this greatly affects how highlights react to the light in the scene).

Each type of GI (generated and received) has its own adjustable values for Strength and Saturation.



In order for the Illumination surface channel to take effect, Global Illumination (GI) must be enabled in Render Settings. ([See *Global Illumination \(Cineware\)*](#).)

Generate GI

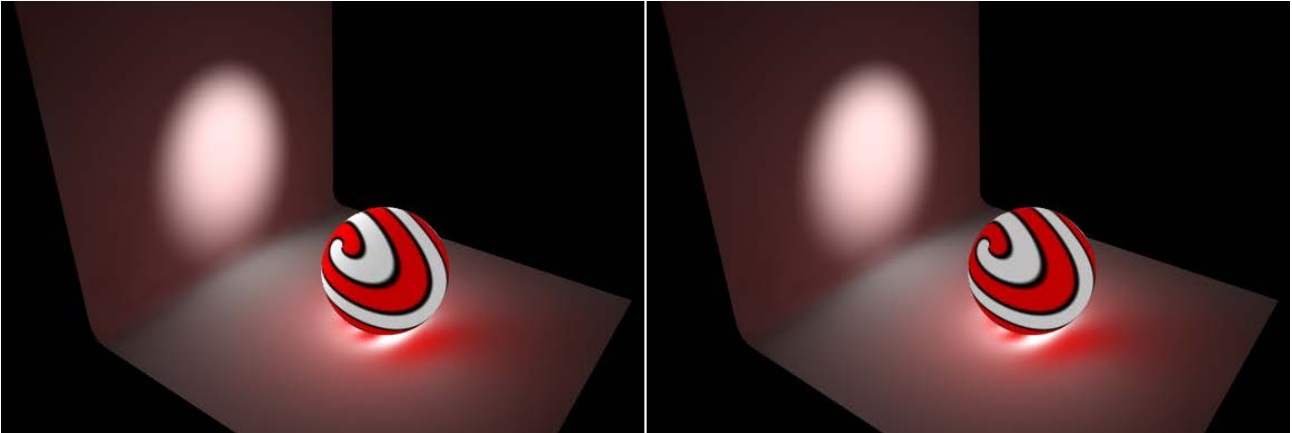
The Generate GI option affects the light created by the surface's Luminance channel as well as any passed on by a given light source.

- Generate GI Strength: Define how strong a given surface will emit.
- Generate GI Saturation: Fine-tune the saturation of textures used in generating GI.

Deactivate the **Generate GI** setting if a given surface should not have an effect on other objects.

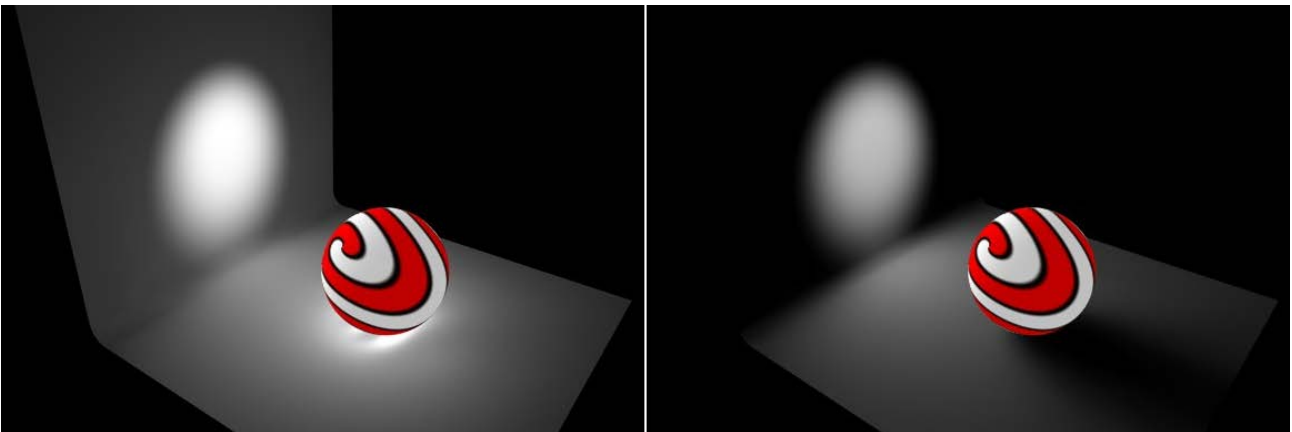
In the image below, the original scene is at the left. A Spot light is used to illuminate the wall. The sphere has been assigned a surface with an active Luminance channel through which the wall is also illuminated in part when the scene is rendered using Global Illumination.

On the right, the **Receive GI** option for the sphere's surface has been disabled. This means that the sphere will no longer receive light reflected from the wall. The sphere's self-illumination remains unaffected.



Below, on the left, another original scene: the sphere generates and receives GI but the **Saturation** value for the Generate GI option was set to 0%. The scene's brightness remains constant but the sphere's color information in the light it reflects is no longer present.

Finally, below right, the image shows the result of a completely disabled Generate GI option for the sphere. The illumination within the sphere is still evident but this illumination no longer affects the rest of the scene. Only the reflected light from the Spot light remains.



If a surface contains an active Transparency channel it can also be used in combination with the **Portal** parameter. This can improve the quality of GI in many instances, for example when sunlight radiates through a window into a room. Transparent surfaces such as windows will be sampled more closely for light passing through it.

Receive GI

Deactivate this option if a given surface should not receive brightness or color from other objects.

- **Receive GI Strength:** Define to what degree a given surface will receive color and brightness of other surfaces.
- **Receive GI Saturation:** Fine-tune the saturation of textures used for GI. Contrary to the previously described Saturation, this setting affects only the receiving of GI, not the radiation of it.

Portal

Activating this option will define the object to which it is assigned as a Portal (make sure you also activate the Transparency channel).

Portals should be placed/fitted at the windows. When placed at windows with a crossbar frame, multiple Portals will increase overall quality.

Portals are usually single-polygon surfaces (e.g. primitive Plane) that are placed where light passes through relatively narrow openings. The direction in which the normals show is irrelevant.

If you do not want to make corresponding changes to the light-reflecting object itself, you can influence the light passing through the Portal by changing the Color or Brightness settings in the Transparency channel.

Polygon Light

If this option is enabled, the way in which a (luminous) object is included in the GI calculation will be optimized. This option should be enabled if the object serves as an Area light.

Caustics

A caustic effect is a light effect made up of refracted and bundled light. A typical example of a caustic effect are the wavy lines at the bottom of a swimming pool. Caustics can also be seen in refractive surfaces, e.g. in a reflective ring. Surfaces that generate caustic effects must be either reflective or transparent.

Caustics generation and reception can be enabled or disabled separately. However, this is only relevant if Caustics has been enabled in the Render Settings and light sources with active caustics generation are present in the scene.

Samples and Radius both affect the quality of the effect. ([See below.](#))

Generate Caustics

Enable this option to activate caustics generation for the active surface. Make sure that either the Transparency (for caustics that result from light breaking through water in a glass) or Reflectance (for caustics resulting from light reflecting from a curved object) surface channels is active.

Note: Only the Generate parameter is relevant when using volume caustics.

- **Generate Caustics Strength:** Use the slider to set the strength of the effect.

Receive Caustics

Enables the reception of surface caustics for the surface

- **Receive Caustics Strength:** Use the slider to set the strength of the effect.

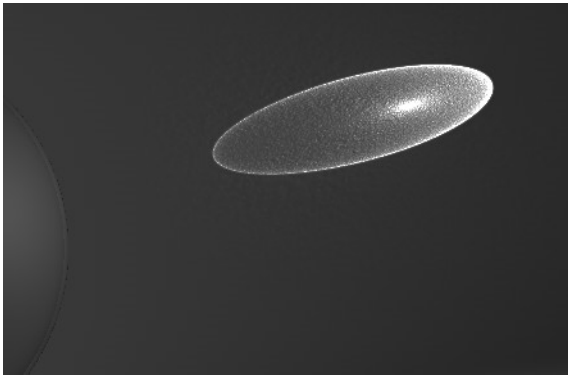
Radius

Since caustics are comprised of photons, the brightness and detail of the effect is determined by the density of these photons.

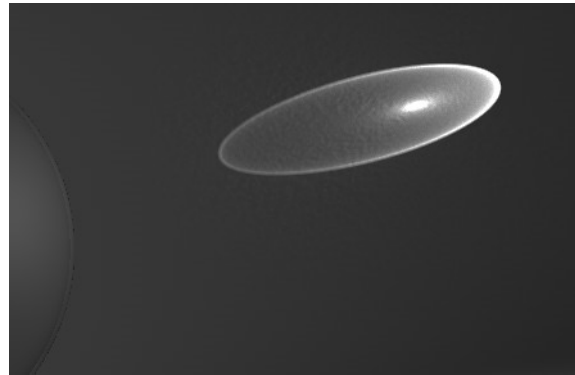
The Radius value defines the radius around the image pixels in which the calculation will scan for photons. The larger the radius the softer, and less accurate the result will be. Smaller radii offer more detailed but also more grainy results.



Radius=1; individual photons can be seen as points of light because they are not interpolated together



Radius=10



Radius=100

Caustics Samples

This defines the maximum number of photons within the Radius that are used to calculate the effect. Generally speaking, the more Samples included in the calculation, the more exact the caustics effect will be. Larger Radius values require longer render times and result in blurred caustic effects.

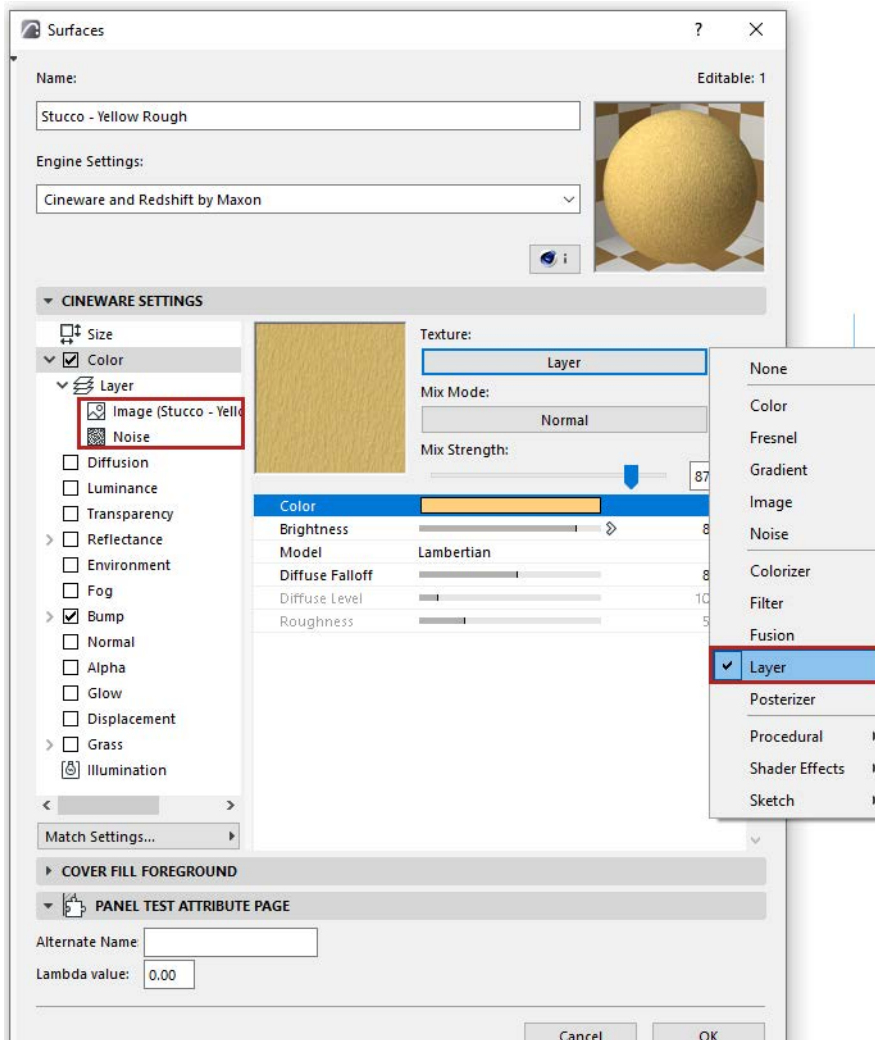
To summarize: more samples per radius means a more accurate image. Increasing the Radius means more blur but a longer render time also.

Textures (Cineware Surfaces)

Most Surface channels enable you to load a texture. Textures add realism to the surface, enabling you to simulate, for example, wood grain.

For a texture, you can select either a **shader** or an **image**.

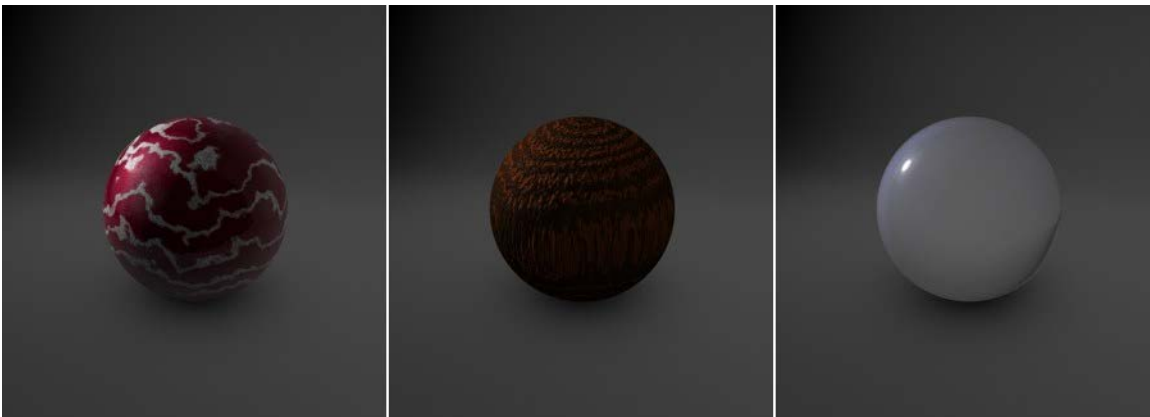
Note: You can also choose **Layer** as a way to combine multiple shaders/images in the same channel. (See [Using Layers to Combine Shaders \(Cineware Surfaces\)](#).)



- An **image** file can be manually created or a photograph (e.g. of real dirt, grime or structured surfaces such as walls or floors). Numerous companies have specialized in creating just such textures for use in 3D. In many cases you can also use your own digital camera to create such images. The image below shows the difference between a surface with only a simple color and one with a bitmap graphic added.

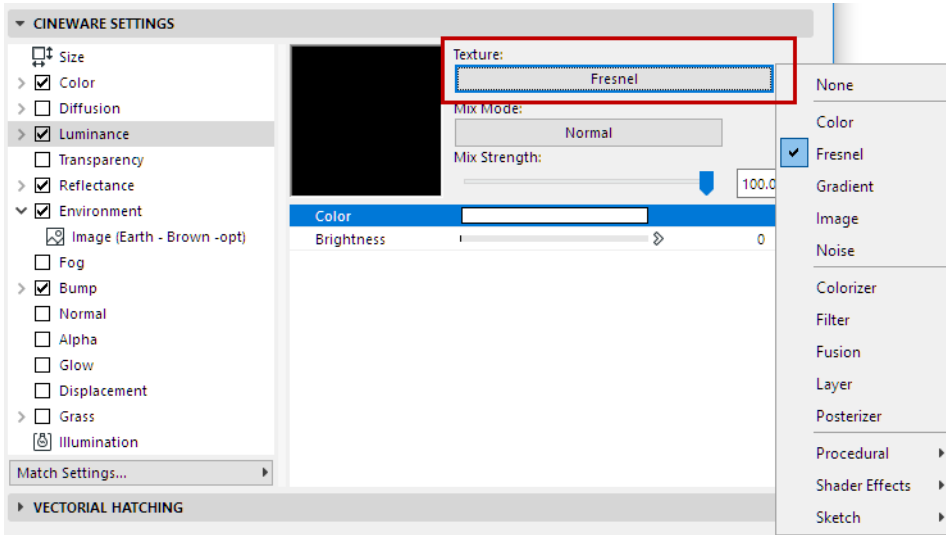


- Alternatively, **shaders** are tiny routines that mathematically calculate a pattern or a certain effect such as light entering and diffusing within an object. Shaders can also be used for the creation of a color gradient or to color correct a loaded image. Shaders can be used in many different ways and are used in some capacity in almost every surface.

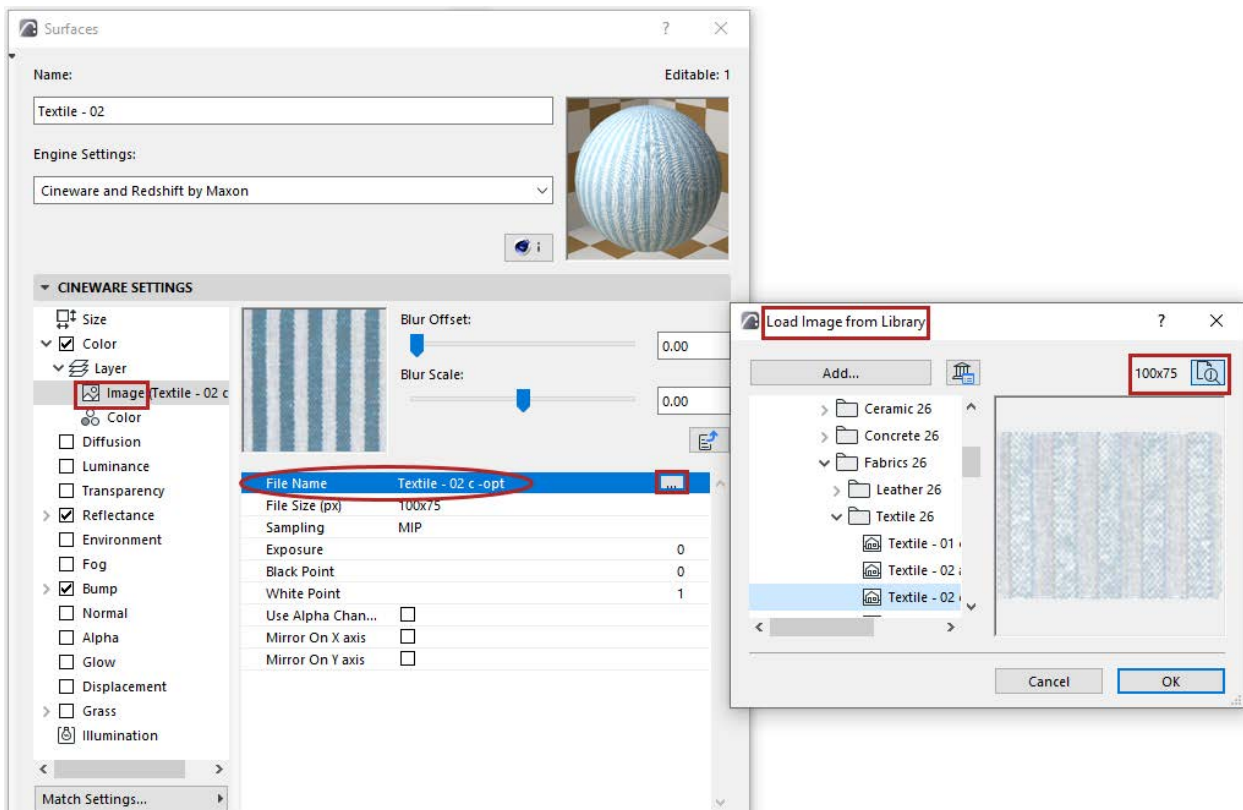


An advantage of using shaders is that you don't have to take into consideration a bitmap's pixel resolution. A pattern generated by a shader looks good close up and doesn't pixellate or blur as a bitmap image might. Therefore, it is recommended that you use shaders whenever possible.

To choose a texture or shader for a particular channel in Surfaces, enable the channel, then click the Texture chooser button and pick one.



When loading a new image texture, browse a tree view of available textures plus previews, including info on their size in pixels.



Related Topics:

[Shaders List \(Cineware Surfaces\)](#)

[Using Layers to Combine Shaders \(Cineware Surfaces\)](#)

[Texture files - naming conventions](#)

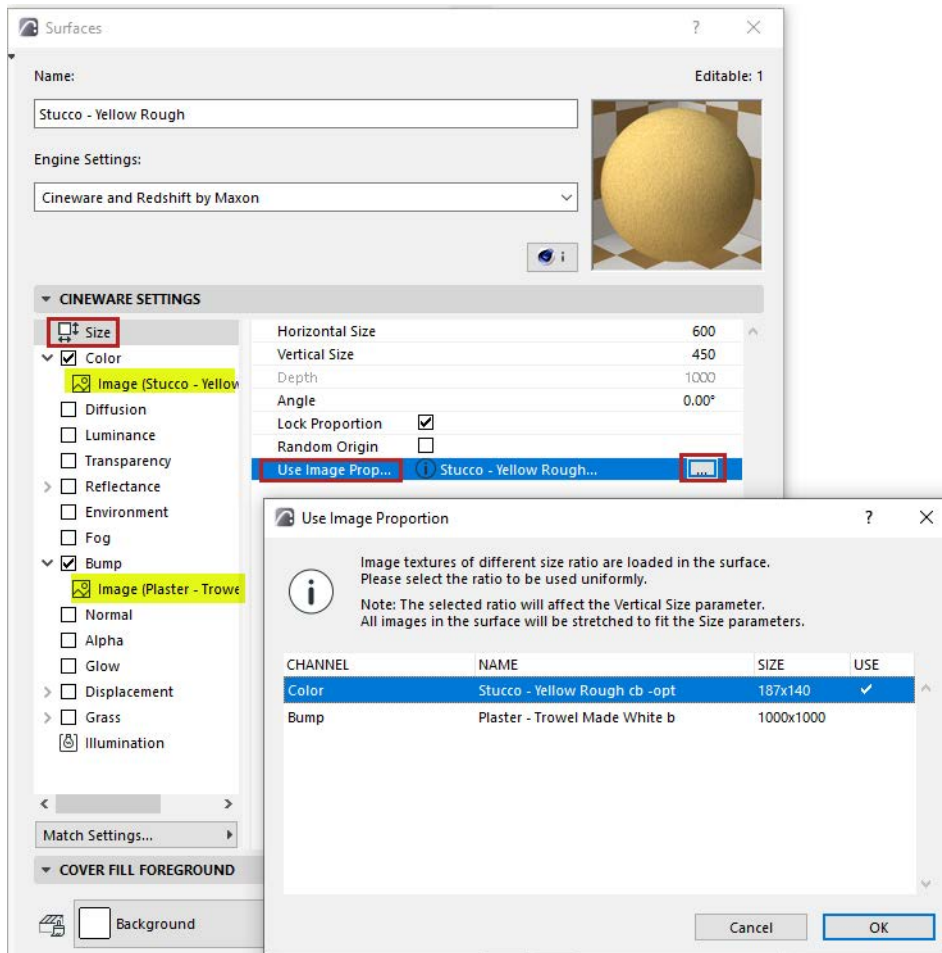
Special Settings for Image Textures (Cineware Surfaces)

[Watch the Video](#)

Special Settings for Image Textures (Cineware Surfaces)

Ensure Size/Ratio Compatibility When Using Multiple Images

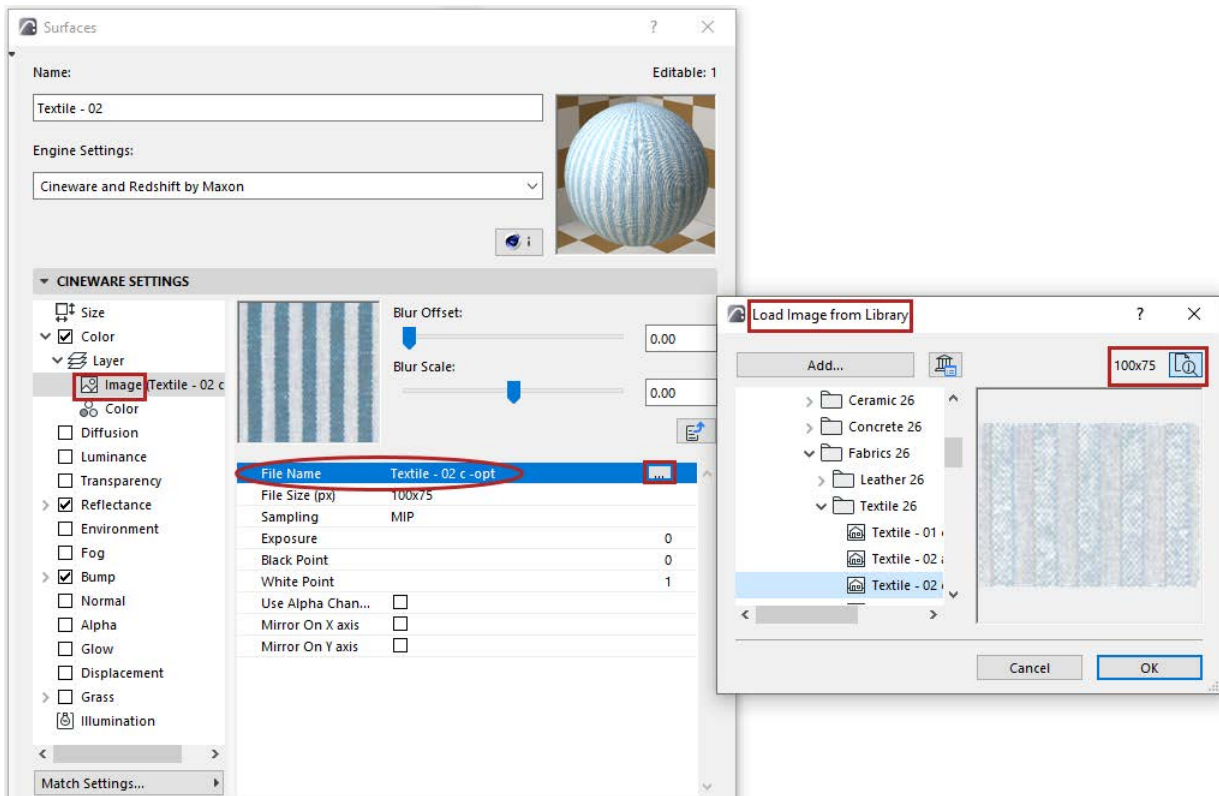
- If the Surface channels use multiple images textures of varying sizes, you get immediate feedback and the chance to select a uniform texture size for all of them



- Similarly, if the image ratios vary, you are prompted to select a uniform texture ratio. All image texture will be stretched to match this ratio. (This does not affect procedurals.) Check the "lock proportions" to maintain those proportions when further adjusting the Horizontal and Vertical size fields.

Image Parameters

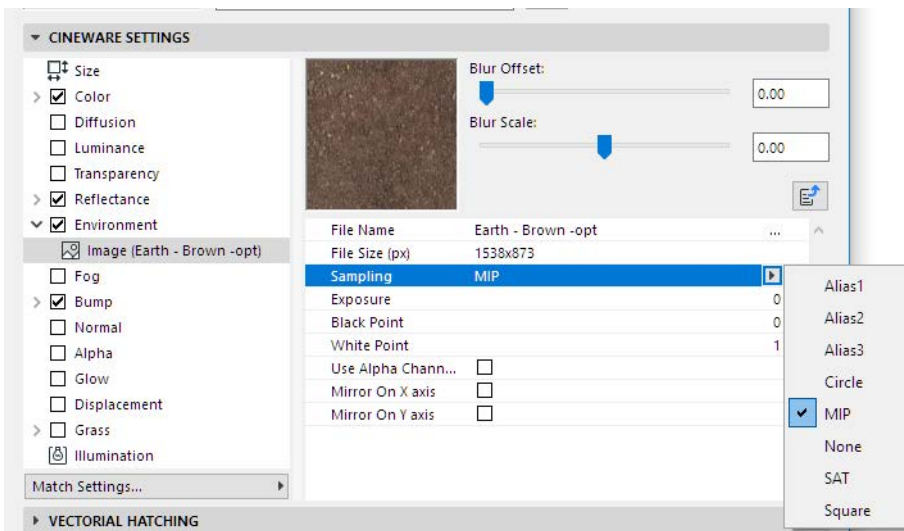
If you have loaded an image as your texture, the following parameters can be set.



Sampling Methods

Use **Sampling** to control how the texture’s pixels are interpolated.

The actual pixel resolution of an image is also scaled during rendering, which depends how close the object to which the surface is assigned will be to the camera when rendered. The options in the **Sampling** control define which algorithm should be used for interpolation.



- **MIP:** Default Sampling Method: **MIP** stands for multum in parvo, which is Latin for many things in a small place. MIP is the default sampling type, and generally results in very smooth blending.
- **SAT:** Highest Quality Sampling Method: The **SAT** option is similar to the MIP option, but more exact. However, the SAT mapping method also requires about twelve times more memory and more time for its

calculations. Furthermore, this method can only be applied to textures with a maximum size of 4000 x 4000 pixels. Therefore, MIP mapping should be used as the default mapping method and SAT mapping should only be used in instances where higher quality renderings are needed.

As a rule of thumb, use MIP mapping initially and switch over to SAT mapping if required.

- MIP and SAT are the best choices for objects in still pictures that extend towards the horizon, such as floors.
- MIP and SAT mapping affect the render time. On the one hand, MIP and SAT mapping take slightly longer to render. On the other hand, you may be able to reduce the antialiasing setting.
- The downside to MIP and SAT mapping is that they require extra memory. MIP mapping needs an extra byte of memory per texture pixel, SAT mapping requires an extra 12 bytes per texture pixel.

Other Sampling Methods

- **None:** When Sampling is set to None, the original texture values are used without interpolation. This method is fast, but often produces poor results and in general is best avoided. Textures tend to look jagged with None interpolation. You can try to reduce the jags by using a high antialiasing setting.
- **Circle** sampling uses a circle of texture pixels (the ones surrounding the intermediate value). Textures that are enlarged at render time tend to look more natural with this type than with the None type. However, straight lines are problematic and tend to look frayed. In addition, the texture is very jagged near the horizon. Circle sampling is, however, a good choice for very small textures (e.g. 3 x 3 pixels), because it helps the pixels to blend softly.

Issues can, however, occur on objects that reach deep into 3D space. For example, a surface at the end of a horizon can end up looking very turbulent.

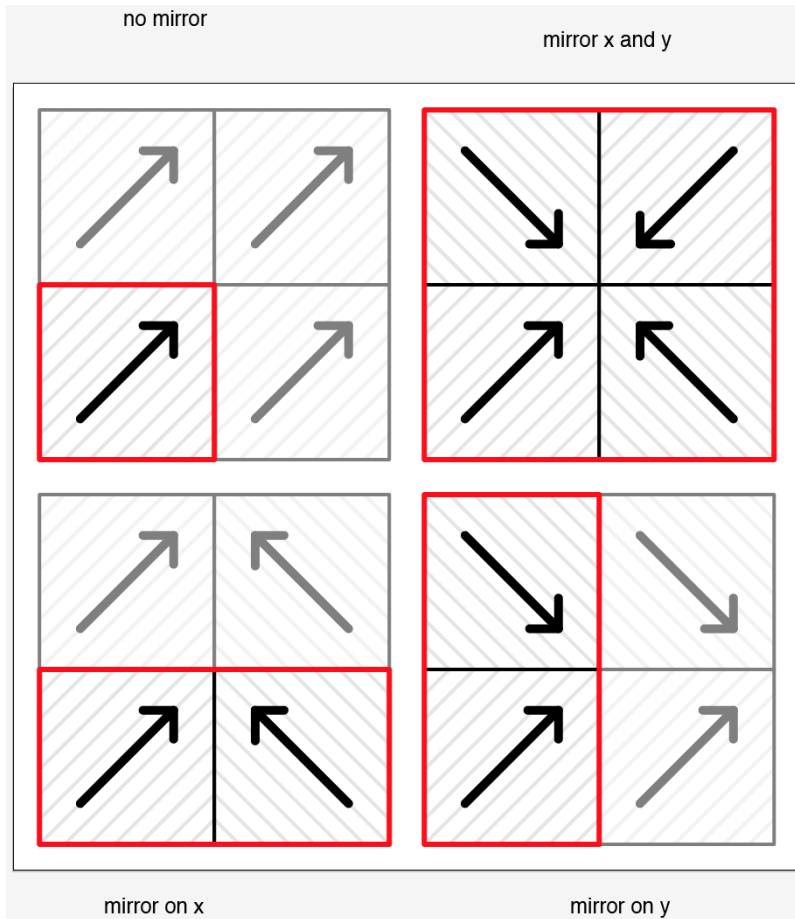
- **Square** uses a square of texture pixels (the ones surrounding the intermediate value). This leads to a softer transition between texture pixels than with None. The picture quality is good.
- **Alias 1, Alias 2** and **Alias 3** blend the texture more strongly than Circle and Square. Alias 3 blends the most, Alias 1 the least. Alias3 can give smoother results than Alias 1, but it also takes longer to calculate. However, even Alias 3 cannot prevent jags near the horizon.

Use Alpha Channel Only

An Alpha Channel is a transparency channel for a given pixel, rendering it black or white or a shade of gray in between; no RGB color channel is used. If you use Alpha Channel only, your image will be rendered in shades of gray.

Mirror on X axis/Mirror on Y axis

Apply a mirror effect to the image.



Exposure

Use this setting to adjust the brightness of HDRI as well as normal images.

Black Point/White Point

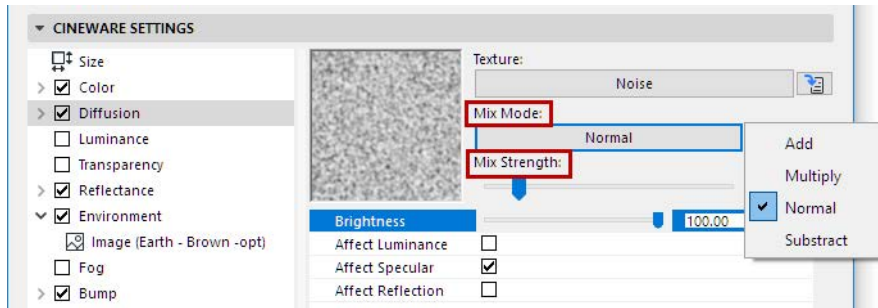
These points define the darkest and brightest points, respectively, and save this information to a histogram. If one of these points is subsequently moved, the dark and light levels will be scaled accordingly.

Mix Mode and Mix Strength (Cineware Surfaces)

Loaded textures can also interact with defined color values, e.g., to colorize a grayscale image or to affect the brightness of an image or shader.

The texture is placed on top of the color.

Their interaction is created and adjusted via the Mix Mode and Mix Strength parameters, below the channel's Texture chooser.



Use these parameters to **mix the color and textures** using one of four modes.

The default mode for most channels is Normal. Not all channels have Mix settings.

- **Normal:** In normal mode, the Mix Strength sets the opacity of the texture.

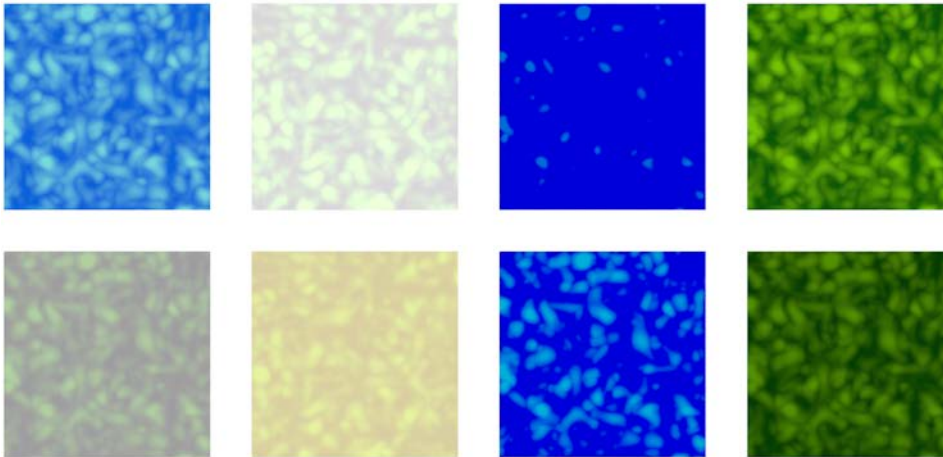
For example:

- If Mix Strength is set to 100%, you will see the texture only (which is on top).
- If Mix Strength is set to 70%, the result is 70% of the texture and 30% of the color. For example, if a texture pixel of RGB 255/0/0 (red) is used with a color value of RGB 255/255/0 (yellow) with the Mix Strength set to 50%, the resultant color is 255/128/0 (orange).

The remaining options, Add, Subtract and Multiply, mix the texture and color according to the defined Mix Strength.

- **Add:** In this mode, the texture's RGB value is added to the color's RGB value. Color channel values cannot exceed the maximum of RGB 255. So if a texture pixel of RGB 0/255/255 (cyan) is added to a color value of 255/255/0 (yellow), the result is 255/255/255 (white).
- **Subtract:** The color's RGB value is subtracted from the texture's RGB value. Thus if a texture pixel is RGB 255/255/255 (white) and the color value is 255/0/0 (red), subtracting with Mix Strength set to 100% gives the result 0/255/255 (cyan).

- **Multiply:** The RGB value of the texture is multiplied by the RGB value of the color. For example RGB 255/128/0 (orange) multiplied by RGB 0/255/0 (green) results in RGB 0/128/0 (dark green).



Mix Mode: From left to right: Normal, Add, Subtract, Multiply
 Mix Strength: Top row = 100%, bottom row = 50%

Mix Strength

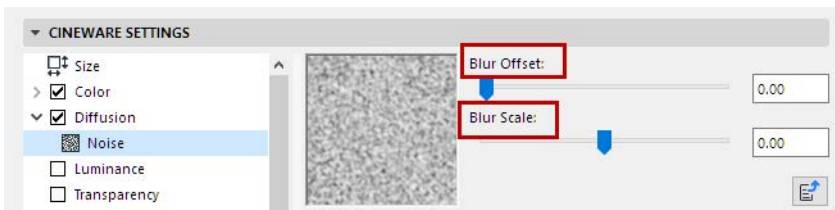
Mix Strength defines the mixing proportion between the texture and color or between the brightness and transparency (depending on which mode is selected) of the texture to be mixed.

Texture Blur Settings

The **Blur Offset** and **Blur Scale** options enable you to blur or sharpen the texture.

- Blur Offset softens a texture.
- Blur Scale: a positive value increases the blur; a negative value weakens it.

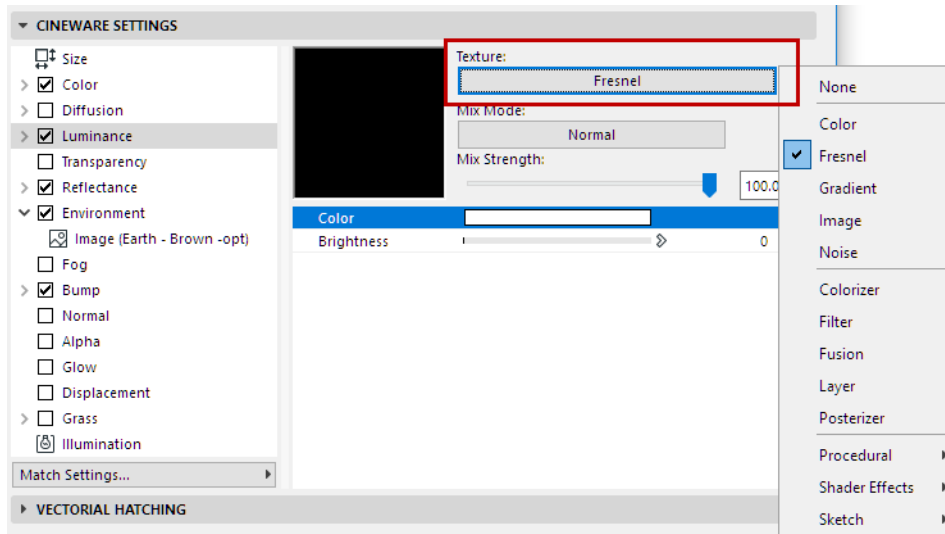
With floors, try a positive blur scale of about +20%. Floors tend to suffer most from perspective distortion, so they require special treatment.



From left to right: blur offset: 0 / 25 / 50 / 75 / 100

Shaders List (Cineware Surfaces)

To choose a shader for a particular channel in Surfaces, enable the channel, then click the Texture chooser button and pick one.



These shaders are tiny applications, most of which offer their own specific settings for modification.

In general, shaders are designed for a specific purpose, e.g. to depict a gradient or surface structure (wood, marble, etc.). Since shaders are made up of color information they can also be multiplied with other color or brightness values via the Mix Mode.

See [Mix Mode and Mix Strength \(Cineware Surfaces\)](#).

Once a shader has been selected, its settings can be accessed by clicking either on its name in the Texture setting or on its preview image below its name.

For a particular Surface channel, you can combine several shaders/textures in a layer, to achieve more nuanced effects.

See [Using Layers to Combine Shaders \(Cineware Surfaces\)](#).

The following is a list of shaders and shader groups with a brief explanation of their parameters:

Color

Gradient

Fresnel

Noise

Colorizer

Filter

Fusion

Layer

Posterizer

Procedural Shaders (Cineware Surfaces)

Shader Effects (Cineware Surfaces)

Sketch Shaders (Cineware Surfaces)

Additional Effects for Shader Layers/Folders (Cineware Surfaces)

Color

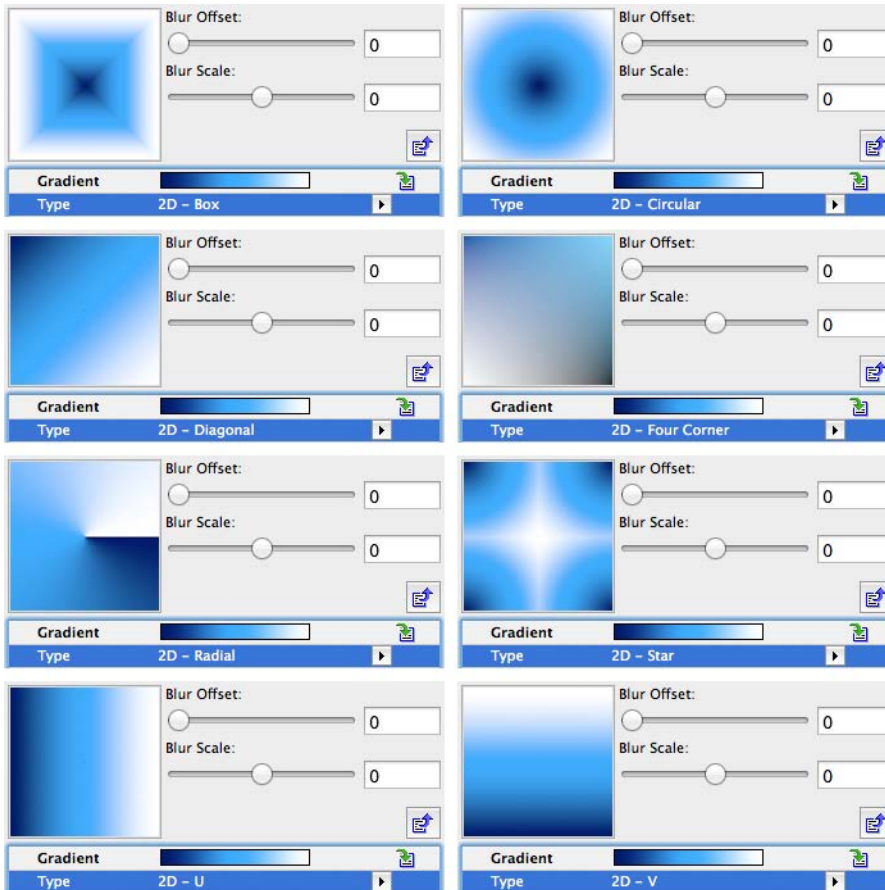
Define a color using the color chooser.

The simplest of all shaders is the Color shader. Only the basic color fields are available for modification. This shader can of course be used within a Layer or Fusion shader in combination with other shaders or textures.

Gradient

Use the Gradient shader to create custom color transitions in different directions and with different patterns.

- Use the **Type** pop-up to choose among several standard patterns.
- Most of the gradients can be rotated using the **Angle** value.



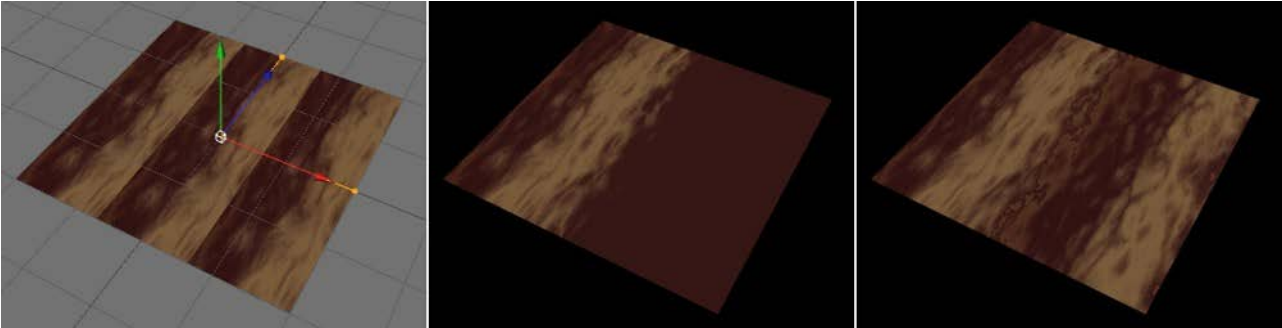
- **Turbulence**: This setting distorts the transition between colors. Setting the Turbulence value to greater than 0% will activate the Octaves, Scale and Frequency options below
 - The Octaves value defines the detail depth of the Turbulence effect.
 - The Scale value defines the density of the effect.

The larger these values are, the more distortion will be created within the gradient.

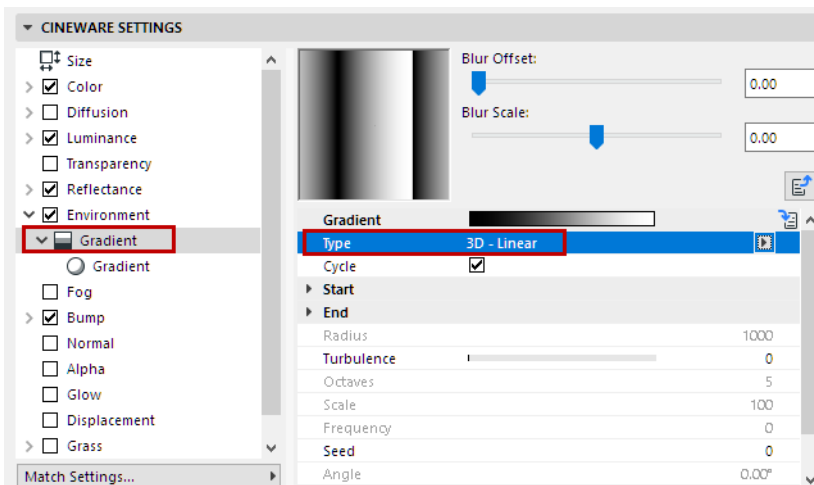
- The **Frequency** value can be used when animating the gradient surface. The value entered here defines the speed with which the turbulence will be modified.
- Modifying the **Seed** value will result in the turbulence being recalculated. This lets multiple Gradient shaders be used in a single surface without generating identical results.

Normally a gradient is limited to the area of a texture tile, i.e. the area displayed in the preview image. If the surface with the gradient should be used on an object multiple times, the **Cycle** option should be enabled. This will cause the surface to continue around the object.

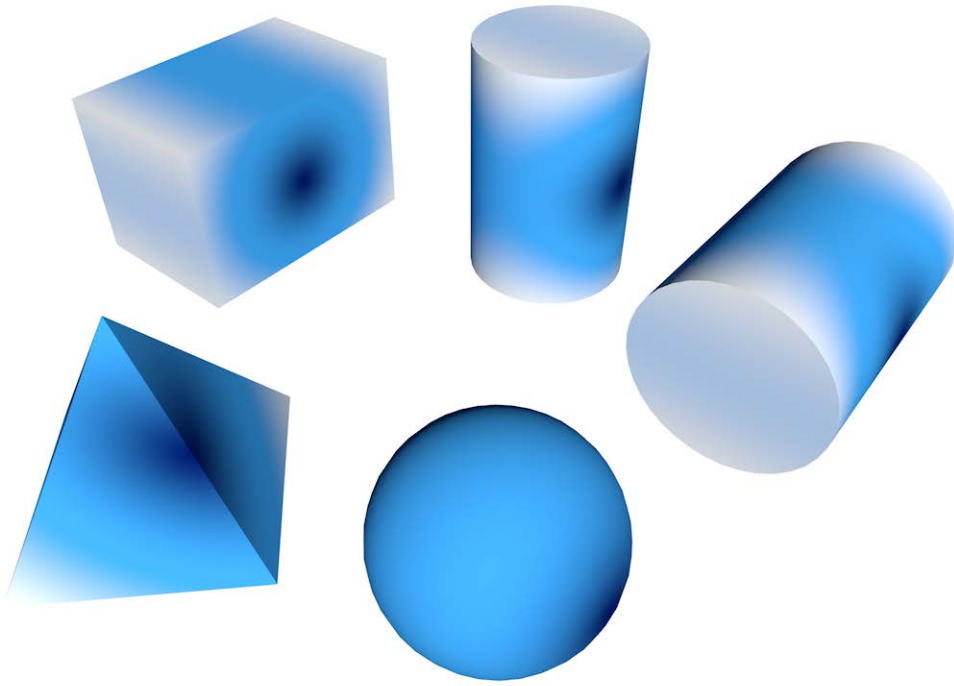
An example of this is shown in the image below. The example at the left shows a surface with a Gradient shader assigned to a simple Plane object. Since the gradient is restricted to the size of a single tile the gradient is repeated across the surface of the object. When the object is rendered (center), only a single gradient is shown that has simply been extended to the edge of the object. Activating the Cycle option will remedy this and the Gradient shader will cover the Plane object entirely (right).



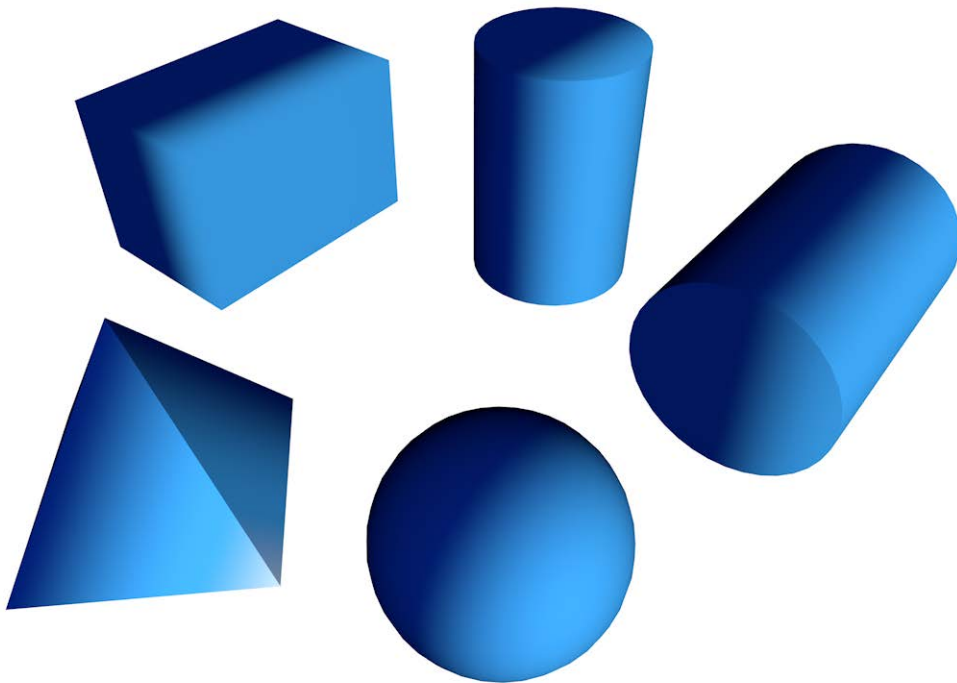
In addition to the normal 2D gradients, 3D gradients are also available. 3D gradients cover an area in 3D space along a given axis and can be used to cover a 3D object.



- The coordinate system used is defined by the **Space** setting
- **Cycle**: Sometimes the needed texture map can become much larger than the original texture. Enable Cycle to tile the shader.
- For spherical or cylindrical gradients, the **Radius** value is used to define the volume. The **Start** and **End** values define the position and orientation of the 3D gradient.



3D Gradient, of cylindrical type (cycle OFF)



3D Gradient, of linear type (cycle OFF)

Fresnel

This shader can make numerous types of surfaces, especially glass, car finish, etc.) look more realistic. The greater the angle at which you view a surface, the more reflective any surface becomes.



Left: a Fresnel shader in the Color Channel.

Right: a Fresnel shader in the Transparency Channel.

The Fresnel shader orients itself according to the angle of the surface to the viewer. The color from the left end of the gradient will dominate if the surface is viewed at an angle; the color from the right end of the gradient will dominate if the surface is viewed frontally.

This behavior reflects natural phenomena. A storefront window, for example, is barely noticeable when viewed frontally but looks like a mirrored surface when viewed from an angle.

Although the Fresnel shader can be used in any surface channel (the Reflectance channel is preferred), it does not affect the Bump or Displacement channels unless used as a secondary channel

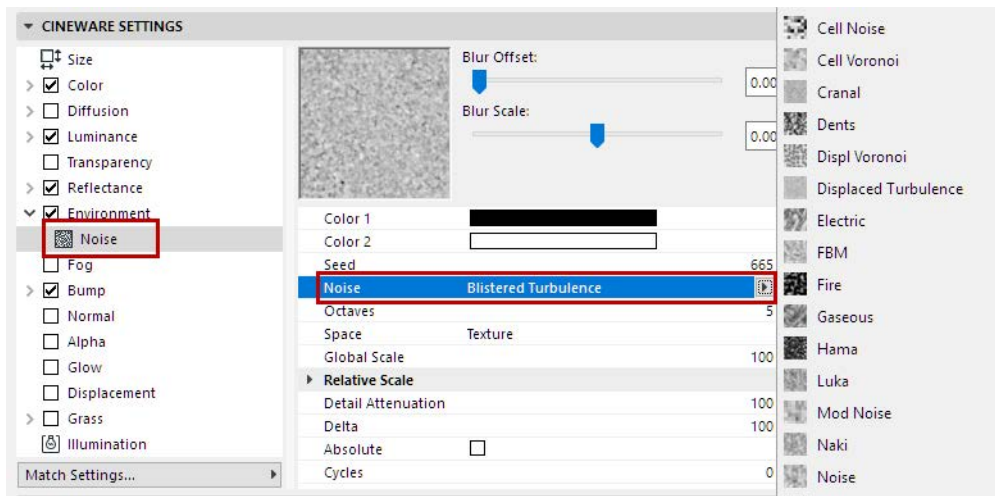
- **Use Bump:** To make the shader affect not only geometry but the shadows on the surface as well, enable the Use Bump option. Bump refers to all surface characteristics that affect the Normals and thereby the object surface, i.e. the Bump and Normal channels.

Enable the **Physical** option to access additional options (IOR- Index of Refraction, Presets, and Invert) for fine-tuning the fresnel shader:

Noise

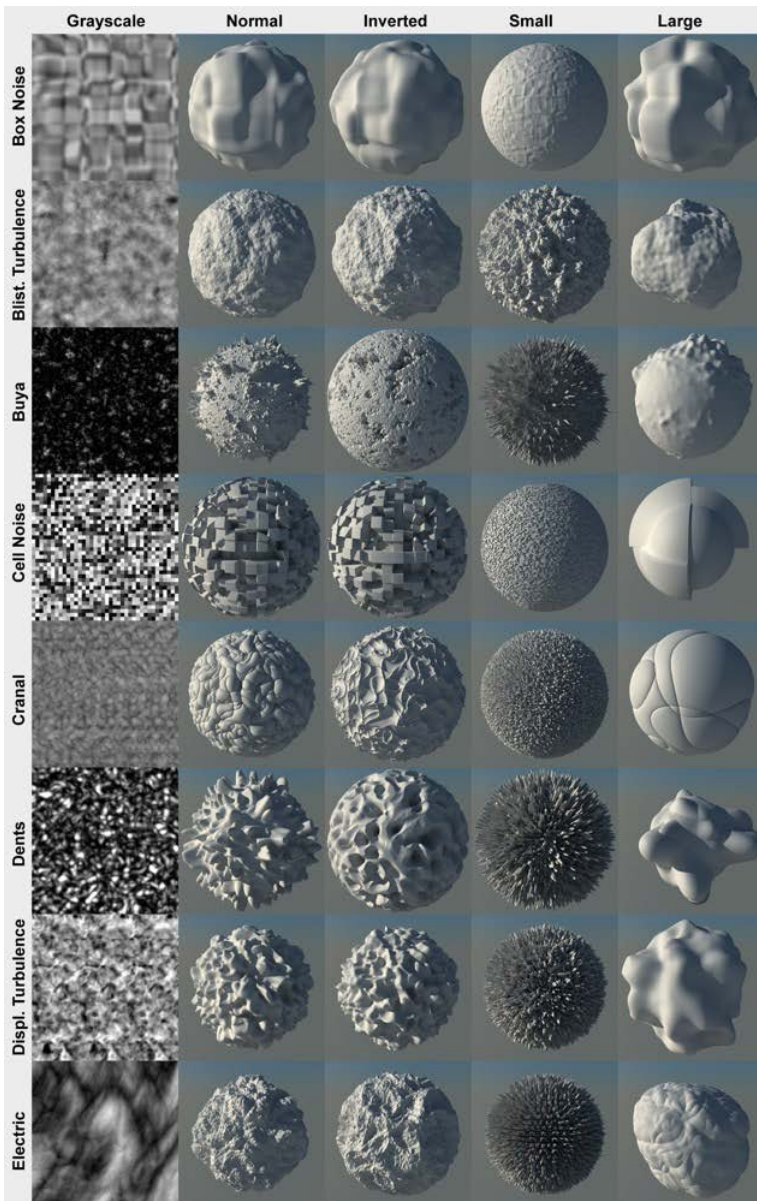
This shader creates a random pattern that can be used, for example, for sun surfaces and stone reliefs. The advantage of 3D noise is that you don't have to worry about mapping or seams in the texture.

Noise is an integral component of any render system because it turns an otherwise far too smooth surface into a more or less randomly uneven surface (e.g. when used in the Bump channel).



Below are examples of some Noise types available in the Noise Shader. Their respective effects are shown using Sub-Polygon Displacement ([see *Sub-Polygon Displacement*, in the *Displacement* channel](#)).

On the left is the Noise Shader as a normal grayscale image; on the right is the same Noise Shader on a sphere, with different Noise scales (Global Scale parameter).



Colorizer

Colorizer takes the value of an input channel and remaps it based on a gradient. This is very useful for coloring shaders such as noise and for coloring grayscale gradients.

- **Input:** Use the Input pop-up to select which of the texture's characteristics should be used when calculating the gradient. The Gradient will then be applied from left to right in accordance with the texture's input intensity.

If, for example, Input is set to Luminance, and the Colorizer contains a black-to-white gradient, all black areas of the texture will be assigned the colors at the left of the gradient. White areas will be assigned the color from the right side of the gradient.

- **Cycle:** If Cycle is enable, then a texture that contains Brightness or Intensity values greater than 100% will be re-calculated to a range of between 0% and 100%.

You can also use a Colorize effect applied to shader layers or folders.

See [Additional Effects for Shader Layers/Folders \(Cineware Surfaces\)](#).

Filter

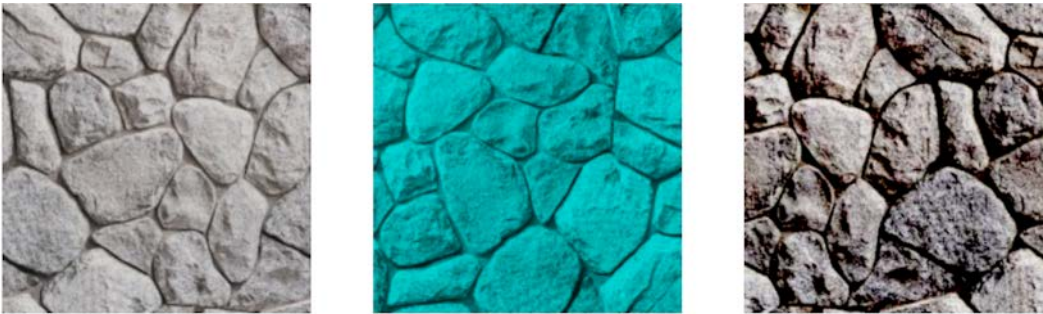
The Filter shader filters another shader or a bitmap.

Use the Filter shader to adjust a shader or bitmap's brightness, contrast, hue, and saturation. You can also clip the color (to reduce the color range and increase the contrast).

The Filter shader is especially useful for adjusting HDR1 textures.

You can also use a Filter effect applied to shader layers or folders.

See [Additional Effects for Shader Layers/Folders \(Cineware Surfaces\)](#).



left = original

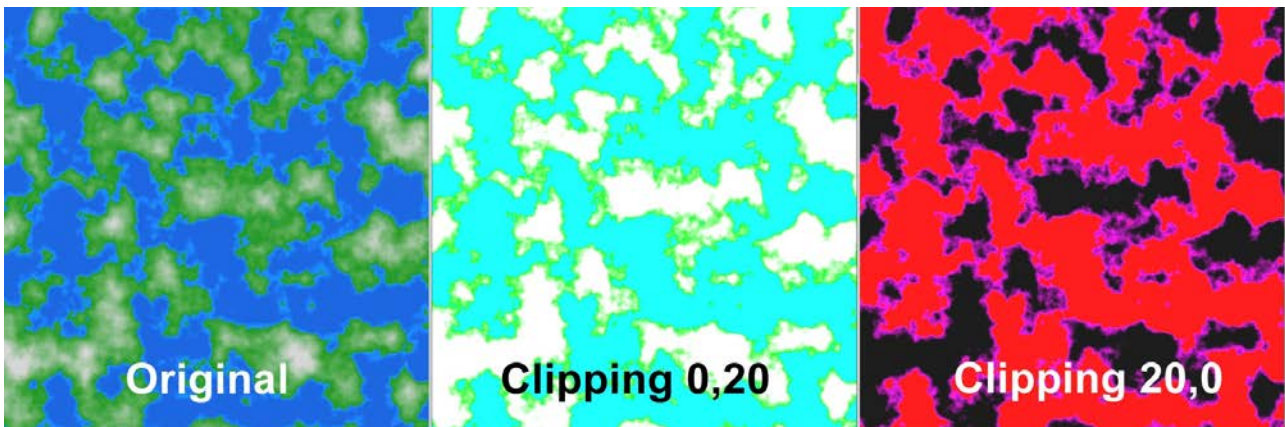
middle = hue edit, colorize on, and increased saturation

right = brightness and contrast adjusting

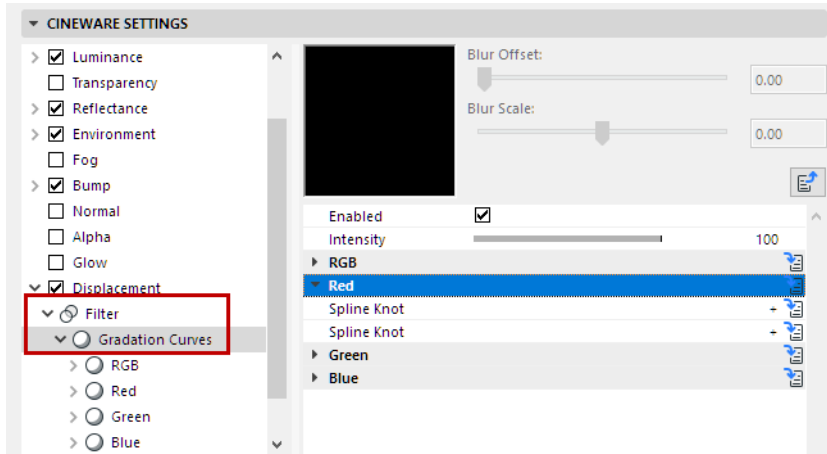
Low Clip/High Clip

These parameters clip or cut colors on two borders. The fastest way to understand how the parameters work is simply to try them.

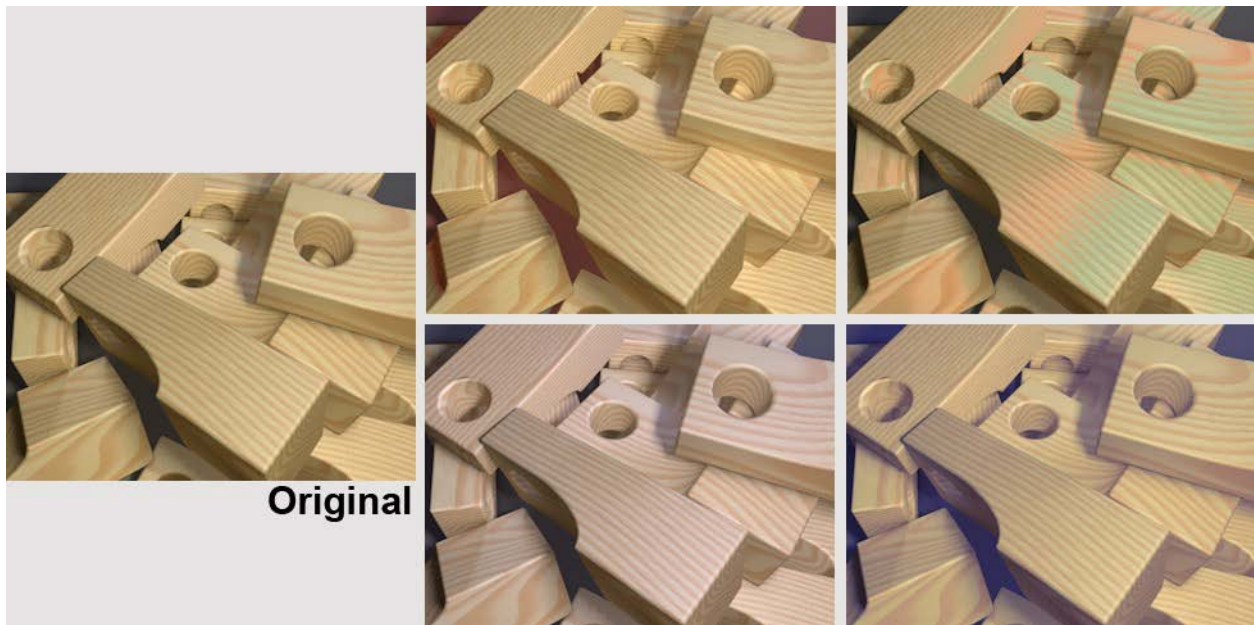
Inverting the **Low Clip** and **High Clip** values can result in the texture being inverted.



Gradation Curves



Gradation curves can be used to precisely modify brightness and color regions of the original image. For example, brighten only the dark regions of an image; or brighter regions can be colored.

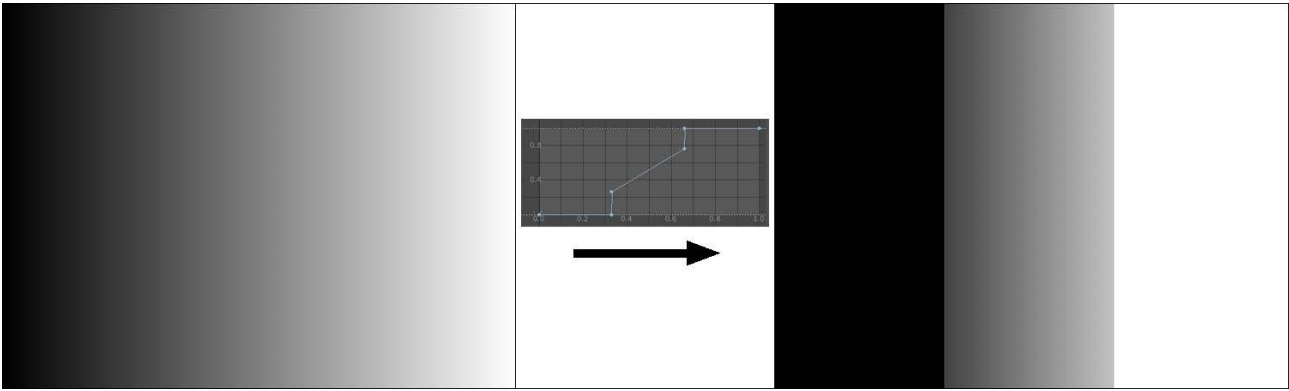


Gradation curves can be used to brighten, darken or color specific brightness regions

Use this curve (in addition to the RGB curves) to define the overall brightness.

How the Curve Works

A gradation curve defines where a new brightness value will be assigned along the X axis. The curve applies a brightness value of 0 to the first third, i.e., black. The second third has a linear gradient that assigns proportional brightness values to the region. The last third has a maximum value, which means it appears white.



The gradient curve is applied to the linear grayscale at the left. The result is shown on the right.

Intensity

Use this setting to define the strength of the Gradation Curve's effect on the shader. A value of 0% will result in no effect, a value of 100% produces the maximum effect.

RGB

Use this curve to define the overall brightness.

This can also be done for individual color channels using the separate **red, green and blue** gradient curves. This makes it possible to color dark or bright regions only:



Dark regions in the center were colored red, and bright regions at the right were colored blue

Fusion

Fusion allows you to combine two textures with a mask using the blending modes standard in photo editing applications, as if they were layers. However, you can use the Layer shader (see below) to achieve the same effect.

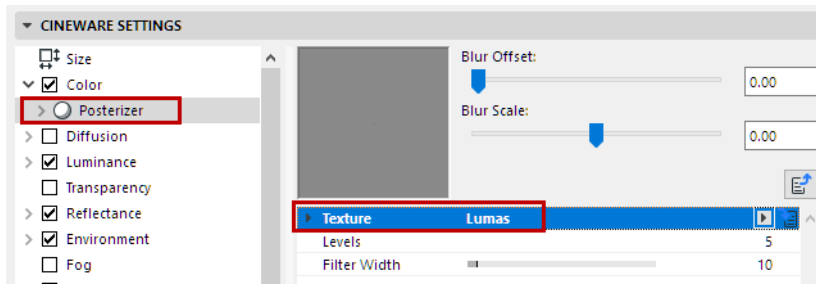
Layer

[See *Using Layers to Combine Shaders \(Cineware Surfaces\)*.](#)

Posterizer

Interesting effects can be achieved by allowing only certain brightness or color characteristics of a given texture to be shown. For example, any number of brightness bands can be arrayed next to one another. The number of color or brightness bands is defined by the Levels value. The Filter Width value serves to soften any hard edges by blurring them slightly.

The Posterizer shader can be used to remove unwanted color transitions or to create a comic look. In the example below, a Lumas shader in the Posterizer shader was applied to achieve the comic look.



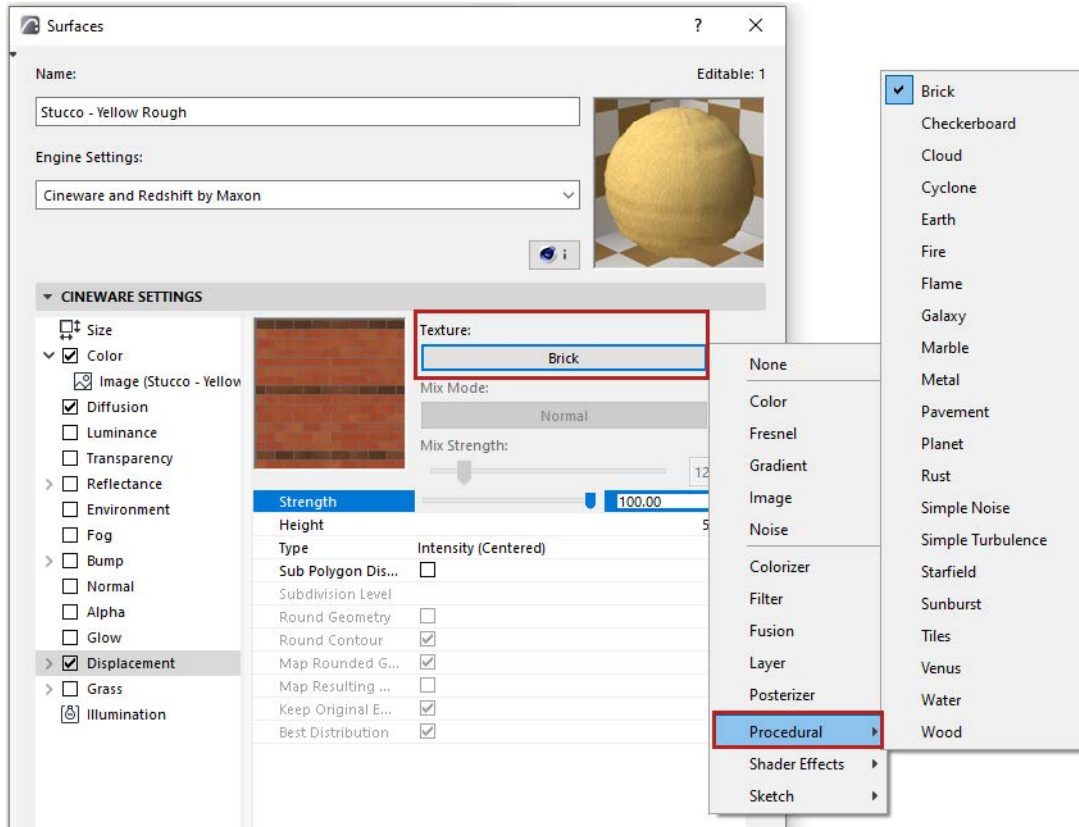
You can also use a Posterize effect applied to shader layers or folders.

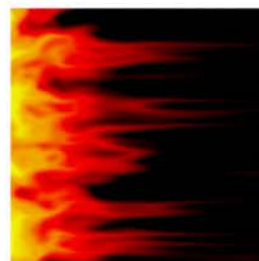
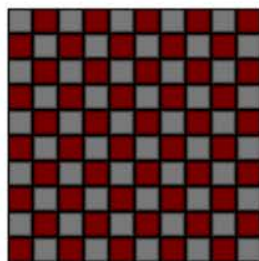
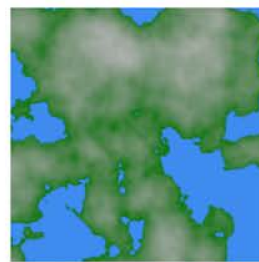
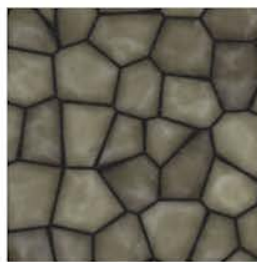
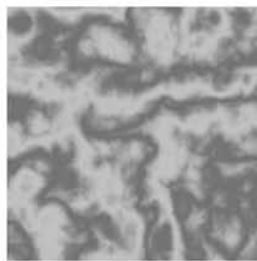
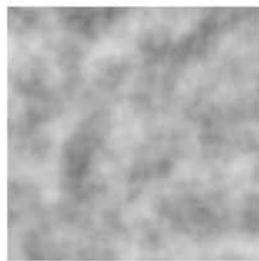
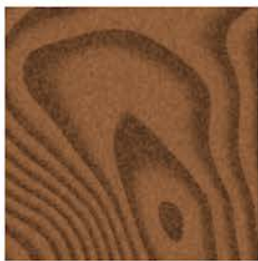
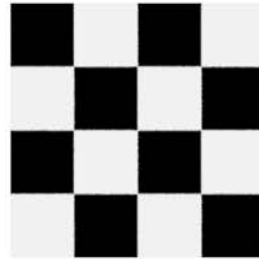
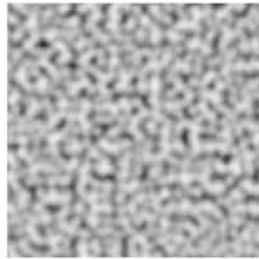
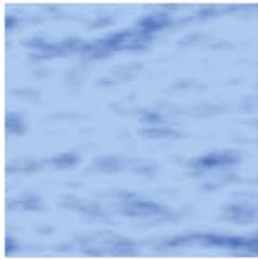
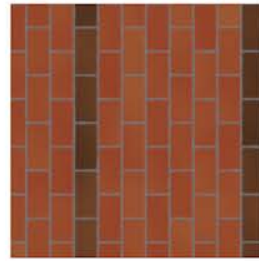
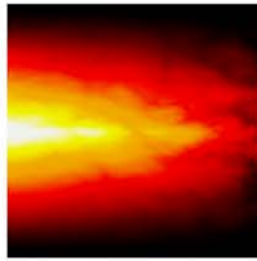
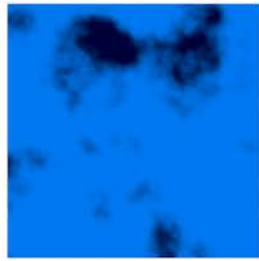
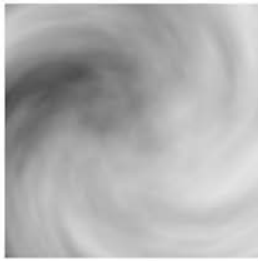
See [Additional Effects for Shader Layers/Folders \(Cineware Surfaces\)](#).

Procedural Shaders (Cineware Surfaces)

A procedural texture is a computer-generated image created using an algorithm intended to create a realistic representation of natural elements such as wood, water, or marble.

Usually, the natural look of the rendered result is achieved by the usage of fractal noise and turbulence functions. These functions are used as a numerical representation of the “randomness” found in nature.

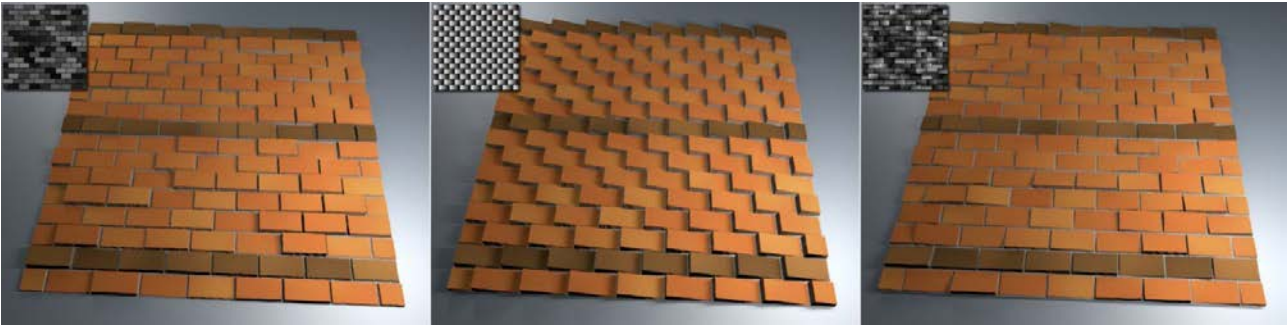




Specific settings are available for each procedural shader.

Brick Shader Displacement Mode

The Brick shader's parameters include a Displacement Mode for achieving more realistic brick walls.



Various Displacement Settings

The Displacement settings have an effect when the shader is used in the Displacement channel (or to a lesser degree in the Bump channel). The Displacement channel generates the necessary grayscale values to achieve realistic-looking results.

The Displacement Mode can be used in combination with the Colorizer-Shader to generate random colors in brick shape. The Brick shader should have similar values to the Color channel with regard to brick size and grout.

Height Variation

Increase this value if the height of the bricks should vary randomly. A value of 0% will produce a perfectly level surface with only the grout lying deeper than the brick surface. The higher the value, the more varied the bricks' heights will be.

Slope Intensity

If the bricks are too uniform with regard to their perpendicular height from the wall, this value can be used to add variation. The brick side defined in Slope Direction will then be raised. A value of 0% will produce a brick surface absolutely perpendicular to the wall. Increasing values will add correspondingly more variation.

Slope Direction

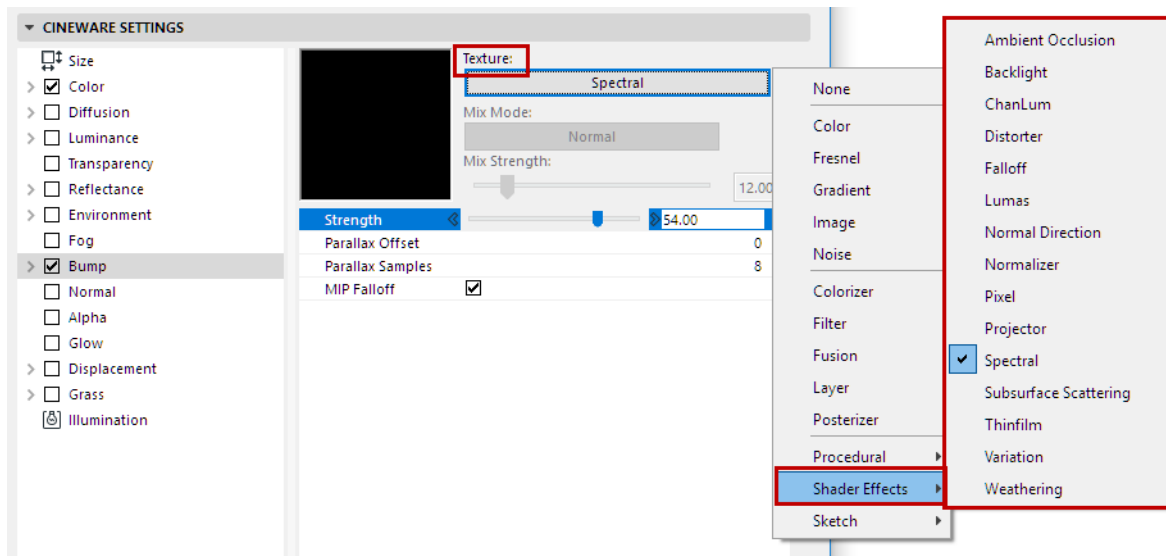
Use this setting to define the brick side that should be raised. If set to Random, a random edge or corner will be raised on each brick.

Seed

This setting can be used to add random distribution of the random values (Height Variation, Slope Direction).

Shader Effects (Cineware Surfaces)

This group of shaders is found in the Shader Effects sub-menu of the Surface channel's Texture pop-up:



These shaders are listed below with a brief explanation of their parameters.

[Ambient Occlusion \(Cineware Surface Channel\)](#)

[ChanLum, Subsurface Scattering, Backlight](#)

[Distorter](#)

[Falloff](#)

[Lumas](#)

[Normal Direction](#)

[Normalizer](#)

[Pixel](#)

[Projector](#)

[Spectral](#)

[Thinfilm](#)

[Variation](#)

[Weathering](#)

Ambient Occlusion (Cineware Surface Channel)

Ambient Occlusion for Cineware can be found in two locations in the interface:

- In Surfaces, as a channel shader (see screenshot above).
- In PhotoRender Settings (Detailed view), as a global effect and calculated throughout the entire scene in Render Settings.

See [Ambient Occlusion \(Cineware Effect\)](#).

AO options in both locations are identical.

ChanLum, Subsurface Scattering, Backlight

These shaders all have to do with collecting light. Some surfaces have the ability to direct light inward, where it is dispersed (e.g. a wax candle or frosted glass).

Normally, light can only shine directly onto the surface of an object and cannot, for example, be transmitted further from the illuminated back side of an object to its front that lies in a shadowed region. Several shaders are available to help remedy this in their own way.

These shaders should be loaded into the Luminance channel. In this channel, the brightness these effects generate can be used as an illumination effect or added to the surface's brightness.

ChanLum

From a defined distance from the object surface outward, the ChanLum shader observes the environment surrounding the object and samples light rays. This light is then used to brighten the surface of the object. This may not be realistic according to the world of physics but is often sufficiently accurate to simulate light that does not penetrate too deeply into a given surface.

Subsurface Scattering

This shader causes light to penetrate a surface where the light is dispersed. If the distance traveled by the light within the object is less than the defined value, the light can emerge again at a different location. This shader is well-suited for use on backlit objects.

Here, a candle that is simply illuminated by a light from above, which is used to simulate the flame. This light source is only able to illuminate the surfaces at the top end of the candle.



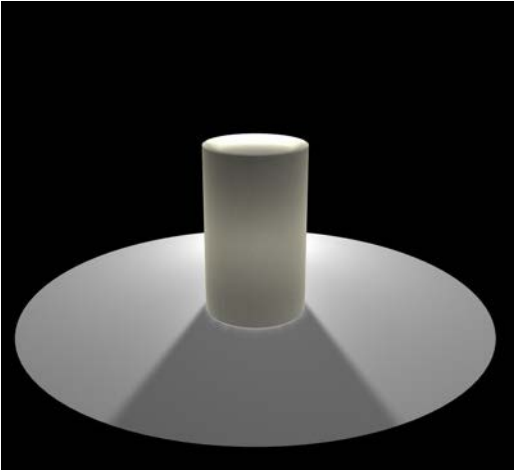
Here is the same scene with a ChanLum shader added in the wax surface's Luminance channel. You can see how the light now continues around the top edge of the candle to also affect the sides. It also looks as if the candlelight were seeping through the edge a little.



Here, a Subsurface Scattering shader was also added to the Luminance channel. The light now affects a large part of the top half of the candle but does not accentuate the edge very well. This type of lighting scenario is less suited for Subsurface Scattering because of the small angle that lies between the angle of view and the rays of light.

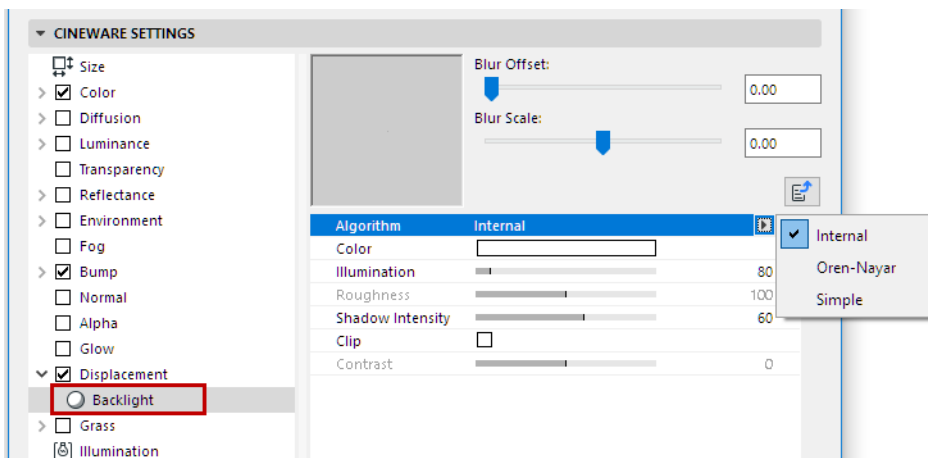


Here, the angle of view is increased by placing the light source behind the candle: now the Subsurface Scattering shader is well-suited: the light actually shines through the candle and the top edge is affected, as we wanted, thanks to the more intense light.



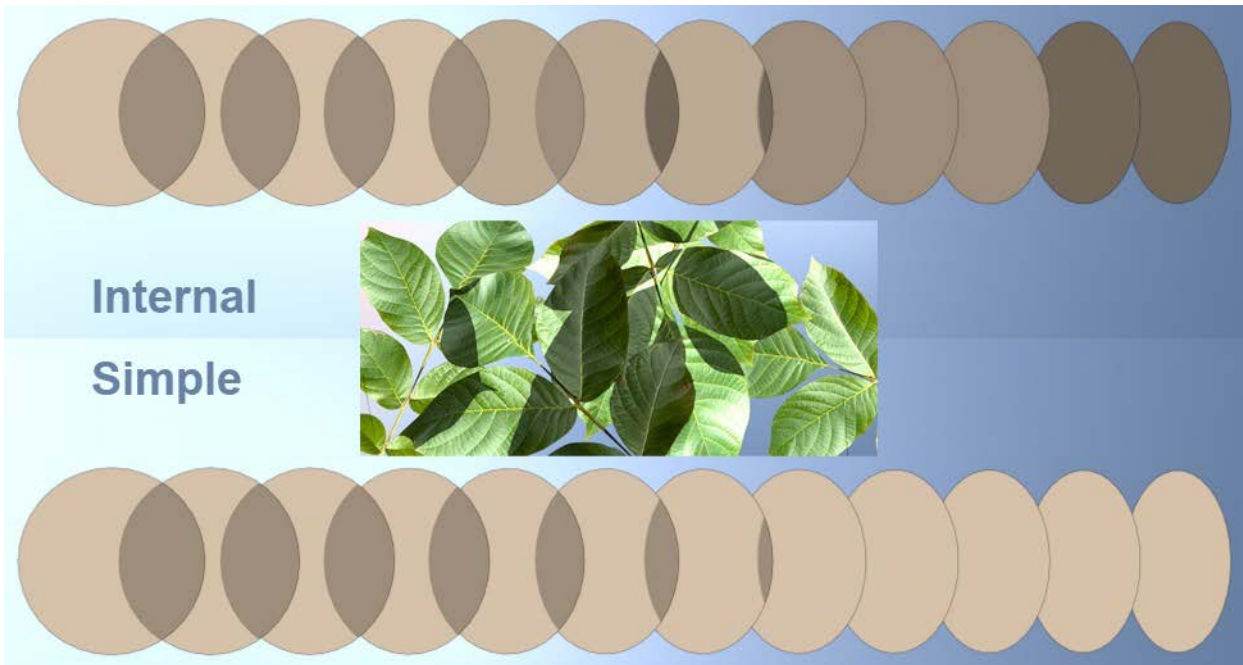
Backlight

Allows you to gather illumination from the back face of an object. This gives the ability to create translucency effects with backlighting, such as rice paper or a thin leaf lit from the opposite side complete with shadowing.



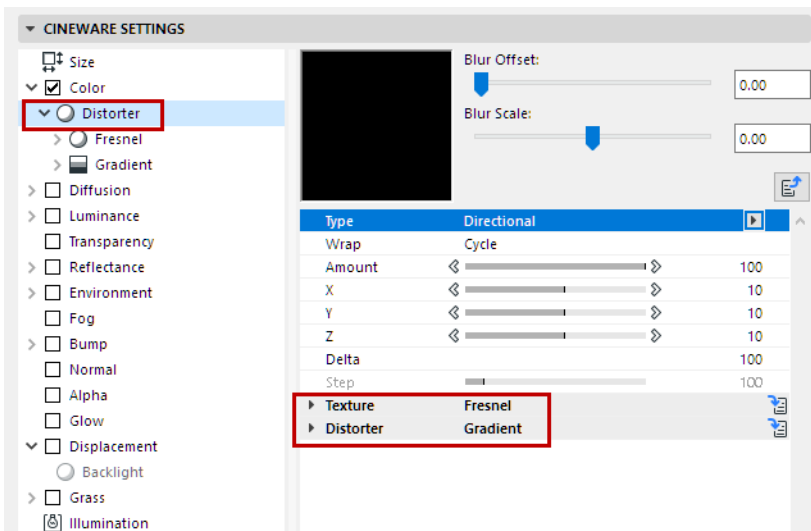
Algorithm: Use this setting to define the shading algorithm for the backlighting.

- **Internal** is equivalent to the common Phong shading.
- For **Oren Nayar**, see [Model](#).
- **Simple** generates no shading at all, so that backlit objects are rendered uniformly bright. This is particularly well suited for very thin surfaces such as leaves, blades of grass, etc.



Distorter

Distorter takes the value of an input channel and distorts it using the value from another texture channel.



Type: The distortion algorithm to be used.

- Directional: The value of the distortion channel is added to the sample coordinate of the texture.
- Bidirectional: The value of the distortion channel is added to the sample coordinate of the texture if the value is between 50-100% and is subtracted if the value is between 0-50%.
- Flow Field: The distortion value is determined by evaluating the flow direction of the distortion texture and then the flow direction vector is used to offset the sample coordinate of the texture.

Wrap: The Wrap function controls what happens to parts of the texture that have been distorted and now lie outside the UV map. These can be ignored (None), repeated (Cycle), cut off (Clamp) or mirrored (Seamless).

Amount: Global distortion amount. 100% means distortion values from 0 to 1 in UV and 0 to 10 in 3D.

X/Y/Z: X is the distortion amount in U for 2D and X for 3D. Y is the distortion amount in V for 2D and Y for 3D. Z is the distortion amount in Z for 3D.

Delta: Delta is a scaling factor to use in sampling the texture for evaluating slope to be used in the bump channel. This allows you to get very sharp bump with minute detail that is not possible with the standard bump delta.

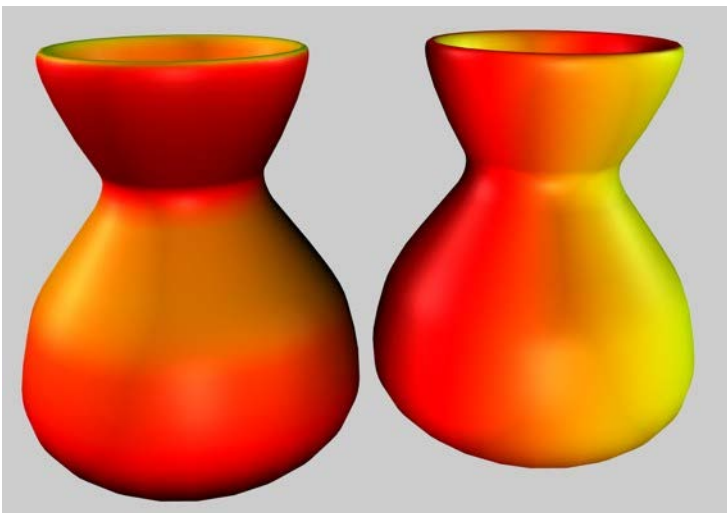
Step: Relative step size to be used in evaluating the flow direction in Flow Field distortion type.

Texture: This is the source image or shader to distort with the Distorter texture. You can bring in and affect any image or channel shader available to Cineware.

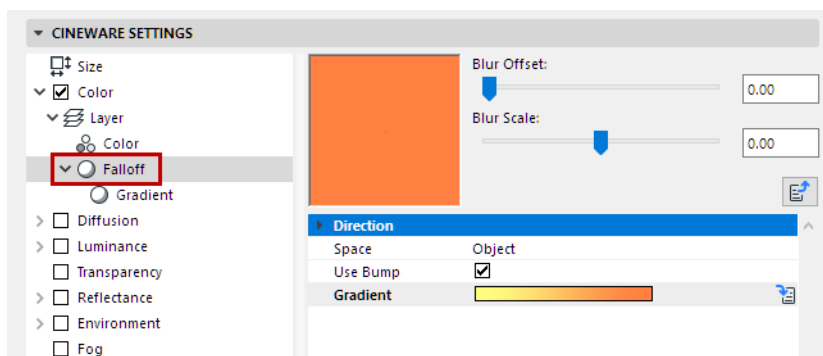
Distorter: The distorter texture is used to distort the texture defined by Texture (see above). The value of the distorter image or shader is used in the distortion algorithm to offset the sampling of the texture.

Falloff

Falloff calculates the falloff between a custom vector and the surface normal. When the vector is the same as the surface normal, the value is 1, and when it points completely away, the value is 0. Falloff then remaps the value using a gradient.



Same shader settings, only with different vectors



Direction [XYZ]: The vector to falloff from in 3D space.

Space: The space to calculate the falloff in.

- **Object:** The vector is specified in object coordinates and the falloff is affected by the object's orientation (and by the orientation of the texture axis). In this way, the falloff sticks to the object as it changes rotations.

- **World:** The vector is specified in world coordinates and is not affected by the object's orientation. This is the most common setting because it allows the object to move and have the falloff stay oriented in a consistent direction.
- **Camera:** The vector is specified in coordinates relative to the orientation of the camera. In this way, up is always up (in the camera view, for example) regardless of camera or object orientation.

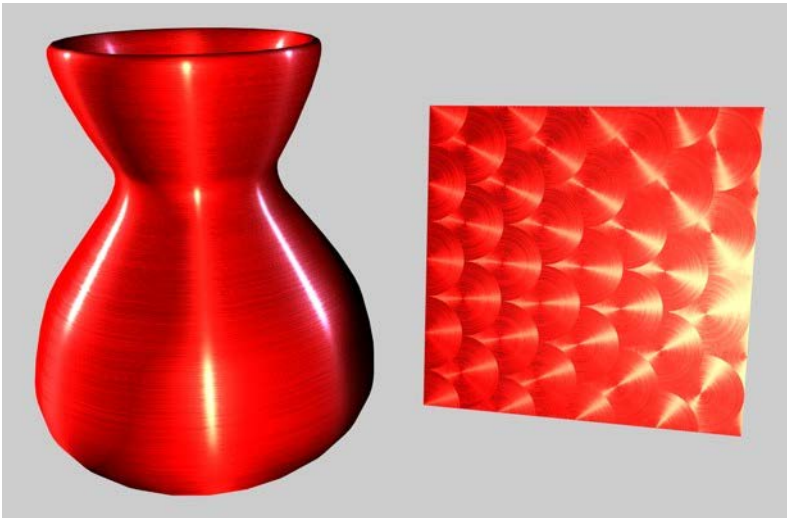
Use Bump: If enabled, the bump normal is used to calculate the falloff. If off then the bump normal is ignored.

Gradient: The gradient used by Falloff to remap the value.

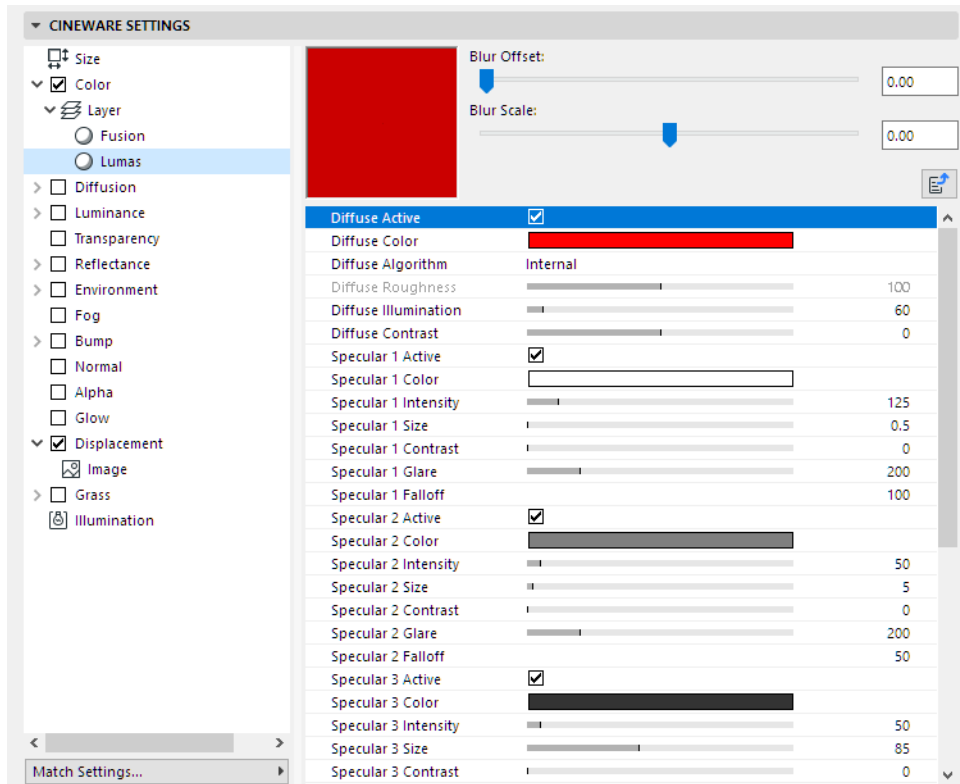
Lumas

Lumas is an illumination shader that includes three specular highlights and the ability to simulate the anisotropic scratches that tend to produce elongated highlights in particular.

An anisotropic effect is the mirroring of a surface, caused by small, inconspicuous scratches. You have probably seen this effect often on metal surfaces that are somewhat older in the form of concentric scratches on the surface



Lumas is useful in combination with Fusion and other channel shaders or images.



- Diffuse Active
- **Diffuse Color:** Sets the base diffuse color of the surface.
- **Diffuse Algorithm:** Specifies which illumination model Lumas will use.
 - **Internal** is the standard Lambertian model
 - **Oren Nayar**, which provides for a roughness variable allowing the user to go from simple surfaces (0 Roughness, identical to Lambertian), to complex rough surface (1+ Roughness, which provides a matte surface, such as linen or dirt).
- **Diffuse Roughness:** Dictates how rough the Oren Nayar illumination model will be. Zero is no roughness while 100% or greater is very rough. Experimentation is required to fully understand this parameter. This option is only available if Oren Nayar is selected in the Algorithm drop-down list.
- **Illumination:** Scales the Color to produce attenuation in the illumination of the surface (this simply means it makes the color darker when the value is close to zero and brighter when the value is closer to 100% or above).

If you are using the Oren Nayar illumination algorithm, you may want to bump up the illumination by about 10% to 20% because it renders darker usually. Appropriate values are between 0% to 200%.
- **Contrast:** Provides contrast for the result of the Color. Zero causes no effect, 0% - 100% causes standard contrast, above 100% causes a wrap contrast (the values that go above 100% wrap around to 0% again), negative values provide an inverse contrast to simulates surface that seem luminescent, such as silver. Appropriate values are -500% to 500%.

Specular Highlight Settings (Lumas Shader)

The specular highlights controlled in the Reflectance channel are added to the surface color. All three specular highlights share the same parameter set, which is described below.

- **Active:** Use this checkbox to individually activate or deactivate each specular light.
- **Color:** Sets the base specular color for the specular component.
- **Intensity:** Scales the Color to produce attenuation in the specular reflection of the surface (this simply means it makes the color darker when the value is close to 0% and brighter when the value is closer to 100% or above). The appropriate range is 0% - 1,000%.
- **Size:** Sets the size of the specular reflection. Appropriate value range for Size is 0.001% - 200%.
- **Contrast:** Provides contrast for the result of the specular sample color. Implements a standard contrast function. Appropriate values are 0% - 100%.
- **Glare:** Works with Intensity by using the edge falloff (multiplied by the Falloff scalar) to change the intensity of the specular reflection. This can be used to add a glare to the surface when a light is reflecting more on an edge or specular effects to reduce the specular reflection when a light causes a reflection closer to the edge. Appropriate values run between 0% - 200%.
- **Falloff:** Utilized to vary the falloff of the center of the object to the edge to affect the Glare intensity. Smaller values cause the specular reflection to reflect the Intensity value more, while larger values make the surface use the Glare scalar more. Experimentation is required to fully understand this parameter.

Anisotropy: Anisotropy is the property of being directionally dependent (as opposed to isotropy, which implies identical properties in all directions.)

Projection: The type of projection is used to define the anamorphic (disproportional) scaling of the specular highlights. It can also define the scratch direction to be used with the reflection and environment convolution groups.

- Planar: A flat XY planar projection.
- Auto Planar: Automatically projects on a plane parallel to the current normal.
- Shrink Wrap: A spherical projection for scaling direction, but uses a separate algorithm for projecting the scratches.
- Radial Auto Planar: Automatically projects on a plane parallel to the current normal.
- Radial Pattern Auto Planar: Creates a multi-origin radial scratch pattern that is parallel to the current normal.
- Radial Planar: Creates a radial scratch pattern originating at the center of a plane parallel to the current normal.
- Radial Pattern Planar: Creates a multi-origin radial scratch pattern that is parallel to the current normal.

Projection Scale: Scales the scratch algorithms that have an actual pattern (Radial Pattern is the only current algorithm of this type to which scale applies).

X Roughness/Y Roughness: Scale the highlights in the X and Y directions defined by the Scratch Projection algorithm. The appropriate range is 0.1% to 10,000%. If the X & Y roughness are the same, the standard internal specular algorithm is used.

Specular 1/Specular 2/ Specular 3: These check boxes specify which specular channels will be affected by the anisotropic scratches.

Amplitude: Scales the effect of the scratch of the specular highlights. The higher the value, the more scratched the speculars will look. The appropriate range is 0% - 100%.

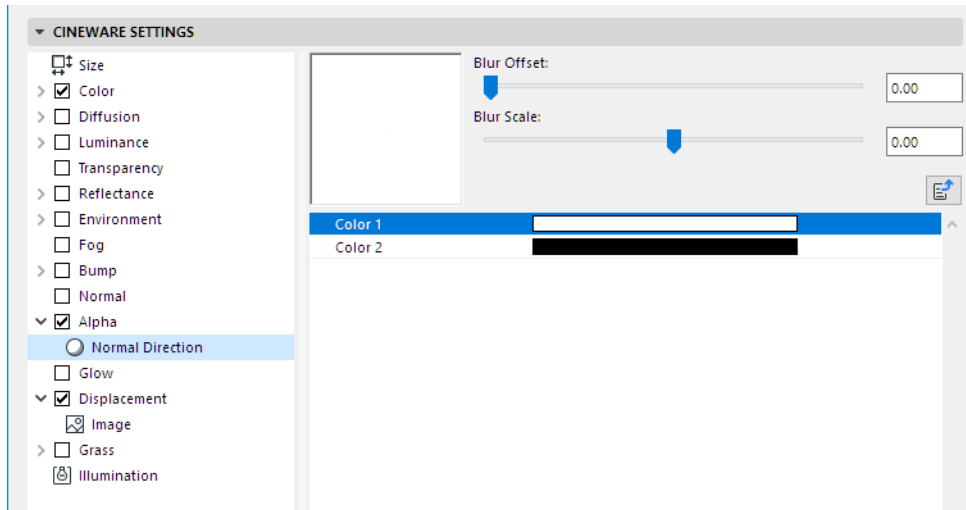
Scale: Scales the scratch pattern itself. This applies to all scratch algorithms.

Length: Defines the scratch length in the space of the scratch pattern. Smoother milled surfaces would use a higher scratch length and rougher surfaces would use smaller values. The appropriate range is 1% - 1,000%.

Attenuation: Scales the amount of detail of the scratches based upon the samples, angle to the camera and the distance from the camera. The larger the values, the greater the attenuation, causing less scratching (better for animation); the smaller the value, the greater the detail (better for still images). The appropriate range is 0% - 1000%.

Normal Direction

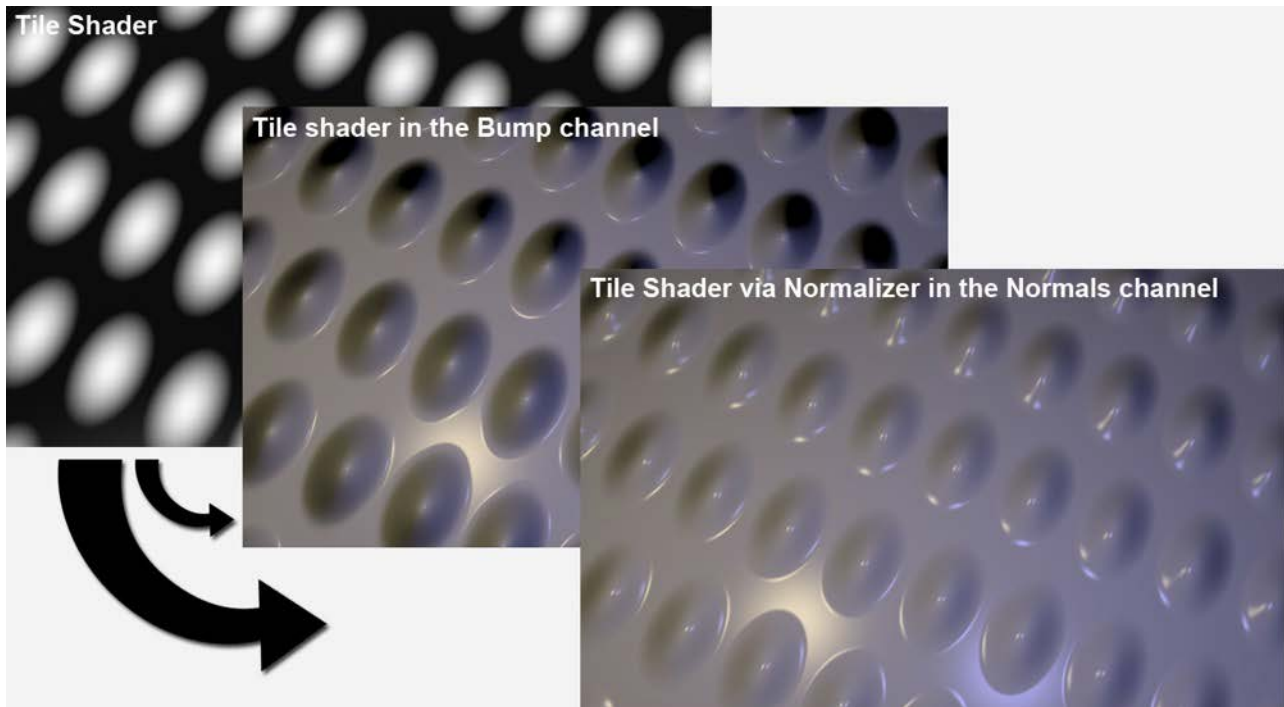
This shader colors an object based on the direction of its surface normals. Surfaces whose normals point towards the rendering ray take Color 1, the other surfaces take Color 2.



Normalizer

Using the Normalizer shader (placed into the Normal channel) you can use common textures that were usually used in Bump maps in the Normal channel. The Normalizer evaluates the contrasts to ascertain the location of edges and uses this information to calculate a Normal map.

As a reminder: Normal and Bump maps basically work the same. Information coded in RGB images affect an object's (geometry) surface Normals when rendered and feign a detailed structure. Contrary to Bump maps, Normal maps can also "alter" the direction of the Normals, which makes it possible to create realistic effects (note the specular light on the bumps in the image below).

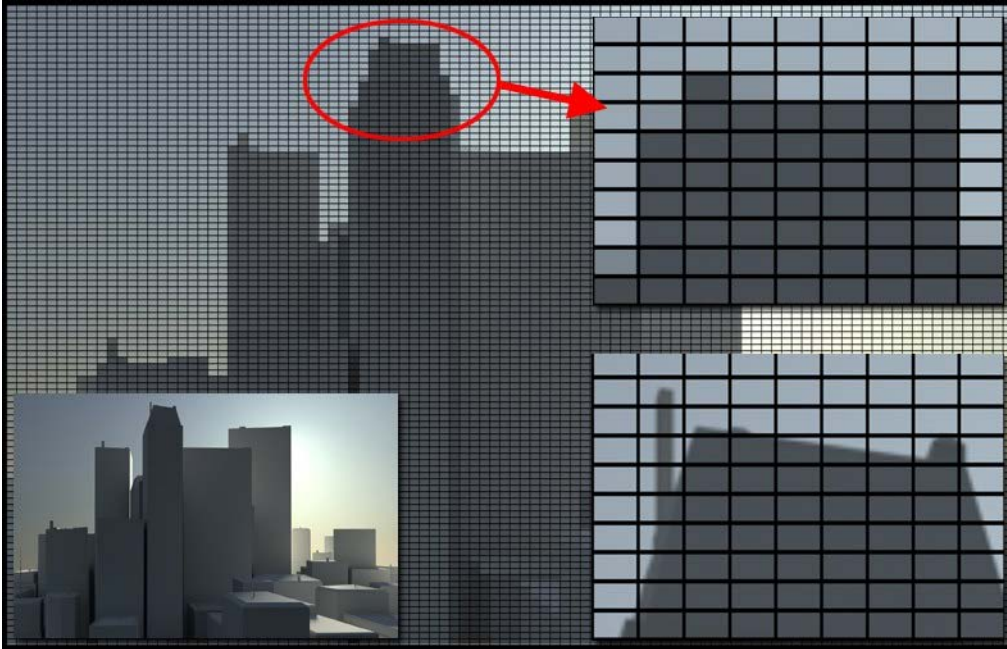


The texture shown at left was placed in the Bump channel and the Normal channel, respectively

Pixel

The Pixel Shader creates a pixel-like distribution of a texture.

Let's say you want to project a texture made up of small areas of the same color onto a surface (e.g. a TFT display). You would want the individual pixels to also look good close up. This is where the Pixel Shader works its magic - it creates a fitting "pixel coarseness" without having to make changes to the original texture.

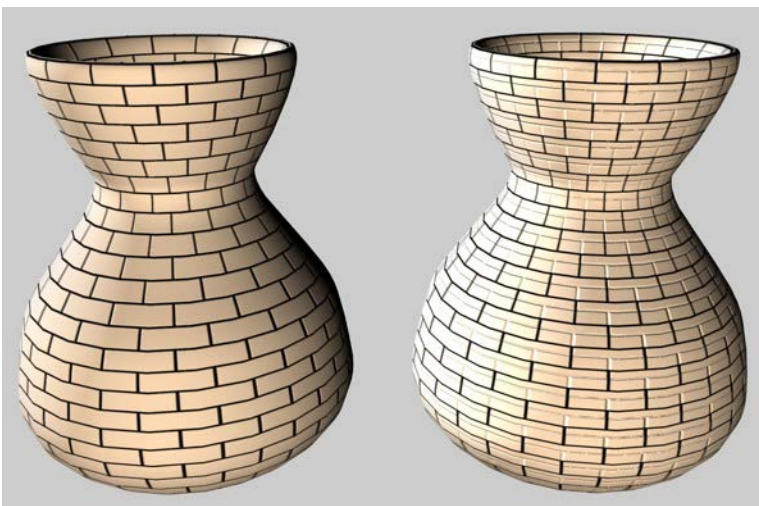


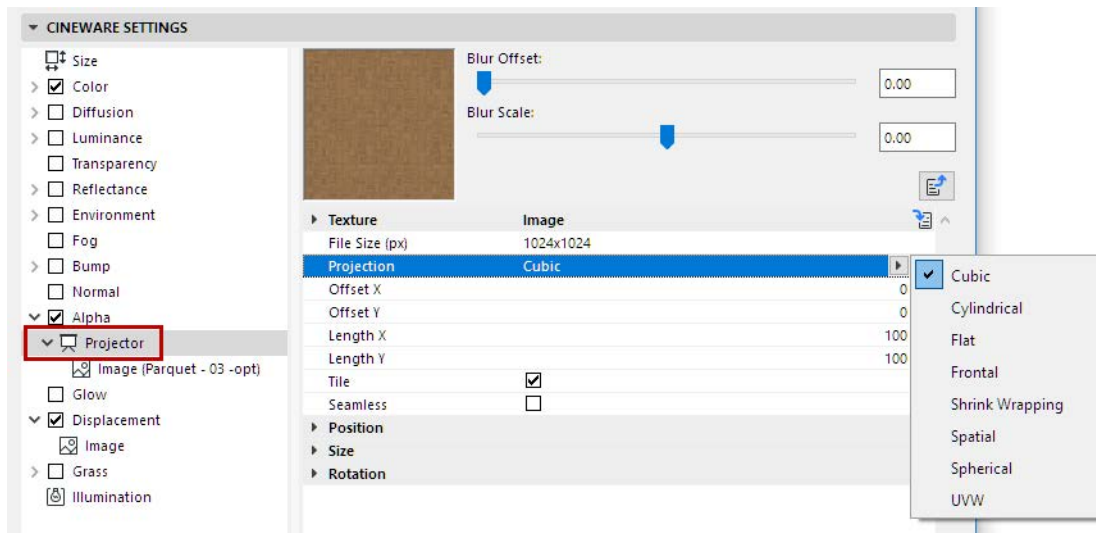
Top right: With Pixel Shader applied; bottom right: Without the Pixel Shader applied

Note: The Pixel Shader only works with 2D Shaders and textures. When using a Noise Shader, for example, the Space option should be set to UV (2D)

Projector

Projector allows you to alter the projection of a shader or image. This is useful for having different mappings in separate channels of a surface, especially when used in conjunction with other shaders such as Fusion. Note that 3D shaders do not work with Projector.





Texture: This is the source image or shader to be projected. You can bring in and project any image or 2D channel shader available to Cineware.

Projection Types:

- Spherical
- Cylindrical
- Flat, Cubic
- Frontal
- Shrink Wrapping
- Spatial
- UVW Mapping

Offset X/Offset Y: Offset of the 2D texture in UV.

Length X/Length Y: Scaling of the 2D texture in UV.

Tiles X/Tiles Y: Tiling of the 2D texture in UV.

Tile: If enabled, the 2D texture will tile in UV the number of times specified in the Tile X and Tile Y parameters.

Seamless: If Tile is enabled, then enabling Seamless causes the tiles to be mirrored so that they tile seamlessly.

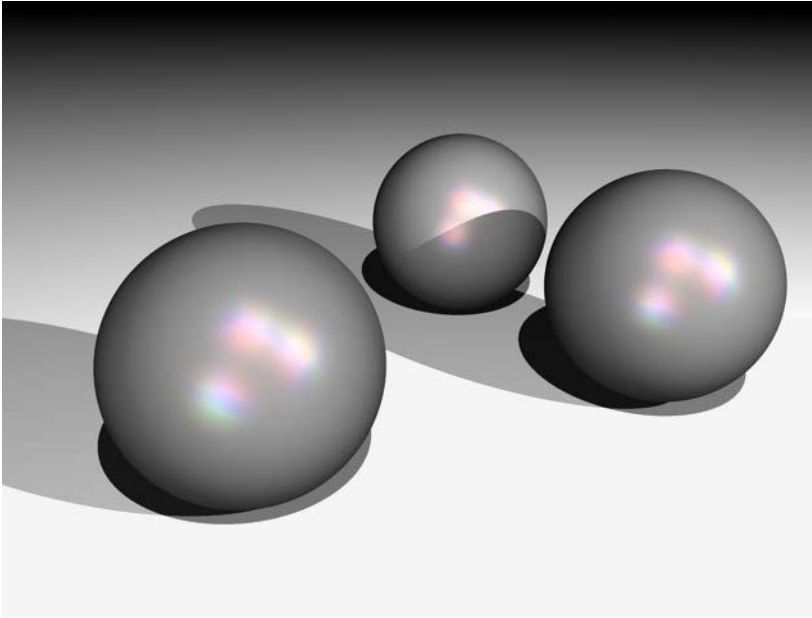
Position (X/Y/X): Offset of the 3D texture space.

Size: Scaling of the 3D texture space.

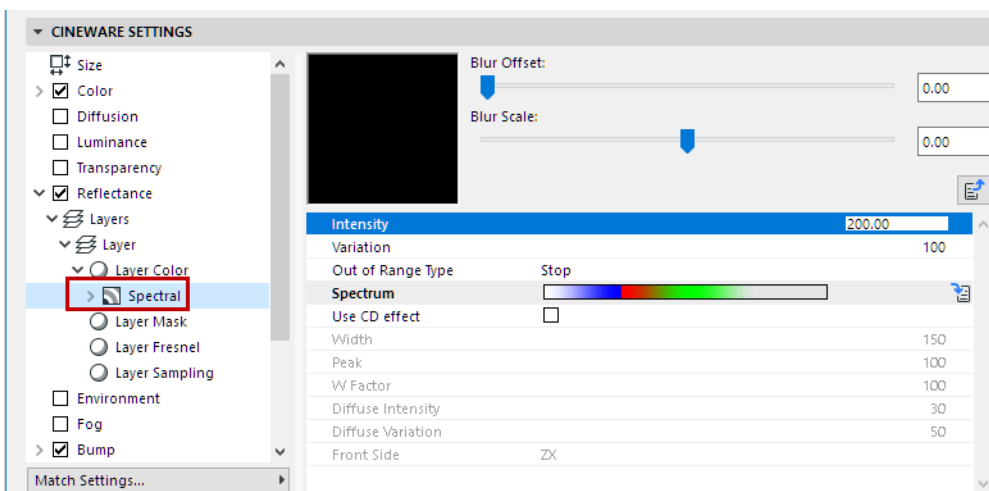
Rotation (Heading/Pitch/Bank): Orientation of the 3D texture space.

Spectral

The Spectral shader is ideal for prismatic or iridescent effects such as shiny pearls.

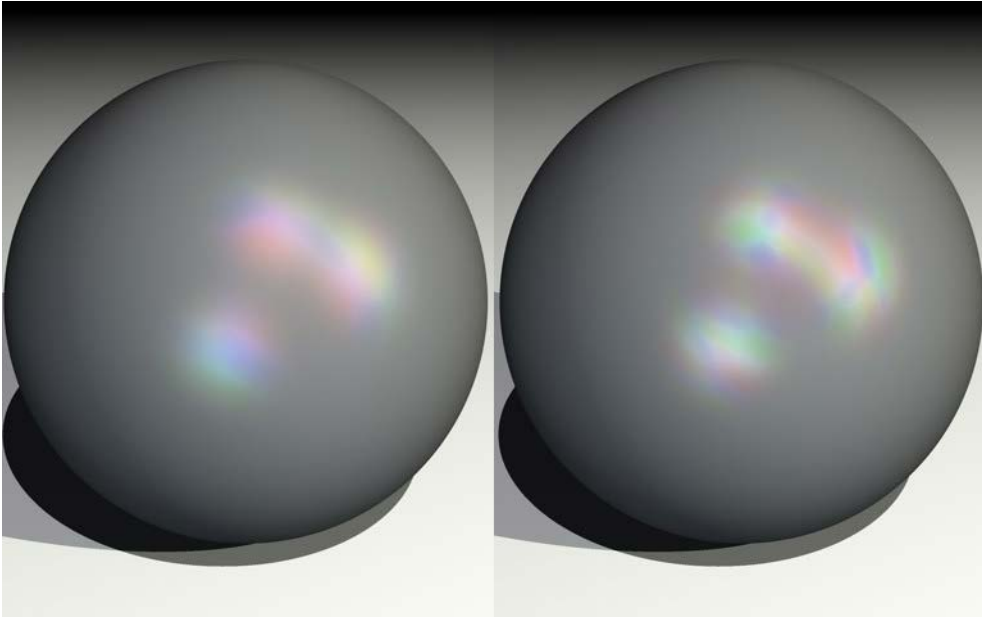


The calculation of this effect takes the camera perspective and the angle between the light source and surface into account. In general, the Spectral shader should be primarily used in a Specular layer (under Layer: Color) in the Reflectance channel.



Intensity: Controls the overall brightness of the color reflexes. A value of 0 means no effect at all, while bigger values result in brighter color reflexes.

Variation defines how many times the gradient is repeated within the defined range.



Variation set to 1 (left) and 2 (right)

Out of Range Type: In combination with the Variation parameter, Out Of Range Type controls if and how the gradient is repeated. Experiment with the **Stop**, **Mirror** and **Tiling** modes to see the effect.

- With Stop, the gradient is applied once only and the last color is used for the outer areas that are out of range.
- With Mirror, the gradient is mirrored when it is repeated to avoid seams.
- With Tiling, the gradient is repeated according to the Variation value.

Spectrum: The Spectrum gradient defines the colors for the spectral reflexes.

Use CD Effect: If you enable this option, the shader is optimized for cylindrical surfaces and you can render wedge-shaped, prismatic color effects, such as those seen on CD surfaces.

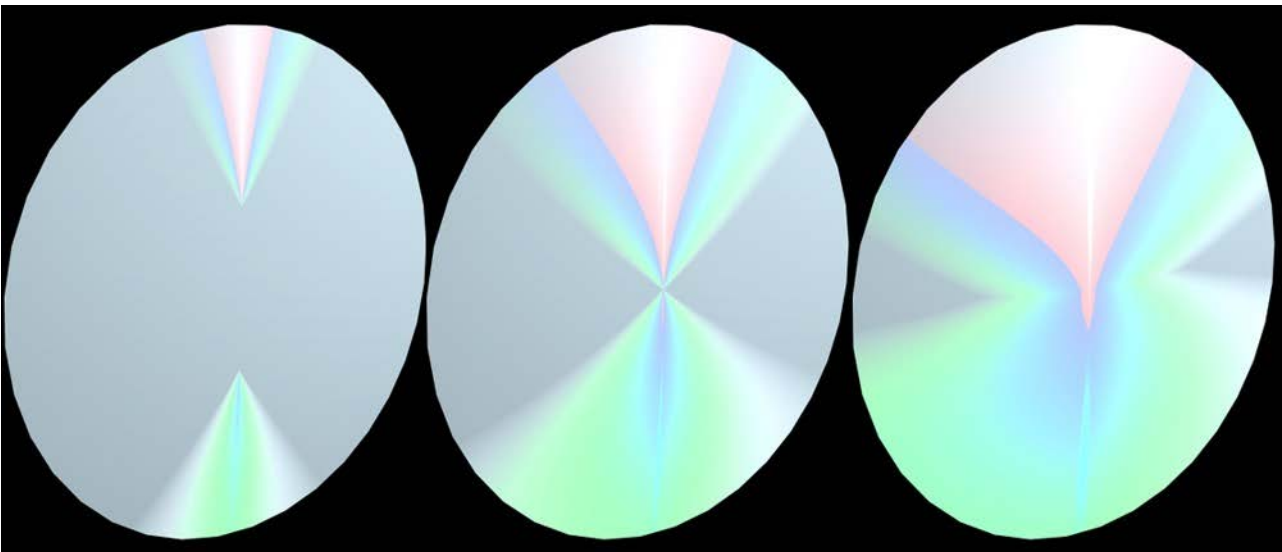
Width: Width defines how far the gradient spreads out.



Left: greater width; right: less width

Peak: Normally the entire rainbow colored spectrum is visible only if the angle between the light source and the camera forms a straight line. Increasing the Peak value allows you to generate a larger color spectrum.

W Factor: Defines the starting point of the prismatic effect, based on the center of the surface. A value of 1 makes the gradient start from the exact center of the surface. Values above 1 offset the gradient away from the center, and values below 1 shift it more towards the center.



W Factor 1.5, 1 and 0.7

Diffuse Intensity: Diffuse Intensity controls the intensity of the spectral effect. Increasing the value results in overblown colors.

Diffuse Variation: Scatters the spectral effect by bringing in random colors from the gradient.

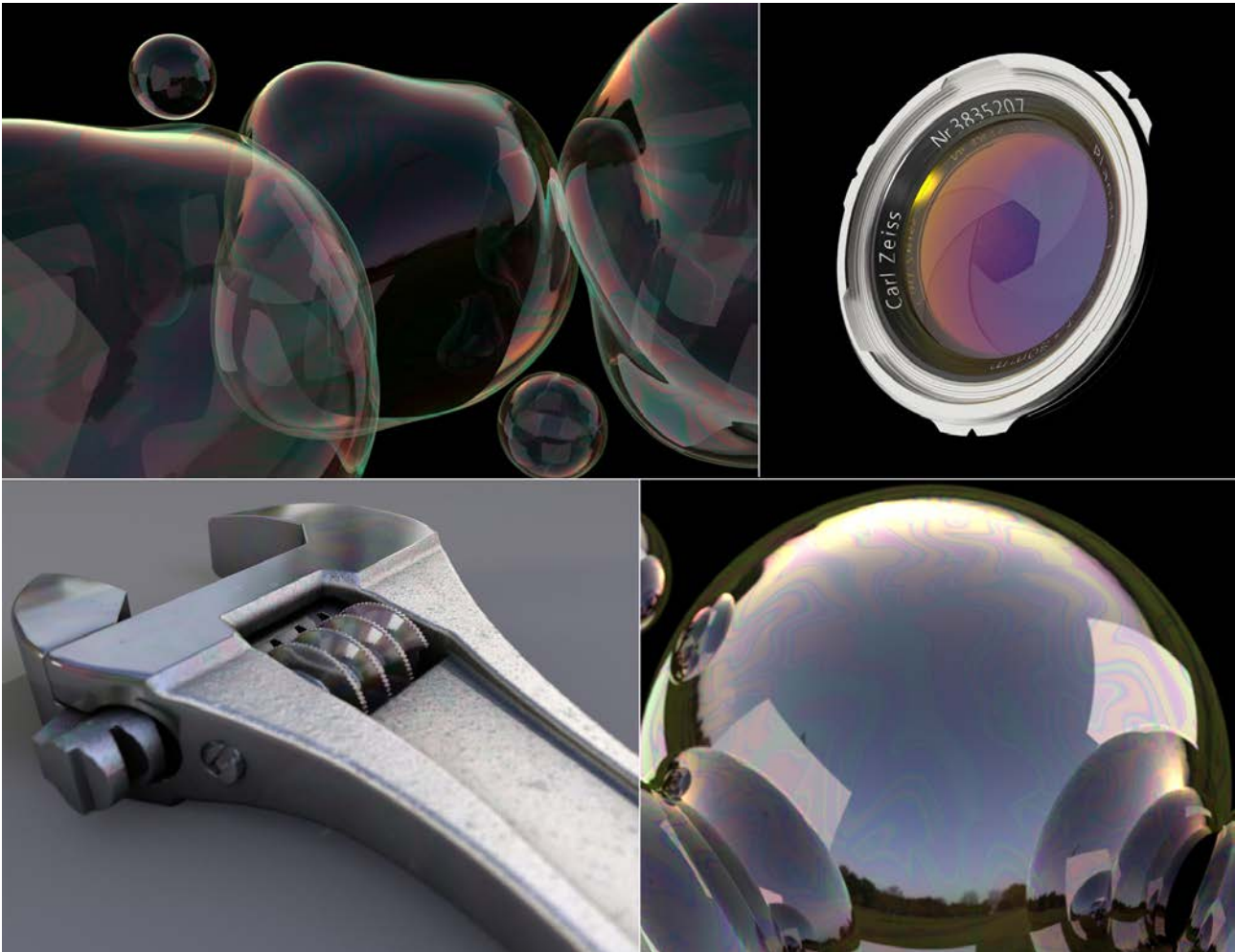


Lesser and greater Diffuse Variation values

Front Side: Defines how the spectral gradient is projected onto the surface, with reference to the object coordinate system. If you can't see the CD effect immediately, you probably need to adjust this parameter.

Thinfilm

Use this shader to simulate a special physical effect referred to as “thin film” or “interference” effect. This effect occurs on very thin transparent surfaces - for example, as when all colors of the spectrum shimmer in surfaces of soap bubbles, oil film on water, mother of pearl.



To render realistic-looking results, you must create realistic lighting setups (e.g., by using a Physical Sky, luminous reflectors).

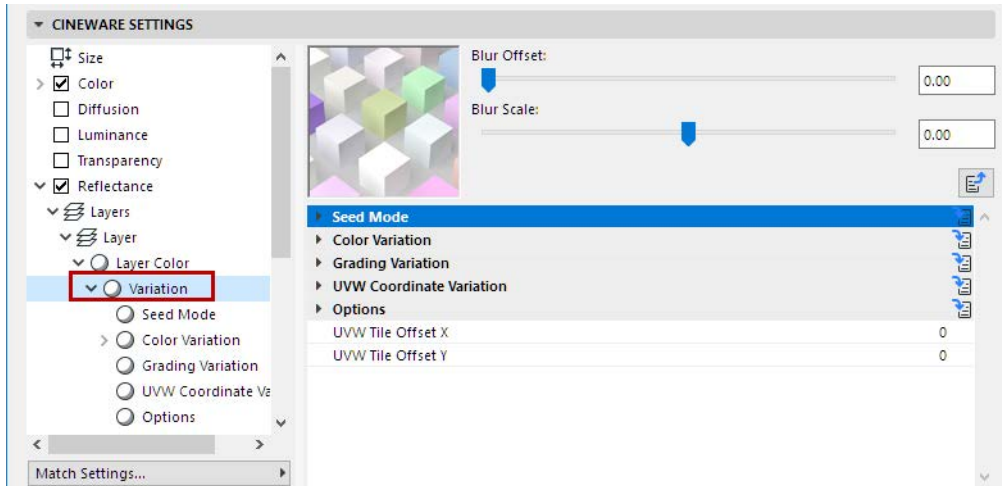
The Thinfilm shader will only work correctly if it is loaded into a Reflectance channel's Texture field (preferably in a Beckmann or GGX layer) in the channel's Layer Color menu. Specular layers should be disabled.

Tips:

- For soap bubbles, the Transparency channel should also be enabled and its Additive enabled.
- Make sure that suitable reflective objects are used; HDRI textures can be applied to a Sky Object or layers with luminous materials can be used.
- This effect can also be used on materials such as leather, rubber and metals, etc.

[Watch the Video](#)

Variation



The Variation Shader allows for easy generation of “randomness” in your scene.

The effect can be applied in a wide range of situations:

- Rock bed: The Variation Shader makes sure all your rocks look unique.
- Foliage looks more natural when all leaves have subtle variations.
- Setup the variation shader on a single car-paint material and apply it to all cars in a parking lot -- the shader automatically gives each car a custom paint job.
- Render books on a shelf: Apply the variation shader to the book cover material to make all the books appear different with minimal effort.

The Variation Shader is designed to work with regular objects and hierarchical tools (Curtain Wall, Stair, Railing). Generally speaking, the effect works best for objects where some variation is desired and it's not worth putting in a lot of effort to manually produce custom textures.

The Variation Shader works by generating random numbers based on a seed value which is automatically generated at render-time. The input texture's properties are then modulated based on the random numbers with user-defined intensity. The effects can be subtle, but also outright garish.

The shader is not restricted to use in the color slot; interesting effects can also be achieved by using the Variation Shader on bump maps.

It also allows for seeding based on the polygon face ID, which makes it ideal for working with dense meshes such as foliage without the need to split the leaves into separate objects.

[Watch the Video](#)

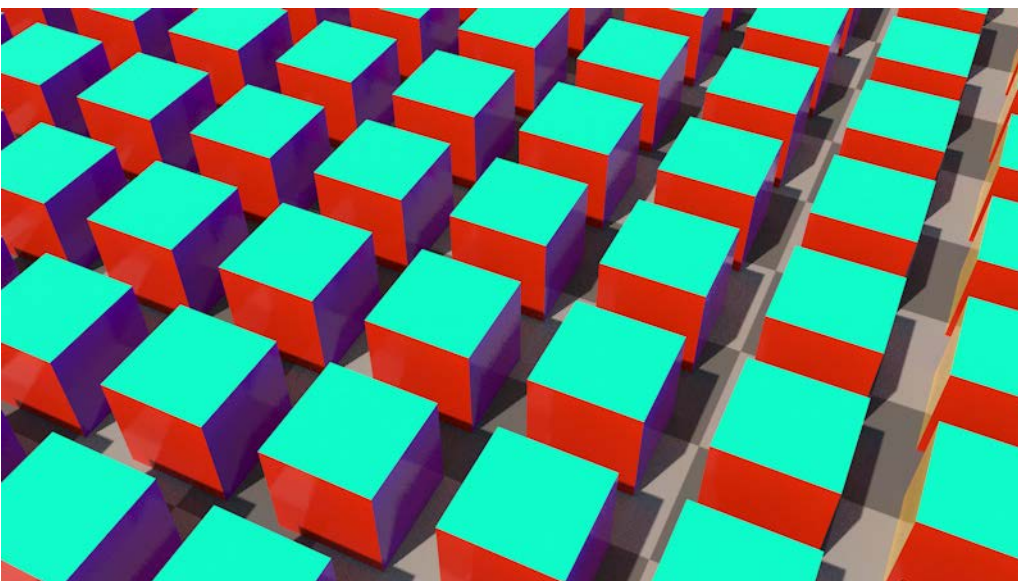
Seed Mode

The seed mode controls which data type the random numbers are based on. Depending on your choice of seed mode, you can achieve different types of effects.

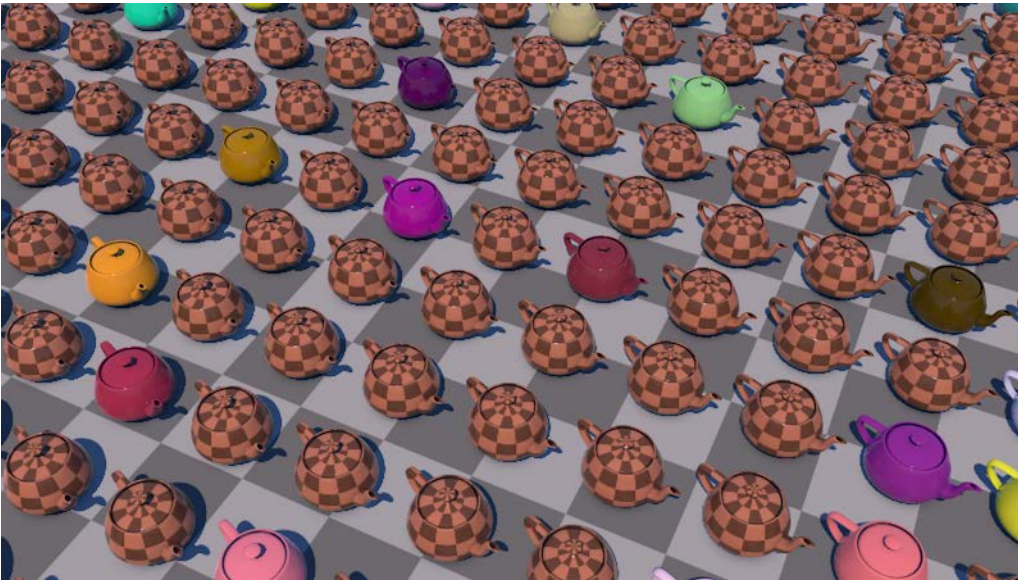
Limitation: Note that adding or removing objects to the scene may alter the seed values. This can be an issue when rendering sequences, as the variation will appear to change randomly from frame to frame.



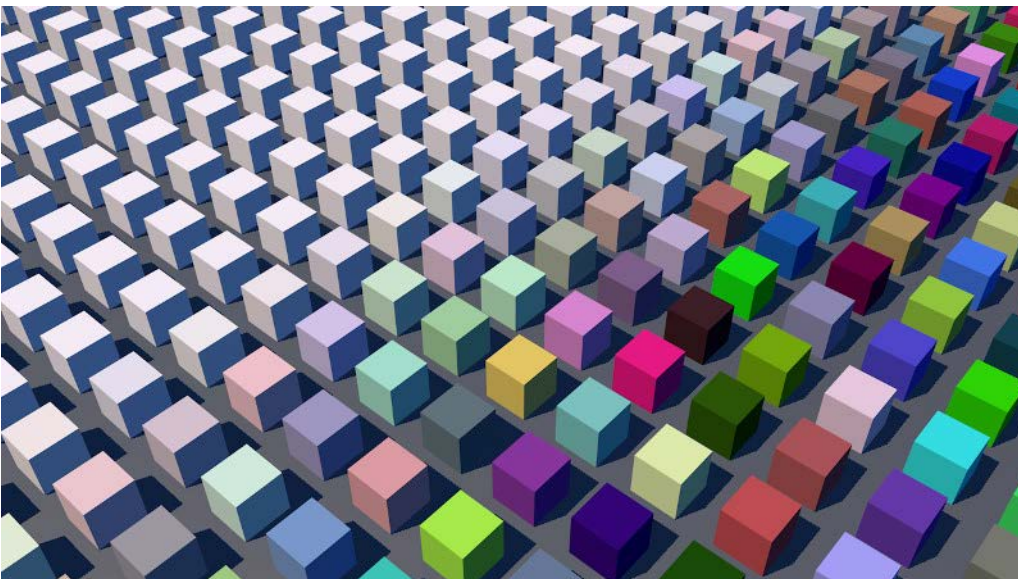
- **Seed:** The seed value controls how random numbers are generated. Use this value to influence the generation of random numbers if you're not happy with the variation that's being produced.
- Object Variation
 - Disabled: No variations will take place for different objects with the same Variation shader applied to them.
 - Object Hierarchy: If the Variations shader is applied to an object that is part of a hierarchy: In this mode, the same color will be applied to all Child objects, i.e., it will not be varied. It will only be varied per hierarchy.
 - Objects: Each object to which a shader is assigned (incl. objects in subordinate hierarchies) will be varied (as long as this is not prevented by one of the following settings).
 - Parent: No variation will be applied to objects at the same hierarchical level and with the same Variation shader if they share a Parent object. Child objects will also not be varied.
- Polygon Variation
 - Polygons: Texture is varied for each polygon.



- Polygon step: It may occur that, for example, a tree's leaves are created using a huge number of coherent polygon groups. For example, a leaf can consist of 6 polygons (to allow it to bend), which is in turn duplicated hundreds of times for a single tree. If the variation should be constant within one 6-polygon group but varied for other 6-polygon groups, define number of polygons for the leaf (in this case 6) and the variation will occur in steps of 6 polygons. However, if only a single leaf has a different polygon count, this function will not work correctly! Set this value to 1 to create variations for each polygon.
- Disabled: Polygons will not be given different variations.
- **Probability:** Controls how likely it is that the shader will apply variation for any given object. At 100% all objects receive variation. At 50%, variation will be applied to only (approximately) half of the objects. When the shader decides to not apply variation, the input texture is returned unchanged.



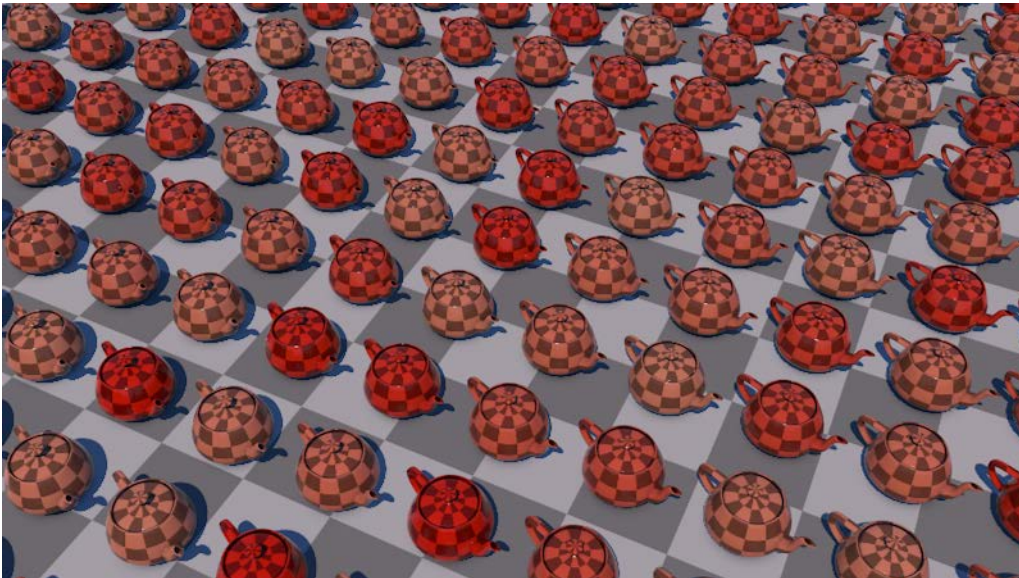
- **Global Mask:** Controls the overall effect of variation based on an input texture.



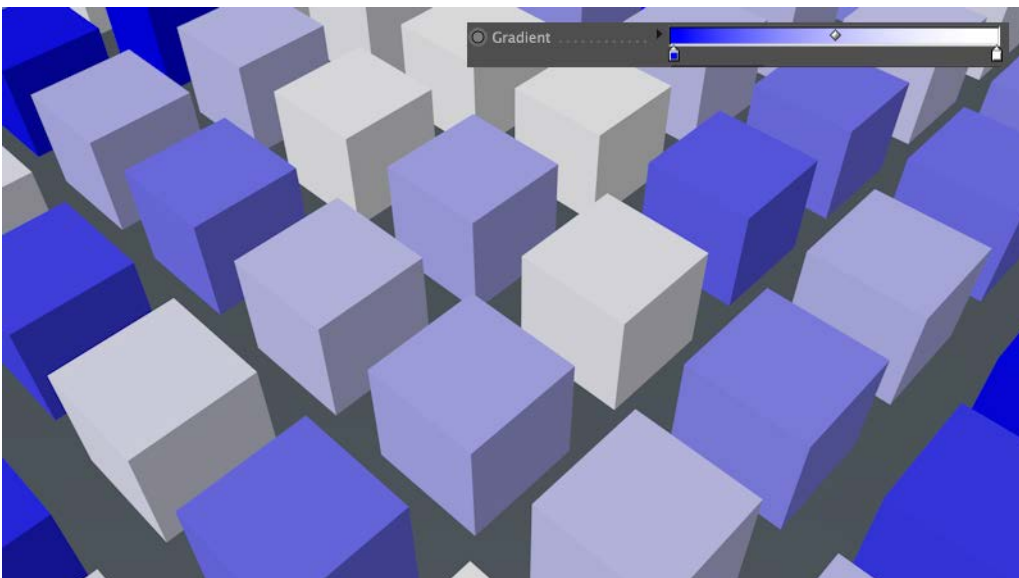
Color Variation

These settings define which textures should be used and which colors should randomly be mixed with them.

- Shaders
 - Active
 - Probability
 - Texture: This is the main input. Plug any shader or texture in here.

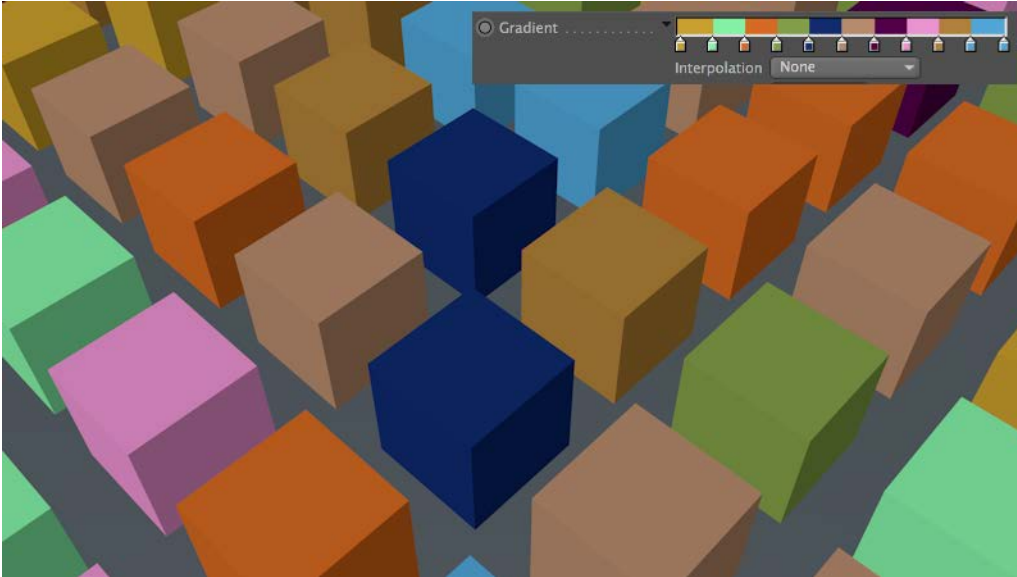


- Secondary Texture. As an example, you could introduce a dead leaf texture to make an otherwise boring looking tree more interesting
 - Secondary Texture Blend. Controls the maximum intensity at which the secondary texture will be applied. This value is randomized based on the seed value
 - Secondary Texture Mode: Pick the blend mode with which the texture is applied.
- Gradient: Define a gradient from which random colors will be picked and applied using the color mode below.

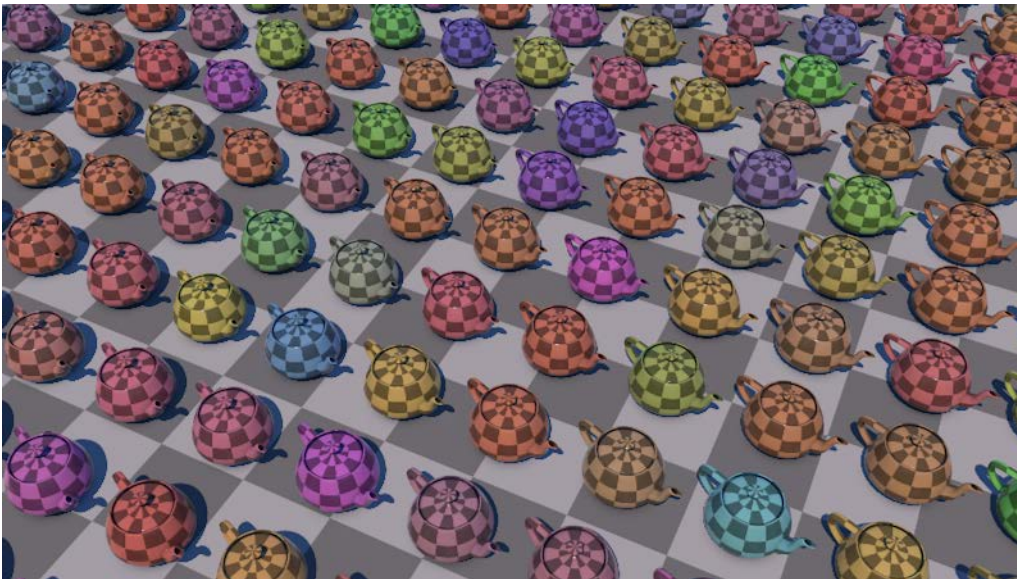


- Gradient Blend: Controls the maximum intensity at which the gradient color will be applied. This value is randomized based on the seed value.
- Gradient Mode: Pick the blend mode with which the gradient is applied.

Note: In “Replace” mode, the Texture Blend setting has no effect; the texture is applied at 100% opacity. This can be useful in combination with the Probability parameter



- Random Color: Generates a random color and blends it with the input texture up to the specified value.



- Random Color Mode: Pick the blend mode with which the texture is applied.

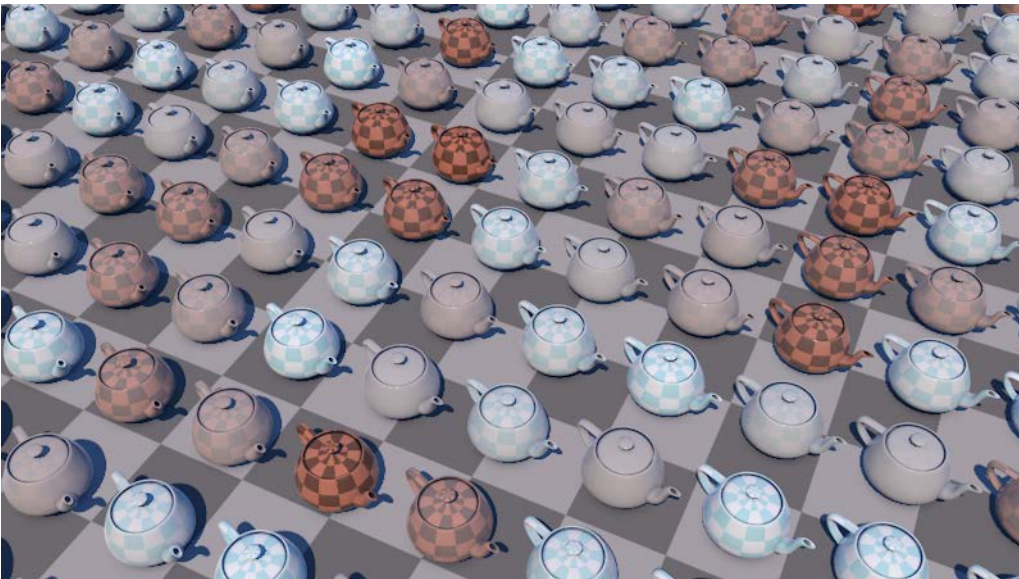
Grading Variation

The color grading options all work in a similar manner: The values control the maximum offsets to the grading parameters. E.g. with a value of 0.5 on the Contrast slider, the shader will randomly adjust the contrast between -0.5 and 0.5.

- Contrast: Randomly adds or removes contrast up to the specified value.

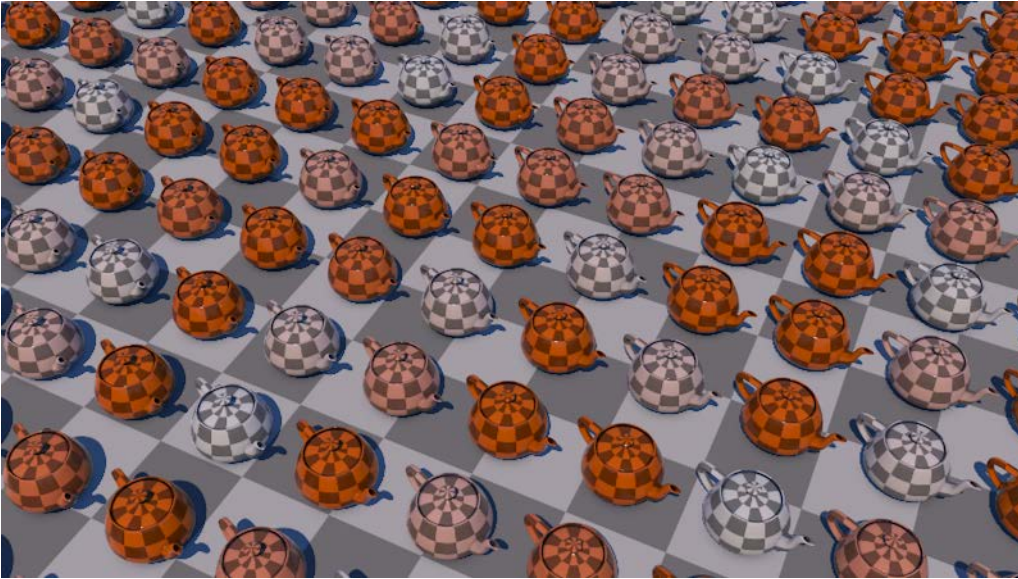


- Gamma: Randomly varies the gamma up to the specified value.
- Invert: Randomly inverts the input texture up to the specified value.



- Hue: Randomly shifts the color hue up to the specified value. Use low values to create natural looking color variation. Higher values makes for very colorful effects.

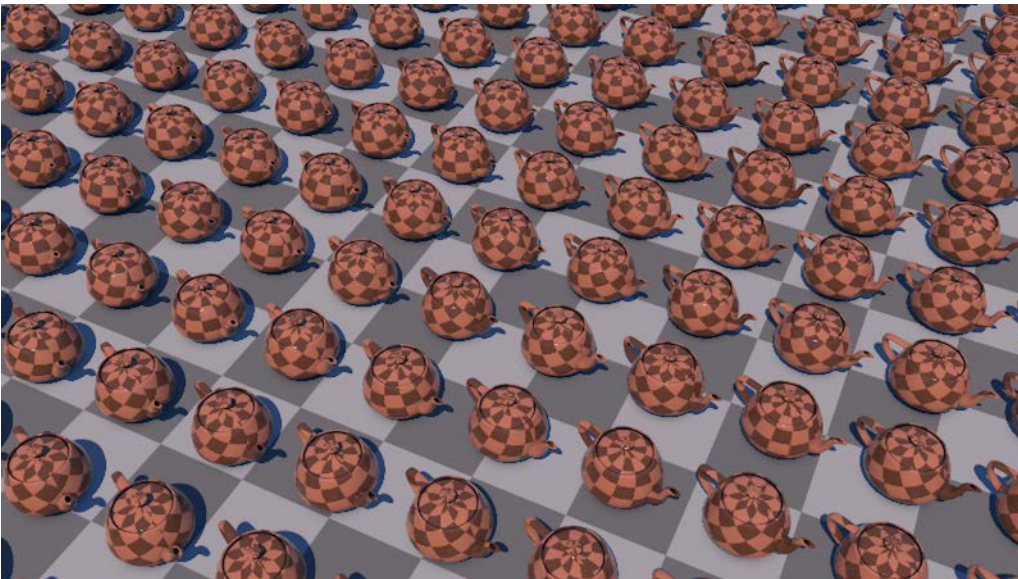
- Saturation: Randomly increases or decreases the color saturation up to the specified value. As with the Hue parameter, use low values to create more natural looking effects.



- Lightness: Randomly increases or decreases the color lightness up to the specified value.

UVW Coordinate Variation

The Variation shader is able to not only mix colors, but also to move and rotate UVW coordinates. This works best with tileable textures.



- UVW Shift: These settings define the maximum random movement of UVW coordinates left/right or up/down.
 - UVW Flip X/Y: These options can be used to randomly mirror UVW coordinates vertically and/or horizontally. The random mirroring takes place prior to any random rotation defined using the following two options.

- UVW Rotate: This value defines the maximum random rotation (by default: 100% = 360°) in each direction for the UV coordinates (rotation center U, V: 0.5, 0.5, i.e., center of the UV tile. Contingent on the next setting, the rotation calculated here will then be set to a fixed, quantized value.
- UVW Quantize Rotation: Define here if the UVW coordinates should rotate steplessly (0 Degrees) or in steps of 45, 90 or 180 degrees. This is dependent on the UVW Rotate value: an even distribution of steps is only achieved at a value of 100%.
- UVW Scale: Use these values to randomly scale the UVW coordinates. The values entered here define the limits in the positive and negative directions: e.g., if the U coordinate is set to 0.5, the UV polygons will be varied between 0.5 and 1.5 times the horizontal U scale.

Options

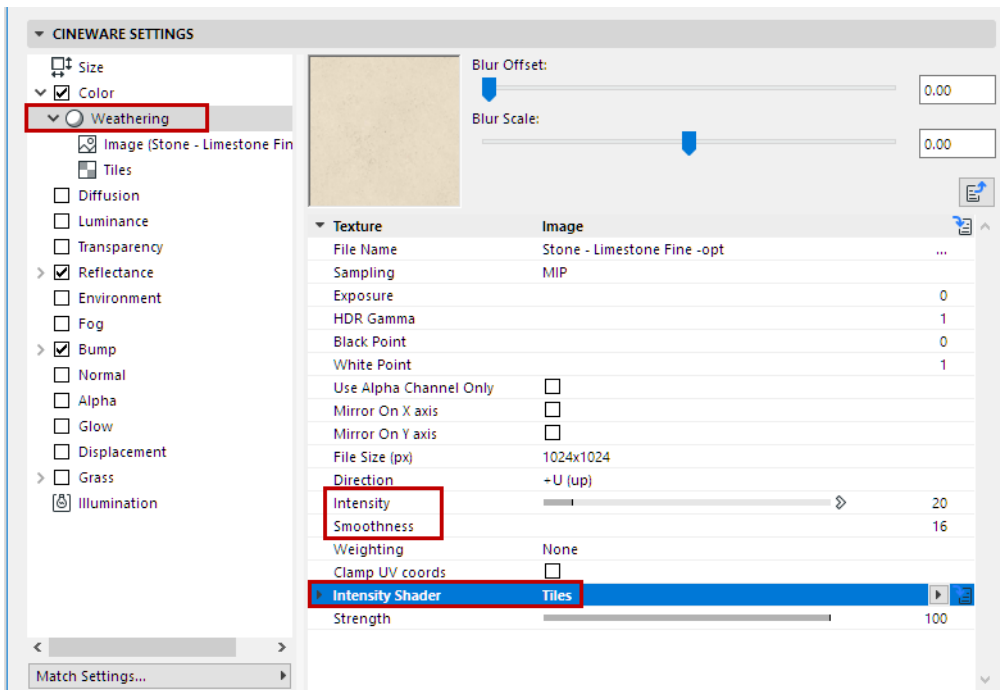
- Clamp One: Ensures that no values higher than 1.0 are returned. When varying colors, it can happen that they turn out “too bright” (RGB values in excess of 255, 255, 255). This is often not desired and can be prevented by enabling this option. However, if you use HDRI images, this option should remain disabled.

Weathering

This Shader helps you lend a weathered look to your objects. You can weather a texture in a specific direction, which makes it look as if rain, wind or other factors affected the surface.



From left to right: Unchanged, completely weathered, partially weathered



The shader's **Intensity** controls the strength with which the texture is smudged, as you can see in the image below in which the value is increased from left to right from 20% (default value) to 50% and 100% respectively.



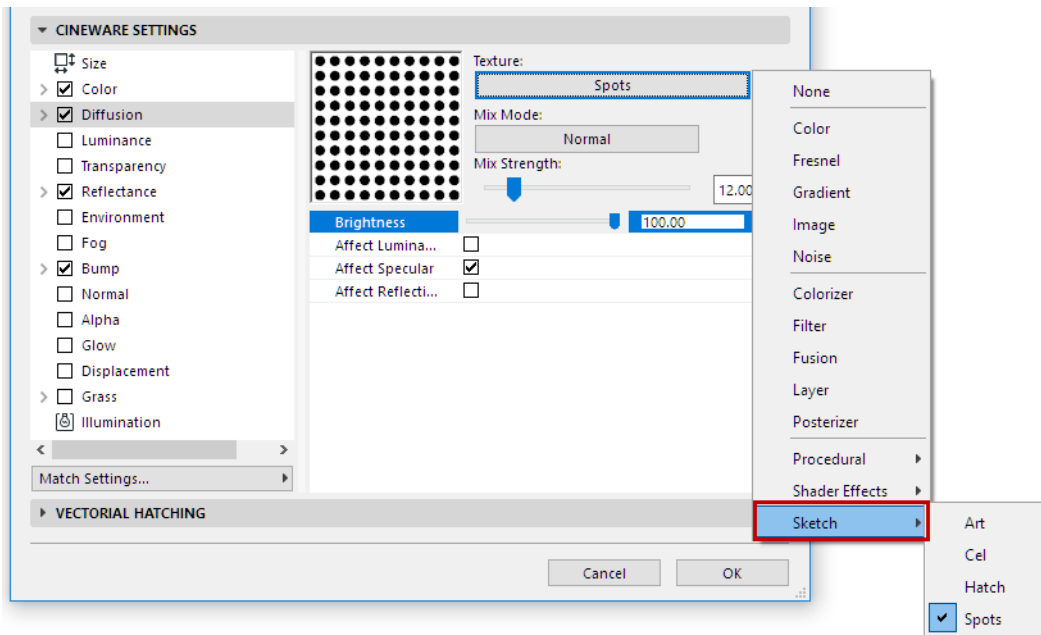
In addition to Intensity, you can also define how smooth the smudged texture should be. The higher the **Smoothness** value, the smoother the effect will be and the more realistic it will look. In the image below, the Smoothness is increased from left to right from 4 to 16 (default value) to 32.



Finally, the **Intensity shader's** brightness value can be used to define the degree of weathering. An image or a Shader can be used. In the image below, a Tiles Shader was used to create a horizontal line structure.

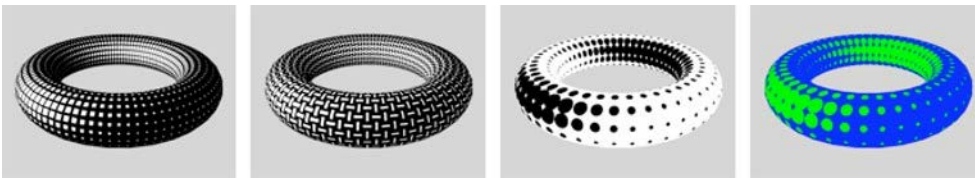


Sketch Shaders (Cineware Surfaces)



Four Sketch shaders are available: Art, Cel, Hatch and Spots.

- The Art shader uses a sphere to calculate the lighting and shading of a given object. Uniform shading can be achieved by varying the surface and color of the sphere.
- The Cel shader combines Manga and Anime shading styles with quantized color shading and freely definable color gradients.
- The Hatch shader applies a freely definable hatch pattern based on any texture you select.
- With the Spots shader, points (or any other element you want to use) shrink in correlation to the brightness of the surface. This is useful for creating halftone effects that are common on low-resolution prints.



Grid / Custom texture / Circle with Invert / With color

You can load Sketch shaders into any surface channel which accepts a texture, but generally they work best in the Luminance channel with all other channels such as Specular disabled.

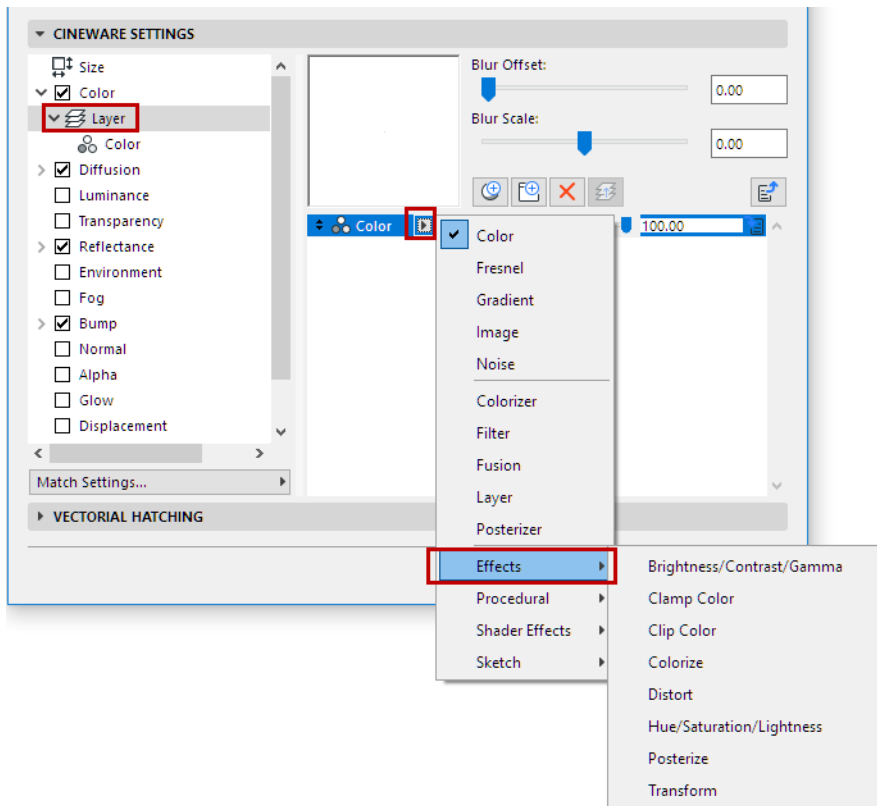
Antialiasing tip: The Art shader, Hatch shader and Spots shader each use a texture to shade the objects. For example, the Hatch shader uses a picture of a stroke or block of strokes to draw hatches onto the objects. Naturally you will want these texture strokes to look smooth and antialiased when rendered, which would usually mean setting the Antialiasing mode in PhotoRendering settings to Best.

[See Antialiasing \(Cineware\).](#)

However, to speed up rendering you can set the Antialiasing mode to Geometry instead of Best and blur the texture slightly using the texture's Blur Offset and Blur Scaling values. Although Cineware will not antialias

textures in Geometry mode, this blurring will produce smooth strokes that look antialiased, and the rendering will be much faster. Ideal for when you are using thousands of hatched lines.

Additional Effects for Shader Layers/Folders (Cineware Surfaces)



The list of Effects (available only from a channel's Layer control) allows you to add various effects to the layers or folders.

See [Using Layers to Combine Shaders \(Cineware Surfaces\)](#).

Most of these effects are based on other Cineware shaders. For example, the Distort effect offers parameters that you will also find in the Distorter shader. For descriptions of these inherited parameters, look up the corresponding shader in the reference guide.

Brightness / Contrast / Gamma, Hue / Saturation / Lightness / Clip Color

See [Filter shader](#).

Posterize

See [Posterizer shader](#).

Colorize

See [Colorizer shader](#).

Clamp Color

The Low Clamp and High Clamp parameters cut off the color values at the given points.

Transform

Use these parameters to rotate shaders or textures and mirror them horizontally or vertically. You can also change the scale and position of the shaders or textures.

Distort

This effect is a simplified Distorter shader that works exclusively with noise.

The Time Scale parameter allows you to animate the noise effect. To animate the noise, set the value higher than 0. The higher the value, the faster the noise will change.

See [Distorter shader](#).

Using Layers to Combine Shaders (Cineware Surfaces)

The Layer shader lets you load multiple shaders or bitmaps into a surface's Texture setting, which will then be rendered together.

You can stack shaders and bitmaps into layers and folders. You can define various blend strengths and modes. Alpha channels are supported. You can apply effects to individual layers or all layers.

To change the order of the layers, drag and drop them to their new positions.

You can also add folders or move images/shaders within the Layer shader itself.

Note: The order of the shaders defines the order in which they are applied to the surface.

Some shaders are available only within Layers.

[See Additional Effects for Shader Layers/Folders \(Cineware Surfaces\).](#)

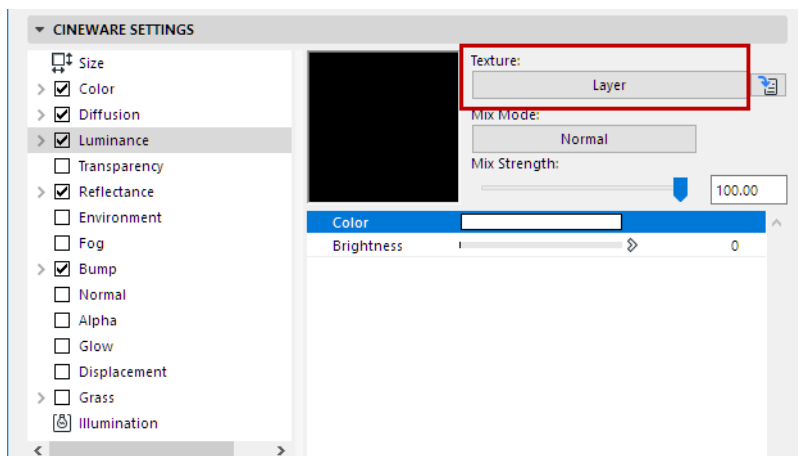
However, you should be aware that the maximum number of parameters that can be used in a surface is 1024.

[See Limit on Number of Surface Parameters.](#)

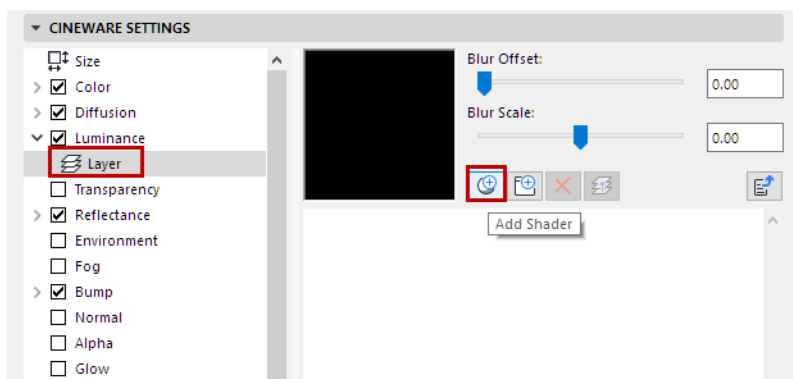
Add Shaders to a Channel Layer

You can add multiple shaders and textures to a particular surface channel.

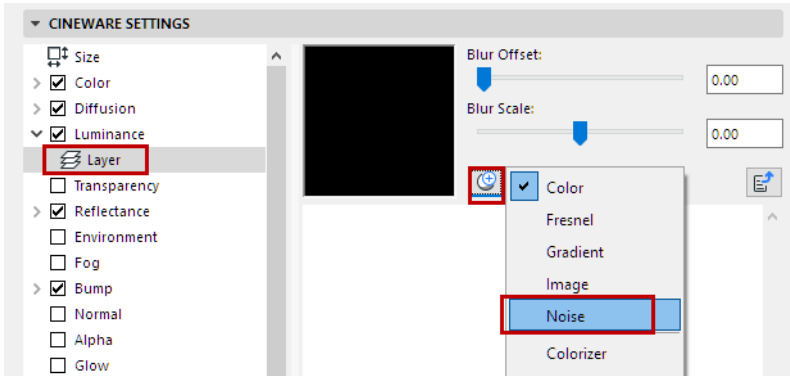
1. To get started, choose Layer from the channel's Texture pop-up:



2. With the Layer item selected or open, click the **Add Shader** pop-up.



3. In the appearing shader row, use the first pop-up to choose a particular shader.

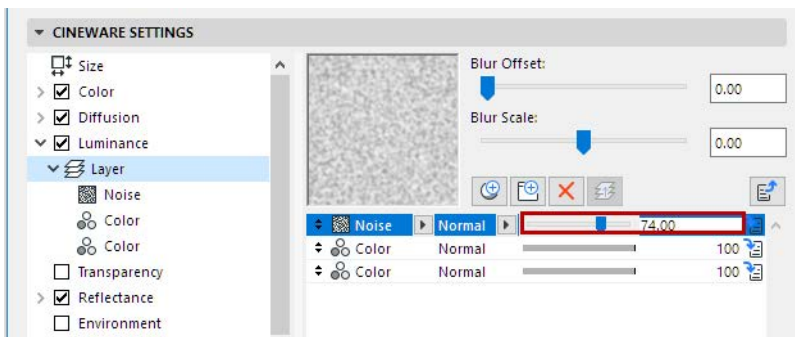


4. Do this for each shader you want to add.

For each shader added to a Layer, you can adjust its Blend Mode and Blend Strength (see below).

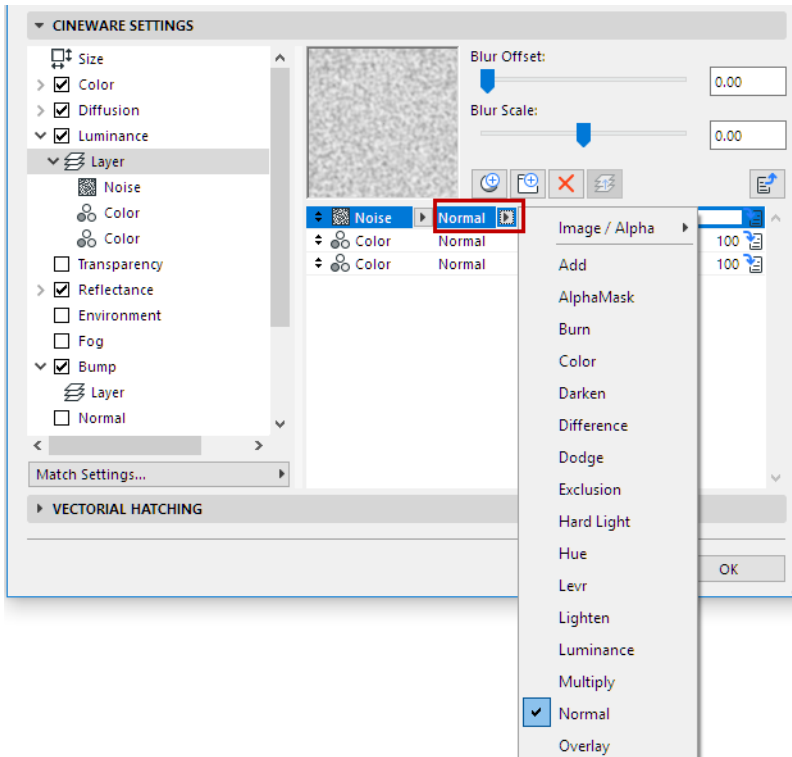
Texture Blend Strength

Select a shader row within a Layer in a surface channel, and use the slider to adjust the shader's Blend Strength.

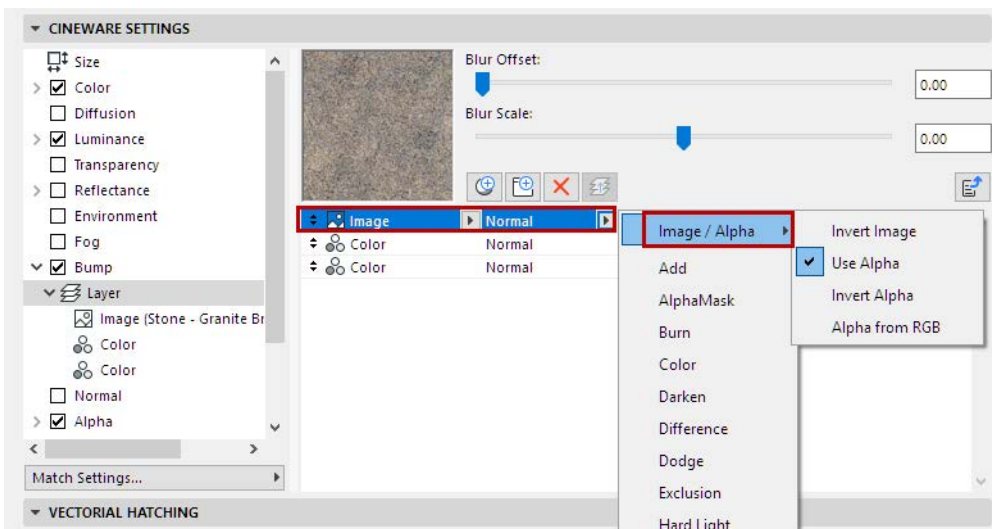


Texture Blend Mode

Select a shader row within a Layer in a surface channel, and click the second pop-up (with the Normal blend mode as the default) to access the Blend Mode definitions.



- Image/Alpha



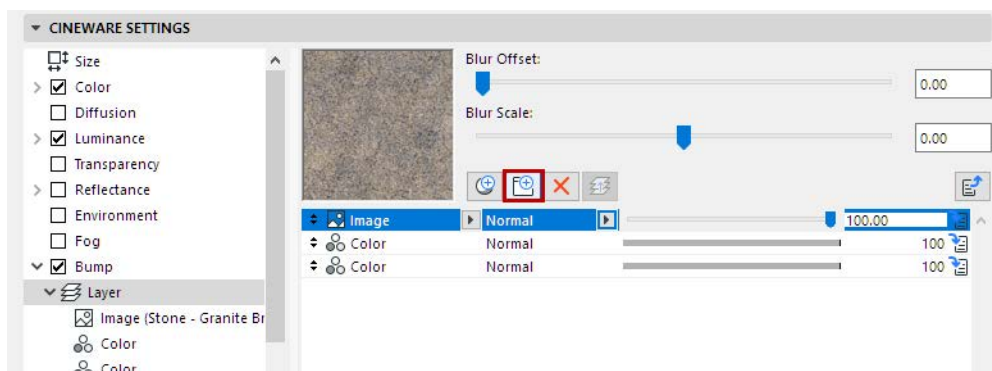
- **Invert Image:** Inverts the image's color values. Black regions are made white and vice versa.
- **Use Alpha:** Toggles any existing alpha channel on or off.
- **Invert Alpha:** Invert the alpha channel, so transparent and solid areas are reversed.
- **Alpha from RGB:** This option is made available only if Use Alpha has been activated in the same menu. Alpha from RGB generates an alpha channel using the grayscale values extracted from a given

bitmap's RGB values. If the bitmap in question also contains an alpha channel, it will also be used when Alpha from RGB is active.

- **Normal:** The background pixel is completely covered by the blend pixel if the strength of blending is 100%.
- **Multiply:** This mode multiplies the background pixel by the blend pixel. Unless the blend pixel is white, the result will be darker than the background pixel.
- **Screen:** This is the opposite of the Multiply blend mode. The background pixel and blend pixel are both inverted, multiplied and then inverted again. This mode generally gives a result brighter than the background.
- **Overlay:** A combination of Screen and Multiply mode, depending on the background color.
- **Hard Light:** This mode works like Multiply mode and Screen mode at the same time. 50% brightness is neutral (nothing happens). Lighter values act like Screen, darker values act like Multiply.
- **Soft Light:** This is a combination of the Burn and Dodge modes. The Dodge mode is used for blend pixels greater than 50% brightness, the Burn mode is used for values below 50%.
- **Dodge, Burn:** The brightness value of the blend pixel is used to brighten (Dodge) or darken (Burn) the background pixel.
- **Darken:** The background and blend pixel are compared and the result is whichever is darker.
- **Lighten:** This is the same as darken except the result is whichever pixel is lighter.
- **Add:** One of the simplest blend modes: the background pixel and blend pixel are added together.
- **Subtract:** The background pixel and blend pixel are added together then white is subtracted (result = background color + blend color - 256).
- **Difference:** The background and blend pixels are subtracted from each other.
- **Exclusion:** This mode is the same as Difference mode except that it creates a softer effect.
- **Hue, Saturation, Luminance:** In these modes, the result is the hue, saturation or luminosity of the blend pixel applied to the background pixel.
- **Levr:** The value of the blend pixel is used to apply contrast to the background pixel.

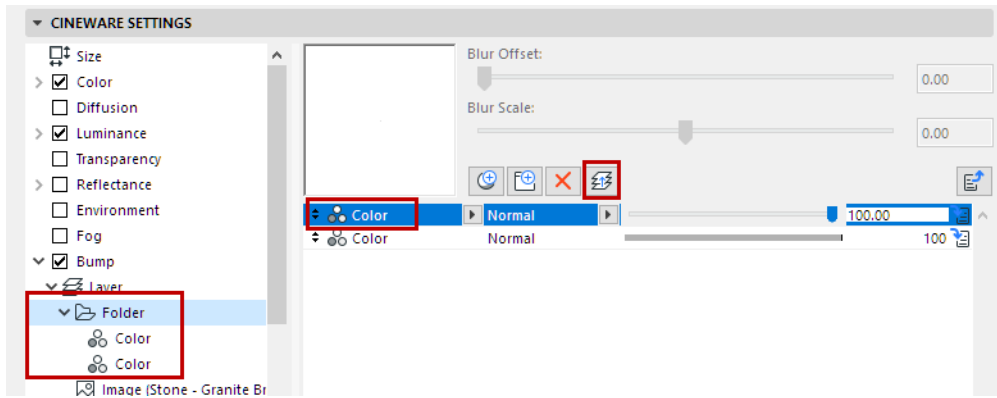
Organize Shaders into Folders

If you have loaded multiple shaders or images into a surface's Texture Layer, you can use Folders to group the shaders. Folders are available only within Layers.



- Click the Folder icon to create a new folder.

- Within the list of shaders on the right half of the dialog box (but not in the tree hierarchy at the left), select any shader, then drag it into the folder. You will see the corresponding hierarchy in the tree structure, too. You can use any number of folders or Layer shaders, and each can contain any number of shaders.
- To remove a shader from a folder or from a Layer shader within another Layer, select the shader and click the Move Shader Up One Level button.

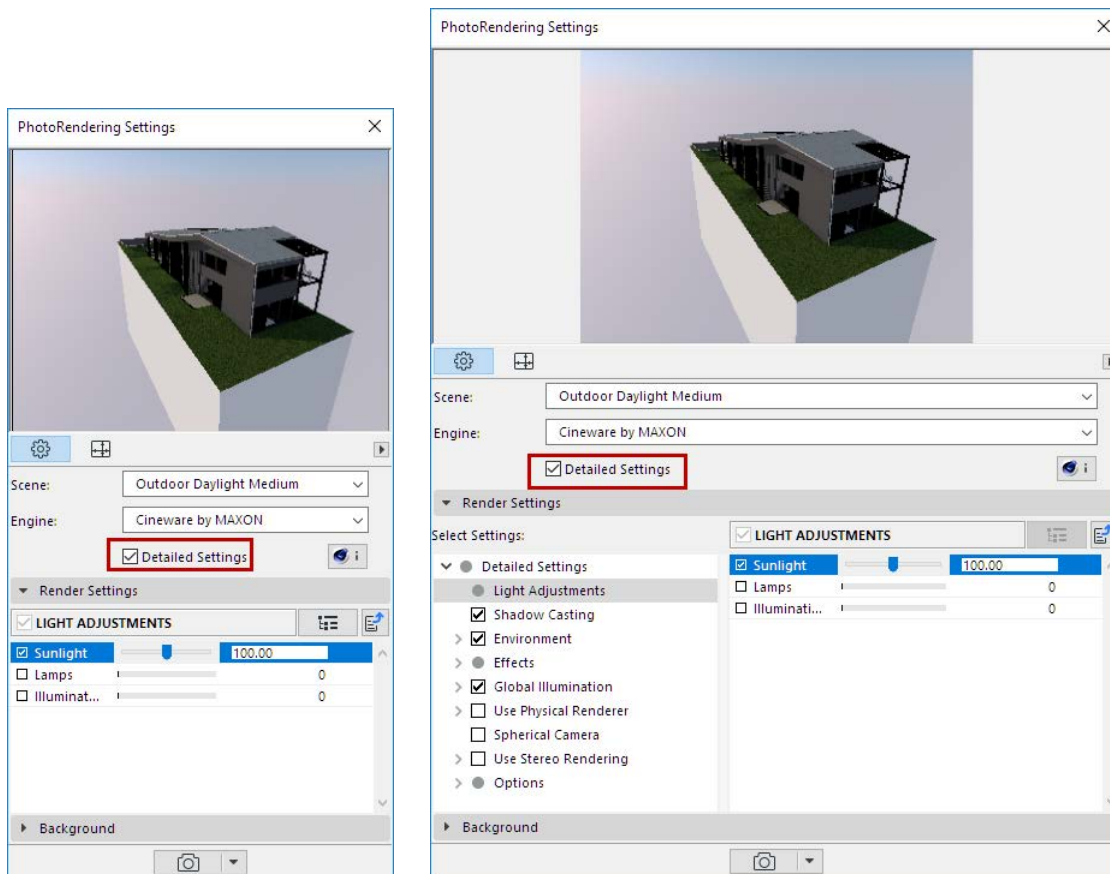


Detailed Cineware Settings

To access the full set of Cineware parameters, go to PhotoRendering Settings (Document > Creative Imaging).

Select the Cineware engine, then check the **Detailed Settings** box.

Use the compact view, or stretch the dialog box to see the full view, with a tree structure of all the settings on the left.



If you are not familiar with PhotoRendering parameters, the detailed view settings can be daunting. If so, the Basic Settings are for you. Uncheck the Detailed Settings box and return to Basic Settings, where a half-dozen parameters are all you need to worry about, and pre-set scenes and environments enable you to create high-quality renderings very easily.

[See Cineware Basic Settings.](#)

Information on Detailed Cineware Settings is found in the following sections:

[General Considerations When Using Detailed Settings](#)

[Navigating in the PhotoRendering Settings Palette](#)

[List of Detailed Cineware Settings](#)

General Considerations When Using Detailed Settings

The following are general considerations related to some of the Detailed parameters of Cineware Settings.

Physical vs. Standard Renderer

Optimize Rendering Time with “General Options”

Consider Global Illumination

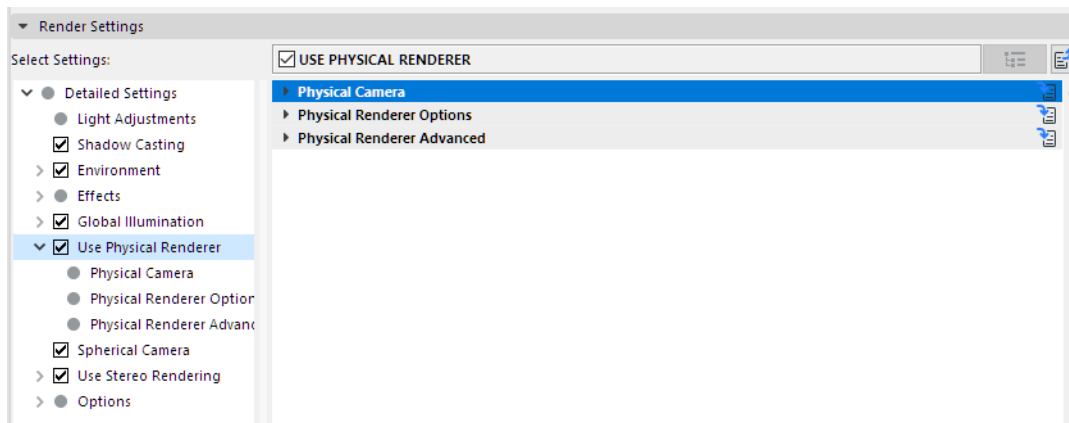
Progressive Rendering Mode

Physical vs. Standard Renderer

By default, Cineware uses a rendering mode called “Standard”, which will give good results for most users.

The alternative is the Physical Renderer, which simulates the working of a physical camera and has corresponding parameters (e.g. shutter speed).

To activate Physical Renderer, check the **Use Physical Renderer** box in the Detailed Settings view of Cineware Settings.



Some considerations when choosing a Renderer mode (Standard vs. Physical):

- If you have a standard-sized building without blurry reflections: Use **Standard** render mode
- If you have a large, detailed model including a variety of surfaces with complex parameters, and/or many blurry reflections: **Physical** render mode will decrease rendering times for these operations
- When rendering bitmap objects, use the Physical renderer

Characteristics of the Physical Renderer

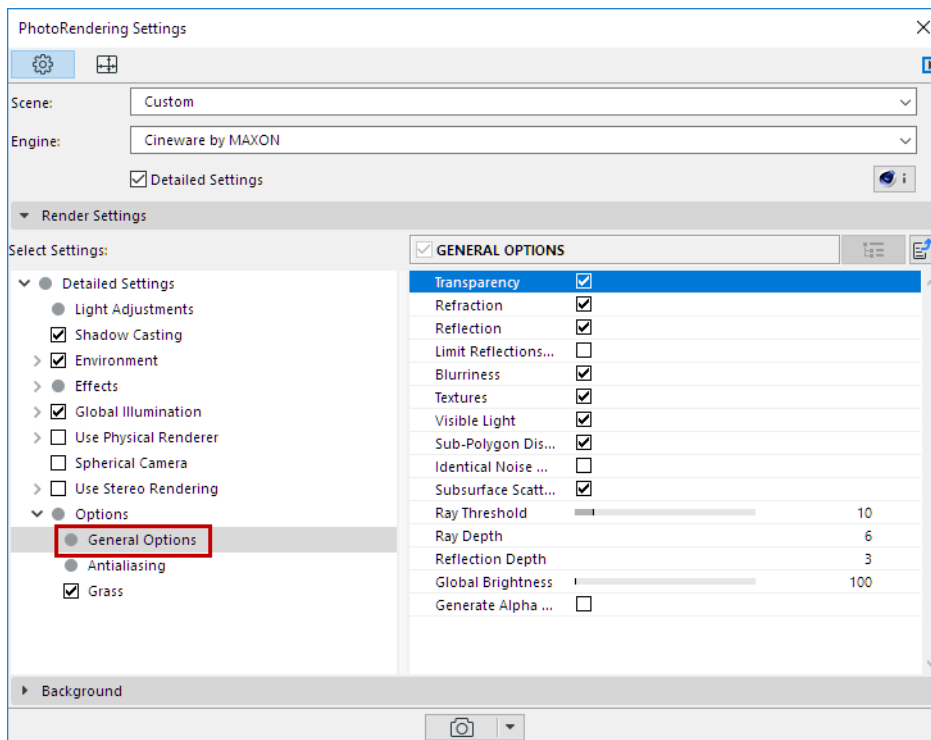
- Tends to create a grainy image, if the settings are not optimized
- Correctly depicts the following photographic effects:
 - Rendered depth of field with corresponding blur effects
 - Vignetting (darkening towards the image edge)
 - Chromatic Aberration (color seams at edges)
- When rendering multiple, combined blur effects (blurry reflections/blurry transparency) the Physical Renderer is faster than the Standard renderer.
- Easier to set up: the Physical renderer does not use Min/Max Samples and Accuracy values in light and Surfaces
- Does not use Antialiasing

- includes different RayTracing Engines to choose from: Physical (legacy), Embree (TurboBoost), and Embree.
- Includes different rendering algorithms to choose from: Fixed/Adaptive and Progressive mode. Using the progressive mode, one can get a grainy but still realistic image immediately after pressing the render button (and after GI, if used), and thus the rendered image becomes progressively less grainy as rendering time passes. ([See Progressive Rendering Mode.](#))
- Depth of Field works like a camera setting (F-stop). (In contrast, Depth of Field in the Standard renderer is a post effect.) This makes the Physical renderer preferable if:
 - you are using Textures with alpha effects and want to combine these with Depth of Field
 - your model contains many transparent surfaces (this can look artificial if rendered as a post effect, as in the Standard renderer).

Drawbacks of the Physical Renderer

- The realistic simulation of physical effects requires more calculation power and more time. It does, however, offer several more settings (which offer great potential for optimization).
- Physical rendering does not work in conjunction with Specular lights or cylindrical lens; grass is rendered much more slowly.

Optimize Rendering Time with “General Options”



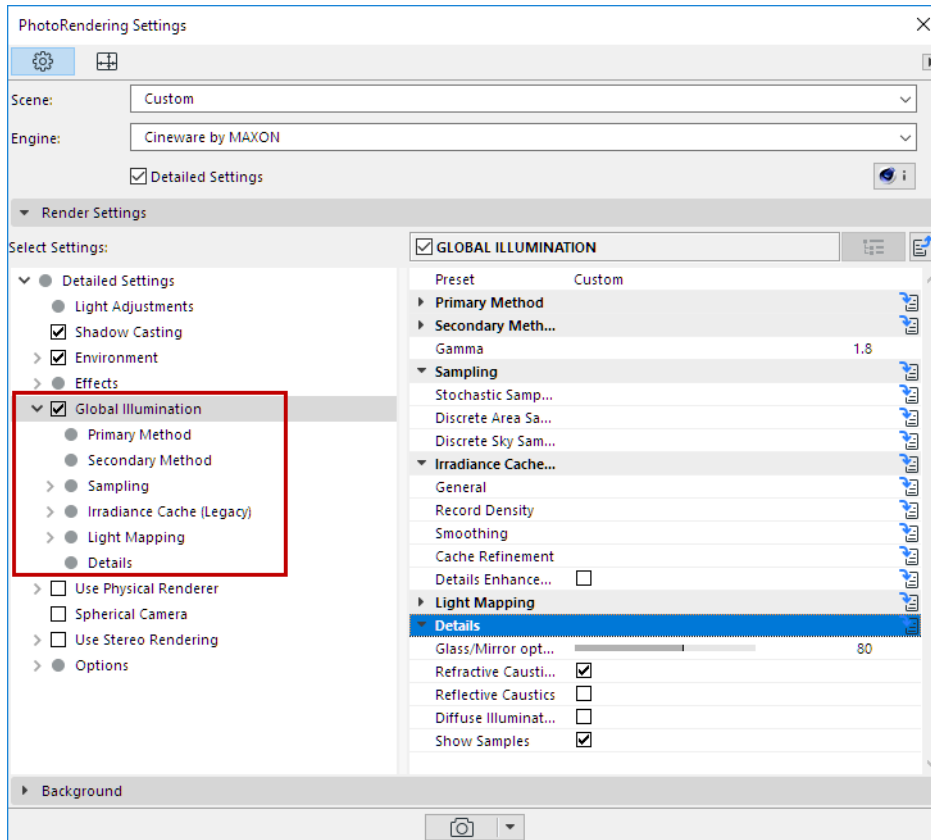
This list of options affect the rendering as a whole: if you uncheck an option here, it is disabled for the whole rendering model, regardless of what else you have set elsewhere. It is worth considering which of these can be turned off so as to minimize render time. For example, you can turn off all blurriness with a single click.

Consider Global Illumination

Global Illumination (GI) is a sophisticated method for achieving realistic lighting effects in your model. The algorithm simulates the effects of not just direct light sources, but indirect light effects - that is, inter-reflections caused by light rays bouncing off surfaces.

With Cineware, you can achieve very high-quality renderings using Global Illumination. However, it can greatly increase render time. Two methods can be used in conjunction: Primary and Secondary methods.

See [Primary and Secondary Method \(Cineware Global Illumination\)](#).



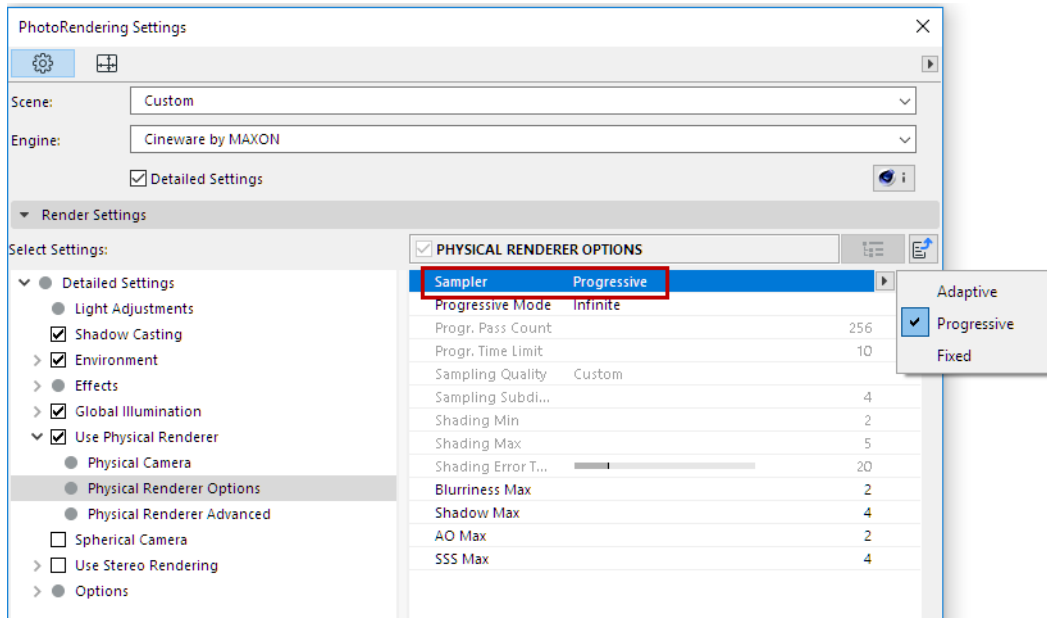
Of the many GI-related parameters, the Diffuse Depth value can make a significant difference in rendering quality. It defines the number of times light reflects in a scene, i.e. how often a “ray of light” is reflected from surfaces.

- The minimum Diffuse Depth value of 1 results in only a direct illumination via flat, light-emitting elements. This will be sufficient for most external scenes, with Physical Sky or HDRI Sky providing a significant light source.
- A Diffuse Depth value of 2 is required to achieve an indirect illumination. A minimum value of 2 will be required for interior scenes.

[Watch the Video](#)

Progressive Rendering Mode

If you are using the Physical renderer, its three sampling settings include the Progressive option: after you start the rendering process, you get immediate basic feedback. Then, as time passes, the image is gradually fine-tuned.



This option, in turn, has three ways to set a limit on the render process:

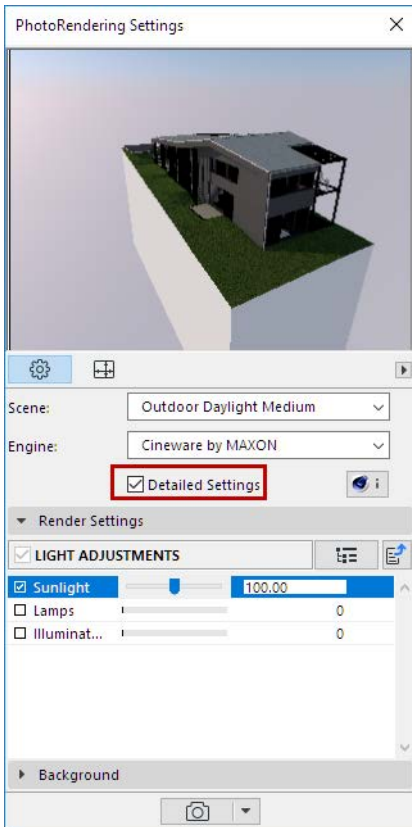
- Infinite: The rendering continues indefinitely until you press Stop.
- Time Limit: The rendering process will stop after a defined time interval.
- Pass count: The rendering process will stop after a specified number of passes.

With Progressive Rendering, the rendering image goes through several progressive passes, while the image becomes visibly less grainy. By using the Infinite option, you may click to stop rendering after a certain time when you feel your image is already fine enough.

In fact, it takes far less than infinity to achieve a desirable result. After a certain number of render passes, it will be almost impossible to see any further changes, since the image is so finely rendered.

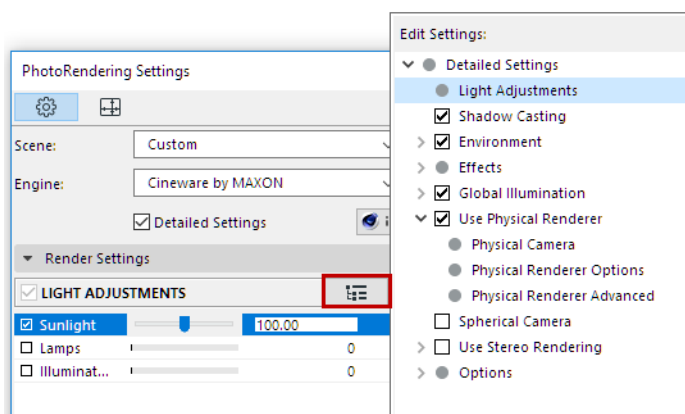
Navigating in the PhotoRendering Settings Palette

To access the full set of Cineware parameters, go to PhotoRendering Settings (Cineware engine) and check the Detailed Settings box.



The appearing dialog box, in compact view, shows only one settings category at a time.

Click the tree view button to access a tree view of all the Settings:

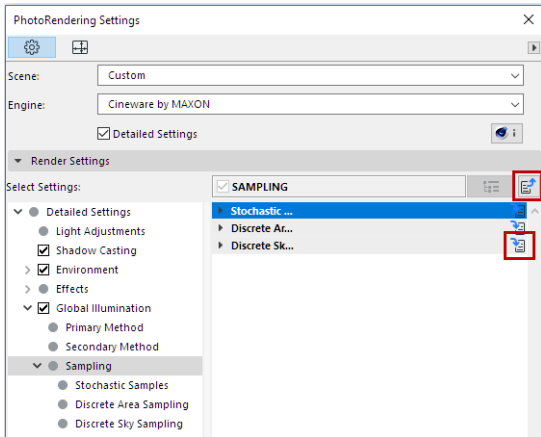


Or stretch the dialog box wide to see the full tree structure as part of the dialog box:

Parameters are often sorted into groups. To be able to edit the full set of parameters for each setting (e.g. Sampling), make sure the setting's checkbox is turned on. Then do one of the following:

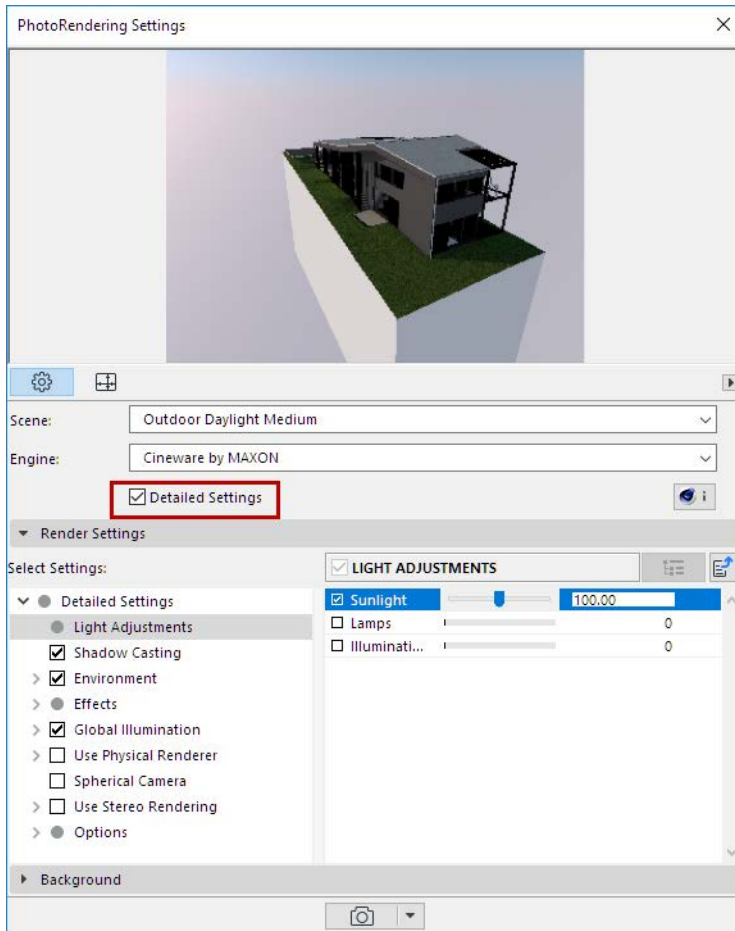
- click the name of the setting in the tree list, or
- click the One Level Down icon for the setting in the parameter list.

To return to the previous view of the Settings list, click the One Level Up icon.



List of Detailed Cineware Settings

The following sections describe the settings available on the Detailed view of the PhotoRendering Settings Palette (for the Cineware engine).



[Light Adjustments \(Cineware\)](#)

[Shadow Casting \(Cineware\)](#)

[Environment: Physical Sky \(Cineware\)](#)

[Environment: HDRI Sky \(Cineware\)](#)

[Cineware Effects](#)

[Global Illumination \(Cineware\)](#)

[Physical Renderer \(Cineware\)](#)

[Spherical Camera \(Cineware\)](#)

[Stereo Rendering \(Cineware\)](#)

The Options category contains the following three parameter groups:

[General Options \(Cineware\)](#)

[Antialiasing \(Cineware\)](#)

[Grass \(PhotoRendering Option\)](#)

Light Adjustments (Cineware)

Use three sliders to set the intensity of three different types of light used in the rendering: Sunlight; Lamps; and Illuminating surfaces.

Each slider can override a particular light source intensity that is set elsewhere in the project.

- **Sunlight.** Can override the Sun's intensity setting. The midpoint of the slider corresponds to 100 percent: this means that the rendering will use the sun intensity exactly corresponding to the Sun object's intensity setting. Adjust the slider to increase or decrease the sun intensity relative to this setting.

Note: If you are using a Physical Sky as your Environment, then this Sun intensity slider affects the sun that is part of that Physical Sky setting. If you are using an HDRI image as your Environment, then this Sun intensity slider affects the Archicad Sunlight settings.

[See *Cineware Basic Settings: Environment Panel.*](#)

- **Lamps:** Can override the Main Intensity settings (set in Lamp Settings) for all individual lamps placed in the project. The midpoint of the slider corresponds to 100 percent: this means that the rendering will use the lamp intensity exactly corresponding to the intensity set for lamp objects placed in the model.

[See *Light Color and Intensity for Lamp Tool Settings.*](#)

Note: These Rendering settings consider the settings that affect the 3D window, but the reverse is not true: the 3D Window does not consider what you adjust in Rendering settings. So a lamp that is turned off in the 3D window cannot be turned on in Rendering Settings.

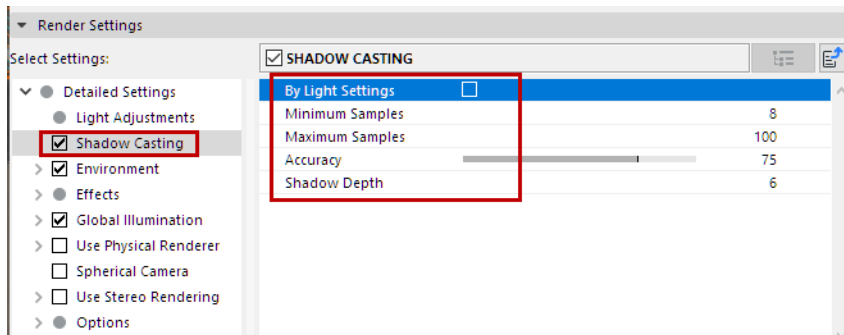
- **Illuminating Surfaces:** This setting overrides the surface-level **Luminance** channel.

Note: All surfaces also have an Illumination channel, which affects Global Illumination. But the Illuminating Surfaces control in Photorendering Settings affects the Luminance only.

Shadow Casting (Cineware)

Check this setting to enable Shadow Casting in your rendering. (If you leave it unchecked, the rendering will not generate any shadows.)

Use the controls below to define Shadow quality:



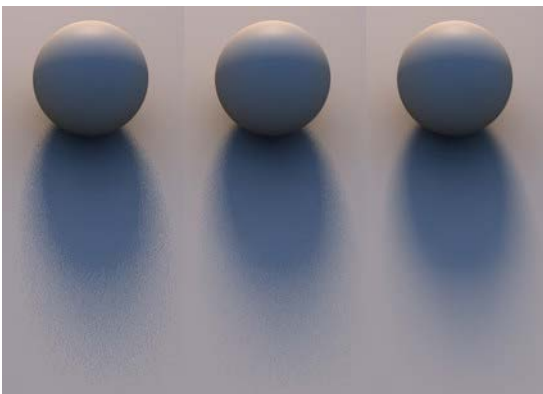
- **By Light Settings** means that the Shadow Casting setting for each individual placed light object (as defined on the Lamp Settings Light Parameters page) will be in effect in this Scene. ([See also Shadow Casting.](#))

If you are not using Light Settings, use the following three settings to control the quality of your Area shadow:

Note: The following three parameters are relevant only if you are using the Standard renderer. They have no effect in Physical rendering mode!

- Minimum Samples
- Maximum Samples
- Accuracy

The more samples, the more homogeneous (less grainy) the rendering will be, and the longer it will take to render. Fewer samples, on the other hand, mean shorter render times.



Minimum Samples and **Maximum Samples** let you set many or few samples, as needed (as well as an interpolation between them).

Of course you could render a scene with the maximum number of samples possible, but this would not make sense: parts of the scene would only require a minimum amount of samples, and the scene would take much too long to render.

The **Accuracy** setting determines how (and how many) samples will be allotted in order to achieve the best result. In the most critical areas, Maximum Samples may be set to its highest value.

The Accuracy setting has the most influence in critical areas, since high values lead to more samples being used. In less critical areas, the Minimum Samples value is applied.

If Minimum Samples=Maximum Samples, Accuracy will have no effect.

Follow these steps to achieve the optimal settings for your scene:

1. Find out where the critical areas in your scene are (grainy areas).
2. Set Minimum Samples and Maximum Samples to the same value and render the scene using Render Marquee Region.
3. Raise the Minimum Samples and Maximum Samples equally until you achieve a good result.
4. Lower Minimum Samples to 25% of the Maximum Samples value and set Accuracy to 50%.
5. Finally, raise the Accuracy setting until you have a result you are happy with.

Also, make a note of these rules of thumb:

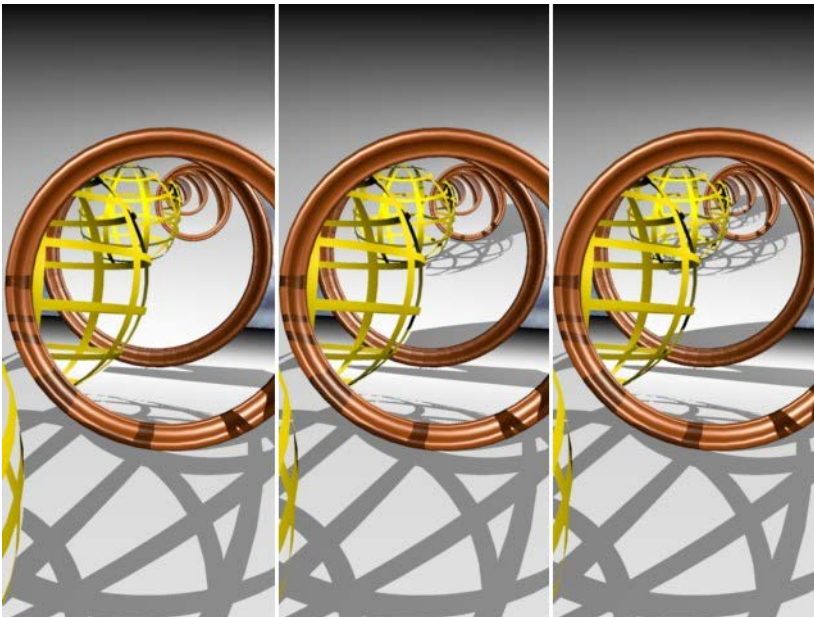
- Quality can be improved the most by raising the Maximum Samples and Accuracy values. A 100% Accuracy will not do you much good in combination with a very low Maximum Samples value.
- No matter which of the three settings you raise, render time will increase accordingly.

Shadow Depth

Shadow Depth behaves analogous to the Reflection Depth. If a surface point lies in the shadow of another object this option will use additional shadow rays to check which shadows should be sent from the object surface in the direction of the light source.

The Shadow Depth setting defines the shadow depth with which visible shadow rays will be calculated. For example, if this value is reduced to 2 no shadows will be calculated for reflective, transparent or refractive rays.

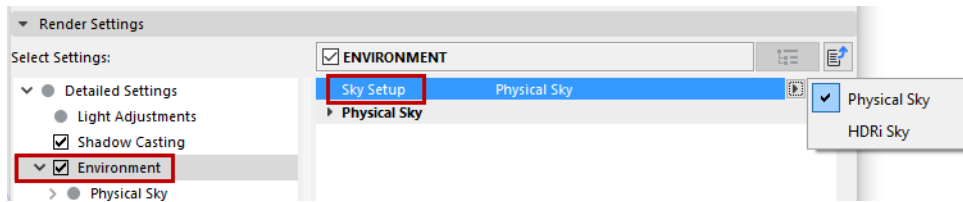
The examples below were rendered with Shadow Depth values of 2, 4 and 8, respectively.



Environment: Physical Sky (Cineware)

The Physical Sky is an entire constellation that includes multiple objects having properties (sun, weather, clouds, fog, rainbow, atmosphere). A Physical Sky provides both a background image and lighting effects. It is “smart” and the parameters are interconnected.

Choose Physical Sky from the **Sky Setup** parameter.



Compositing

The following effects are on by default, and their respective parameters are defined in the weather presets. However, you can turn any of them off, or leave just one of them on, to achieve certain effects and/or optimize rendering time.

- **Sky Visibility:** If you turn this off, the Physical Sky parameters still affect the rendering, but you cannot see the sky itself.
- **Sky Luminance:** You can turn on Sky Luminance with the other effects off: this will give you the light effect of the Physical Sky without anything else.
- **Sky Reflection:** If only Sky Reflection is on, then you can, for example, see the environment reflected on a glass building, without rendering the environment itself.
- **Sky Refraction:** Turn on/off the Physical Sky’s refraction effect.

Time and Location

Do one of the following:

- **Use Archicad Sun Position:** Checking this box will reset the Time & Location according to the sun location. Use this to fine-tune the weather preset you have chosen to match the conditions at the project location.

[See also Note on Sun Settings in Archicad.](#)

- Manually enter the Date/Time and Location (in geographic coordinates) that will affect your rendering scene.

Sun

The sun is an Area light source. The following parameters can be used to define the characteristics of the sun light source. These have no influence on the look of the sky, on the Archicad project sun, or any Sun Object.

Use Warm Colors: This setting increases the “warm” colors in the sun.

Intensity: Defines the intensity of the sun. If, for example, your scene is over exposed due to strong surface reflections, adjusting this value can help.

Saturation Correction: Defines the sun’s color saturation. If you want white sunlight, enter a value of 0% here.

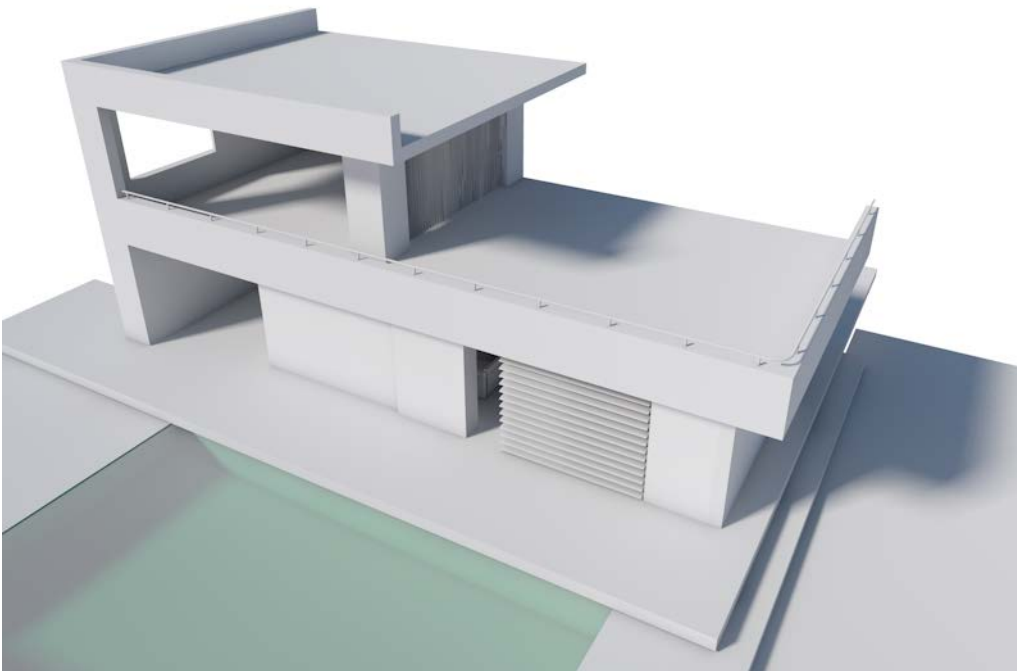
Hue Correction: Depending on the value used, any color in the spectrum can be generated. Great for creating that extraterrestrial look.

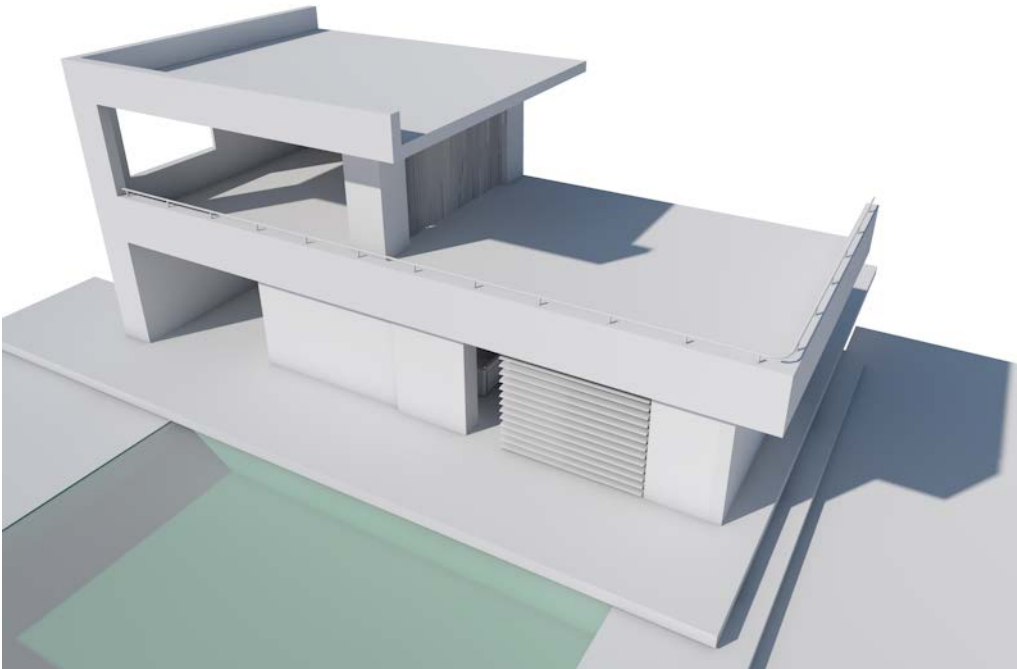
Gamma Correction: Defines the sun's gamma value. This value can be used to prevent over exposing a scene even if the Intensity value is too high. Animating the Gamma Correction parameter for day to night animations can help reduce the brightness of the noon sun.

Size Ratio: Defines the size of the visible sun.

Note that this also has an effect on the Area shadows. If you want a hard sun shadow, enter a value of 0% here.

In the first picture below, a higher Sun Size Ratio causes the shadows to be softer, which corresponds more closely to diffuse lighting situations in real life.





As Seen Intensity: Defines the visibility of the sun. Does not otherwise affect emitted light, etc. This parameter is dependent on the Intensity parameter. If Intensity is set to 0%, setting the As Seen Intensity to, for example, 1000% will not make the sun visible.

Use Custom Colors: Enable this option if you want to create a custom color using the color field below.

- The **sun color** will have no effect on 2D Clouds.

Lens flare: Enables or disables all lens effects.

- **Glow Intensity:** Lens glow and lens reflection. Defines the intensity of the lens glow. Set this value to 0% if you do not want any glow to be seen.
- **Flare Intensity:** Set this parameter's value to 0% to disable the flare altogether. Otherwise you can enter a different value to define its intensity.
- **Distance Scale:** If you encounter problems with the light sources integrated into the sun or moon (e.g. if lens flare or lens glow are displayed incorrectly), simply reduce their distance using this setting.

Shadow

Density: Set Shadow density using the slider. A value of 50% will make the shadow semi-transparent. Values in excess of 100% can also be entered.

Shadow Color: Defines the color of the shadow cast by the sun.

Transparency: Check this box to have transparent objects cast a shadow.

The following three parameters are relevant only if you are using the Standard renderer. They have no effect in Physical rendering mode!

- Minimum Samples
- Maximum Samples
- Shadow Accuracy

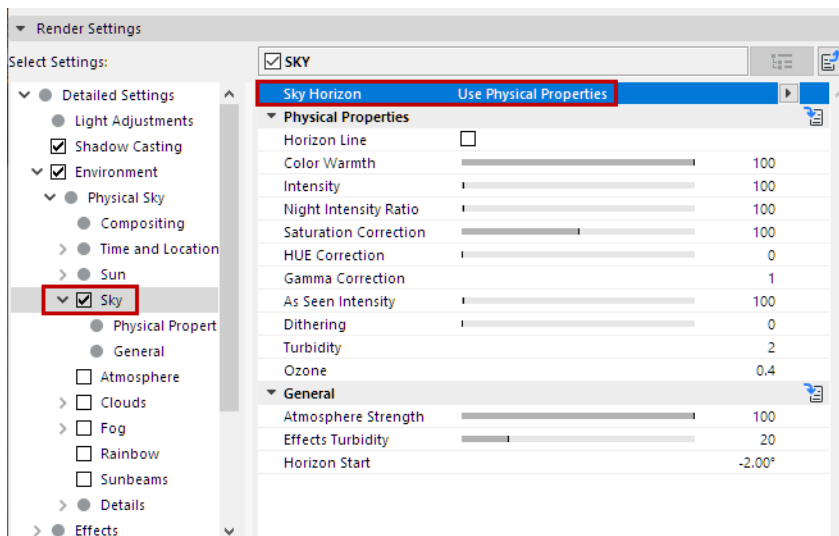
These parameters function just like the Light parameters (with Area shadow) of the same name and pertain exclusively to sun Area shadows. (See [Shadow Casting \(Cineware\)](#).)

Sky

The Sky shader is an accurate simulation of the sun's and sky's natural color and brightness with regard to their current location and the time of day.



- **Sky Horizon:** Choose either Physical Properties, Custom Gradient, or None. If you choose Physical Properties, the following additional parameters are available:



- **Horizon Line:** If enabled, the sky will be clipped at the bottom at the horizon line, where it will be colored black. Otherwise the southern hemisphere will be given the color (and brightness) of the horizon.
- **Color Warmth:** Physical Sky works internally with spectral colors that have to be converted to RGB colors for rendering. This setting increases the “warm” colors in the sky (i.e., less turquoise in a blue sky and more brown in a morning or evening sky).

- **Intensity** [0..10000%]: Defines the intensity of the sky's brightness. This affects the light used by the sky for GI calculation and the direct (most often blue) illumination of objects. The Intensity only regulates the visible intensity (without the influence of GI).
- **Night Intensity Ratio** [0..10000%]: Defines the brightness with which the night sky is rendered as well as the brightness of the light it emits. This parameter is dependent on the Intensity parameter. If Intensity is increased, the Night Intensity Ratio will increase accordingly, without the value itself being increased.
- **Saturation Correction** [0..200%]: Defines the sky's saturation. If, for example, the sky is too blue, simply lower the Saturation Correction value. A value of 0 will result in a grayscale sky.



At left a lower Saturation Correction value. At right, a higher Saturation Correction value. Note how the sky's blue color at the right is much more intense on the walls and the sofa.

- **Hue Correction** [0..100%]: This parameter can be used to create extraterrestrial sky colors. The entire color spectrum is available for you to use.
- **Gamma Correction** [0.1..10]: The internal brightness range for the sky is much larger than can be displayed on your monitor. Use the Gamma Correction value to define the brightness range from brightest to darkest values.



- **As Seen Intensity** [0..10000%]: Defines the brightness of the rendered, visible sky only (also dependent on the Intensity parameter). The GI calculation will not be affected!
- **Dithering** [0..100%]: If color banding (sudden color transition) appears when the sky is rendered, increase this value to add a slight noise to the sky's gradient, which will hide such banding.
- **Turbidity**: Many natural color phenomena in the sky are caused by various combinations of light and water or other particles in the atmosphere. In Cineware, this is called "turbidity".
The smaller the value, the clearer the atmosphere will be. Higher values will result in correspondingly in more foggy or smoky, as well as more colorful, effects.
 - **Tip:** In case the sky is too bright or too dark, the Turbidity and Atmosphere Strength (see below) parameters will have the greatest influence in rectifying this.

- **Ozone:** In the real world, ozone filters ultra violet sunlight. In visual terms, it filters yellow and red sunlight (i.e., the Sun option must also be enabled). Therefore, the higher the Ozone value the more bluish the sunlight will appear (in the real world, this takes place at dusk and dawn). The color of the sky itself remains unaffected.

General

- **Atmosphere Strength** [0..100%]: The atmospheric strength affects the brightness of the atmosphere (the same effect as the Turbidity parameter, described above). Lower values will cause the atmosphere to fade to black; larger values will fade to white (if the Turbidity value is high enough).



Atmosphere Strength value reduced to 50%

- **Horizon Start** [-89..89°]: Varies the height of the horizon. If, for example, you have a scene in which you do not have an endlessly large Floor element, you may find that you can see under the horizon. If this happens, enter a negative Horizon Start value and the sky color will be stretched or compressed accordingly.

Atmosphere

Note: The default values here are generally adequate and you will rarely have to adjust them.

This effect can be described as follows: you look at a mountain panorama and notice how mountains in the distance seem to be increasingly covered by a bluish haze.

This atmospheric effect (also called “atmospheric perspective”) is caused by two main factors:

- Absorption of light
- Dispersion of sunlight and skylight

Intensity: Defines the brightness of the atmospheric haze. Larger values quickly result in an unrealistic, overexposed effect.

Horizon Fade: This parameter helps you mix the color of the atmosphere with that of the sky, creating an atmospheric perspective effect. A value of 100% will result in only the horizon color being used.

Saturation Correction: Adjusts the atmosphere’s saturation. Smaller values result in a grayish, colorless haze.

Hue Correction: The atmospheric haze can contain any color, depending on the value entered here.

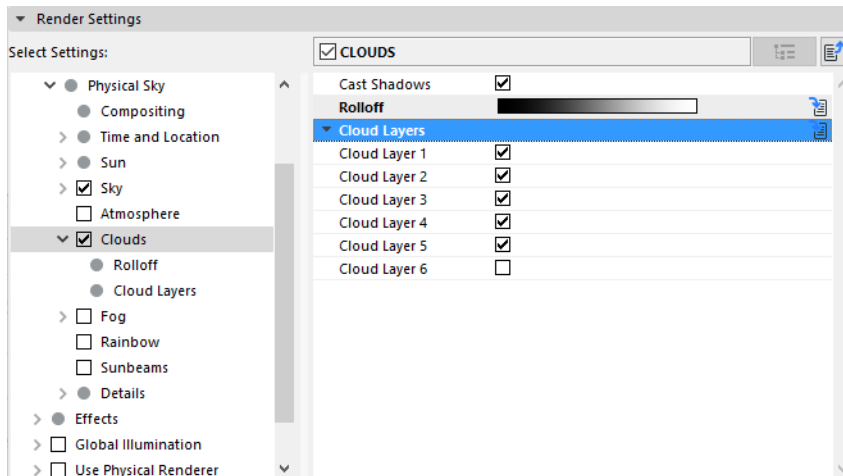
Gamma Correction: This setting can be used to increase the atmospheric effect depending on the lighting situation. It can, in certain instances, help improve the overall look of the effect.

World Scale Ratio: This algorithm assumes that 1000 meters in the real world are equal to 1000 units in Cineware. This ratio can be modified using this setting. Values greater than 100% will increase the atmospheric effect, e.g. 500 Cineware units are equal to 1000 meters in the real world.

Dithering: If color banding (sudden color transition) appears when the atmosphere is rendered, this value can be increased to add a slight noise to the sky's gradient, which will hide such banding.

Clouds

Use the Clouds controls to edit the look of your 2D clouds.



A special method is used to project 2D clouds onto the celestial sphere to give a realistic impression of cloud cover that stretches across the entire firmament.

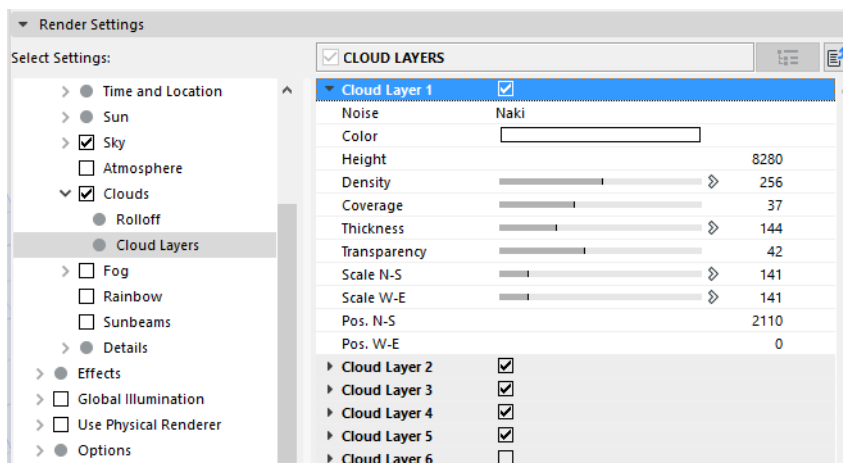
Cast Shadows: Defines whether or not 2D Clouds should cast shadows.

Rolloff: This grayscale gradient causes the sky to fade to the cloud color near the horizon, thus giving the cloud cover a realistic look as it approaches the horizon.

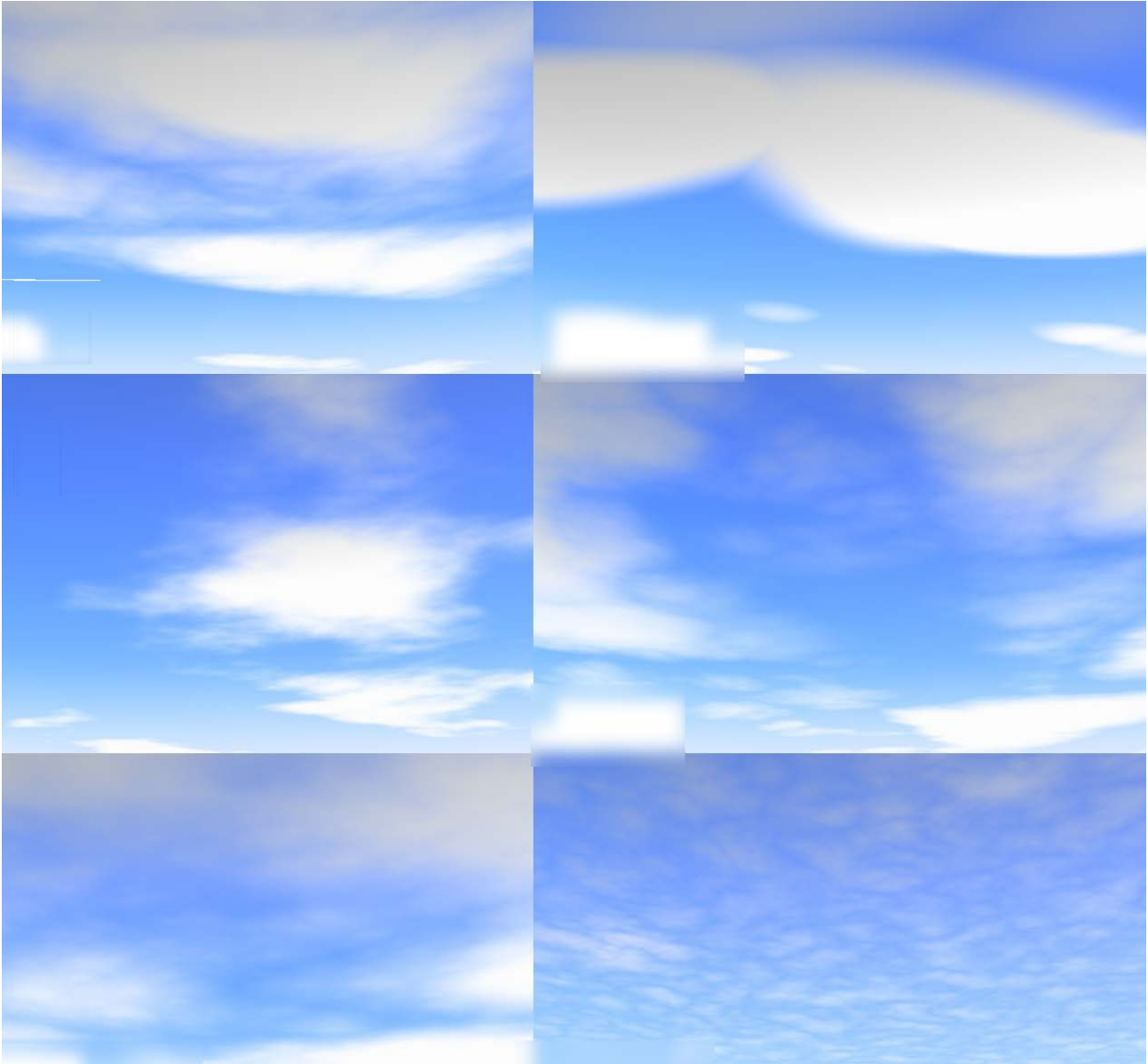
[See also *Adjusting Gradient Parameters*.](#)

Cloud Layers: The levels can be turned on or off via the Levels 1-6 options.

2D Clouds can be made up of up to six cloud layers, each of which can be turned on or off separately. Each layer offers the same settings.



- **Noise:** Noise turns an otherwise smooth surface into a more or less randomly uneven surface. Here you can select the type of noise with which your 2D clouds should be created.



- **Color:** Define the color of your cloud here; its color will be independent of the sun's color or that of other sources of light. Only the cloud's brightness will be adjusted to match the sun.
- **Height:** Set the height of each cloud layer. If you take a look out of the window you will notice that clouds often consist of several layers. Cirrus clouds can, for example, be placed on a layer that is higher (since they reach greater heights in reality as well), and clouds that are darker, rain-filled, and more foreboding can be placed on a lower layer.
- **Density:** Density defines the contrast of the clouds. If a low value is used, the clouds will be very transparent, especially towards the edges. The higher the value, the more homogeneous the clouds will be.

- **Coverage:** This is the most important cloud setting. With it you define the degree of cloud coverage. Very low values result in no cloud coverage; very high values result in a completely overcast sky.



Left, low Coverage value; right, higher Coverage value

- **Thickness:** Thickness is the measure of the degree to which the sunlight will be dispersed or absorbed. Lower values result in the clouds being brightened more by the sunlight, higher values result in darker clouds.
- **Transparency:** Transparency regulates the degree to which 2D clouds cast shadows onto other objects in the scene. Enter a value of 100% if no shadows should be cast.
If, in addition to the sun, you add light sources that cast shadows, make sure they are placed at a height greater than 10,000 m
- **Scale N-S; Scale W-E:** These settings let you scale the noise pattern in the North-South (N-S) or West-East (W-E) directions. You can easily create long, thin, stretched cloud formations by scaling them in one direction only.
- **Position N-S; Position W-E:** Use these settings to move your clouds in the North-South (N-S) or West-East (W-E) directions.

Fog

A fog system has been integrated into Physical Sky - a real volumetric fog, based on diverse types of Noise shaders that render less homogeneously and therefore more realistically.

Note: Using fog can increase render times: the denser the fog, the more time it will consume.

The fog stretches infinitely in a vertically adjustable fog bank.

If you already have Sunbeams activated, any added fog will add much more calculation time.

- **Color:** This is the color the fog should assume. This color will not be affected by other colored light sources, although the sun's color can be multiplied with the fog's color (if the Illumination Intensity setting described below is set to greater than 0).
- **Start Height/End Height:** These settings determine where the fog should begin and end (starting from the ground, Y=0).

The fog bank, which extends infinitely on the X- and Z-axes, must at least be limited in its vertical extension. Otherwise you would not be able to see anything in the resulting soupy fog.

You can determine how quickly density is lost between the Start Height and End Height settings by adjusting the **Density Distribution** (see below).

- **Max. Distance:** Use this setting to optimize render times. Roughly speaking, this distance is calculated from the camera's point of origin, looking into the scene. Only the fog that lies within this range will be calculated.
- **Density:** Density is the measure of the fog's density, i.e. the higher the value, the more opaque the fog.
- **Density Distribution:** Regulates the density between the Start and End settings.

Rainbow

A rainbow is created by light refracting through raindrops. A rainbow's position is dependent upon the position of the sun. To that end, remember the following rules:

- A rainbow's apex always lies exactly opposite the sun.
- The lower the sun's position, the higher a rainbow's apex will lie.

As in reality, there always exists (even if not always visible) a second, less visible mirrored (in color) rainbow with a larger radius.



Two types of rainbows are available (here shown with a high Max. Strength value)

Spatially, a rainbow will always be arranged in front of all Sky objects (even clouds).

Max. Strength: Use this control to adjust the rainbow's transparency. The second rainbow will be created according to the properties of the first. The lower the value, the more transparent the rainbows will be.

We suggest you use unobtrusive, transparent rainbows - less is more.

Turbidity Dependent: In reality, if there's no rain (or moisture in the air), there's no rainbow. Physical Sky regulates the density of the moisture in the air via the Turbidity setting (see above, at Environment > Physical Sky > Sky > Physical Properties). So, if you want to link your rainbow to this setting, activate this option. If Turbidity is set to 0, for example, no rainbow will be created.

If you activate Turbidity, the following options will be made available:

Min. Turbidity Threshold/Max. Turbidity Threshold

Use these controls to adjust the rainbow's turbidity.

Example: You set the first control to 9 and the second to 50. This means no rainbow will be visible as long as Turbidity is less than 10. If Turbidity is greater than 50, the rainbow will be rendered to full strength. Between 9 and 50, the rainbow will be rendered in correlation with the strength of Turbidity.

First Rainbow Inner and Outer Angle/Second Rainbow Inner and Outer Angle

The precise definition of rainbow angles is not needed here. Simply view the difference of both angles as the measure for each rainbow's width.

If you think rainbows are too narrow, try entering inner and outer values of 30° and 50°, respectively, for the first rainbow.

Start Clipping/End Clipping

Imagine an animated airplane approaching from the distance out of the rainbow. In the distance, the airplane is behind the rainbow; as it approaches it will eventually cover the rainbow. Such effects are possible using these settings.

Objects lie in front of the rainbow up to the Start Clip point; starting at the End Clip point, the objects lie behind the rainbow.

Sunbeams

Sunbeams shining through the clouds can be a beautiful and useful effect.

To make the sunbeams visible, make sure of the following:

- From the point of view of the camera, the sun should be mostly hidden behind the clouds.
- The cloud cover must be structured so that sunbeams can pass through them, i.e. with intermittent holes or clearing.

Using fog ([see Fog](#)) in addition to sunbeams can increase render times.



Sunbeams shine through a hole in the cloud cover

Turbidity Dependent: If, as in reality, you want to let atmospheric particles affect your sunbeams, simply activate this option. This links the visible sunbeams to the Turbidity setting (Environment > Physical Sky > Sky > Physical Properties). The more hazy the air (atmosphere), the better the sunbeams can be seen. If Turbidity is set to 0, no sunbeams will be visible.

Intensity: Here you can adjust the brightness of the sunbeams

Min. Brightness: Use this control to determine at what minimum brightness value (internal) the sunbeam will become visible. Select a high value if you want to make only the brightest sunbeams visible, a low value if you also want to make weak sunbeams visible. The brightness of the rendered sunbeams is defined by Intensity.



Left: low Min. Brightness value; right: high Min. Brightness value

Start Distance/End Distance

These settings, calculated from the camera's point of origin, let you define an area within which sunbeams will be depicted.

Sample Distance: Everywhere that physical effects are simulated (e.g. a ray of light loses intensity over a particular distance), points of measure (samples) are defined that calculate the level of intensity at a given location. The more samples you define, the more precise the result will be. Longer render times will also result.

The same applies to Sample Distance: The lower the value, the more precise the sunbeams will be rendered (and the render times will increase as well).

Details

Use these controls to add additional details to your Physical Sky.

Show Moon: Defines whether or not the moon should be shown.

- **Scale:** Here you can vary the size of the moon.
- **Bright Intensity/Dark Intensity:** Use these settings to define the bright side of the moon (side lit by the sun) and the dark side of the moon (side not lit by the sun). If you want a full moon, set both settings to 100%.
- **Distance Scale:** If you encounter problems with the light source integrated into the moon, simply reduce its distance using this setting.

Show Stars: Check this box to display stars.

- **Star Min. Magnitude:** The Magnitude value represents the apparent brightness of stars on a firmament. Use this control to determine the stars' minimum magnitude at which they should be visible. The following rules apply:
 - Low value: Only the brightest stars will be projected
 - Maximum value: All stars in the internal database will be projected.

- **Resize Stars with Magnitude:** Depending on the magnitude (brightness) of the stars, individual stars will be scaled, thus adding even more realism. If this option is not active, all stars will have the same size.
- **Brighten Stars:** Use this control to adjust the brightness of the projected stars: The higher the value, the brighter the stars.
- **Star Radius:** Use this control to adjust star size
- **Show Constellations:** Check this box to display constellations.
- **Constellation Color:** Constellations can be displayed in any color. If you set anti-aliasing mode to Best in Render Settings ([see *Antialiasing \(Cineware\)*](#)), the constellations (as well as the Grid - see below) will be smoothed when rendered.
- **Grid Width:** The celestial sphere is similar to the earth being divided along longitudinal and latitudinal lines, though here they are referred to as azimuth and height. The grid can be made visible by setting Grid Width to 10°, for example. Azimuth and height will then be represented by a line every 10°.
- **Grid Color:** Defines the color of the grid.
The grid will be smoothed if you set anti-aliasing mode to Best in Render Settings ([see *Antialiasing \(Cineware\)*](#)).

Show Planets: Enable or disable the display of our solar system's planets (Mercury, Mars, etc.).

Sky Dome Light: In addition to the sun (which itself is a "normal" internal light source) an area light source that is (almost always) blue exists that simulates the bluish light emitted by the atmosphere. This light can now be disabled (this bluish light is sometimes not desirable for scenes rendered without GI).

Generate GI

[See *Illumination \(Cineware Surface Channel\)* for information about Global Illumination.](#)

Note: The GI emitted pertains only to the sky, not any other elements, i.e. Fog, Rainbow, etc.

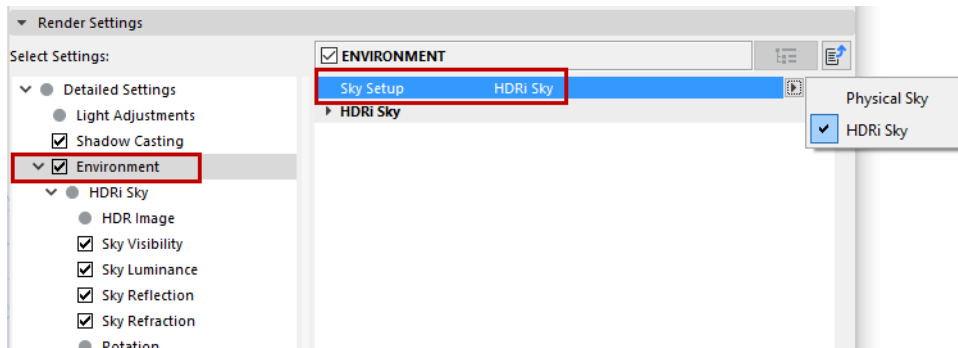
- **GI Strength:** Defines the strength of the generated GI.
- **GI Saturation:** Defines the amount of saturation of the light emitted by the sky, i.e. how blue (or orange for early evening scenes) the scene should be rendered.
- **GI Cloud Influence:** Defines the degree to which the 2D Cloud color should affect the GI. A value of 0% means that the cloud color will be ignored entirely and only the sky itself will be used.

Environment: HDRI Sky (Cineware)

HDRI Sky is an image which provides a background picture and ambient light source for the rendering.

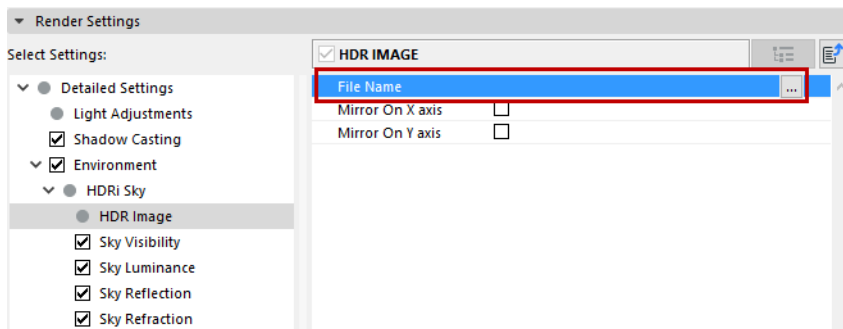
- An HDRI image provides diffuse light and blurred shadows (no sharp shadows)
- The HDRI Sky Setup does not include a separate Sun. To use the effects of the Archicad sun, make sure that the Sunlight is turned on in Cineware Basic Settings (see *Light Sources*). If you increase the Sun slider value, this will also increase shadow sharpness.

From the Sky Setup parameter, choose HDRI Sky.



HDRI Image

In HDRI Image, click the File Name parameter to select an image:



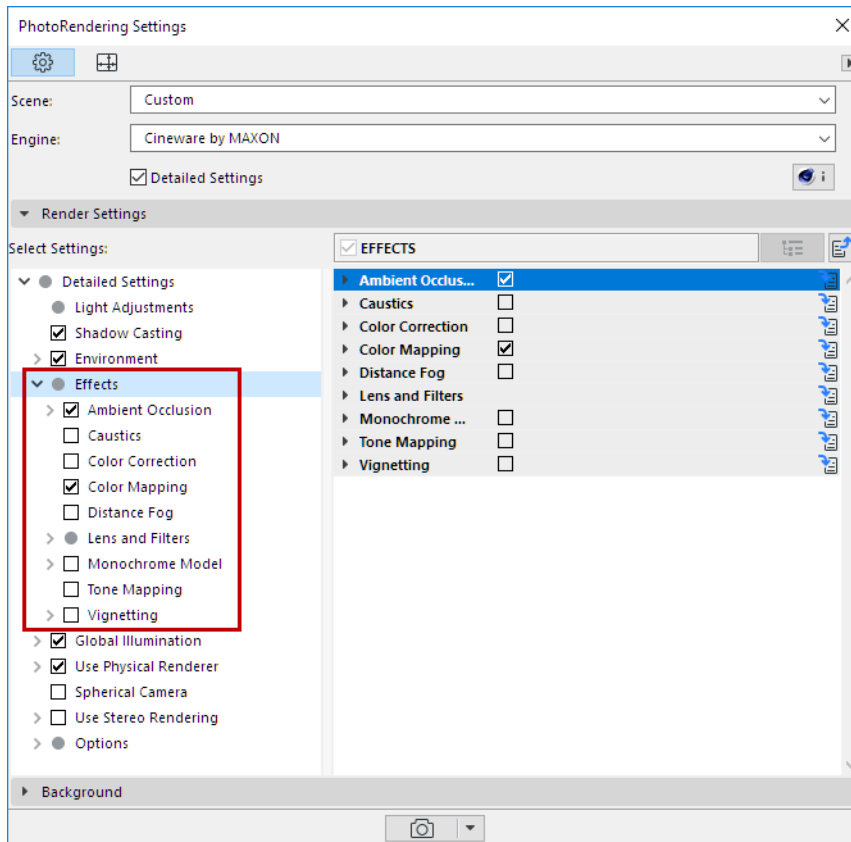
Use a predefined image from the library ([Himg] HDF Images folder), or create your own. (If you create your own, you must use a spherical view, otherwise it will be distorted.)

- **Mirror on X axis/Mirror on Y axis:** Apply a mirror effect to the image.
- The following effects are on by default. However, you can turn any of them off, or leave just one of them on, to achieve certain effects and/or optimize rendering time.
- **Sky Visibility:** If you turn this off, the HDRI Sky parameters still affect the rendering, but you cannot see the sky itself.
 - **Exposure:** Use this setting to adjust the brightness of HDRI as well as normal images.
 - Note: This will have an effect only on the visible, reflected and refracted HDR Sky, but not on the one used for the Sky Luminance.
 - **Black Point/White Point:** These points define the darkest and brightest points, respectively, and save this information to a histogram. If one of these points is subsequently moved, the dark and light levels will be scaled accordingly.

- **Sky Luminance:** You can turn on Sky Luminance with the other effects off: this will give you the light effect of the HDRI Sky without anything else.
 - Use the **Color/Intensity** controls to vary these luminance parameters for the HDRI Image.
 - Use the **Blur Offset** and **Blur Scale** controls to affect the blurriness of the image's luminance effect.
- **Sky Reflection:** If only Sky Reflection is on, then you can, for example, see the environment reflected on a glass building, without rendering the environment itself.
- **Sky Refraction:** Turn on/off the HDRI Sky's refraction effect.
- **Rotation:** Use these controls to rotate the HDRI Sky image.

Cineware Effects

The following effects (available in the Detailed view of PhotoRendering Settings for the Cineware engine) are for optional use, to enhance your rendering. They provide extra scope for fine-tuning your scene. They function as post effects - that is, their effects are added to the scene after it is calculated by the rendering engine.



Ambient Occlusion (Cineware Effect)

Caustics (Cineware Effect)

Cel Renderer (Cineware Effect)

Color Mapping (Cineware Effect)

Color Correction (Cineware Effect)

Depth of Field (Cineware Effect)

Lens and Filters (Cineware Effect)

Distance Fog (Cineware Effect)

Monochrome Model (Cineware Effect)

Tone Mapping (Cineware Effect)

Vignetting (Cineware Effect)

Ambient Occlusion (Cineware Effect)

This effect is available in the Detailed view of PhotoRendering Settings for the Cineware engine.

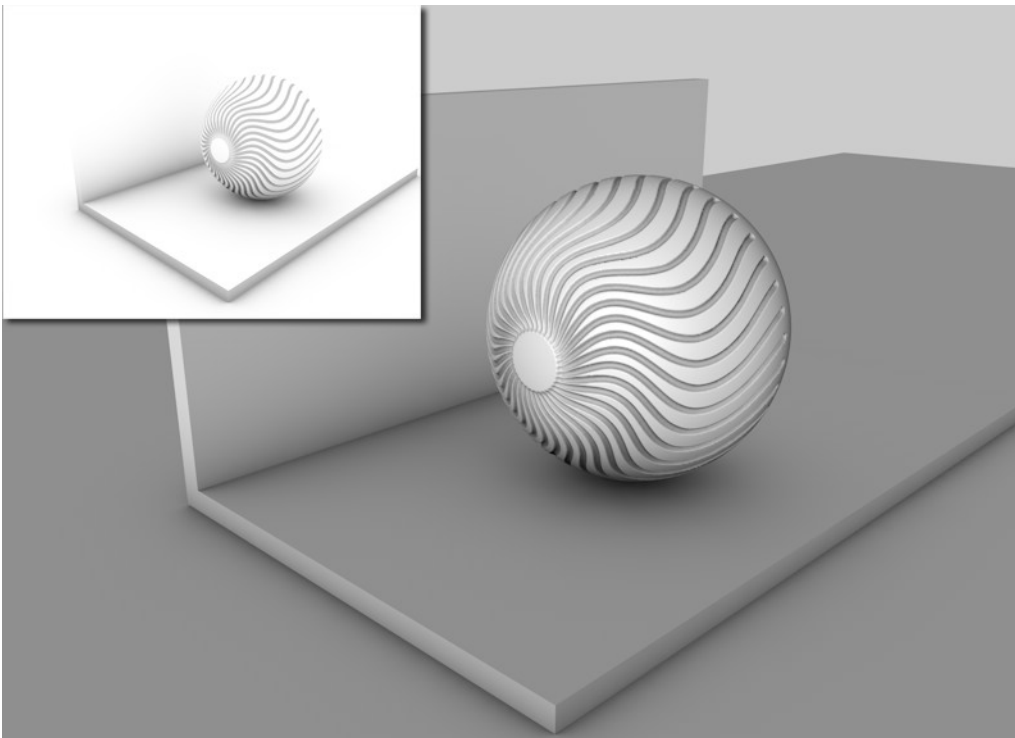
Ambient Occlusion (AO) determines the degree to which each visible surface point is exposed and darkens it accordingly. Ambient Occlusion (AO) offers, within certain limits, a fast alternative to GI.

[Watch the Video](#)

Note: As an alternative to using AO as a global effect on the whole Rendered scene, Ambient Occlusion can be applied as a shader effect to specific Surface Channels.

[See Ambient Occlusion \(Cineware Surface Channel\).](#)

Let's say your scene is surrounded on all sides by a sky. The AO shader will determine to what extent each visible area sees the sky.



Corner areas, holes, and areas between objects placed very close to each other will see less of the sky and will therefore be darkened in accordance with the AO settings.

Two types of Ambient Occlusion calculation are available:

- the usual “brute force” method, which checks the environment’s visibility for each individual pixel
- the faster method via Cache, which only checks certain points’ visibility and interpolates all others in-between.

The latter works internally, similar to the Irradiance Cache GI mode, and is controlled using similar settings. The advantage of using this method is that the AO calculation is much faster.

[See Cache, below.](#)

Color

Use the Color option to define the color gradient that AO assigns, dependent on exposure. Normally, it will be a simple custom black to white gradient but other colors can also be defined.

Be aware of the fact that these gradients will be treated as grayscale, depending on the material channel.

[See also *Adjusting Gradient Parameters*.](#)

General

- **Minimum Ray Length:** Determines how the gradient defined in Color will be rendered between exposed and non-exposed areas. The closer the Minimum Ray Length value is to the Maximum Ray Length value, the further the gradient will be pushed towards the edges, which are defined by the maximum ray length.

This value shouldn't really be tampered with. Leave it set to its default setting of 0.

- **Maximum Ray Length:** This value defines to which distance the surfaces see each other. Where geometry meets, as with the floor and wall or the sphere and floor, even small rays will fall upon surfaces, which is enough to darken those areas.

If high values are used, a much larger distance will be included within which objects will be able to see each other. This results in a softer, more homogeneous darkening that will cause longer render times. In general, lower values are recommended.

- **Dispersion:** During each AO calculation, several rays (Samples) will be emitted for each point within a virtual hemisphere in the scene. These samples check to see if any geometry lies within the Maximum Ray Length. Dispersion determines to what extent these samples will be taken into account on the hemisphere's surface. A value of 0% takes only the hemisphere's zenith (vertically over the sphere) into account. A value of 100% will take the entire hemisphere into account.

Accuracy, Minimum Samples and Maximum Samples (Ambient Occlusion)

These settings are responsible for AO quality.

- Low quality is accompanied by grainy results. This is not necessarily bad. In fact, in some cases, this can be of high aesthetic value.
- If you prefer homogeneous, seamless transitions, you should select higher quality settings, which will also increase render times.

In simple terms: Samples are required to render AO. The more samples that are used, the more homogeneous (less grainy) the rendering will be - the longer the render times will be as well. Fewer samples, on the other hand, result in shorter render times.

Of course a maximum number of samples can be calculated for the entire scene. This would take very, very long to calculate and it does not make any sense since a scene contains many areas in which a relatively small number of samples would suffice.

That's what the **Minimum Samples** and **Maximum Samples** settings are for. They are used to affect the critical and less critical areas of your scene in different ways.

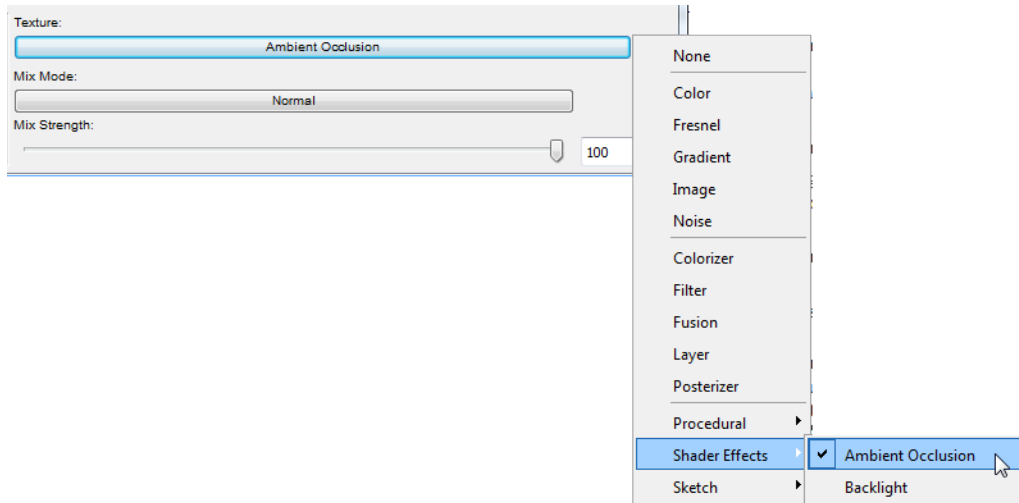
The **Accuracy** setting determines where and in what amount samples must be distributed in order to achieve the best possible result. In critical areas, the maximum amount of samples will be used.

Hence, the Accuracy setting has the most influence in critical areas (since higher values in these areas lead to more samples) and less influence in less critical areas, which use the Minimum Samples value.

- **Contrast:** Use this setting to adjust the AO effect's contrast. Negative values can also be entered!
- **Use Sky Environment:** AO can work without a separate light source if the Physical or HDRI Sky is used as Illumination. If the Environment's Sky checkbox is active, the reflected sky color will be multiplied onto the final AO image.

Note: For this to have an effect, the Ambient Occlusion texture must be activated in the Environment or Luminance surface channels.

See [Ambient Occlusion \(Cineware Surface Channel\)](#).



- **Evaluate Transparency:** When this option is active, transparency will be analyzed on the basis of the Transparency or Alpha surface channels. In doing so, the degree of transparency will be taken into consideration: A semi-transparent sphere will still cause some AO in spite of this option being active.
- **Self Shadowing Only:** When this option is active, separate objects will not see each other, they will only see themselves.

Cache

Because AO works similar to Irradiance Cache, the settings are for the most part the same. In most cases, you will only have to adjust the **Record Density** setting.

Record Density

Record Density defines the subordinate parameters automatically. You should modify the subordinate parameters separately only if the AO is consistently wrong or if it does not appear to be precise enough.

Enable Cache

- If enabled, the AO cache (described briefly in the section below) will be used.
- If disabled, AO will work as in older versions of CineRender (Cineware) (pre-AC20): The environment's visibility will be calculated for each pixel.

How the AO Cache Works

During rendering, several pre-calculations take place (pre-passes), during which the project is analyzed for the regions most important for the camera ("shading points" in corners, concave regions, etc.) for visibility of the environment and to calculate an AO value:

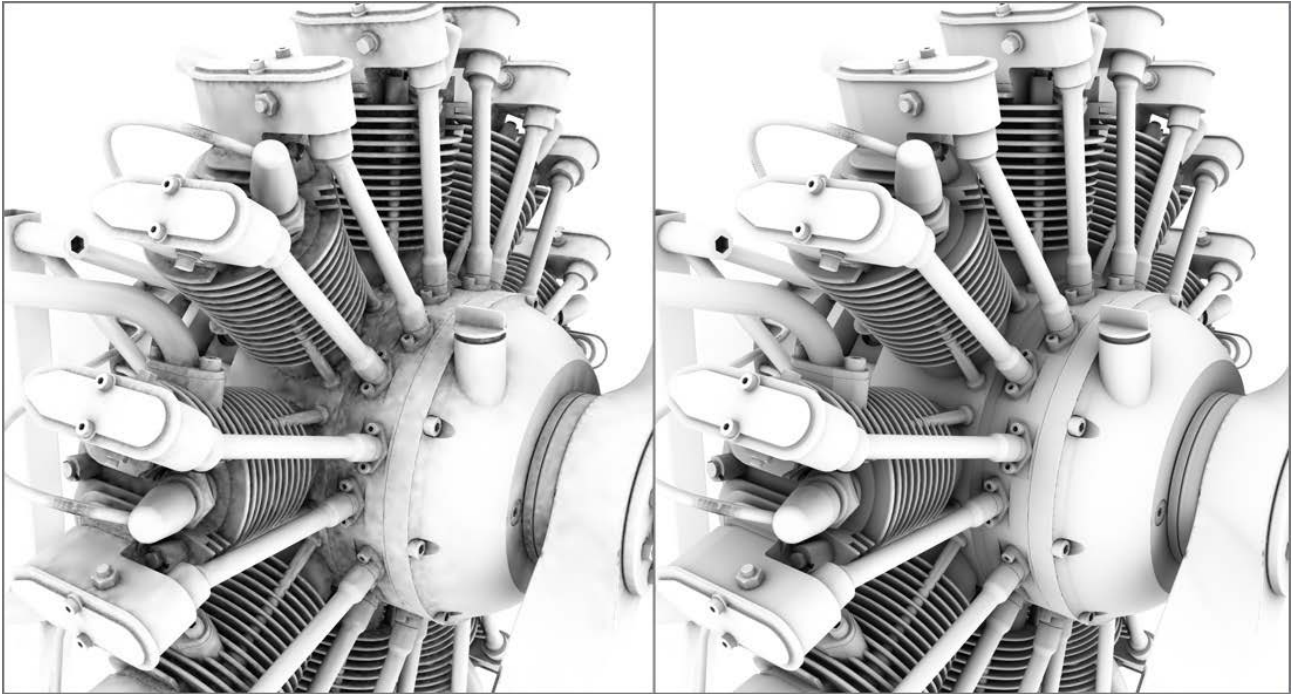
All AO values will be cached to an AO cache and can be saved as a file for later use.

In a second step, the selectively located AO values will be interpolated and smoothed.

The following settings primarily serve to disperse the shading points.

Samples

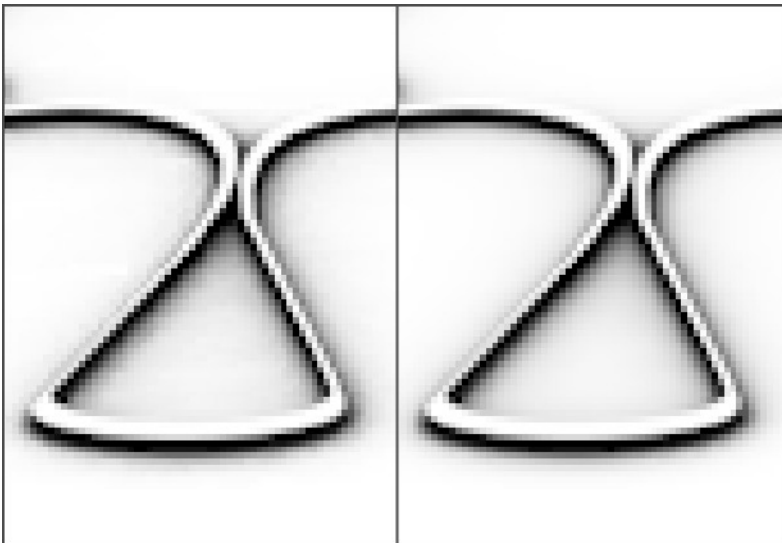
This is the number of samples that are emitted hemispherically from each shading point. If the AO looks spotty, this is the value that should be increased.



Left: small Samples value. Right: larger Samples value. Model by Steen Winther.

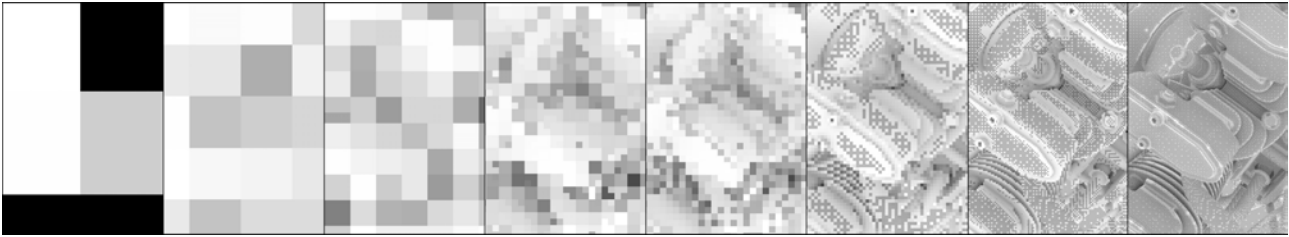
Min Rate/Max Rate

These two settings can, for the most part, be ignored. The effect they have is barely visible. The “worst” setting for Min. Rate and Max. Rate (-8 for each) differs only slightly from their “best” settings (-8 and +4, respectively (see image)).



8/-8 (left) and -8/+4 (right) for Min. Rate and Max. Rate, respectively

Nevertheless, their function should be mentioned: While the pre-passes are being rendered (the phase in which the squares are at first large and become increasingly smaller) the dispersion of the render density is defined. This is an adaptive process that places particular emphasis on the critical regions. The difference between the Min. Rate and the Max. Rate defines the number of pre-passes.



-7 and 0 for Min. Rate and Max. Rate, respectively

A value of 0 will result in full image resolution (pixel size 1*1) and a value of -1 will produce a pixel size of 2*2, -2 = 4*4 and so on. The Min. Rate value should therefore be less than or equal to the Max. Rate value. Positive values can also be used, which will allow caching in the sub-pixel regions (for example, this could be useful, in conjunction with Subpolygon Displacement, if details get lost).

Density/Minimum Spacing/Maximum Spacing

These three settings work together to define the general shading point density at critical and non-critical areas:

- Density: The general shading point density in consideration of the following to settings.
- Minimum Spacing: The shading point density at critical areas (e.g., corners).
- Maximum Spacing: The shading point density at non-critical areas (e.g., planar surfaces without elements that block the view). Various values for this setting can be seen if you take a look at the GI's Density setting. The shading point dispersion is exactly that of the AO cache dispersion.

Smoothing

All of the settings just described revolve around the placement of shading points and their calculation.

AO was ascertained selectively at numerous locations within the project. The selective dispersion must be turned into a flat dispersion for rendering. A smoothing algorithm does this as follows: For each pixel to be rendered on an object's surface, the AO cache is checked for those instances nearest to that pixel and their values will be interpolated for that pixel.

In brief: Values that are too low will result in spotty AO; higher values will result in more homogeneous renderings (but even these ignore Samples that are too low). Generally speaking, higher values take into consideration more shading points for the rendered pixel.

Paradoxically, extremely large values will also lead to spotty results.

Screen Scale

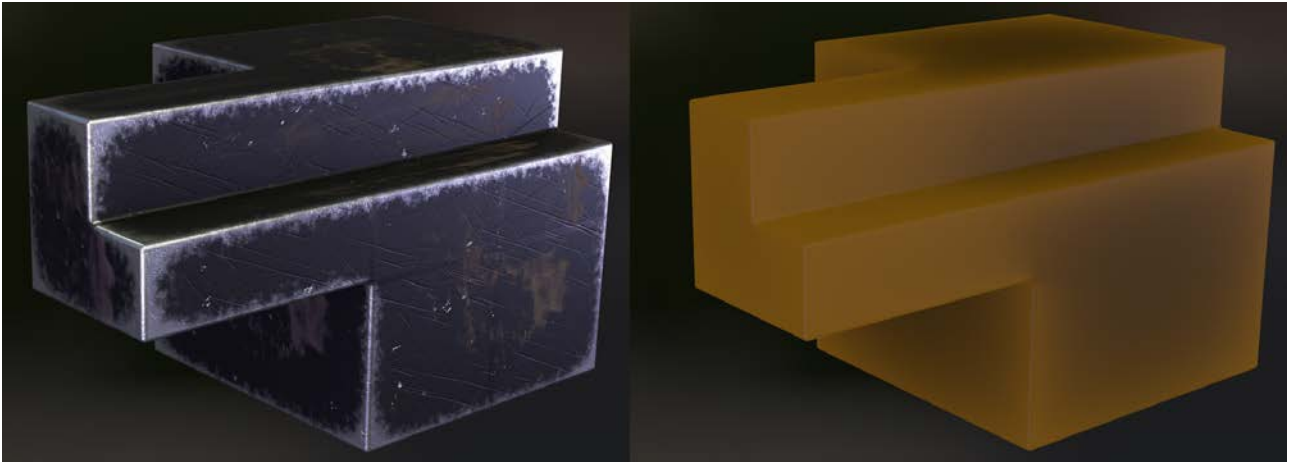
If disabled, the output size for rendering is irrelevant for ascertaining the shading point density. It will remain constant, regardless if you render at a resolution of 80*80 or 3000*3000. For the former it would be too large and for the latter, too small.

If enabled, the shading point density will be linked to the output size for rendering. This will result in a more flexible and adaptive shading point density.

Invert Direction

If this option is enabled, the AO effect will be inverted. Instead of corners and holes, outward pointing edges and corners/peaks will be found.

If exposed surface should, for example, have a different appearance than the remaining surfaces, place the AO shader into the Alpha channel: the respective surface will only have an effect on outward pointing edges and corners/peaks.



This function works best when applied as a shader effect. When enabled as a PhotoRendering effect, only simple coloration of exposed regions can be rendered.

Caustics (Cineware Effect)

Caustics are patterns of focused light created on surfaces and within a visible light beam.

This effect is available in the Detailed view of PhotoRendering Settings for the Cineware engine.

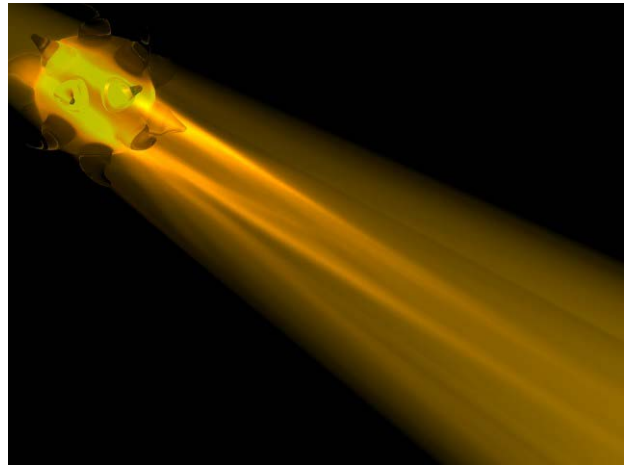
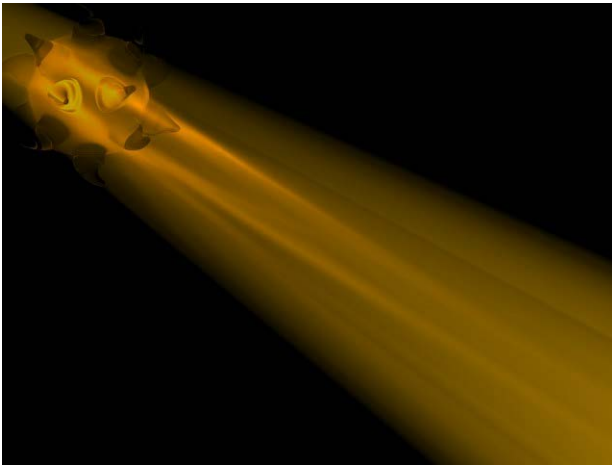
Note: The Caustics effect can also be enabled for Lamp objects.

See [Caustics at Lamp Tool Settings](#).

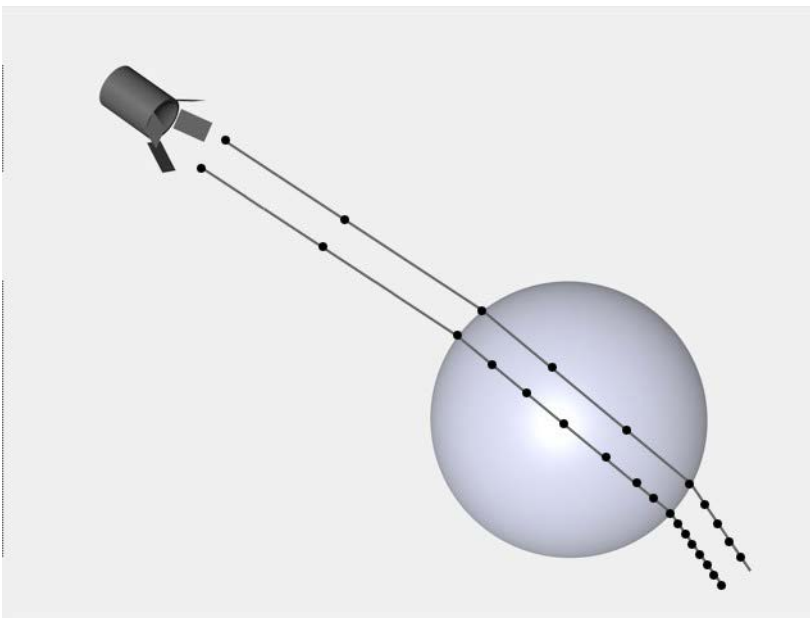
Surface Caustics: Enable this option to switch on surface caustics.

Volume Caustics: Enable this option to switch on volume caustics

Strength: This sets the general brightness of the caustics effect. Here, the strength on the left is normal; the strength on the right is high.



Step Size: With volume caustics, photons are saved in the photon tree at regular intervals as defined by this value.



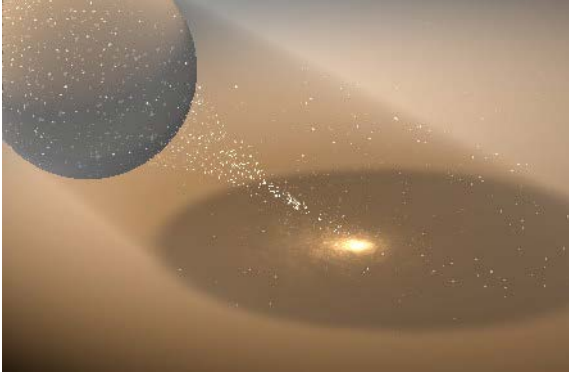
For example, if the volume caustic effect is 100m long, you might set Step Size to 5m so that 20 different values are saved for each photon.

- Lowering Step Size increases the accuracy of the effect but increases the render time also.

- A lower Step Size produces a brighter effect. This is because more values are saved in the photon tree over the same distance.

Sample Radius: This specifies how close photons must be to one another in order to be interpolated. Higher values tend to produce better results but take longer to render.

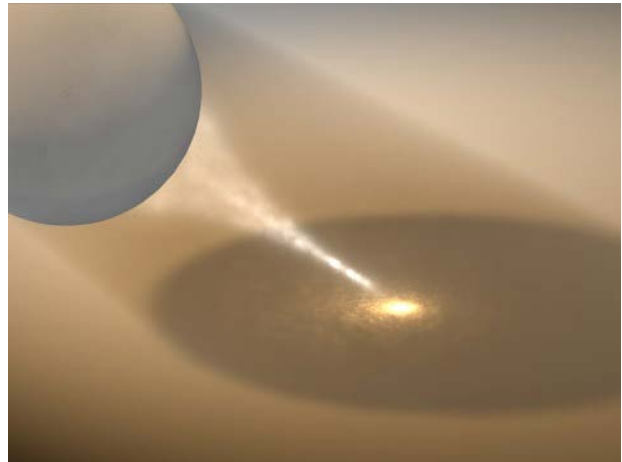
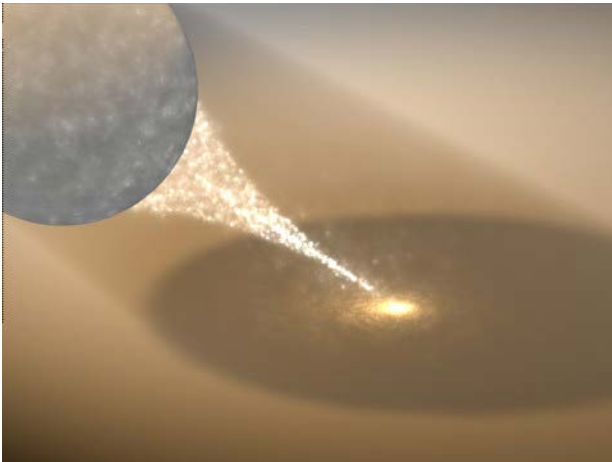
If you notice individual spots or dots in the render, increase the Sample Radius.



Samples: The Samples value represents the maximum number of photons within the sample distance that will be used to calculate the volume caustics.

If Samples is set too low (here, the image on the left), individual photons can be seen as small dots.

As a rule, higher values (here, the image on the right) produce a more accurate effect but add to the render time.



Recompute:

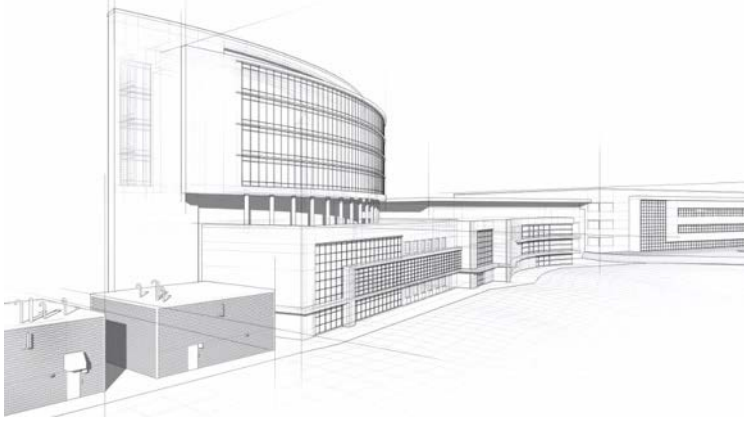
With Recompute, the caustics data from a previous render can be reused to speed up rendering. This saves the time it takes to calculate the caustics effect, but keep in mind that a new calculation will be required if you make major changes to the scene such as moving a caustics-generating light to a new position. Use the Recompute drop-down list to choose when caustics should be recalculated.

- **First Time:** The scene's caustics will be calculated for the first render only; future renders will reuse the data.
- **Always:** Caustics data is always recalculated when you render.
- **Never:** Caustics are not recalculated when you render.

Save Solution: If you enable this option, the caustics data is saved when you render. Depending on the Recompute setting (above), the saved data can be reused to speed up future renders.

Cel Renderer (Cineware Effect)

This effect is available in the Detailed view of PhotoRendering Settings for the Cineware engine. Using this post effect, you can render pictures in a cartoon style. This is a useful effect in architectural renderings.



The render time increases linearly with the number of polygons.

Color

If this option is disabled, objects will usually be rendered with black outlines on a white background.

When Color is enabled, all objects will be rendered using a reduced color palette (depending on the Quantize and Color Steps settings, below) and black outlines on a black background. This gives the rendered subjects a cartoon-like feel.

Illumination

This option is available when Color is enabled.

If this option is enabled, the shading of objects will be affected by the illumination. Shadows will also be rendered cartoon-style.

If Illumination is disabled, each object will have a monotone color.

Also, shadows will not be rendered with the option disabled.

Outline

When enabled, an outline will be drawn around the silhouette of individual objects. If both the Color option and the Outline option are disabled, all you will see is the background color. With Color enabled, the outline will bring out the individual objects and give them a cartoon-like feel.

You can change the color of the outline using Edge Color.

Edge Color

Use this to change the color used for the Outline — even if the Color option is disabled.

Background Color

This setting changes the background color provided that Color is disabled. When Color is enabled, the option has no effect.

Quantize

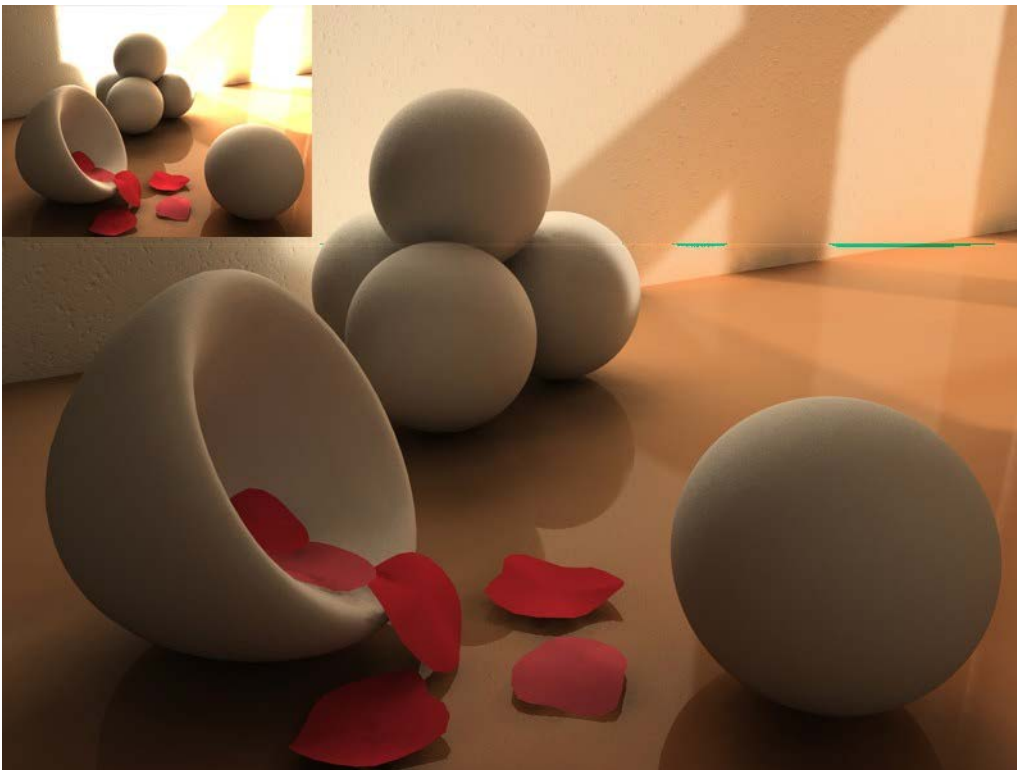
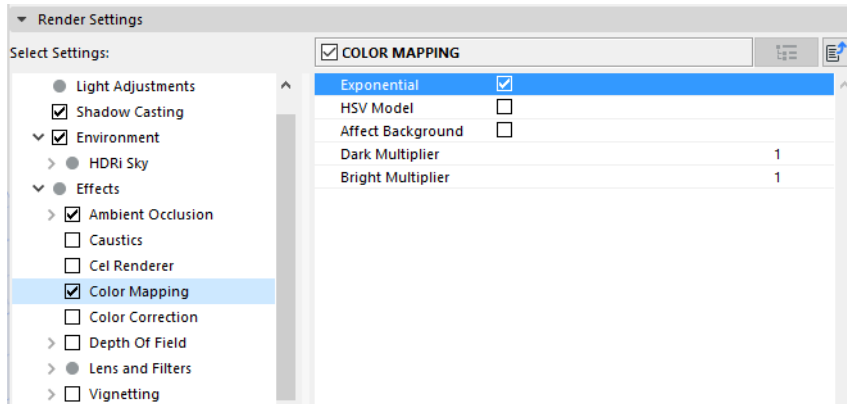
To render the cartoon effect with color steps, enable the Quantize option and set steps to the number of **color steps**.

Color Mapping (Cineware Effect)

This effect is available in the Detailed view of PhotoRendering Settings for the Cineware engine.

An image rendered with GI (and low ray depth) is very bright in some areas and very dark in others. A more balanced allocation of color and brightness can help make the lighting more consistent.

One solution is to edit the internal rendering's interpretation via Color Mapping.



[Watch the Video](#)

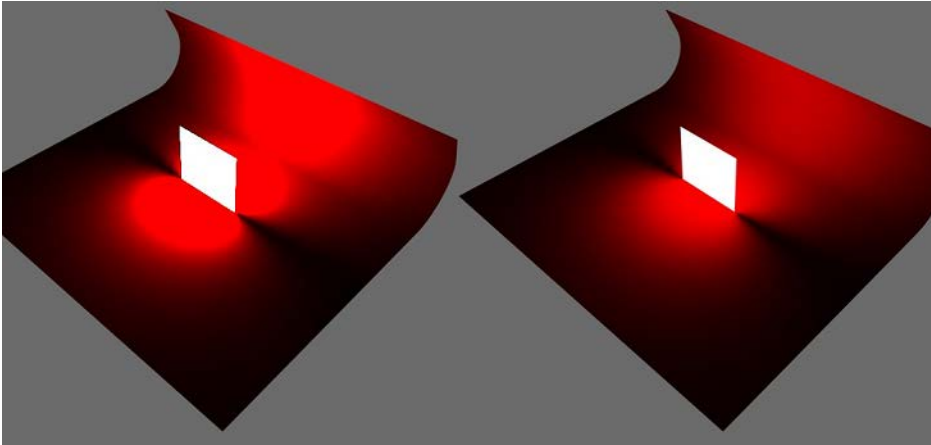
Although these settings look inconspicuous, you should take the time to try them all out.

As a rule:

- Exponential and HSV Model should be left active.
- Use the Multipliers to fine-tune your image.

Exponential

The color dispersion will be controlled exponentially, not linearly.



HSV Model

If this option is active, the HSV (Hue, Saturation, Value) model's colors will be converted and only the V (brightness) component will be affected. The colors will remain identical, e.g. a dark blue will not be turned into a purple.

Affect Background

Here you can define whether or not Color Mapping should affect the background (if you are using one - such as Sky or Background.)

Dark Multiplier/Bright Multiplier

These settings let you strengthen or weaken dark and light colors.

Color Correction (Cineware Effect)

This post effect lets you make common color corrections right away without having to switch to a different application.

This effect is available in the Detailed view of PhotoRendering Settings for the Cineware engine.

[Watch the Video](#)

Saturation

Defines the amount of hue in a given color or in the colors of an image, i.e. how brilliant a color/colors will appear.

A Saturation value of 0 means no change in the hue of the original colors.

A Saturation value of -100 will result in an image being reduced to grayscale values.

A Saturation value of +100 brightens the colors.

Brightness

Makes an image evenly brighter or darker.

Contrast

Higher Contrast values will darken dark regions and brighten bright regions of an image. This amplifies the differences between regions with different brightness values.

Lower Contrast values will lighten dark regions and darken lighter regions of an image, which will result in an optical “flattening” of the image.

Exposure

Exposure is used to adjust the brightness of HDRI and also normal images.

Gamma

The Gamma value adjusts the brightness values (e.g. a bright pixel can be made much brighter, or darkened).

Red/Green/Blue

Use these options to enable or disable each color channel.

Invert

Inverts the image’s color values. Black regions are made white and vice versa. RGB images are inverted by subtracting the maximum pixel color value (i.e. from 255 for 8-bit color depth) for each channel (red, green, blue).

Black Point/White Point

These values define the darkest and brightest points, respectively. If one of the values is modified the total value will be scaled accordingly.

Depth of Field (Cineware Effect)

This effect is available in the Detailed view of PhotoRendering Settings for the Cineware engine.

Note: This Depth of Field parameter applies for the Standard Renderer only, and is considered a post effect. If you are using the Physical Renderer, its own Depth of Field settings will apply: [see Depth Of Field](#).

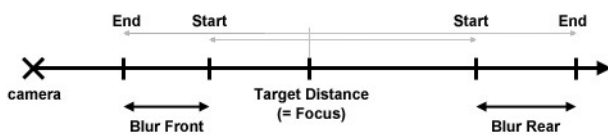
Depth of field is the region of a rendered image in which objects will appear blurred because their distance from the camera (either far or near) causes them to be out of focus.

A large depth of field is when a large region in the distance is displayed, whereas a small depth of field is when a relatively small region is displayed in focus.

Here, the depth of field values decrease, from right to left:



Use the Depth of Field (DOF) value to define which part of an image should be in focus and which out of focus. Either the rear region (DOF rear) or the front region (DOF front) can be blurred. You can also focus on the center region of a scene. In this case both the foreground and the background will be blurred. If neither of the options is enabled, the camera will behave as before: all objects will be in focus.



General

Define the type and strength of Blur you want to add.

- **Blur Strength:** This parameter defines the general strength of blurring. Other parameters modify this strength such as Distance Blur and Radial Blur. Note that if you set Blur Strength to 0% there will be no blurring at all, even when the other blur parameters are set to high values.
- **Use Distance Blur:** The Distance Blur value refines the strength of blurring. For example, if you set Blur Strength to 80% and Distance Blur to 50%, the strength of full blurring will be 40% (50% of 80%).
- **Use Radial Blur:** Radial blur gets stronger from the picture's center outwards.

- **Use Autofocus:** This option can be enabled to simulate the Autofocus feature of a real camera. The object in the center of the view will become the focal point, i.e. the object in the center will be perfectly sharp.
- **Autofocus:** So that the object does not have to be exactly in the center, you can enter an Autofocus tolerance value. The maximum value of 100% allows the object to be anywhere in the view. With lower values, the object must be closer to the view's center to trigger the autofocus.
- **Front Blur/Rear Blur:** In the gradients, black represents full sharpness, white represents full blur (i.e. the Distance Blur value) and gray is for intermediate blur strength (the brighter the gray, the stronger the blur).

Lens Details

Using the settings on the Lens Details page, you can simulate the popular movie effect where objects that are brightly lit in places slip out of focus. Artifacts will appear in the bright areas. Use the Lens Details to adjust these artifacts.

Lens Sharpness: The sharpness of the lens artifacts.

Lens Intensity: The brightness of the lens artifacts.

Lens Shape: Choose the shape of the lens: Circle, triangle, rhombus, pentagon, hexagon, heptagon, octagon or nonagon.

Lens Rotate: Controls the orientation of the lens artifacts.

Tint

Using these parameters, you can add tints that take the scene's depth into account. For example, you can add a blue tint to your renders that gets stronger towards the horizon (aerial perspective); simulate the orange tint of the horizon on the sunset; or recreate the deep, dark cold blue of a moonlit scene. You can also simulate the orange tint of daylight film used to photograph interiors and the blue tint of interior film when it is used to photograph exteriors.

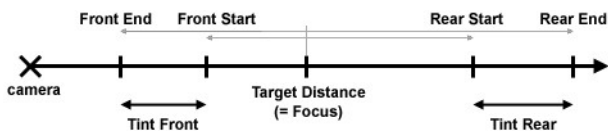
Use Tint

- Front Color
- Rear Color

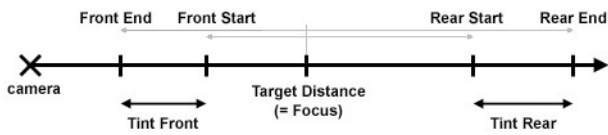
Use Ranges/Use Camera Ranges

There are three possible states. The camera's focus is always taken into account.

- Use Ranges disabled. Creates a tint that is close to reality.
- Use Ranges enabled, Use Camera Ranges enabled. This works in the same way as described for Distance Blur. Use a camera to help define the tint ranges.



- Use Ranges enabled, Use Camera Ranges disabled.



Front Start/Front End/End Start/End End

The tint range is defined by the values Front Start, Front End, End Start and End End (not the camera parameters) based on the camera's focus. This enables you to define the tint independently of the Depth of Field settings.

Depth of field limitations

Problems may occur in places where blurred objects are seen in reflections or behind transparencies.

- If glow is calculated after depth of field, inaccuracies may arise.
- If glow is calculated before depth of field, the depth of field will overwrite the glow.
- The Specular post effect does not work in conjunction with depth of field.
- DO NOT activate depth of field if Alpha Channel is activated in the image output settings. Otherwise the alpha channel will not be rendered correctly.

Lens and Filters (Cineware Effect)

This effect is available in the Detailed view of PhotoRendering Settings for the Cineware engine.

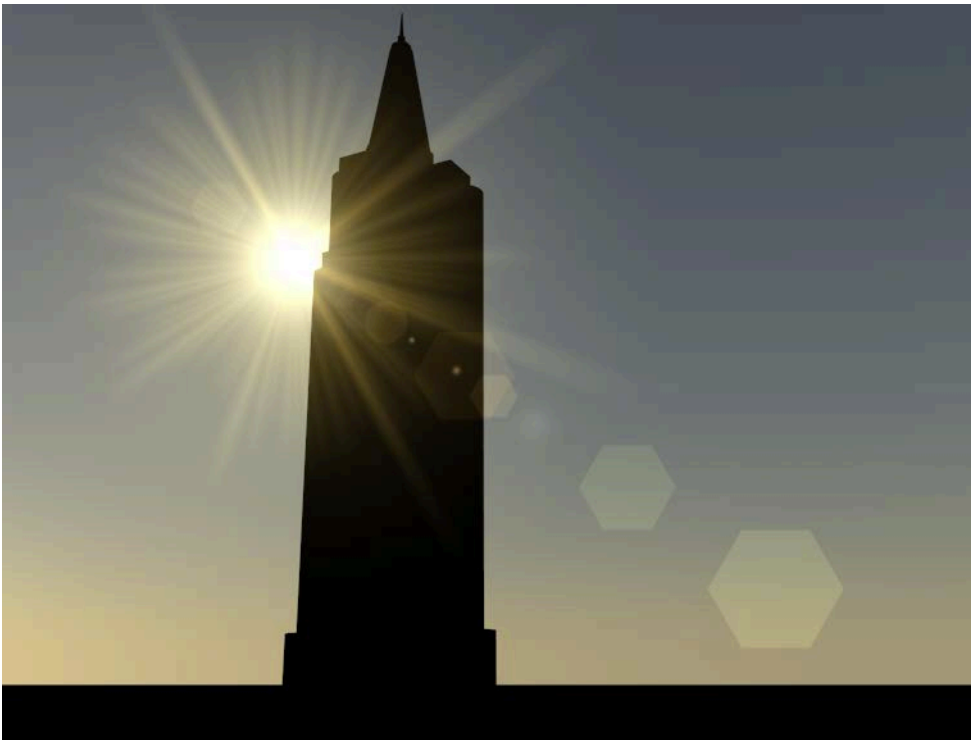
Cylindrical Lens

This option enables you to pan across a much larger area than would be possible using other projection types - for example, to achieve a panorama view of up to 360°.

- **Horizontal Field of View:** Use this setting to define how much of the horizontal view should be taken into account for calculation. A value of up to 360° can be entered.
- **Vertical Size:** Use this setting to define the size of the parallel camera. Higher values will enlarge the portion of the image shown and will compress the image.

Lens Flare

Turn this on to render lens flare effects if they are used in your scene.



See [Lens Flare](#) in *Lamp Tool Settings*.

See also *Lens flare* in the *Physical Sky's Sun settings*.

Median Filter

Removes peaks in color values from the image.

Use **Strength** to increase or decrease the Median Filter effect.

Object Glow

Turn this on to render glow effects if they are used in your scene.

See [Glow \(Cineware Surface Channel\)](#) in *Surfaces*.

Sharpen Filter

This filter emphasizes transitions in the image. This reinforces the edges.

Use **Strength** to increase or decrease the Sharpen Filter effect. Try using the filter with 30% strength.

Soften Filter

Each pixel is balanced with its neighbor to produce a softer transition.

Use **Strength** to increase or decrease the Soften Filter effect.

Distance Fog (Cineware Effect)

This effect is available in the Detailed view of PhotoRendering Settings for the Cineware engine.

[Watch the Video](#)

Color, Strength

These parameters define the color and brightness of the fog.

Distance

The shorter the distance, the thicker the fog.

Environment fog fills the entire screen, stretching to infinity. Distance refers to the fog's intensity by specifying the distance over which a light beam will lose its intensity completely. As the light loses intensity, the fog color is added. If, for example, you have entered a value of 500 for Distance, a light beam that starts off with 100% intensity will reduce to 20% after traveling 400 units; at the end of a further 100 units, the light will have faded out completely, giving way to the fog color.

Beams that penetrate the fog beyond the limit defined in Distance are absorbed completely by the fog color — if you enable environment fog, you cannot see a sky or a background image.

Affect Background

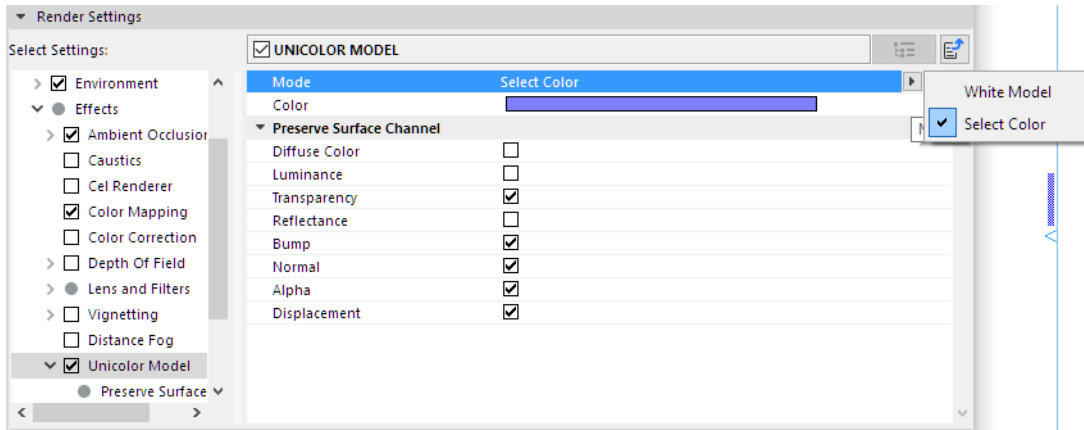
Use this setting to determine whether or not existing Sky Objects or backgrounds should be affected by the fog. If this setting is not active, fog will affect only normal object geometry, and the sky or background will be visible, as if no fog were present.

Monochrome Model (Cineware Effect)

This effect is available in the Detailed view of PhotoRendering Settings for the Cineware engine.

Check the Monochrome Model box to apply a single color to the model, replacing individual surface colors.

- Choose **White Model** for this effect, or
- Select **Color** from the color chooser.



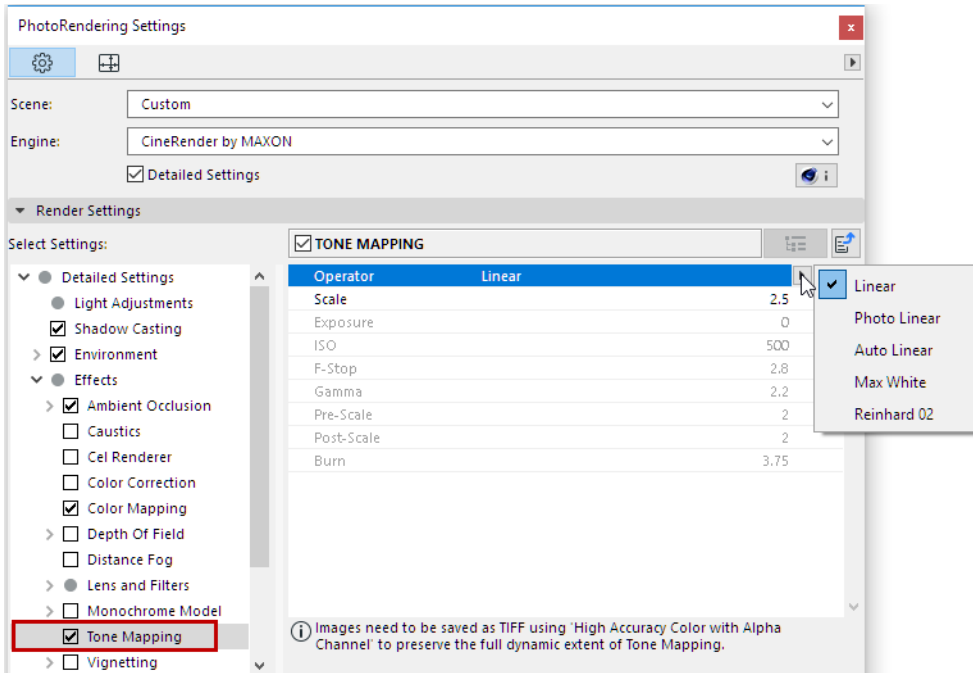
Preserve Surface Channel

Check the Surface Channel effects that you want to include in the model display for rendering.

[Watch the Video](#)

Tone Mapping (Cineware Effect)

This effect is available in the Detailed view of PhotoRendering Settings for the Cineware engine.

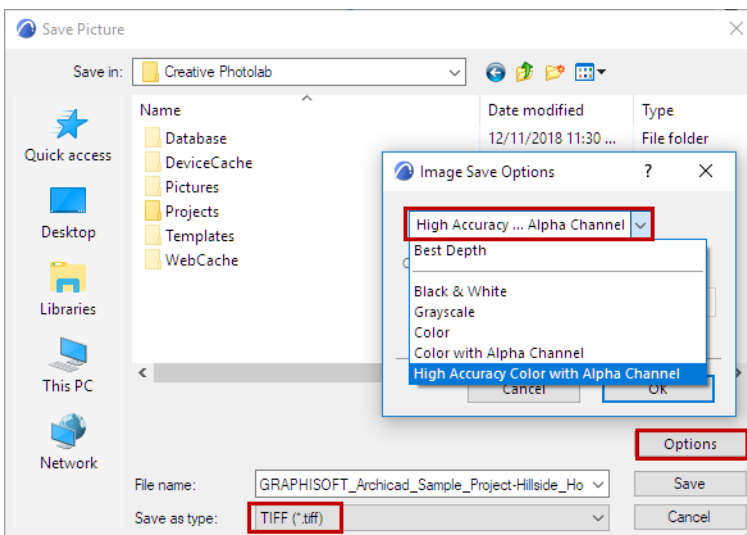


Cineware always renders internally using 32 bits. These color values must be interpreted in order to convert these to 8 or 16 bits for commonly-used output devices. This is done by a series of algorithms (operators) that use different methods.

Note that Tone Mapping permanently modifies the image. If you are planning to perform a color correction in another application (e.g., Photoshop, After Effects, Nuke, etc.) you don't have to perform tone mapping in Cineware.

Save as TIFF

To get the full effect of Tone Mapping, you must save the Render in TIFF format, and adjust the Image Save Options to "High Accuracy Color with Alpha Channel."



Operator for Tone Mapping

When choosing an Operator, it is best to test which one works best for the scene at hand.

Generally speaking, the Reinhard method is the most flexible method. It is based on photographic principles.

Operator: Linear Modes

The linear modes brighten or darken all regions of the image uniformly. This means that some image regions can be drowned out.

- **Linear:** This is a simple mode that multiplies all color values with the Scale factor. Both dark and bright regions will be brightened or darkened.
- **Photo Linear:** These common camera settings can be used to adjust the results.
 - **Exposure** (the larger the value, the brighter)
 - **ISO** (the larger the value, the brighter (this setting represents the film/sensor sensitivity))
 - **F-Stop** (the larger the value, the darker)
 - **Gamma:** Can be used to increase (larger values) or decrease (smaller values) the image's mid-range brightness (which is actually not linear behavior if the value is not equal to 1).
- **Auto Linear:** This operator can be compared to a camera's automatic exposure: an average brightness value for the image is determined internally. Based on this, correct values for Exposure and F-Stop will be defined, using Photo Linear formulas. Subsequently, a gamma correction can be made using the Gamma setting.

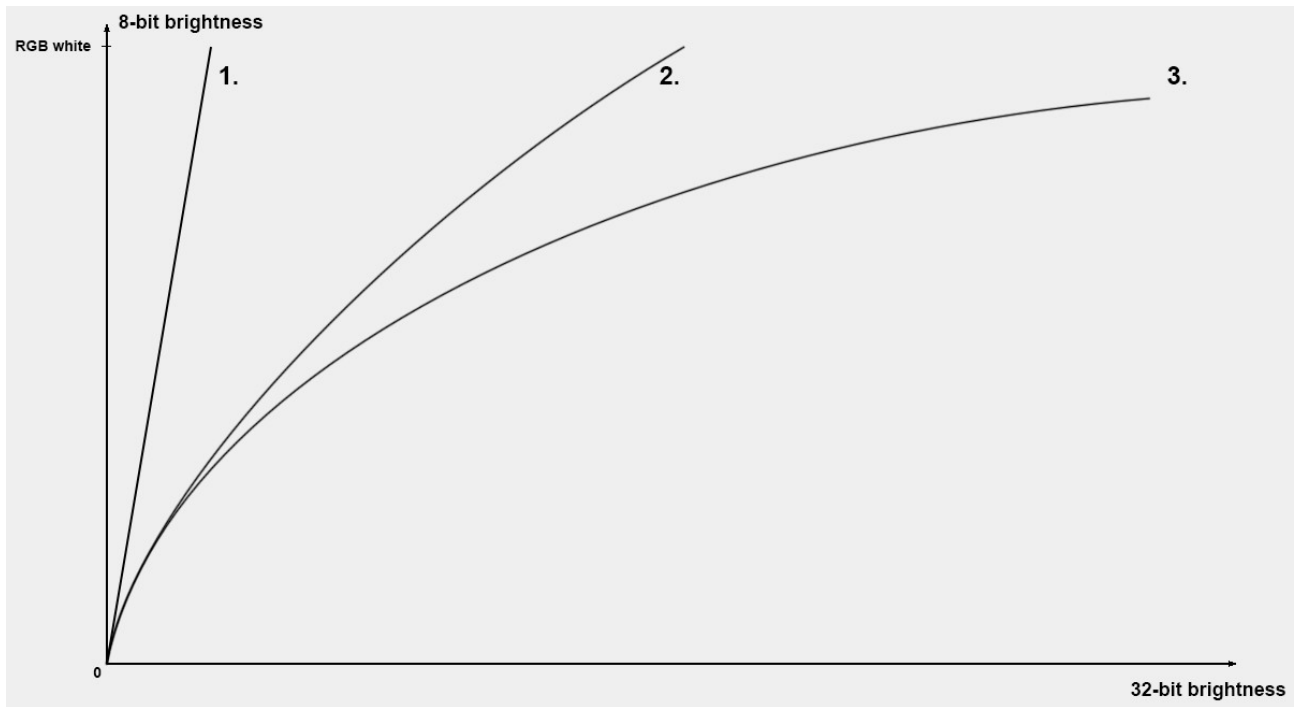
Max White

This option determines the brightest point and sets it to white. The remainder of the image will be "scaled" relative to this point. If there are very bright regions in the image, darker regions will be made much darker or even black.

Reinhard 02

The Reinhard operator is based on tone mapping techniques used in photography. As you can see in the image below, bright regions are compressed in particular. This tone mapping method is the most widely used method for rendering.

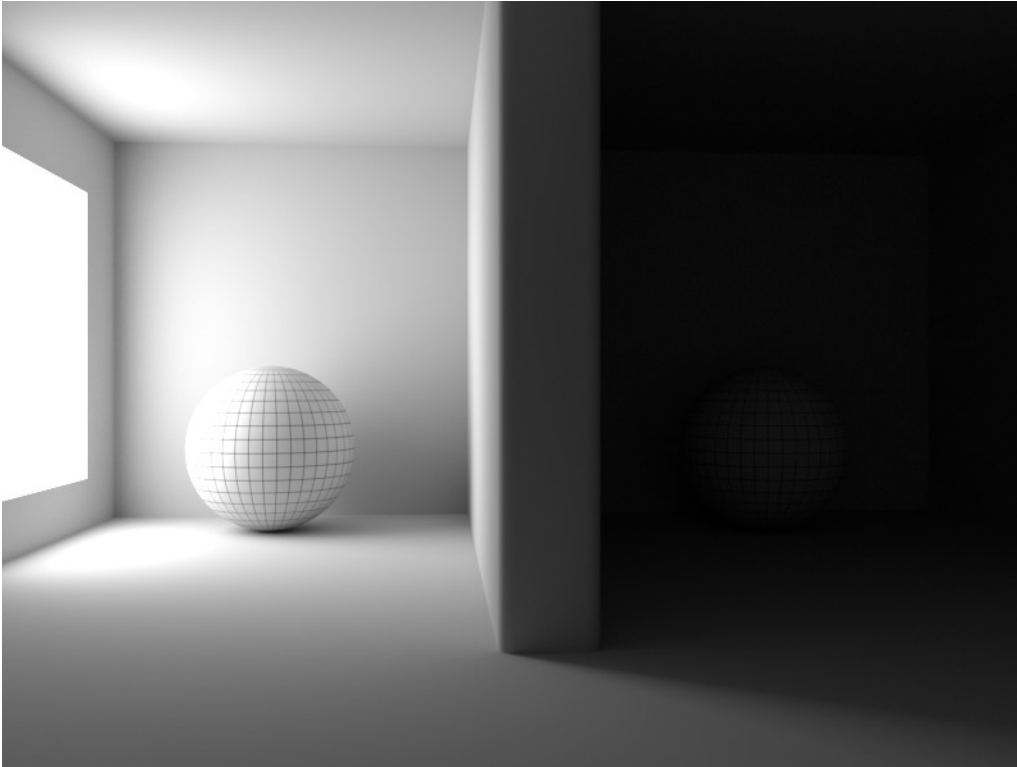
Pre-Scale primarily affects darker regions (higher values can be used to brighten shaded regions) and the **Post-Scale** is used to adjust brighter regions, and even blends between linear and exponential curve progression.



Three Reinhard curves with different Burn values from 1 to 3.

Tone Mapping Example

The following scene is rendered without Tone Mapping:

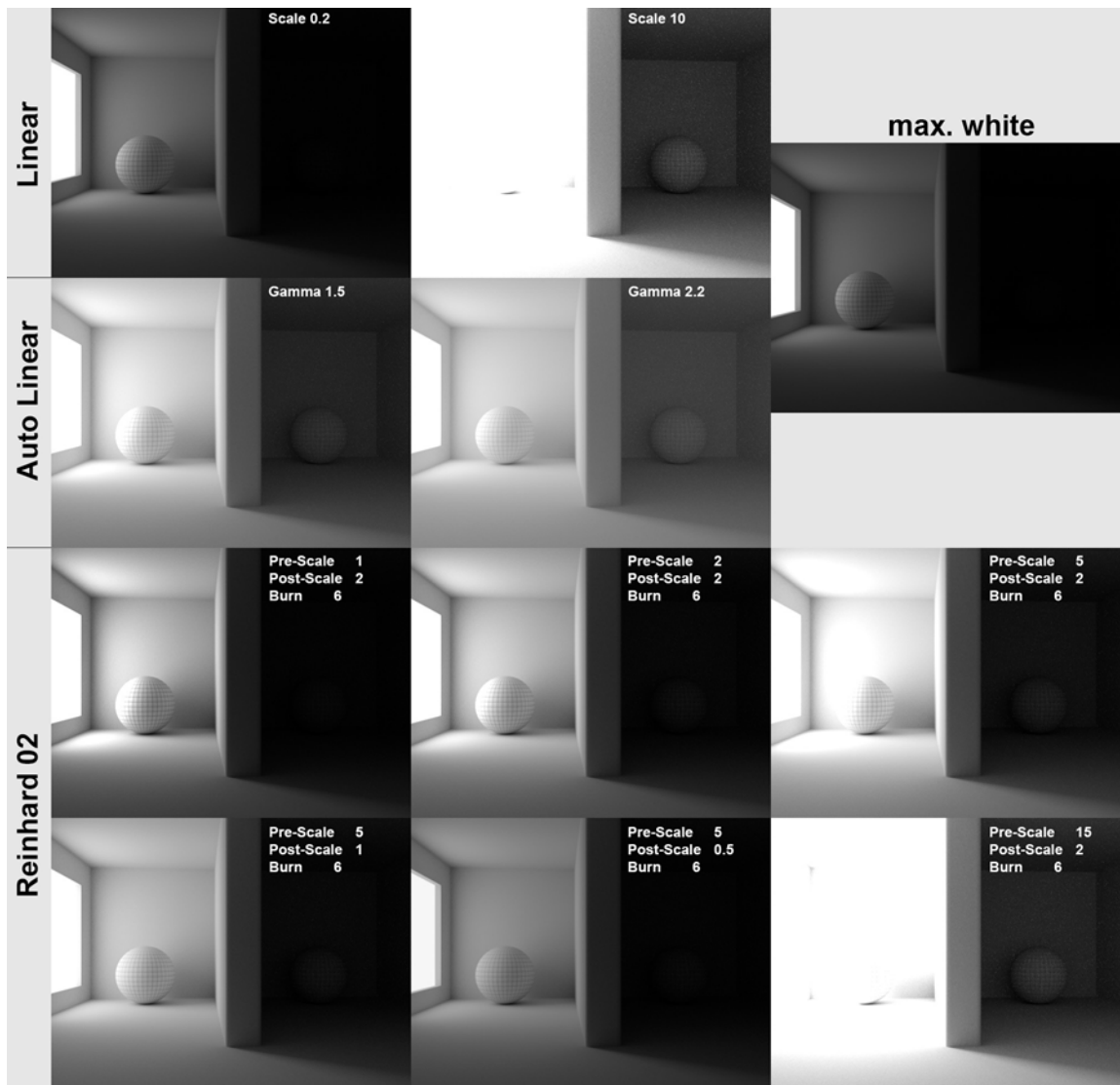


The luminous material at the left has a brightness of 250%, a sphere lies at the right in the dark.

As you can see, there are over-exposed and very dark regions.

The image below illustrates how some of the operators work. (Photo Linear is not used here because it behaves similarly to the other linear operators.)

As seen in the image, Auto Linear works best (in combination with Gamma): Very bright regions are darkened and dark regions are brightened. Reinhard doesn't work very well with this scene: very dark regions cannot be brightened adequately. The Reinhard operator works best for scenes with even lighting and no very dark regions (e.g., interior room renderings).

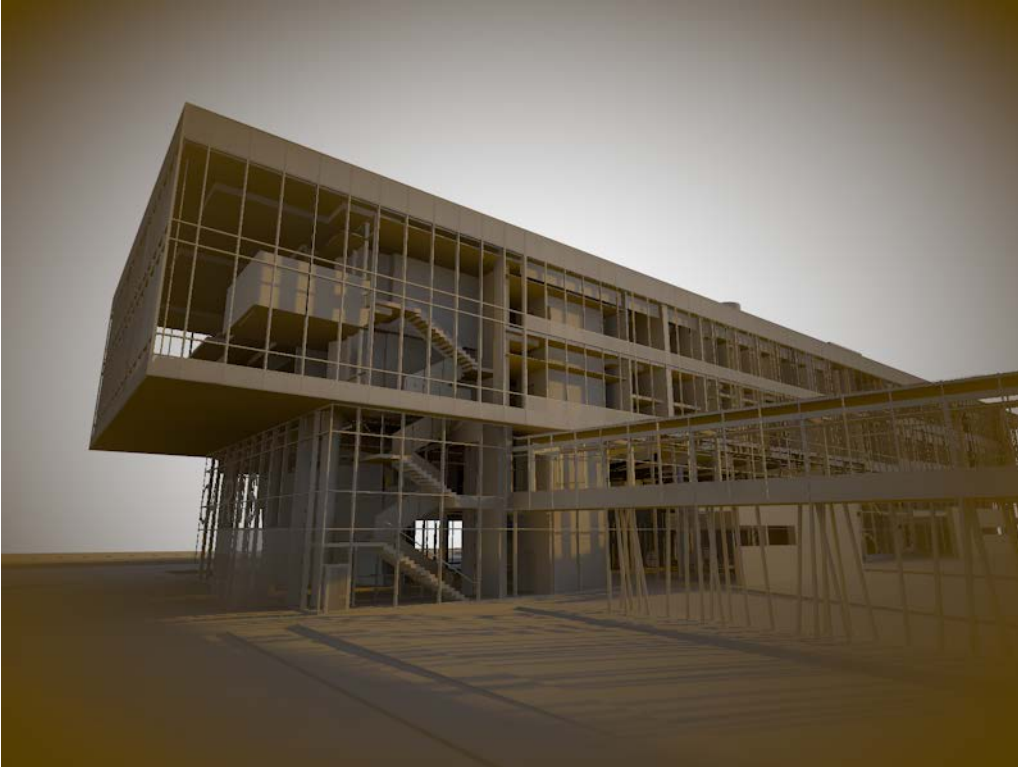


The linear operators don't work well with the example scene, because they either darken bright regions, which makes dark regions even darker, or they brighten dark regions, which over-exposes brighter regions.

Vignetting (Cineware Effect)

This effect is available in the Detailed view of PhotoRendering Settings for the Cineware engine.

Vignetting is the effect of darkening an image to its edges (light that hits the lens axis at an angle does not fill the shutter opening entirely, due to the way the shutter is constructed).



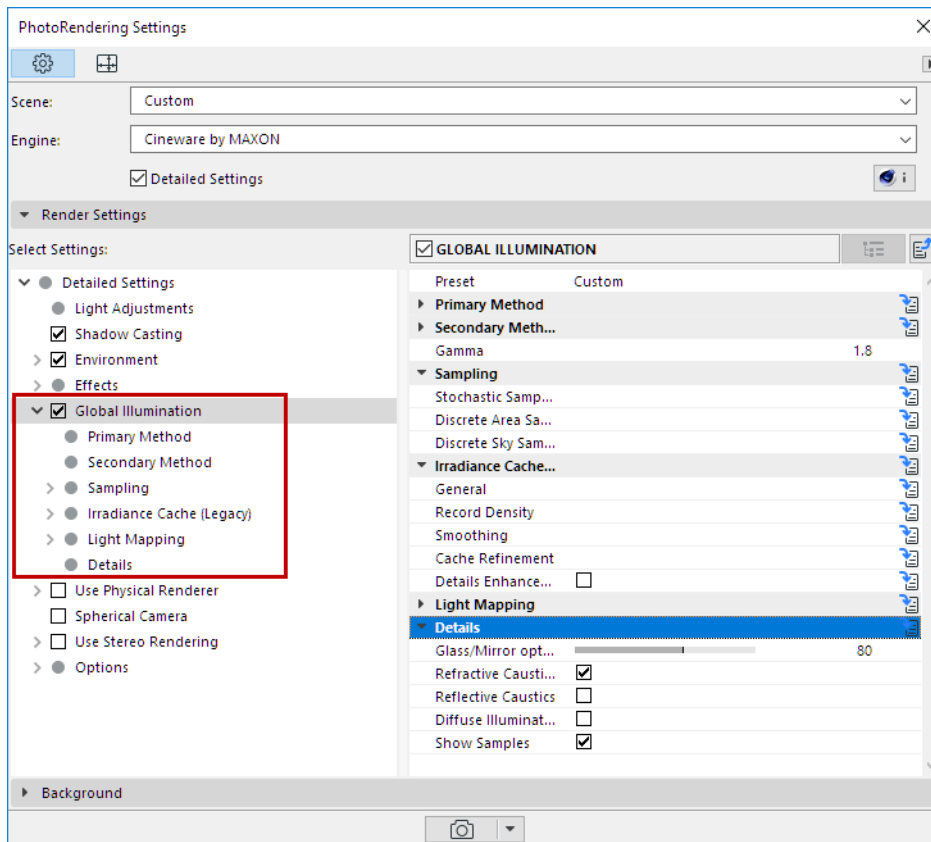
See also [Defining Color and Brightness Values](#) and [Adjusting Gradient Parameters](#).

Global Illumination (Cineware)

This effect is available in the Detailed view of PhotoRendering Settings for the Cineware engine.

Global Illumination (GI) is a sophisticated method for achieving realistic lighting effects in your model. The algorithm simulates the effects of not just direct light sources, but indirect light effects - that is, inter-reflections caused by light rays bouncing off surfaces.

With Cineware, you can achieve very high-quality renderings using Global Illumination. However, it can greatly increase render time.



Preset (Cineware Global Illumination)

Primary and Secondary Method (Cineware Global Illumination)

Sampling (Cineware Global Illumination)

Radiosity Maps (Cineware Global Illumination)

How Does QMC Work?

Irradiance Cache (Legacy) (Cineware Global Illumination)

Details (Cineware Global Illumination)

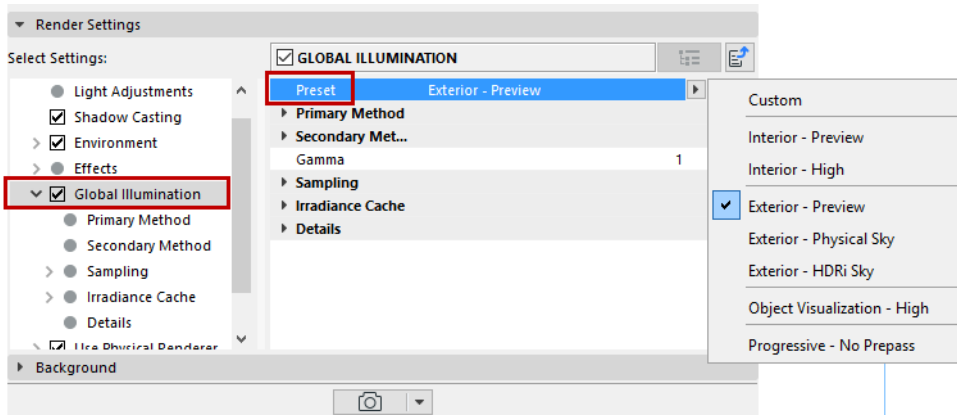
[Watch the Video](#)

[Watch the Video](#)

[Watch the Video](#)

Preset (Cineware Global Illumination)

Presets are available for the optional Global Illumination effect, in the Detailed view of PhotoRendering Settings for the Cineware engine.



Since there are so many possible combinations of GI methods, this Preset option lets you save GI settings for specific types of Projects and Scenes.

It's important to know how your Scene is set up. You should differentiate between two main types:

- **Interior spaces** are mostly illuminated by fewer, smaller light sources: light that falls through windows, and lamps that emit light within a limited radius. Interior spaces are therefore more difficult to calculate for GI.
- **Exterior spaces** are basically set up under an open sky. Skies emit light evenly from a large surface, which makes it easier for GI to calculate exterior scenes.

These presets should only serve as a starting point. These settings can then be fine-tuned to optimize render quality or render time.

List of Interior Presets

- **Custom:** As soon as you modify a GI setting, this option will be activated.
- **Interior - Preview:** Uses Irradiance Cache + Radiosity Maps. A fast preview with reduced Diffuse Depth.
- **Interior - Preview (High Diffuse Depth):** Uses Irradiance Cache + Light Mapping
- **Interior - Preview (Small Illuminants):** Uses Irradiance Cache + Light Mapping
- **Interior - High:** Uses Irradiance Cache + Irradiance Cache
- **Interior - High (High Diffuse Depth):** Uses Irradiance Cache + Light Mapping
- **Interior - High (Small Illuminants):** Uses Irradiance Cache + Light Mapping

Notes on Interior Presets

- **Preview:** These have reduced values, which can be calculated quickly and are well-suited for test renderings.
- **High:** Designed for higher quality renderings. However, it may be necessary to increase or decrease individual values to achieve the desired render speed/quality.
- **High Diffuse Depth:** Refers to the effects of numerous light reflections, which produce a much more realistic result with endlessly higher "sample depth". Projects will tend to be rendered brighter; dark corners will also be brightened. This effect is created in conjunction with Light Mapping

- **Small Illuminants:** Illumination will mainly be created by smaller lights that are defined as Portals and Polygon Lights. IR+Light Mapping in combination with Force Per-Pixel is the best method of ensuring that small lights unfold their full potential.

List of Exterior Presets

- **Exterior - Preview:** Uses Irradiance Cache + Radiosity Maps. A fast preview with reduced Diffuse Depth.
- **Exterior - Physical Sky:** Uses Irradiance Cache + QMC.
- **Exterior - HDR Image:** Uses Irradiance Cache + QMC.

Notes on Exterior Presets

- **Preview:** These have reduced values, which can be calculated quickly and are well-suited for test renderings.
- The difference between **Physical Sky** and **HDR Image:** Force Per-Pixel is enabled for the latter. An HDR image assigned to a Sky object is evaluated precisely for rendering, which will more precisely render multiple shadows - if they are present in the HDR image (note that the Diffuse Depth value may have to be increased).

Object Visualization

Object Visualizations are well-lit and are most often centered in the composition, which means that they generally require few light reflections.

- **Object Visualization - Preview:** Uses Irradiance Cache + Light Mapping
- **Object Visualization - High:** Uses Irradiance Cache + Irradiance Cache.

Physical Renderer's Progressive Sampler

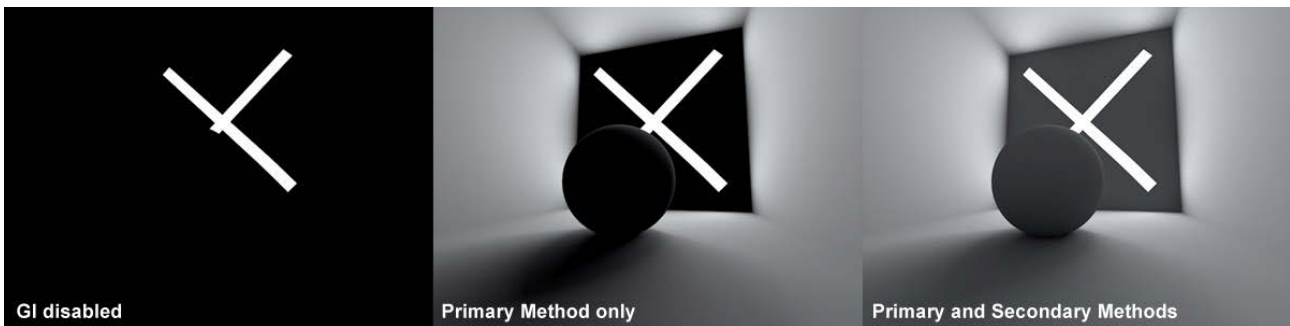
- **Progressive - No Prepass:** Uses QMC + QMC.
- **Progressive - Fast Complete Diffuse:** Uses QMC + Light Mapping

These two options are designed for use with the Physical Renderer's Progressive Sampler setting. You can quickly render rough images whose quality improves progressively and lets you check the overall render quality without long waiting periods until the image is completely rendered.

Primary and Secondary Method (Cineware Global Illumination)

GI in Cineware works in two parts:

- The **Primary Method** calculates the effect of Diffuse Depth 1, which is
 - the light emitted by polygon lights
 - or the light emitted by illuminated surfaces (via real light sources or the Physical Sky) without further reflection. The latter case results in the typical GI lighting with indirect reflection of light (e.g. a red sphere illuminated by a white light reflects red light).
- The **Secondary Method** calculates the brightness of surfaces (that do not have to lie in the camera's field of view) that are brightened by multiple light reflections.

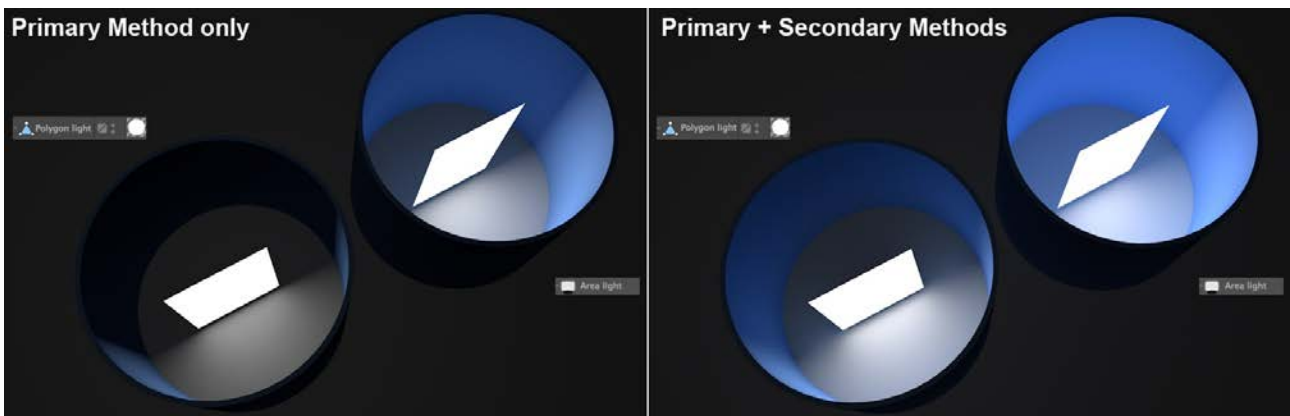


Primary Method only brightens surfaces that are directly lit.

Secondary Method is responsible for additionally reflected light.

In this example, only the Primary Method was used on the left picture. Both Primary and Secondary methods were used on the right picture.

In each picture, the left uses a polygon light, the right uses a normal light:



The results of both methods will be added to make up the overall GI when rendered.

The **Primary Method** is most important method for the render quality and therefore higher-quality methods such as QMC or IR should be used. These also take correspondingly longer to render! For light that is reflected back-and-forth, methods can be used that are of “lesser quality” and render faster (such as Radiosity).

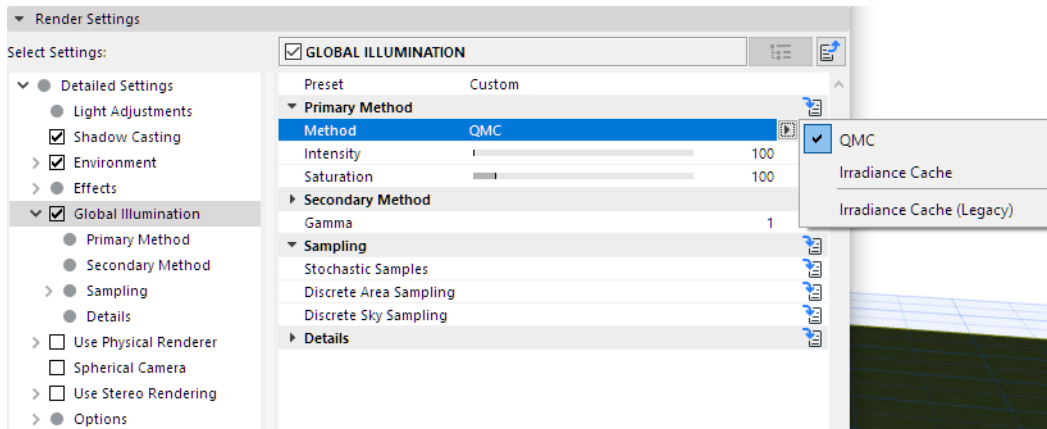
Depending on the project's setup, different methods can then be combined. Use the predefined **Presets**.

See [Preset \(Cineware Global Illumination\)](#).

Tip: The two-part GI calculation already existed in previous versions: With Mode set to IR and Sampling set to Radiosity Maps, this corresponds to the current Irradiance Cache Primary Method and Radiosity Maps Secondary Method.

In the following you will find a brief description of the individual methods.

Options for Primary Method



- **Quasi-Monte Carlo (QMC):** QMC is the most precise but slowest method. Animations are flicker-free (if not free of noise) when QMC+QMC is used.

For details, see [How Does QMC Work?](#)

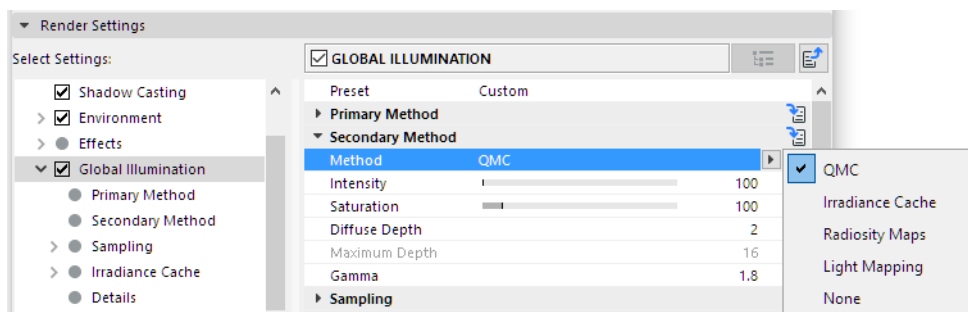
- **Irradiance Cache:** A simplified, fast method of ascertaining the most important areas of a given project, calculating GI at these locations, and interpolating. Animations tend to flicker if the values are too low.

For details, see [Irradiance Cache \(Global Illumination\)](#).

- **Irradiance Cache (Legacy):** This is the Irradiance Cache from Cineware versions prior to V20. It has been maintained so older Projects can be rendered with the same render results.

For details, see [Irradiance Cache \(Legacy\) \(Cineware Global Illumination\)](#).

Options for Secondary Method



- **Quasi-Monte Carlo (QMC):** QMC as a Secondary Method is best when used as IR+QMC for exterior scenes, and most precise - and slowest - as QMC+QMC.
- **Irradiance Cache:** IC as Secondary Method works well for interior spaces with small lights defined as Portal or Polygon lights. Make sure to reduce the Samples value in combination with QMC+IR. Internally, a much greater number of QMC samples is used for IR, which can increase render times very dramatically.

- **Radiosity Maps:** Radiosity Maps as Secondary Method are well-suited for fast preview renderings because of their low Diffuse Depth (less reflected light).

For details, see [Radiosity Maps \(Cineware Global Illumination\)](#).

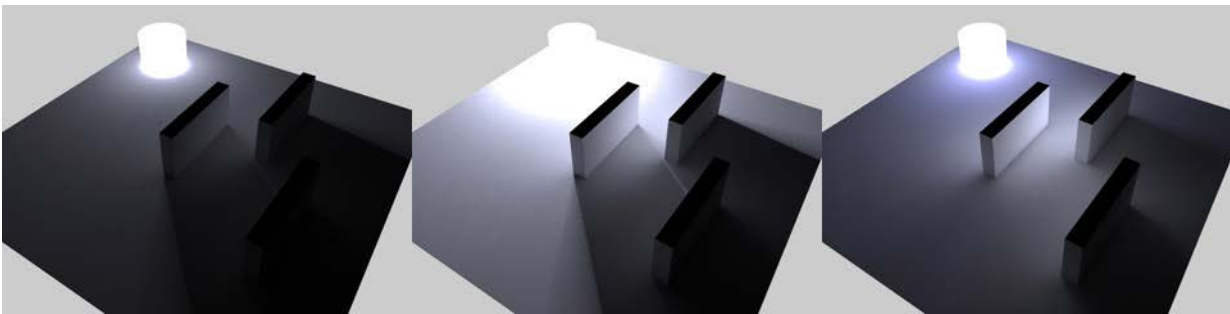
- **Light Mapping:** Light Mapping as Secondary Method works very well when rendering interior spaces where a lot of light is needed, which can be very quickly supplied by the high number of light reflections from Light Mapping.

For details, see [Light Mapping](#).

- **None:** Disables the secondary GI calculation. This represents a Diffuse Depth of 1.

Intensity (Primary and Secondary)

Use these parameters to adjust the GI brightness based on the number of light reflections. The Primary Intensity parameter affects the regions being illuminated directly; the Secondary Illumination parameter affects reflected light.



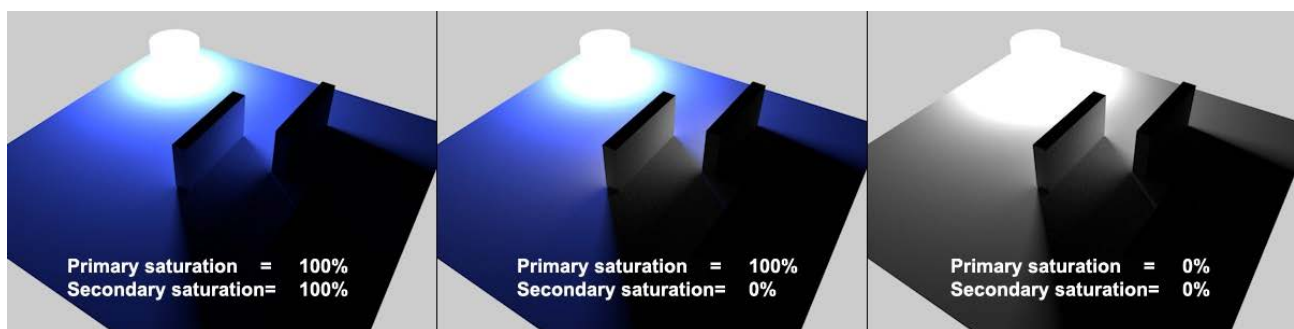
*From left to right, the following Primary/Secondary Intensity values:
100%/100%, 300%/100%, 100%/500%*

Saturation

Defines the color saturation used within the GI calculation separately for the Primary and Secondary Method GI functions. This is especially useful if the Physical Sky produces shadows that are too blue. Try reducing the Saturation value for the Secondary Method.

If other methods produce too little saturation (as is sometimes the case with IR/QMC as a secondary method), the Saturation value can be increased.

Note that if the Primary Method's Saturation setting is set to 0%, no colors will be available for the Secondary Method.



*A single blue Polygon Light lights the scenery, with various Saturation settings.
Note how the Secondary Method at the center emits colorless light.*

For Radiosity Maps, the Secondary Method's Saturation will only affect real Area lights (not Polygon Lights) and/or Physical Sky (which is given a real light source via the Sun light).

Color saturations can also be modified for surfaces ([see also Illumination \(Cineware Surface Channel\)](#)). Both saturation parameters represent a global, project-wide saturation setting.

Diffuse Depth

Of the many GI-related parameters, the Diffuse Depth value (which can be defined only for the Irradiance Cache or QMC Secondary Methods) can make a significant difference in rendering quality. It defines the number of times light reflects in a scene, i.e. how often a "ray of light" is reflected from surfaces.

Greater Diffuse Depth values will result in moderately longer render times (the difference between a value of 1 and 2 is greater than that between 2 and 8) but the dispersion of light will become increasingly homogeneous, brighter and more realistic. However, the effect at values greater than 3 in a normal scene will become less and less noticeable and the rendered result will simply become brighter.

- The minimum Diffuse Depth value of 1 (as in the following image) results in only a direct illumination via flat, light-emitting elements. This will be sufficient for most external scenes, with Physical Sky or HDRI Sky providing a significant light source.



- A Diffuse Depth value of 3 (as in the following image) is required to achieve an indirect illumination, i.e. light reflected from other surfaces. A minimum value of 2 will be required for interior scenes.



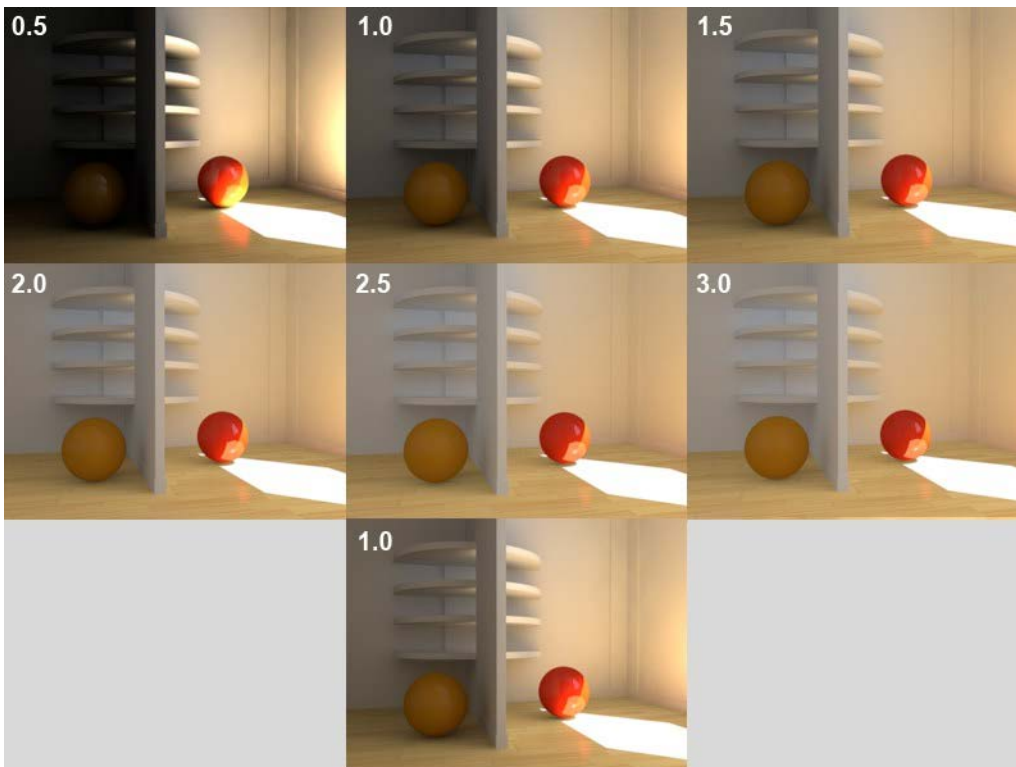
Note that gamma correction can, within limits, be used to compensate for lower Diffuse Depth values.

Tip: When “real” light sources are used, indirect illumination can already be achieved with a Diffuse Depth value of 1 because the objects illuminated by the light source will be recognized as a luminous object.

Gamma

This gamma value only affects the indirect GI lighting. Gamma values define how the internally rendered brightness values should be displayed in RGB mode. Simply stated, a progression from the darkest (black) to brightest (white) is defined.

This lets relatively dark renderings (e.g. resulting from a low Diffuse Depth value) be brightened. But be careful - high gamma values will reduce contrast and “flatten” the overall image (values ranging from 1 to 3 have proven to be most effective; in some instances higher values may be necessary). Values less than 1 will darken the image, whereas values greater than 1 will brighten an image.



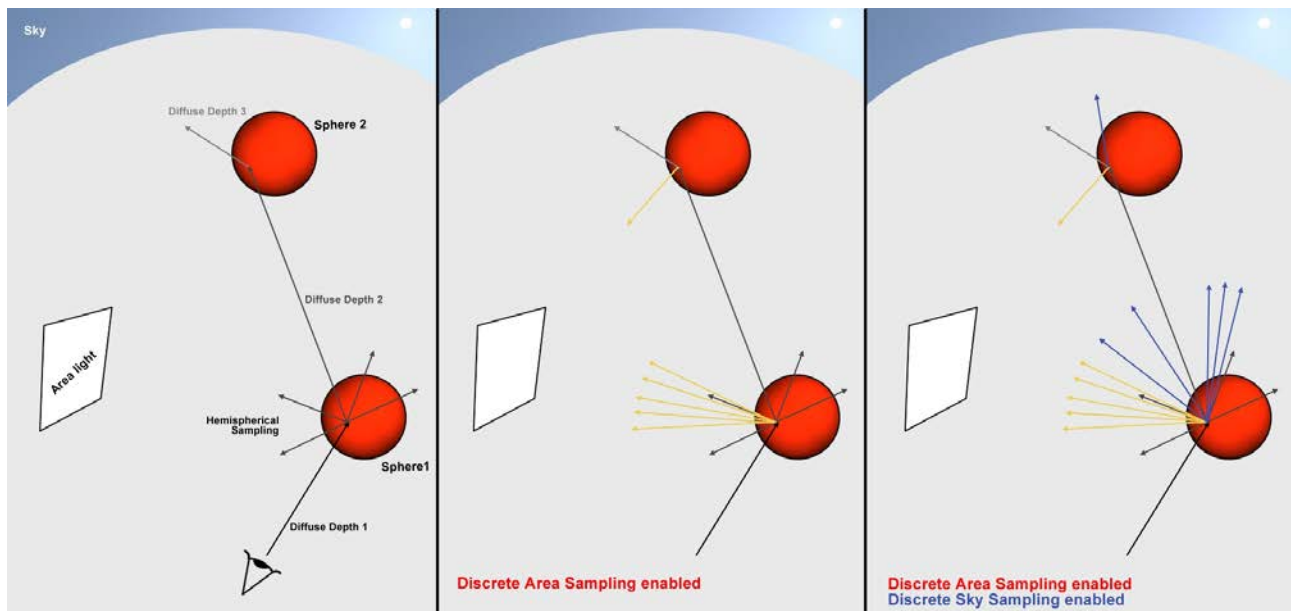
Sampling (Cineware Global Illumination)

This setting is available, as part of the optional Global Illumination effect, in the Detailed view of PhotoRendering Settings for the Cineware engine.

The Sampling settings here only affect the GI sampling: How many samples should be sent and where in order to gather light from the environment? Samples are sent out hemispherically from the shading point (also called the “primary sampler”).

The Sampling settings affect the GI sampling for the QMC and IR Primary Method (if these are also used for the Secondary Method, only a fraction of the value defined here will be used):

The image below shows how the QMC GI Mode works.

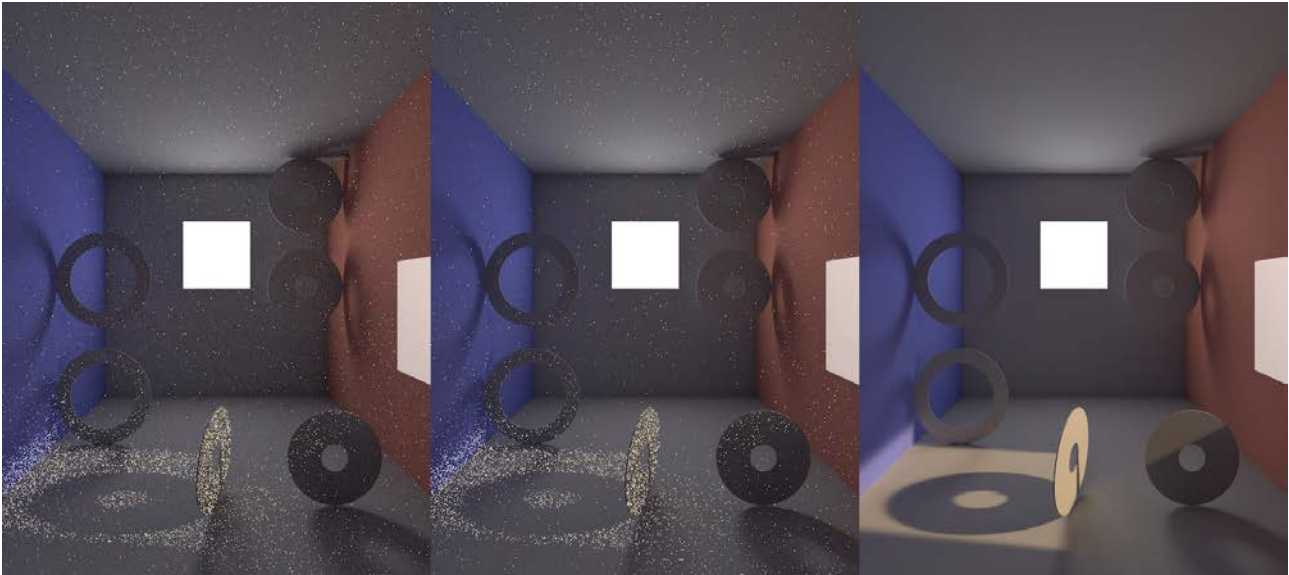


In the image above:

- **Left** (Discrete Area Sampling and Discrete Sky Sampling disabled): At the first point of contact, several samples are sent out in a hemispherical direction (this is exactly what the Hemispherical Sampling option is meant for).
- **Center** (Discrete Area Sampling is enabled and Discrete Sky Sampling is disabled): In addition to the hemispherical sampling, several samples are also sent out in the direction of Portals and (polygonal) lights.
- **Right** (Discrete Area Sampling and Discrete Sky Sampling enabled): In addition to the hemispherical sampling and the Area light sampling, additional samples are sent out in the direction of the sky.

The Sampling settings described below primarily define if and how many samples are created on Sphere 1.

In the following image, an Area light was positioned at the rear of the room. A window in the right-side wall looks out to a Sky object with an HDRI assigned to it (it can render hard shadows!). No other light sources have been placed in the project. The images were rendered with the QMC Primary Method (with the IC Method, you would see spots instead of graininess in the image).



- **Left** (Discrete Area Sampling and Discrete Sky Sampling disabled): This is not a viable option for this particular Project. The image itself is very grainy (the result of random samples hitting the Area light). The shadows are even more grainy. Bright points are also visible, the result of very few random samples hitting the (HDRI) sky's sun.
- **Center** (Discrete Area Sampling enabled and Discrete Sky Sampling disabled): The general graininess has been greatly reduced because additional samples are being sent to the Area light (each point visible to the camera is analyzed, which results in a highly defined shadow for the Area).
- **Right** (Discrete Area Sampling and Discrete Sky Sampling enabled): In addition to the hemispherical sampling and the Area light samples, additional samples are sent in the direction of the sky. This abolishes the bright points, because every point visible to the camera is precisely analyzed with regard to the sky's influence (including its effect on the remaining project elements).

In general, you can leave Discrete Sampling enabled. Disable it only for special instances, to intentionally exclude one of the Sampling modes. The difference in render time will be noticeable if no Area light or sky is present in the project.

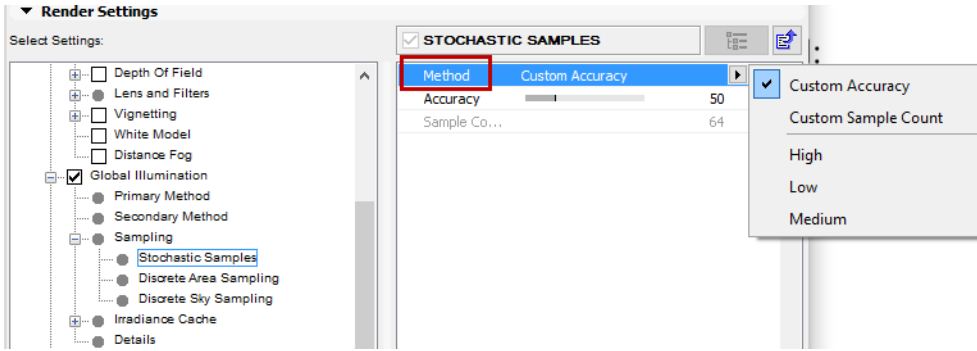
The description of the following settings applies to the **QMC Primary Method**. The same applies to the IR method, except that sampling takes places for each shading point, rather than for each pixel.

Stochastic Samples

Method: Two methods of defining sample count are used:

- An automatic determination controlled by one of the quality settings (Low, Medium, High, Accuracy)

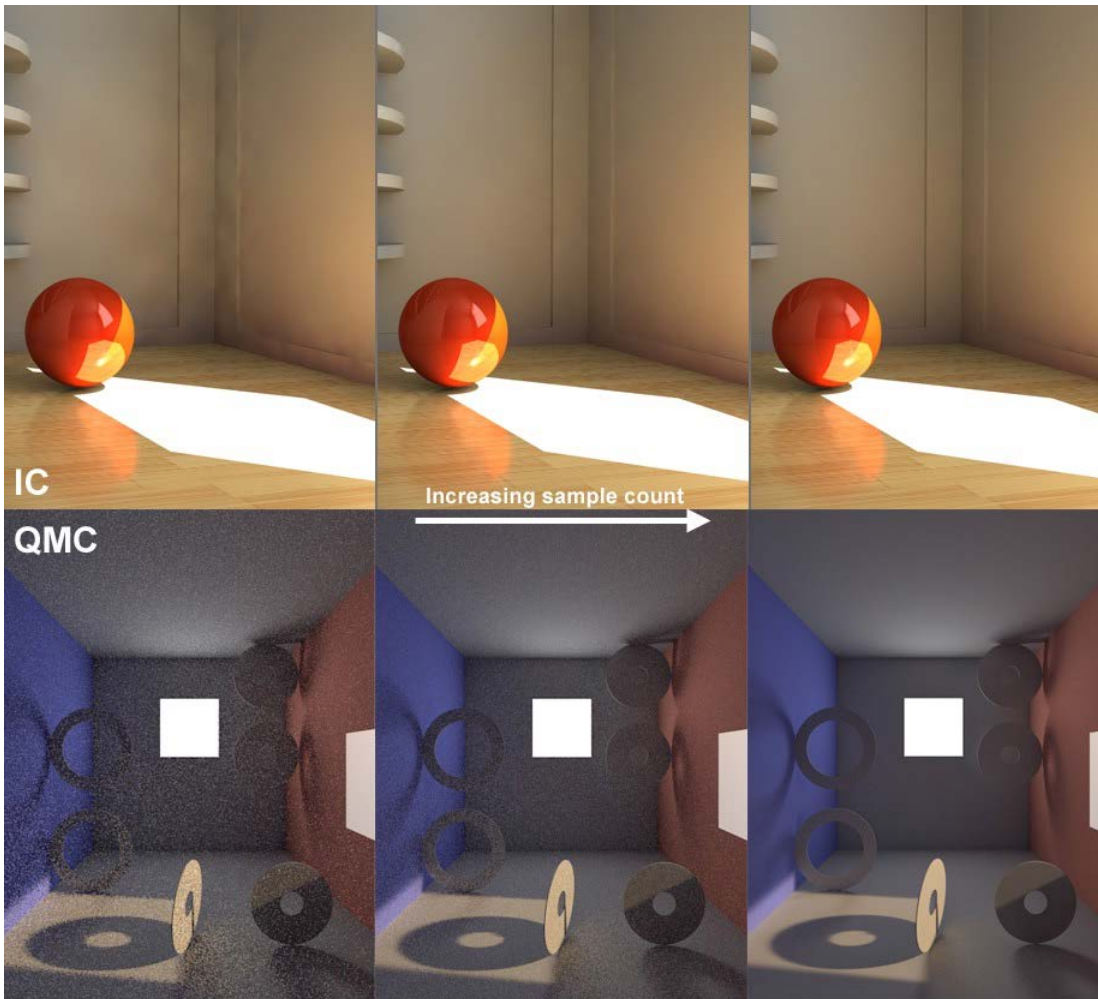
- A fixed number of samples defined by the Sample Count setting



- Selecting **Custom Sample Count** lets you manually define the Sample Count value
- Selecting **Custom Accuracy** lets you manually define the Accuracy value

Accuracy: Use this setting to define an optimized sample count. The optimal count depends on the Project (and in IR GI Mode, on the other Irradiance Cache settings as well) and of course the defined Accuracy value.

Sample Count: This setting defines the fixed number of samples to be used. A higher value produces a correspondingly better render quality (for QMC this can be seen in the graininess; for IR the number of spots is reduced).



Increasing Sample Count (left to right)

This rendering Sample Count will also be used by the Discrete Area and Discrete Sky Sampling if they do not have a custom count defined (see below).

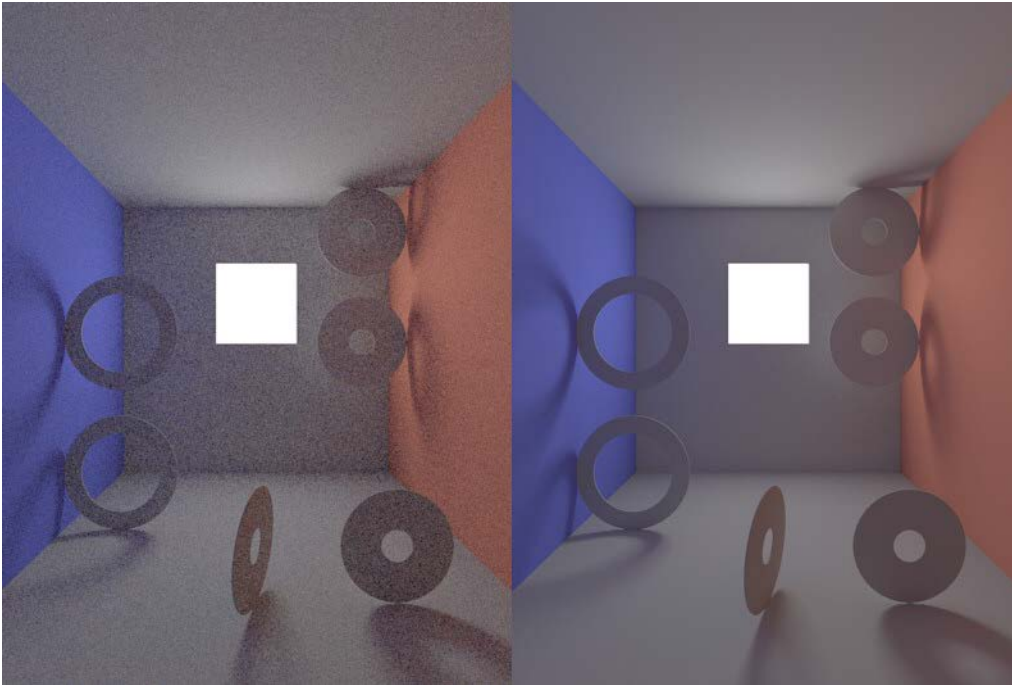
Discrete Area Sampling

Check **Use Area Sampling** to enable this method.

In order for this sampling type to work, the **GI Area Light option** must be enabled in the Surface's Illumination channel.

[See Illumination \(Cineware Surface Channel\).](#)

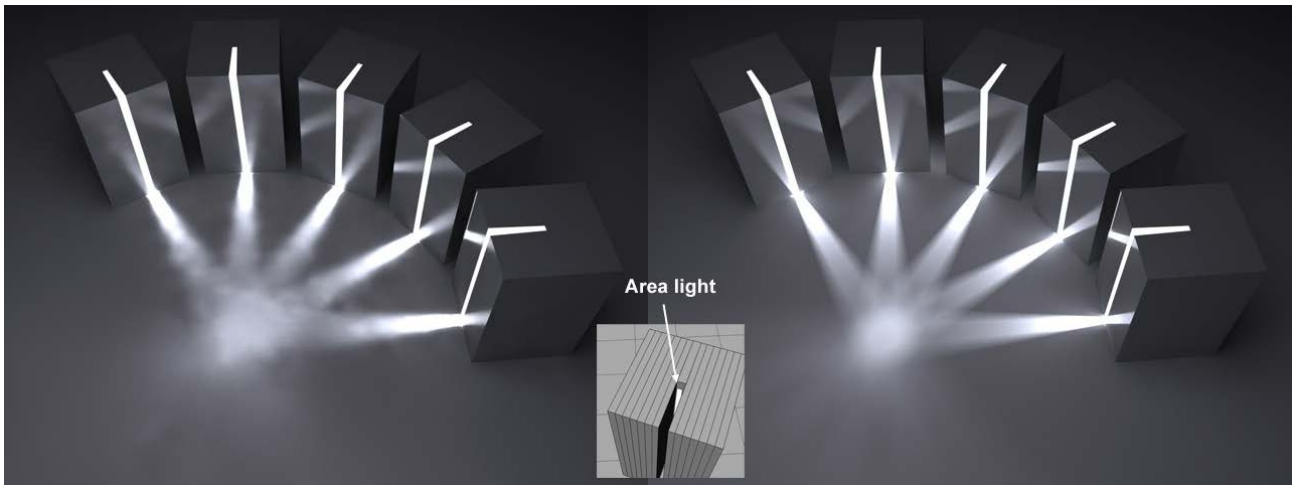
This sampling method sends additional samples to (polygon) Area lights. This will emphasize them disproportionately, which will have a major effect on the quality of the GI.



At center: Discrete Area Sampling disabled, at right: enabled

Note that even if you disable this option, the Area lights will NOT be omitted from the GI calculation. These lights will merely receive no special attention and will be hit randomly by the hemispheric sampling (with a correspondingly grainy result).

Force Per-Pixel: This option is only advantageous in IR Primary Method. Normally every light will be taken into consideration for the cache when an Irradiance Cache is created. However, this does not work if you have very small, bright Area lights. The result will be spotty images. If you enable the Force Per-Pixel option, the calculation of the Area lights will be split from the cache and calculated separately (as the QMC method does by default) for each potential pixel (object surfaces but no backgrounds or skies, for example).



At left, Force Per-Pixel disabled, at right: enabled

Custom Count/Sample Count

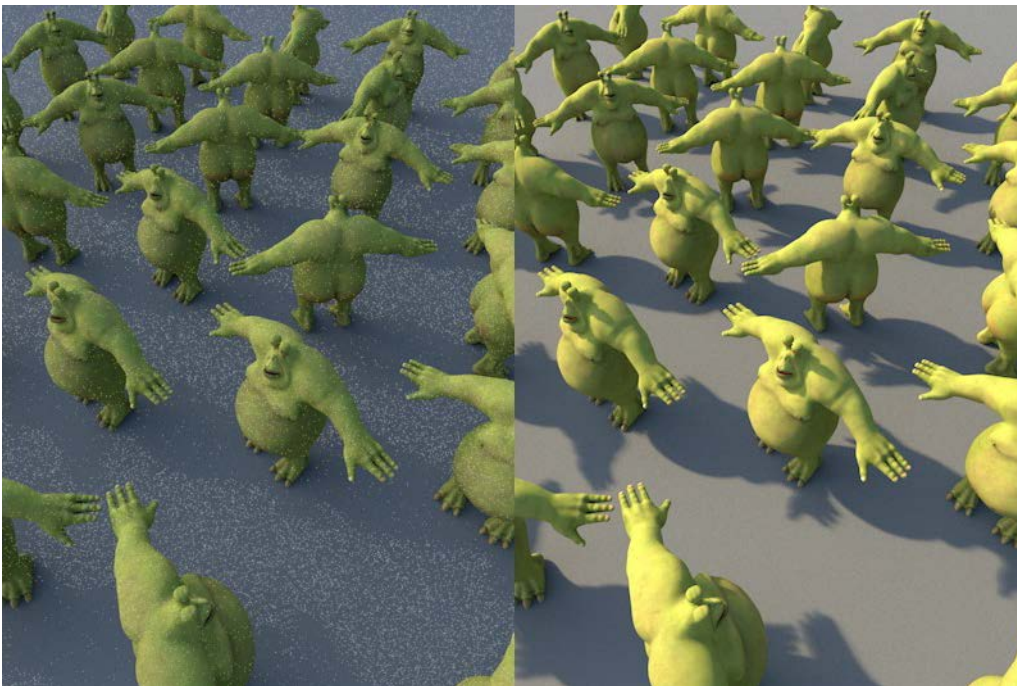
Use these fields to define a custom sample count. If Custom Count is disabled, the same number of samples will be used as defined in Stochastic Sampling.

Discrete Sky Sampling

Check **Use Sky Sampling** to enable this method.

This sampling mode takes into account the sky in particular (e.g., the Physical Sky or a HDRI Sky).

A sky map will be calculated internally during rendering, which will then concentrate the additionally created samples primarily on the brightest regions during rendering. This means that HDRI textures with enough contrast can cast shadows with locally very bright regions.

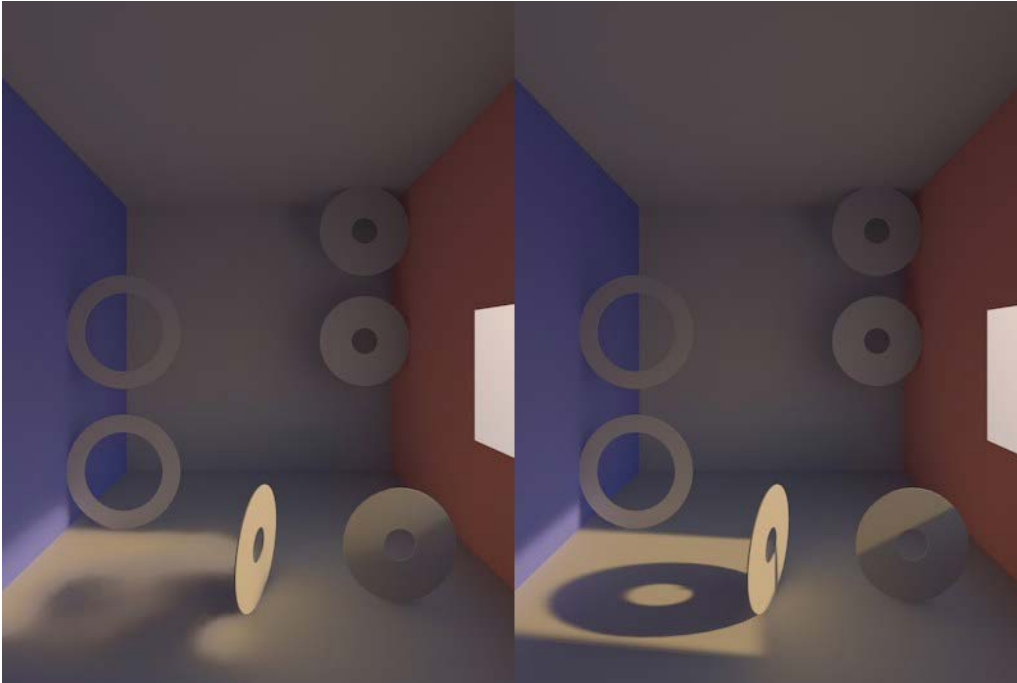


*The only source of illumination here is the HDRI texture on a Sky object.
Note the relatively hard shadow (option disabled at left).*

If this option is disabled, the sky will NOT be omitted from the GI calculation. The sky will merely receive no special attention and will be hit randomly by the hemispherical sampling (the extremely bright sun will produce a grainy image).

Force Per-Pixel: This option has advantages only for the IR Primary Method. When an Irradiance Cache is created, the sky will normally be sampled and taken into consideration for the cache. For bright regions (sun) or those with smaller surfaces, this method has limits and will result in spotty renderings.

If you enable Force Per-Pixel, the calculation of the light emitted by the sky will be split from the cache and calculated separately for each pixel that comes into question (object surfaces but no backgrounds, skies, etc.), which is how the QMC method works by default.



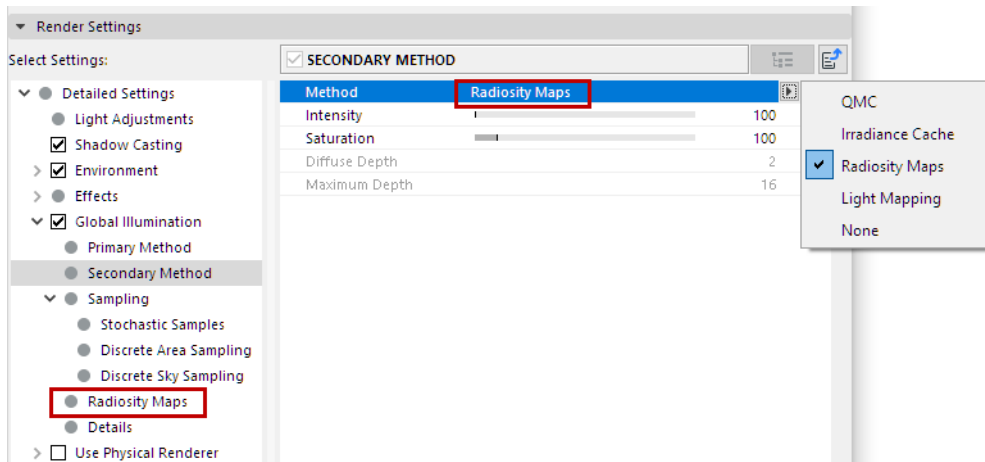
*A Sky object with an HDRI texture applied to it is located behind the window.
At left, Force Per-Pixel disabled, at right enabled.*

Custom Count/Sample Count

Use these fields to define a custom sample count. If Custom Count is disabled, the same number of samples will be used as defined in Stochastic Sampling.

Radiosity Maps (Cineware Global Illumination)

This setting is available, as part of the optional Global Illumination effect, in the Detailed view of PhotoRendering Settings for the Cineware engine.



When using the Secondary Method, Radiosity Maps are a quick and easy to use function that is particularly well-suited for preview renderings.

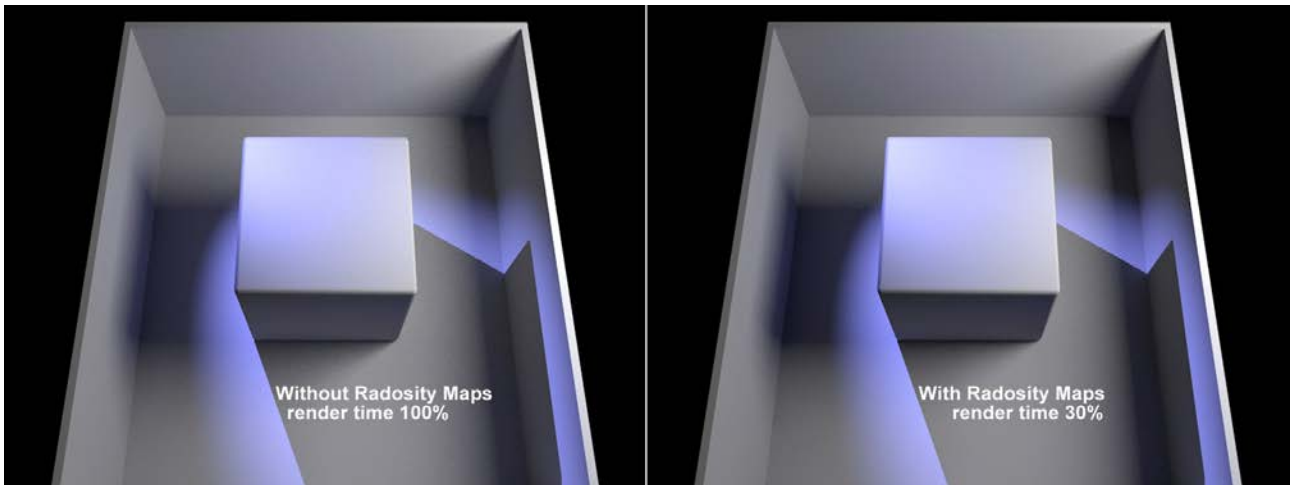
Simply put, illumination (light sources, Area lights, sky) on polygons is calculated internally as special textures (Radiosity Maps) during rendering, prior to the actual GI calculation. These Radiosity Maps are then used during the actual GI calculation for faster rendering. This method has several advantages and disadvantages:

Advantages:

- GI calculation is faster
- Radiosity Maps can be saved and re-used.

Disadvantages:

- The actual diffuse depth (number of diffused light reflections) is 1 (2 in the case of Area lights and/or skies), which darkens the rendering. To some degree, this can be offset by increasing the Gamma value.
- More memory is required
- When using simplified geometry (e.g., single-polygon walls without thickness), light can seep through. This can be prevented by modeling more realistically, i.e., giving walls a thickness.



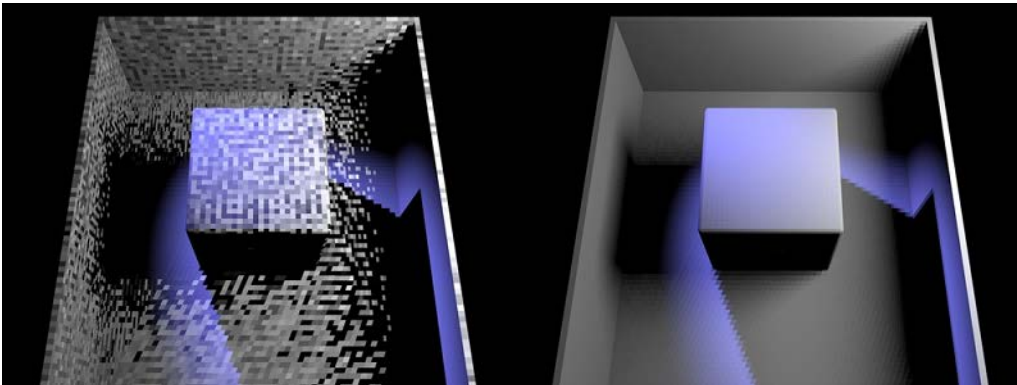
In the example above, the render time was greatly reduced while achieving a similar result. The use of Radiosity Maps darkens (reduces diffuse depth) the corners and shadows somewhat.

Tips:

- Radiosity Maps can be made visible (set the Mode control to “Shading”).
- Radiosity Maps should have a light dispersion that is as homogeneous as possible.

Map Density

Use this setting to define the Radiosity Map resolution. The higher the value, the smaller the Texels will be (the small squares for which a uniform color/brightness is ascertained) and the higher the quality of the Radiosity Map will be (with correspondingly longer render times and increased memory requirements).

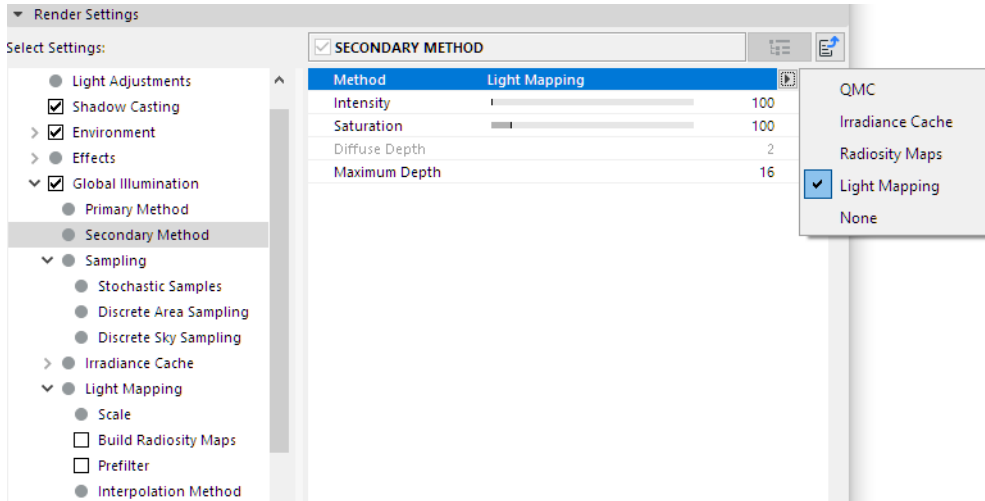


At left a Radiosity Map of poor quality, at right of better quality

You can make Texels visible without complicated calculations by setting the Mode to Texels and rendering the Project.

If the Texels are too large and light seeps through, reducing the size of the Texels can help alleviate this problem.

Light Mapping



Simply stated, the Light Mapping method works as follows: A series of samples are emitted into the scene from the angle of view of the camera. These samples are often reflected (depending on the Maximum Depth value), unless the sample doesn't first strike the sky or doesn't strike anything at all. The colors calculated when geometry is struck are evaluated. Entire sample chains result, which can be calculated very quickly (also for high numbers of reflections) and in consideration of other sample chains - faster than all other GI methods. The calculated colors are saved in a cell pattern (or as a file, if desired, which can be re-used later) and then made available using the Primary Method, which itself uses the Light Map with a sample depth greater than 1 when gathering light (samples).

Note that the rendered image will more often be brighter due to the high sample depth, which is higher than that of other GI methods. To compensate for this brightness, reduce the Intensity value.

This method has both advantages and disadvantages:

Advantages:

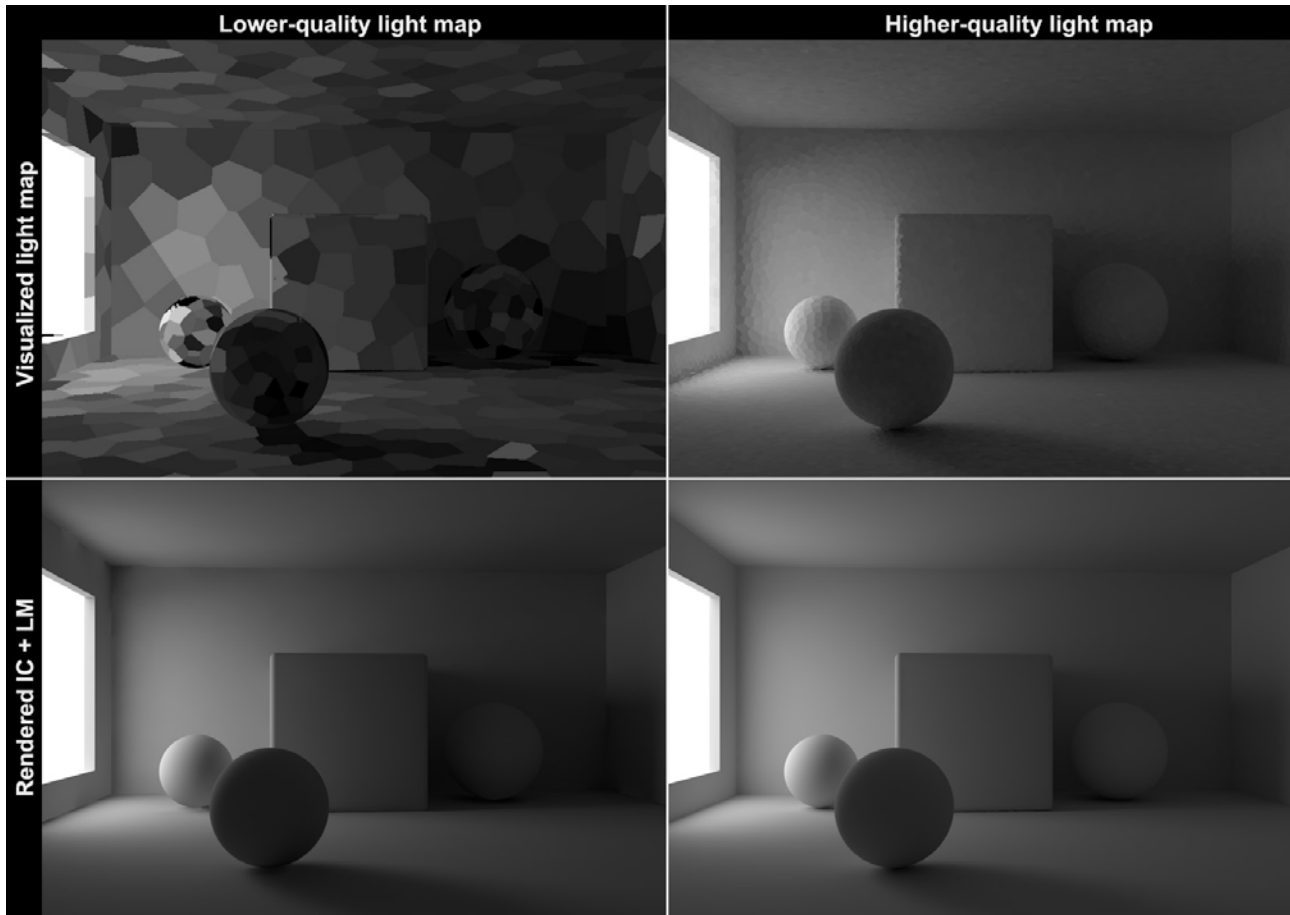
- Very fast GI calculation (with very high sample depths)
- Light Maps can be saved and re-used to a certain degree (these are dependent on the angle of view)

Disadvantages:

Light leaks can occur (these can be minimized by reducing the Samples Size value and by not using interpolation. Using thicker objects instead of single polygon surfaces also helps).

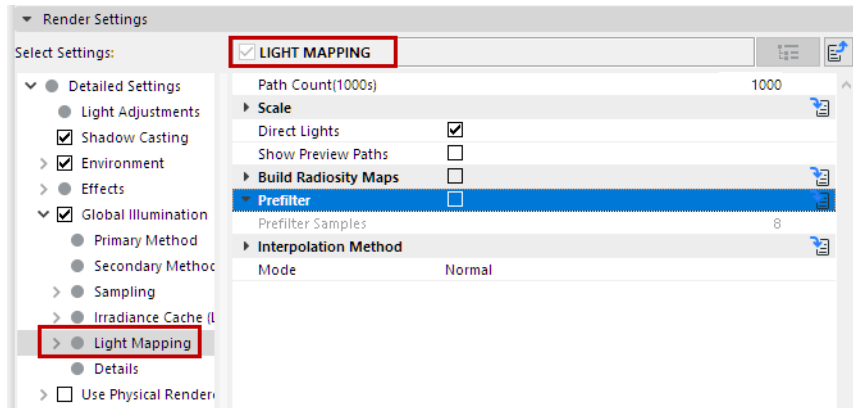
Light Map Characteristics

First, we will take a look at what a good Light Map looks like (you can make a Light Map visible by setting the Mode to Visualize):



At the top left and top right are poor and better quality Light Maps, respectively. Good Light Maps have a homogeneous light progression; in contrast, Light Maps with poor quality appear heterogeneous. When rendered, the difference is not as apparent, because the Primary Method takes numerous Light Map samples with numerous samples and produces median values. Nevertheless, the Primary Method will deliver sub-optimal results, even with the best settings, if the initially calculated Light Map was of poor quality. This can be seen at the bottom left of the image, where flickering occurs in the regions around the window and beneath the left ball.

Light Mapping Settings

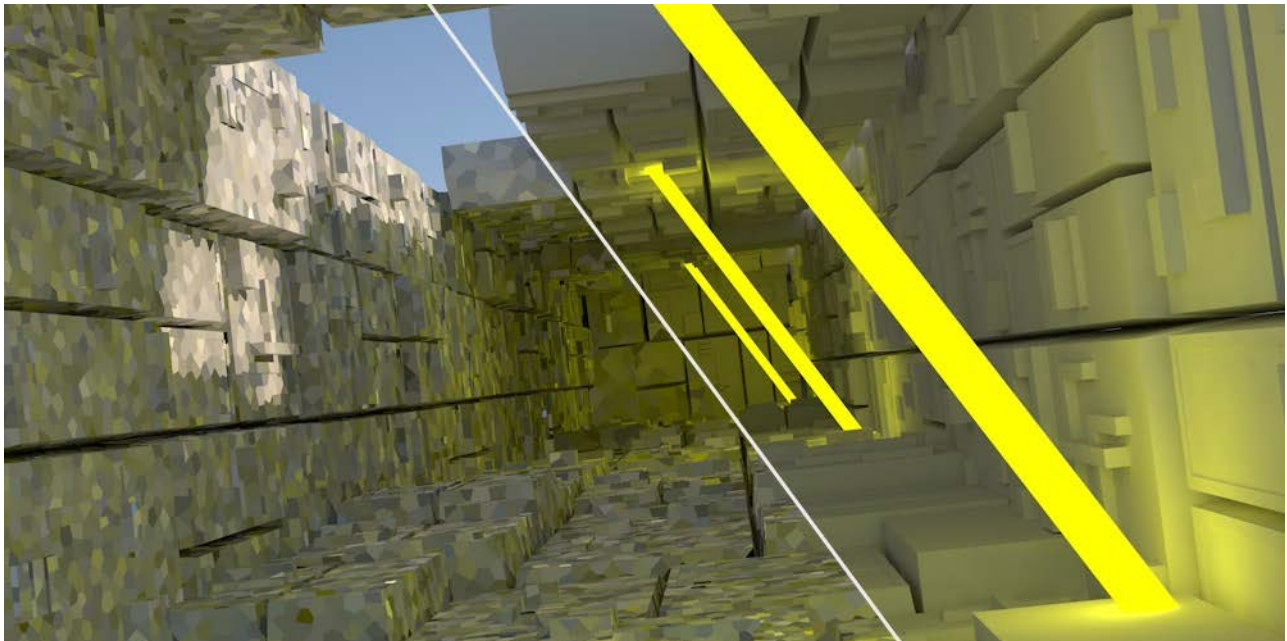


The **Prefilter** and **Interpolation Method** functions can be used to remove the cell pattern and for smoothing (both render very fast), in order to achieve the most homogeneous light dispersion

Path Count (x1000)

Next to the Record Density setting, Path Count (1000s) is the most important setting for adjusting Light Map quality.

The Path Count (1000s) value (which is multiplied internally by 1000) defines the number of samples that should be calculated for the entire scene. A sample chain with a depth corresponding to the Maximum Depth value will be generated.



Left: Smaller Path Count (1000s) values. Right: Larger values.

The higher the number, the more homogeneous the light dispersion will be and the longer the corresponding render times will be. Also, the higher the Path Count, the more samples will be used per cell element and the random color divergence (at the top of the image, a sample coincidentally strikes a black seam) of neighboring cells will be smaller

Samples Size



Left: Larger Sample Size values, Right: smaller values

Use these values to define the cell size. The smaller the cells, the more accurate the result will be with regard to details. Cells that are too large will quickly lead to light leaks and are less precise with regard to details (shadows will be lost here and there). Depending on the Scale defined (see below), Sample Size can be defined as absolute (World) or relative (Screen).

Scale

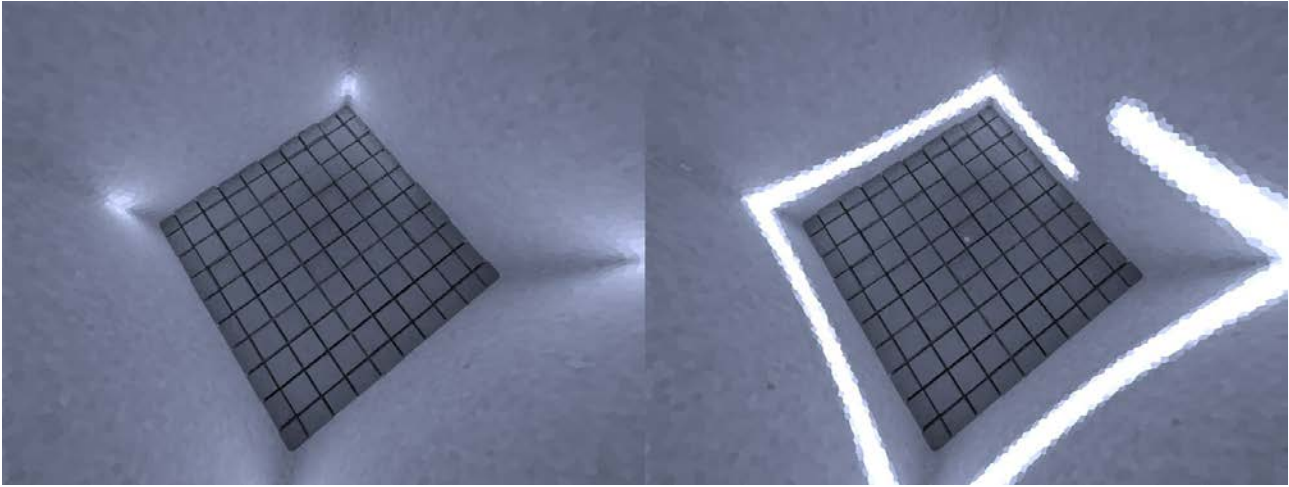
Select from the following options:

- **Screen:** The cell diameter is defined as a fraction of the output size. A value of 0.1 represents 10 cells wide. The cell depth will decline for geometry in the distance. This setting is affected by several algorithms, which use other criteria (e.g., very small Sample Size values will produce larger cells and geometry such as spheres will have smaller cells) to dynamically determine the cell size.
- **World:** The Sample Size values can be output as absolute in the world coordinate system. Sample Size will represent the approximate diameter of a cell, which means that the cell density will appear greater on distant geometry than on geometry that lies nearer.

Direct Lights

Enabling this option will speed up rendering for projects with a high number of real light sources. For GI calculation, surfaces illuminated by light sources will be placed directly into Light Maps:

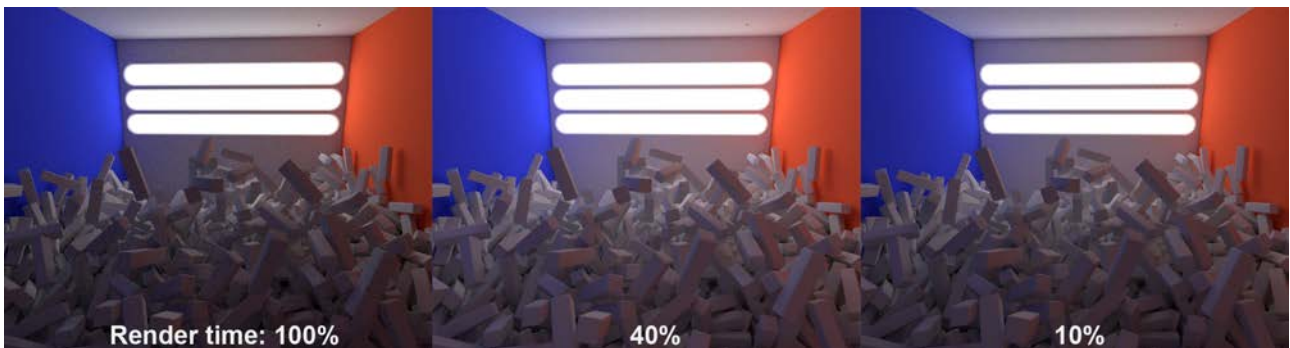
Enabling this option will speed up rendering for projects with a high number of real light sources. For GI calculation, surfaces illuminated by light sources will be placed directly into Light Maps:



Left: Direct Lights disabled; right: enabled. The strip of light is the light emitted by 120 Spot lights.

The gain in render speed can be quite substantial, depending on the scene (simply put, light source information gathered during Light Map calculation is subsequently re-used by the GI Primary Method). Very good results at moderate render times can be achieved when using QMC+LM.

The following image was rendered using QMC+QMC (Record Density of 8) on the left, QMC+LM (Direct Lights disabled) at center, and QMC+LM (Direct Lights enabled) at right:



Note how much faster and better Direct Lights renders. The Light Map's high Record Density produces a brighter and more realistic image.

Show Preview Paths

This option does not affect the Light Map directly. If enabled, the progress of the samples just calculated will be shown during calculation, then compiled according to Sample Size in a cell and averaged.

Build Radiosity Maps

Enabling this option is like switching on a turbo charger. If enabled, the Light Map will be calculated and it will then be converted to a Radiosity Map, which will be used internally for rendering. This dramatically reduces render time while basically maintaining the same level of quality (both with IR+LM and with QMC+LM).

- Disadvantage: Radiosity Maps require a lot of memory for the saved cache on the hard drive as well as RAM. Problems can occur with complex projects. Rendering a Light Map with the right settings is also very fast.

Map Density

This setting works like the Map Density Radiosity Map setting of the same name, but the sampling is much faster. This is where you adjust the texel size.

See [Map Density for the Radiosity Map](#).

Sampling Subdivisions

This setting works like the Sampling Subdivisions Radiosity Map setting of the same name, but the sampling is much faster. This is where you adjust a type of “antialiasing” for the texels.

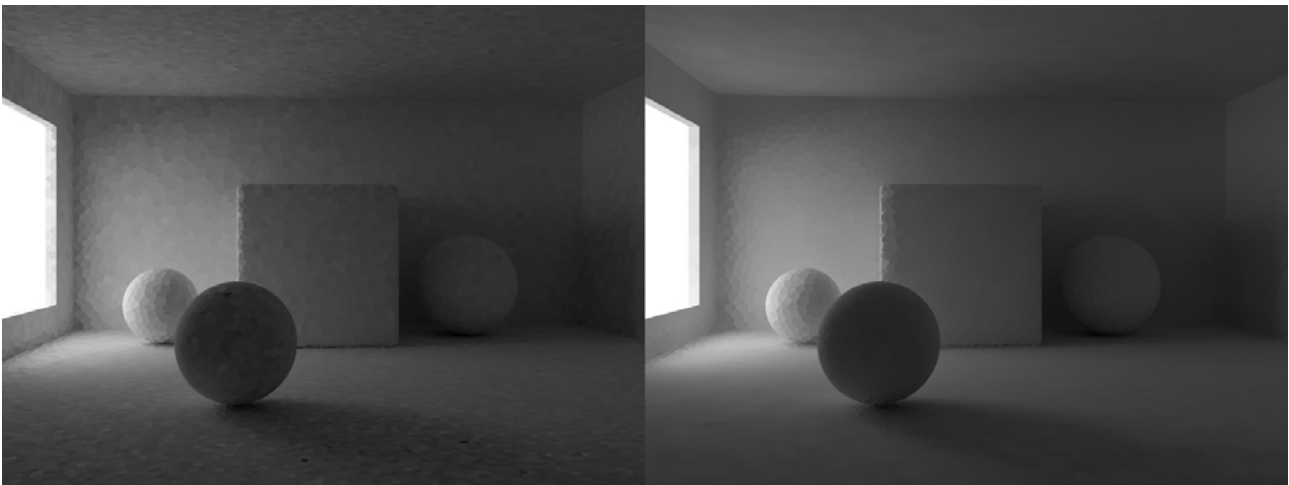
Prefilter

Check this box to enable the Prefilter option.

The Prefilter ensures that a heterogeneous, spotty Light Map (or Radiosity Map) is converted to a more uniform map before they are used for rendering or one of the following interpolations.

This is done per cell. Depending on the settings, the colors of several neighboring cells will be averaged and then assigned to the cell. This process is calculated very fast and has basically no effect on render time.

However, note that a type of blur effect takes place that can swallow details and lead to light leaks (which can be compensated for by improving the Path Count (1000s) and Sample Size settings).



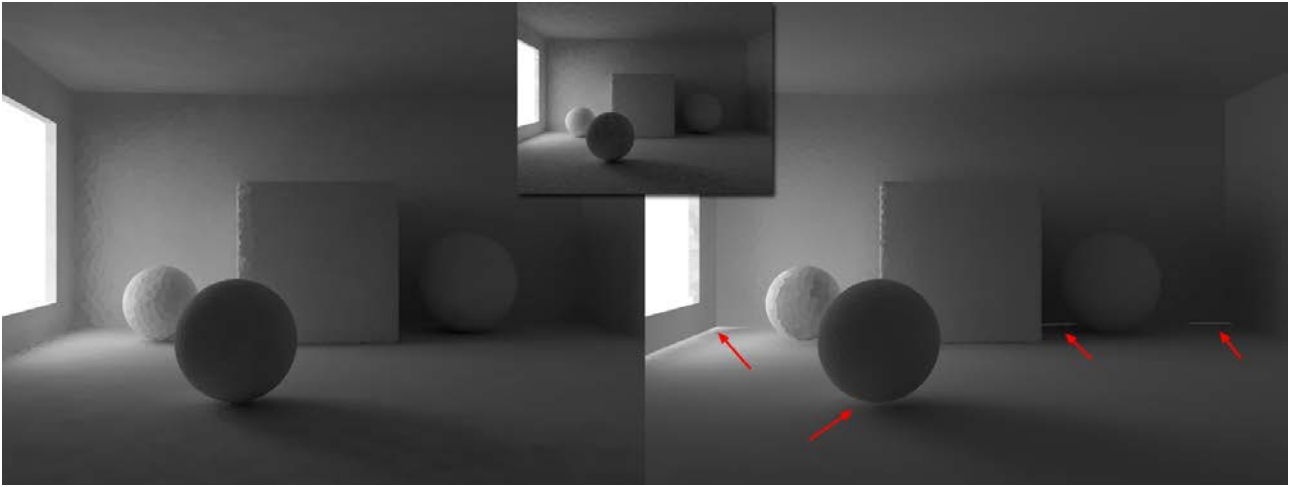
Left without Prefilter, right with Prefilter.

Note: Both the Prefilter and the effect of the interpolation methods are always calculated anew during rendering.

Prefilter Samples

Use this setting to define the size of the radius for the current cell by averaging surrounding cells.

Values that are too large will swallow details and lead to light leaks.



At left a small Prefilter Samples value, at right a larger value.

Note that contact shadows and light leaks are present on the right.

Interpolation Method

During rendering, the Light Map's (or Radiosity Map's) cells are actually interpolated so the cell structure is dissolved if the settings' values are high enough. This will produce even brightness progressions.

Even better results can be achieved in combination with the Prefilter. However, interpolation requires a corresponding amount of additional render time and correspondingly more light leaks will result with larger interpolations.



The pre-filtered Light Map on the left has interpolation added at the right.

Method

Choose a method for planar interpolation of discontinuous color gradients (cells):

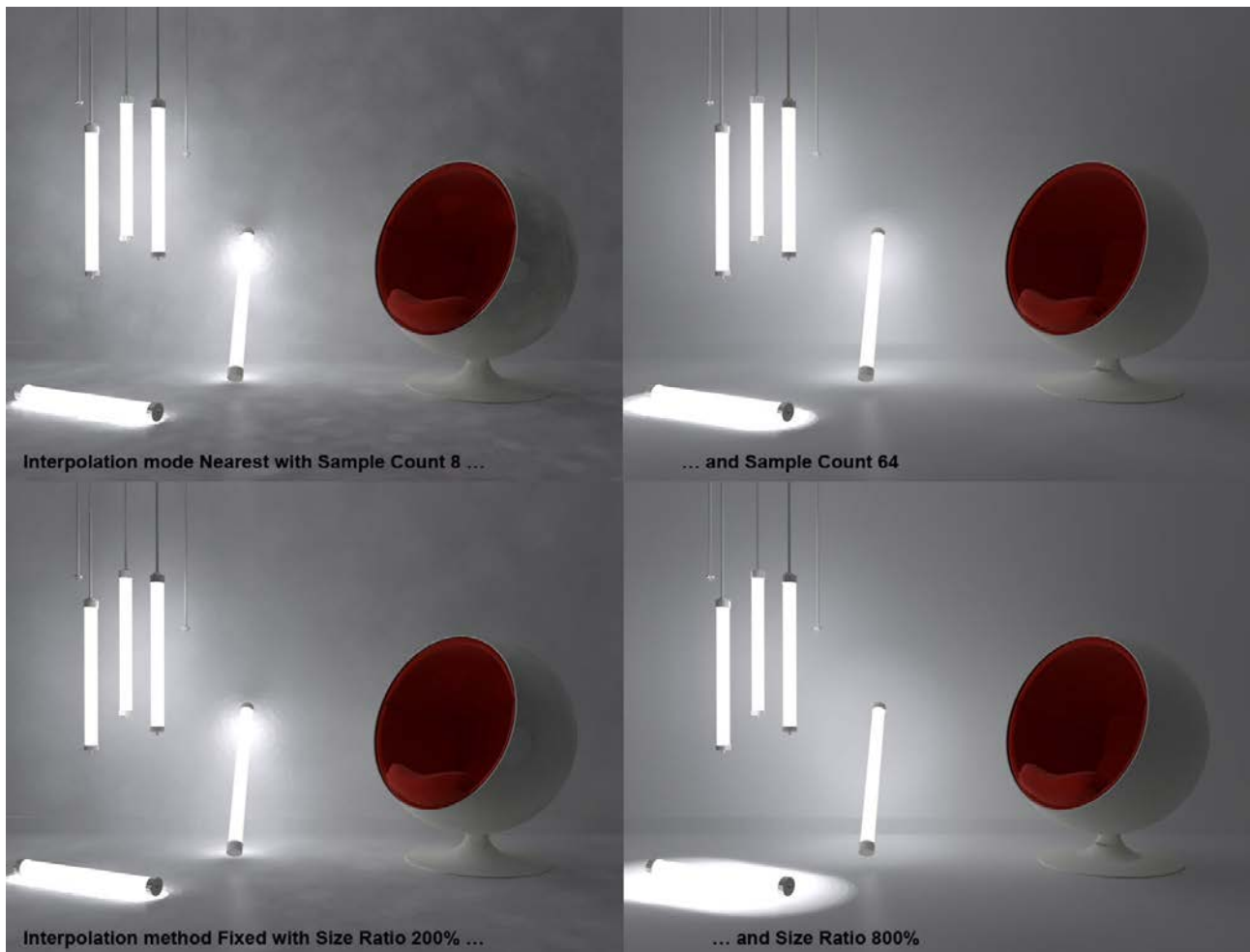
- None: No interpolation will be made (the calculation is very fast); light leaks will be minimized but the Primary Method for GI will see the cells. Sufficient pre-filtering can help.

- Nearest: A specific number of neighboring samples (defined by Sample Count) is ascertained and their colors are averaged. This is not an absolute value, because it also considers the Samples Size value. The sample density will reduce the radius in which the samples lie.
- Fixed: In conjunction with the Size Ratio value, a fixed distance around the point to be calculated is ascertained within which samples are obtained. This method produces the most “blurred” results.

See the image below for an illustration of these effects. (For illustration purposes, Prefilter was not used):



Prior to interpolation.



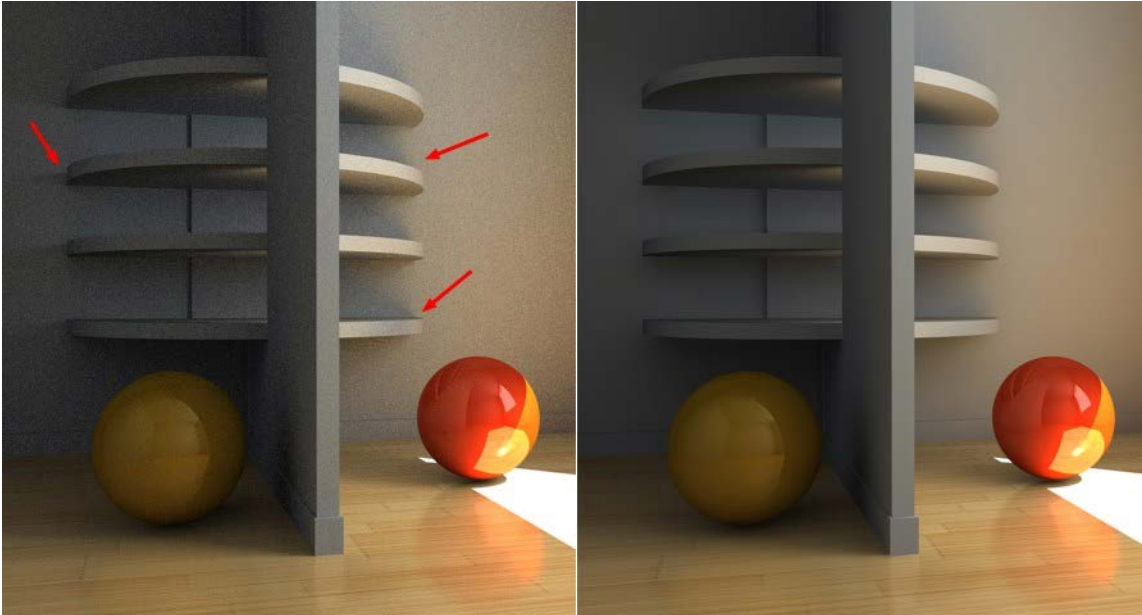
Various interpolation methods and settings.

Mode

Select which Light Map should be displayed:

- Visualize: The Primary Method for GI will not be calculated, and only the Secondary Method will be displayed. This mode is suitable for fine-tuning a Light Map prior to rendering.
- Normal: The final rendering must ALWAYS be done in Normal mode

How Does QMC Work?



Left, rendered with QMC+QMC, right with IR+IR.

Note the increased level of detail (shadows) in the QMC rendering.

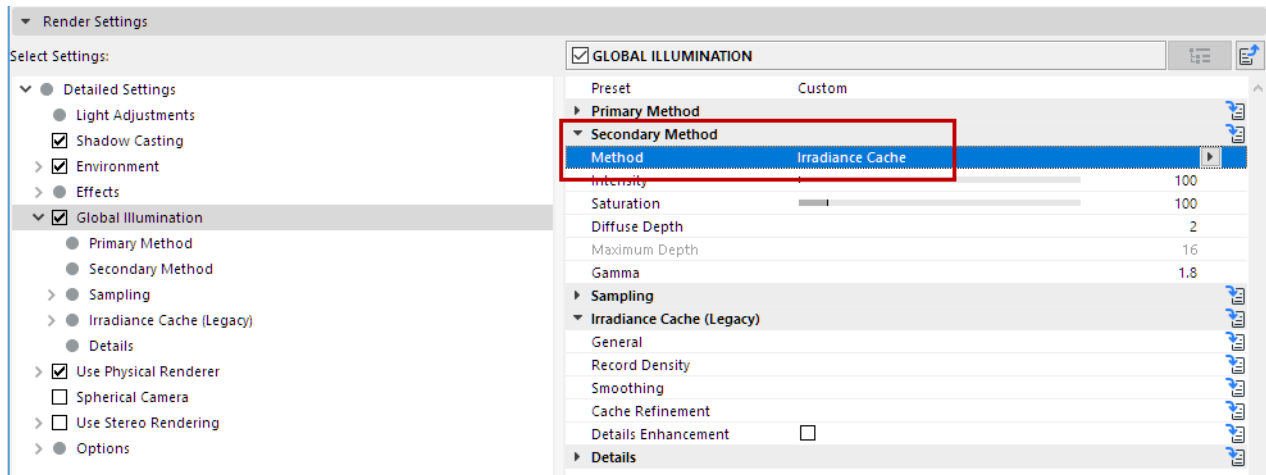
The QMC method of Global Illumination calculation works under a so-called “brute force” principle. This means that, for each object pixel in the image, a definable number of rays (“Sample Count”) can be “beamed” in a hemispherical pattern into the scene. This is not an adaptive process, and render time will definitely not be reduced.

Its advantage is that it offers the most precise render results. Small details in shadows and shading can be rendered out that would otherwise not be noticeable if the IR method were used.

Disadvantages of using QMC as primary method:

- The render times are maxed out (much longer than with Irradiance Cache) and, since brightness and color values based on a finite and randomly distributed number of rays (“Sample Count”) is ascertained for each pixel, the images will have a slight graininess to them that can only be compensated for by increasing the sample count, which in turn increases render time accordingly. Using Polygon Lights/ Portals (indoor scenes, primarily illuminated by light falling through windows) can noticeably improve render quality (which also saves render time).
- No caching takes place (as well as no pre-pass calculation). Hence, a lot of CPU power is needed (e.g., render farms) to render animations at an acceptable level of quality.

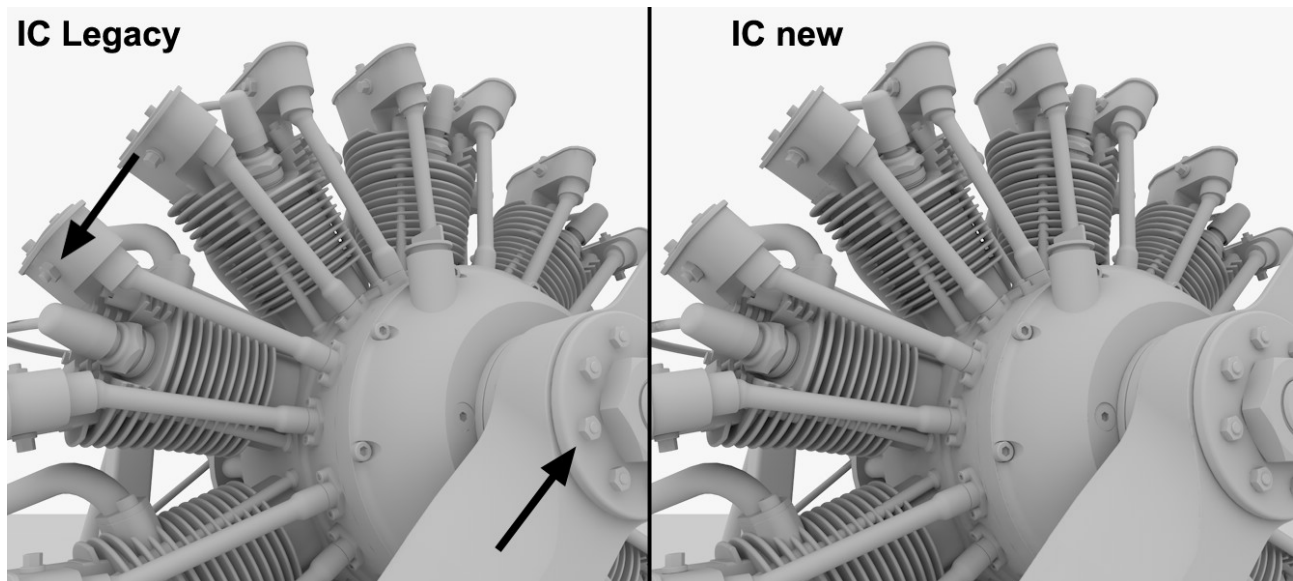
Irradiance Cache (Global Illumination)



A new Irradiance Cache method was introduced as of Archicad 20. This method bears the following advantages compared to other methods:

- Small details such as contact shadows - i.e., those produced where objects meet, at corner edges, etc. - are of much higher quality (with the right settings in the quality of a pure QMC rendering).
- New algorithms speed up rendering.

The Irradiance Cache (Legacy) method is still available. See [Irradiance Cache \(Legacy\) \(Cineware Global Illumination\)](#).



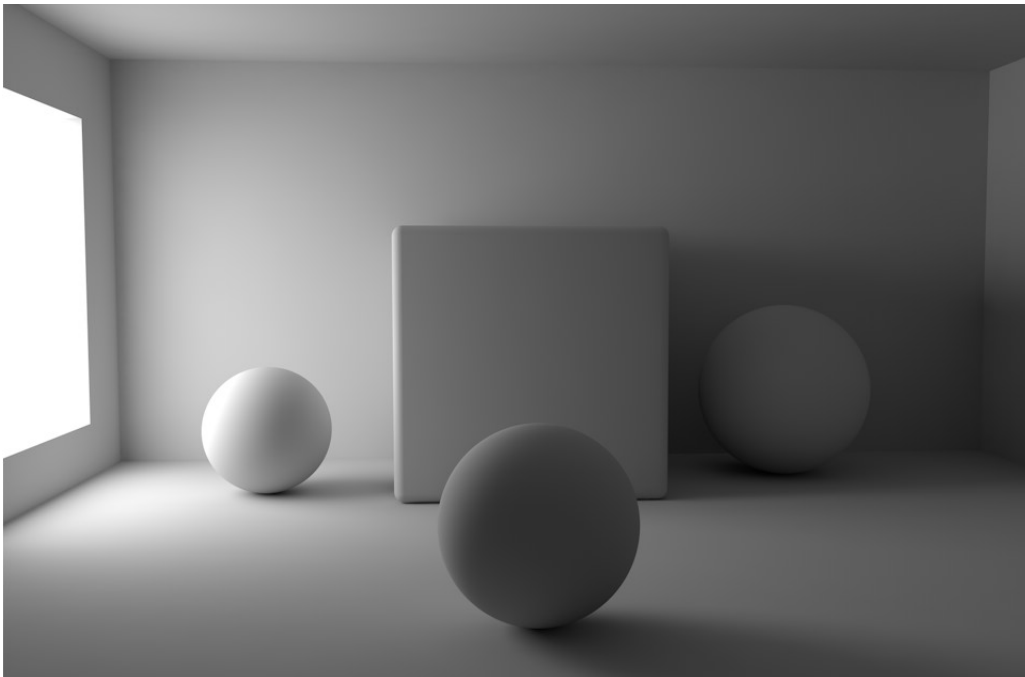
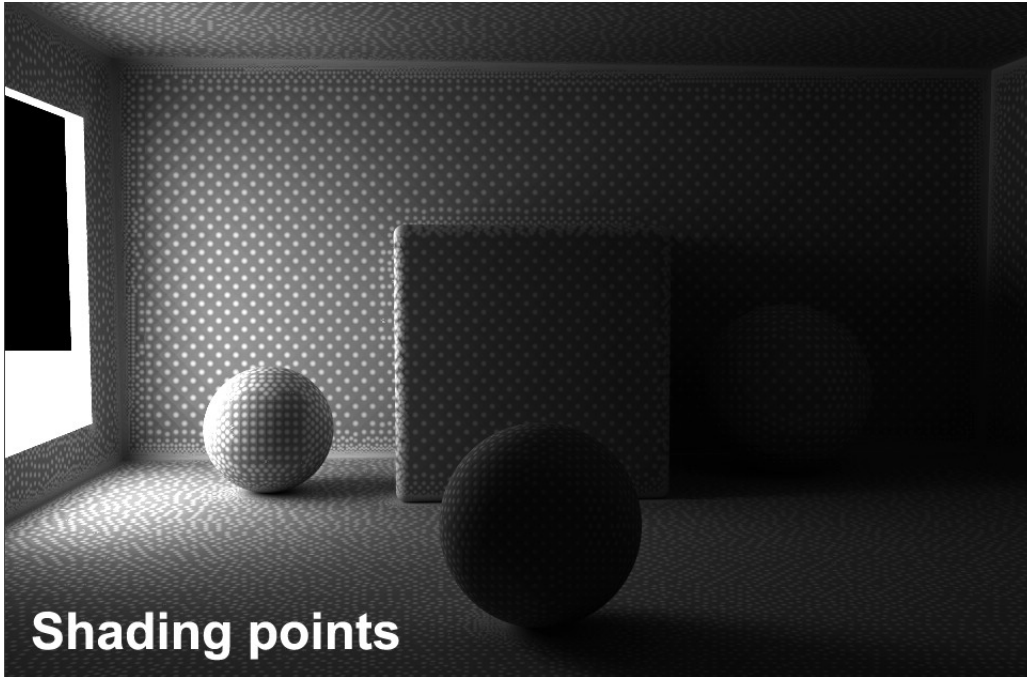
Improved rendering of contact shadows (e.g., in the regions marked in the image on the left).

Model by Steen Winther.

Note: All images on this page were rendered using Irradiance Cache, both as Primary and as Secondary method.

What is an “Irradiance Cache?”

Several pre-calculations (“pre-passes”) are made during IR rendering in which the scene is analyzed in order to ascertain the most important indirect lighting regions (the “shading points”, which are the points displayed during pre-passes; these become very apparent when using lower Density values). [See below for more details:](#)



Brightness and color values for the shading points (upper image) are interpolated to ensure a homogeneous dispersion of light (lower image).

The brightness and color values for these shading points will be saved in the Irradiance Cache as so-called “entries”. These Irradiance Cache entries are interpolated during final rendering to supply pixels that lie between shading points with indirect light.

Disadvantages

- When interpolating between the limited number of shading points, details with regard to light and shadow can be lost (however, not nearly as drastically as with the Legacy method). In this regard, QMC still has the advantage.
- Remember that the QMC mode always offers the best GI quality with regard to light and shadow dispersion (and unfortunately the slowest). Irradiance Cache always attempts to produce a result as closely as possible to QMC.
- Generally speaking, Irradiance Cache tends to flicker most if very bright and small Polygon Lights are used. Large, homogeneous light sources (e.g., a sky used for illumination from which light is emitted evenly from many sides) are especially IR-friendly.

Notes:

- The following settings can also be found in the Ambient Occlusion's settings. The basic principle with which they function is the same (AO, however, only has a Record Density depth of 1).
- If you get spotty results, these can most often be eliminated by using a higher Record Density value. Improving the Secondary Method's settings will help.

Record Density

Most of the following settings will only be used for fine-tuning. For most instances, the Record Density settings Low, Medium and High in conjunction with corresponding values for the settings below will suffice. The Preview mode offers a quick preview of the final result. Custom will be made available as soon as the Record Density value is changed manually.

Generally speaking, the Record Density value must only be modified if, for example, a Polygon Light's (e.g. Portal's) hard shadow should be shown (and you don't want to use the slow Force Per-Pixel).

Min Rate/Max Rate

When rendering with Irradiance Cache, several pre-passes will first be calculated (the phase in which squares appear that successively become smaller). During this phase, the dispersion of shading points will be ascertained. This is an adaptive process in which particular emphasis is placed on critical regions: corners, shadow edges, etc. The difference between Min Rate and Max Rate defines the number of pre-passes. A value of 0 produce a full resolution image (pixel size 1*1) and a value of -1 will produce a pixel size of 2*2, -2=4*4 and so on. Logically, the Min Rate value should always be less than the Max Rate value. Positive values are, however, also possible, which allow sub-pixel cache entries (this can be useful for Subpolygon Displacement if details are lost).

In the end, these settings are not as important as they were for previous IR. Render times differ only nominally and as long as the values are kept reasonable (Max Rate not less than 0), the render results will not differ very much, either. If negative values are used for Min Rate, and Max Rate is set to 0, you're pretty much on track.

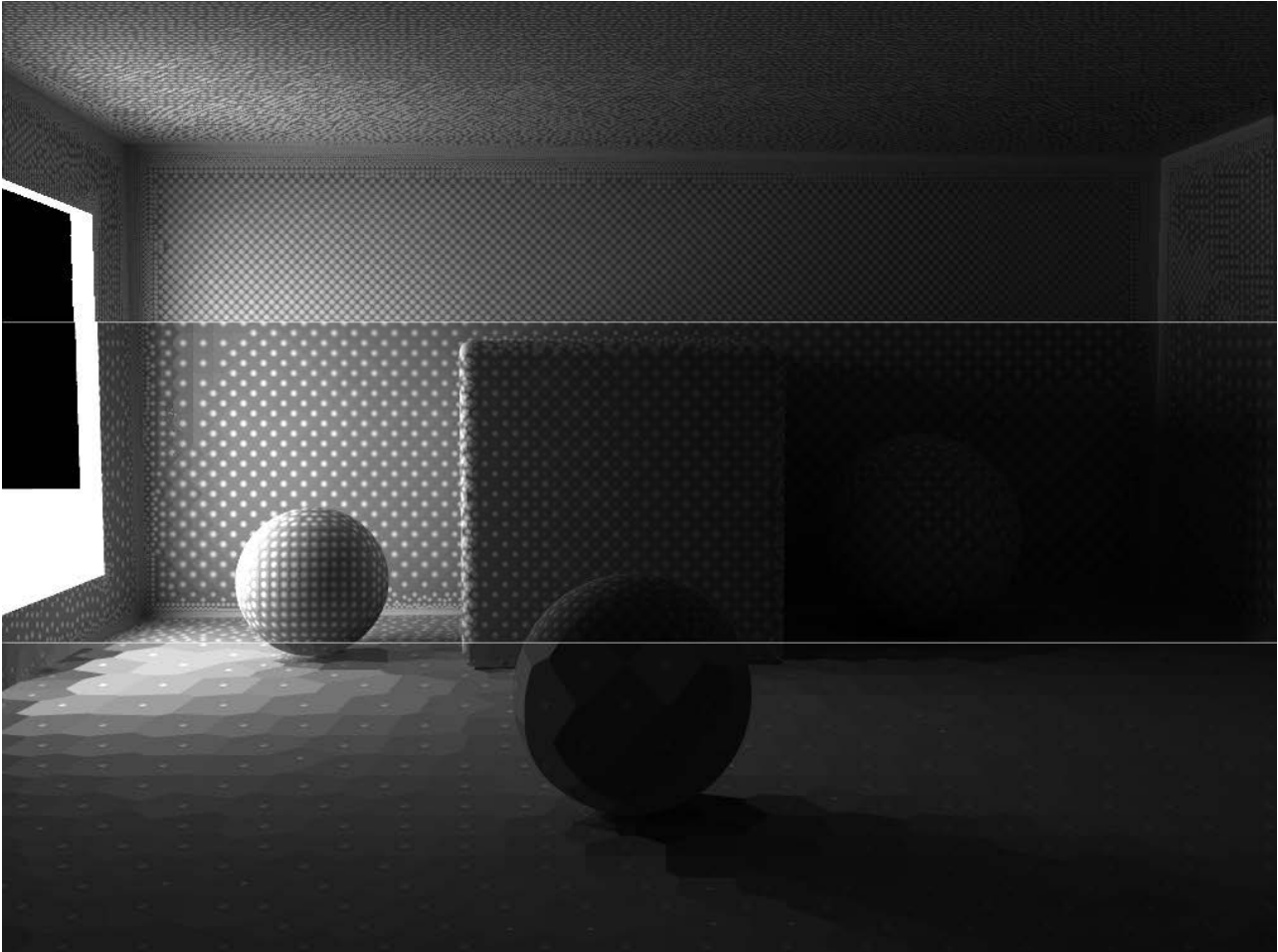
Density

Minimum Spacing/Maximum Spacing

Density: General shading point density. The Density value adjusts the general dispersion of shading points in consideration of the following two settings.

Minimum Spacing: Shading point density in critical areas (e.g., corners, shadows).

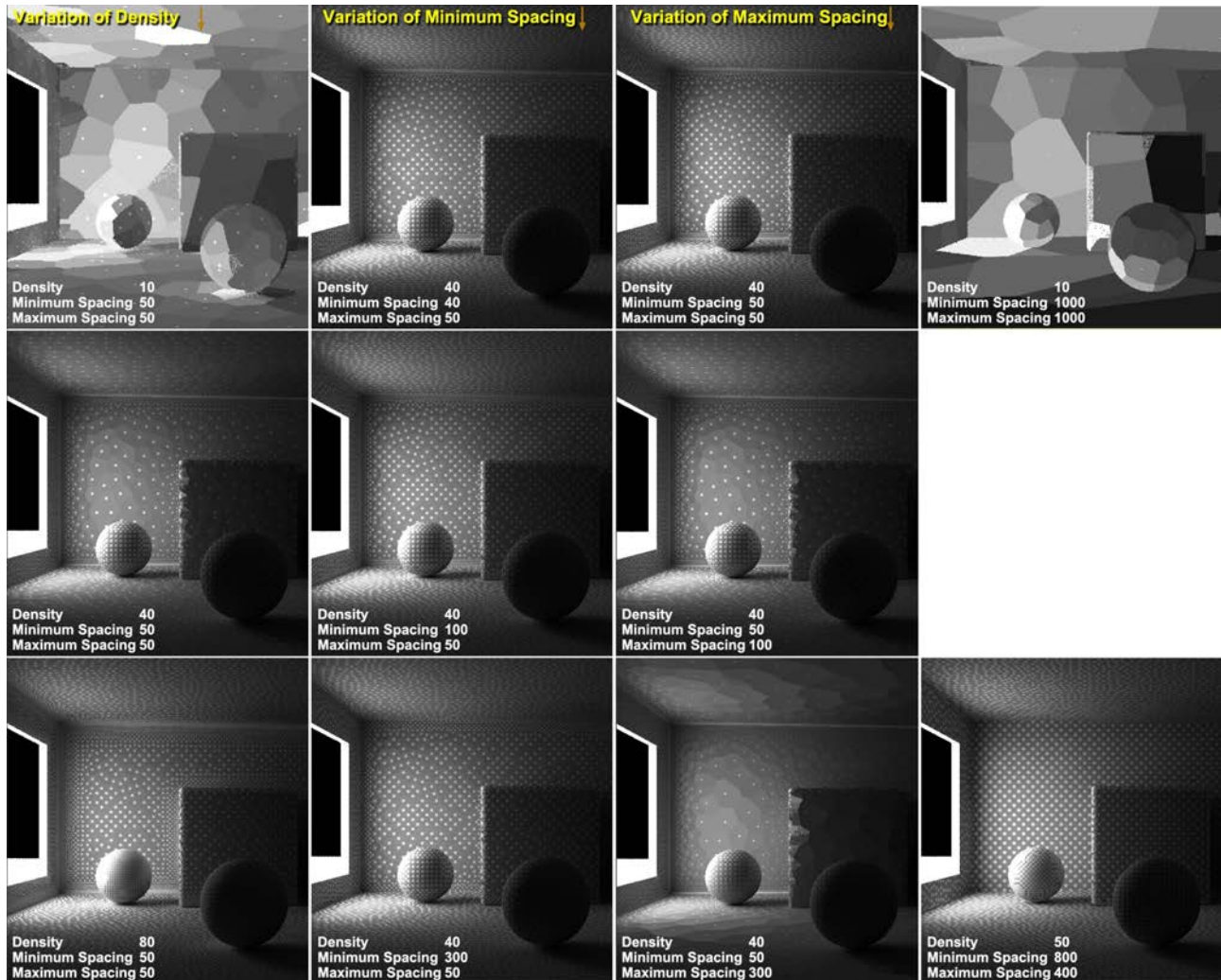
Maximum Spacing: Shading point density in non-critical areas (e.g., plane surfaces).



Increasing Density values from bottom to top

It makes sense to describe these three settings together because all three work together to define the shading point density in critical and non-critical areas:

The following image shows the effects of the above-mentioned settings:



Different variations of the Density, Minimum Spacing and Maximum Spacing settings

Tips for a Cleanly Rendered Image

- Use a high number of shading points in critical areas
- Use a reasonable density value in the remaining areas
- Make sure that each shading point has as many Samples as possible

Smoothing

In general, you will rarely need to modify the Smoothing setting.

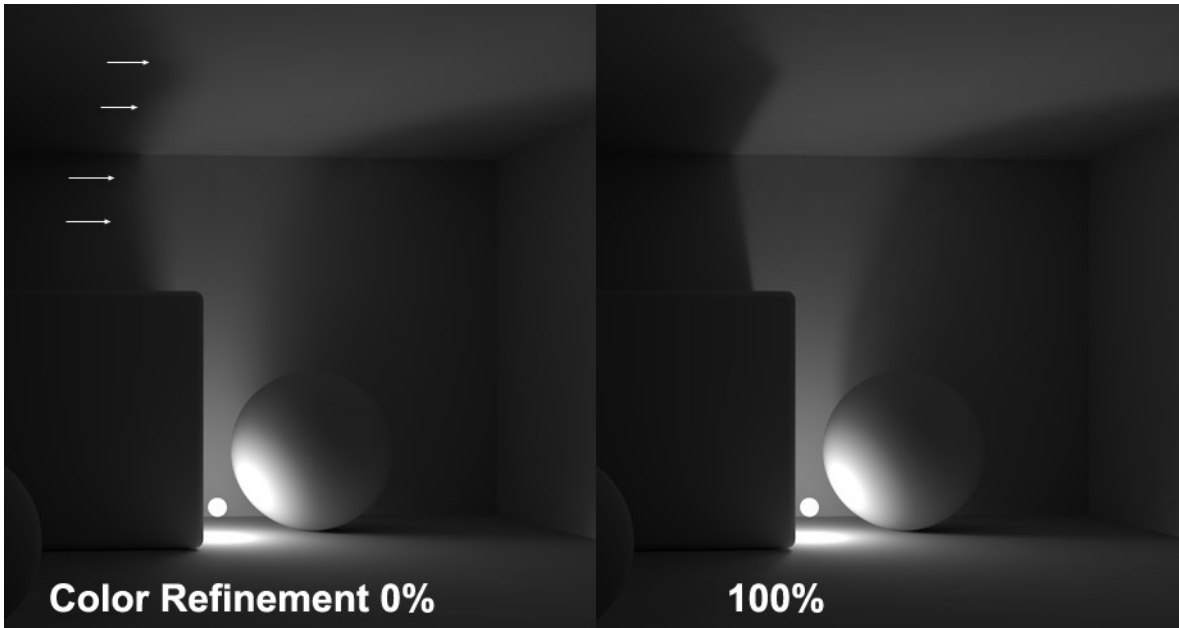
All previously described settings involve the positioning and calculation of shading points, and indirect lighting is ascertained at numerous locations. This point-by-point dispersion of brightness must be converted to a planar dispersion during rendering. A smoothing algorithm does this in the following way: For each pixel to be rendered for a given object, Irradiance Cache is scanned to find entries in their close proximity to interpolate their brightness and color values.

The higher the Smoothing value, the more shading points will be used for interpolation to render a given pixel. This setting represents a threshold value that determines whether or not a given cache entry in its proximity will be used. Smaller values generally produce a sharper (and often more spotty) result, whereas larger

values will interpolate across larger areas, which results in a more uniform illumination but also means that small details will be lost.

Color Refinement

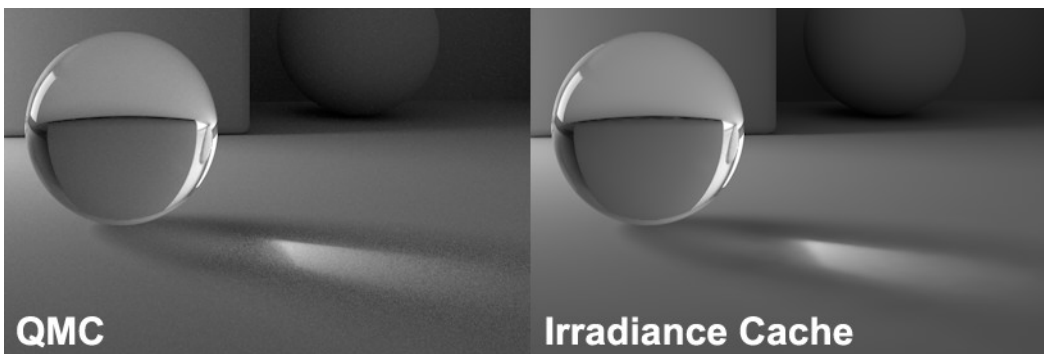
Increasing this setting's value will improve the render quality in regions in which GI lighting changes abruptly (e.g., GI shadows on bright Polygon Lights). What happens is that additional shading points will be generated (with a corresponding increase in render time).



A higher Color Refinement value produces a more defined cast shadow

Important: You should increase the Record Density enough so the dispersion of light is homogeneous, otherwise spotty regions will result in regions in which GI lighting changes abruptly.

GI Caustics also benefit from higher Color Refinement values:



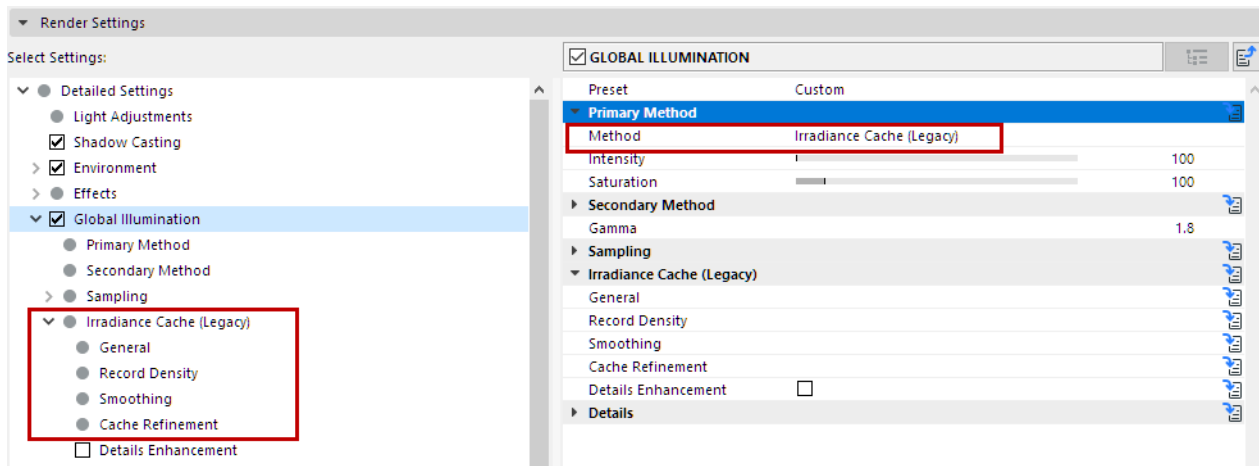
Caustics rendering (right), which took a fraction of the time to render compared to QMC

Screen Scale

With Legacy version of Irradiance Cache, the shading point dispersion was done independently of the size of the rendered image, i.e., the same pre-pass was used for an image 80 x 80 pixels as for an image with a resolution of 1024 x 768. Enabling the Screen Scale option will adjust the shading point density to the image's resolution. This means that correspondingly more cache entries will be created for an image with a very large resolution than for an image with a very small resolution. Hence, smaller images will render faster and larger images will render more slowly, which also means that more details will be visible.

Irradiance Cache (Legacy) (Cineware Global Illumination)

These settings are available, as part of the optional Global Illumination effect, in the Detailed view of PhotoRendering Settings for the Cineware engine.



These parameters are available if GI Method is set to Irradiance Cache (Legacy).

For more information, see [Options for Primary Method](#).

Generally speaking, the predefined values in these additional settings do not have to be modified.

General

Interpolation Method

Different algorithms can be used to calculate the transition between light and dark regions and between color values between the Shading points.

- **Least Squares:** This function essentially calculates a curve based on a limited number of points.
- **Delaunay:** In contrast to the Least Squares interpolation method, only neighboring records and not those located further away will be included in the interpolation for rendering. However, in order to achieve homogeneous dispersion of light, the Record Density, a.k.a. shading point density, should be very high. The Record Density parameter offers several Delaunay preset options that can be used for this purpose (see *Record Density, below*).

The Delaunay option is in fact intended for use in special circumstances and is not really suited for use in “normal” renderings.

- **None:** If this option is selected, no smoothing will take place.
- **Weighted Average:** This method of interpolation works similarly to the Least Squares method, except that it only interpolates between values (whereas the Least Squares method can also extrapolate, i.e. for brighter or darker results than those represented by the defined values). This method can prevent artifacting when using low-quality GI settings. Furthermore, this method will render faster than the Least Squares method. Disadvantage: The dispersion of light is less homogeneous than when using the Least Squares method.

Distance Map

If, in relation to the total number of image pixels, only very few Shading points are interpolated, primarily soft shadows will be created and many details will be lost or smoothed too much. This can, for example, lead to light seeping through polygons. For example: in the form of sunlight seeping through an outer wall and being

visible somewhere in the room. Enabling the Distance Map option can prevent this from happening - but will also increase render times accordingly.

Check Record Visibility

Similar to the Distance Map functionality, this option is also designed to block light from seeping through polygons. Enabling this option will include objects that do not lie directly in the camera's line of view.

Record Density

This value defines the density and dispersion of the Shading points in the prepass phase. The information that these Shading points gather about the Stochastic Samples will then be interpolated between the Shading points to create soft transitions. The more Shading points that exist, the more exact the transition between light and dark regions in the scene will be - and the longer the Prepass and total render time will be.

Method: This parameter contains a set of predefined parameters that are by default optimized for most uses. Only under special circumstances should you change the option to **Custom** and modify these parameters yourself.

- **Preview:** As the name suggests, a preview will be rendered quickly - with correspondingly "poor" values that make a quick rendering possible. These "poor" values will result in many GI details being lost or in faulty rendering.
- **Low/Medium/High/High (Details):** These four options are optimized for use with the Least Squares interpolation method and result in differing levels of quality.
- **Low (Delaunay)/Medium (Delaunay)/High (Delaunay):** These three options are designed for use with the Delauney interpolation method. This interpolation method requires a very dense dispersion of shading points.

Min Rate/Max Rate

When the program creates an Irradiance Cache, it initially starts with a lower image resolution (Min. Rate) and progresses to the final IR resolution (Max. Rate). A value of 0 results in full resolution (pixel size 1*1), a value of -1 results in a pixel size of 2*2, a value of -2 = results in a pixel size of 4*4, and so on. The Min Rate value should logically be smaller than the Max Rate value. Positive values can also be applied, which will allow cache Records in the sub-pixel range (can be useful for SubPolygon Displacement if details are lost).

These resolutions apply only to IR calculations. Since the Irradiance Cache can be scaled relatively easily (i.e. lower IR resolution at larger image resolution), good results can often be achieved even when using smaller resolutions. Hence, these two parameters offer great potential for shortening render times, especially for brightly illuminated, low-detail scenes.

Radius

This parameter defines the maximum distance between shading points. The lower the value the more densely the points will lie together. This parameter primarily affects non-critical regions of the scene such as flat, clear surfaces. The effect of this parameter is also dependent on the Density Control value.

Minimum Radius

This parameter defines the minimum distance between shading points. It primarily affects the critical regions of the scene such as corners, edges, etc. The lower the value the more densely the shading points will lie at these regions. This parameter works in proportion to the Radius parameter above, i.e. if this value is halved, the Minimum Radius value will be halved as well.

The Minimum Radius parameter primarily affects areas in which details are important (e.g. subtle shadows). However, too many shading points in these areas can cause problems.

To make subtle details visible, use the Details Enhancement parameters instead.

[See *Details Enhancement*, below.](#)

Density Control

In contrast to the previous two parameters that primarily affect critical or non-critical regions, this parameter affects the shading points globally throughout the scene. The higher the value the greater the density.

Use Proximity Correction

This parameter is responsible for having neighboring shading points “assist” each other in critical regions and pass on information regarding the proximity of geometry. New shading points are created and calculated during this process.

This behavior can be disabled here, which will save some render time but also result in poorer render quality (especially at corners and edges). This option should be disabled if you are using a Max. Rate value of less than 0. Otherwise unnecessary calculations will result.

Smoothing

Increasing the Smoothing effect results in more lost details, but also in a more homogeneous dispersion of light.

- **Method:** Use the pop-up to define the level of smoothing. Select **Custom** to manually define the Records and Scale values below.

The Records and Scale values limit the number of shading points to be used.

- **Records:** The Records parameter defines, for each pixel to be rendered, the maximum number of surrounding Records in the Irradiance Cache that should be included in the interpolation of color and brightness for that pixel. If, however, the Scale value is so low that not enough Records can be included in the interpolation, it may result in fewer Records being included.

The lower the parameter value, the fewer Records that will be included in the final interpolation and the less homogeneous the rendering will turn out. Higher values will result in more smoothing but also take correspondingly longer to render.

- **Scale:** This parameter serves to spatially limit the Records included in the interpolation process. The larger the value, the more Records will be included and the softer the interpolation will be - and the longer it will take to render. Alternatively the Records parameter can be used for this purpose.

Cache Refinement

The previously described parameters all dealt with the placement of shading points based on a scene's geometric properties. But what if a coarse, dark GI shadow disturbs the entire image?

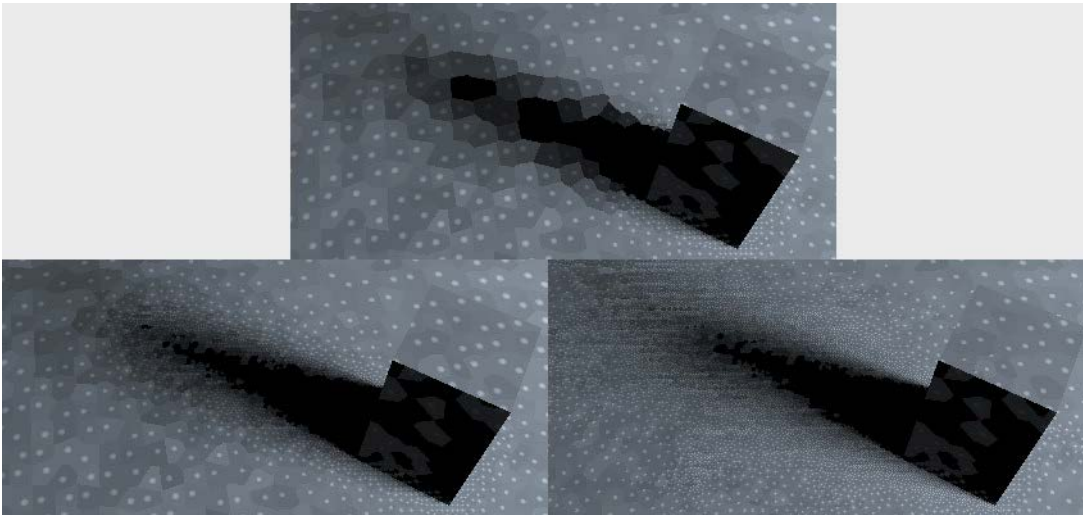
This is where the Cache Refinement parameter comes in: It compares the Records in the Irradiance Cache and generates additional Records (i.e. shading points) in areas with high contrast (brightness and color) in order to refine and render these areas more precisely.

Tip: Refining the cache can greatly increase the number of Records in the Irradiance Cache, which will not necessarily improve the render quality. Especially when applied in conjunction with incorrect smoothing, this can lead to very grainy results. Hence, only refine the cache if absolutely necessary.

Higher Cache Refinement settings will lead to longer render times but not necessarily to better render quality.

Passes [0..4]: Use this setting to set the frequency with which the cache should be defined. Each new pass includes the results of the previous pass and refines it further by creating additional shading points in critical areas.

Color Threshold: This value defines the degree to which (neighboring) cache Records can deviate from each other with regard to their color (intensity) before additional shading points (“Samples”) are added. The lower the value, the lower the deviation threshold will be and the more samples will be added.



Top: Prior to cache refinement. Bottom: After cache refinement.

Left: smaller Color Threshold, right: larger Color Threshold value.

Cutoff: This value includes differences in intensity. The lower the value, the greater the differences between Records have to be for the color correction to apply. A value of 0 will turn cache refinement off.

Strength: This parameter is used to adjust the cache refinement’s overall sample density. A value of 0 will turn cache refinement off, whereas larger values will increase the number of shading points (“Samples”) correspondingly while taking into account Color Threshold and Cutoff values.

Details Enhancement

The Irradiance Cache method renders noise-free and much faster than other sampling methods, but entails compromises on details that lie in shadowed areas. This can result in the omission of finer details, such as tile grouting, during the smoothing process. Enabling Details Enhancement will counteract this by sending additional samples specifically to these areas. The effect is similar to Ambient Occlusion.

In a nutshell: Subtle geometry details are accentuated.

In order to save render time and avoid over-emphasizing finer details, either Ambient Occlusion or the Details Enhancement option should be used, but not both. Using both can make fine details visually too dominant. Since enabling the Details Enhancement option will send additional samples into the scene, you may also want to consider reducing the Record Density value.

The Details Enhancement parameters include options to counteract a characteristic of the Irradiance Cache - the “blurring” (smoothing) of details such as subtle shadows. The enhancement of details uses the QMC Sampling method in critical areas (for each relevant pixel) like corners, edges, cavities, etc. The Details Enhancement functionality can be seen as a special type of Ambient Occlusion whereby indirect light is included.

Note that the internal Irradiance Cache will be calculated differently (the algorithm will be aware of the subsequent detail enhancements and will calculate the critical areas differently) if this option is disabled. However you do it, the Details Enhancement will be calculated separately for each rendered image, i.e. re-using a saved cache will reduce render times.

If you apply Details Enhancement, you can lower the other Irradiance Cache settings (especially those pertaining to Record Density).

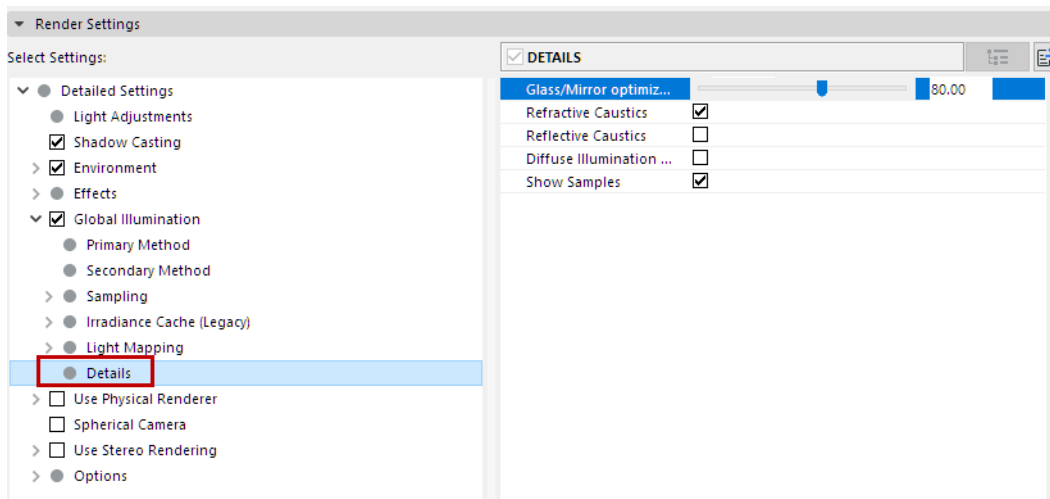
- **Details Enhancement:** Check this box to enable the Details Enhancement parameter.
- **Adaptive:** Enabling this option can be an advantage in some circumstances, e.g. several small areas are more grainy than others. Additional samples will then be rendered in these areas.
- **Estimate Secondary:** This mode renders faster and offers good results for most applications. However, certain areas will be rendered too dark or too colorful as opposed to a pure QMC rendering (GI Mode QMC), which in most cases doesn't matter at all. If desired, this behavior can also be disabled (note that grainy renderings can result, in which case the Quality Ratio value should be raised).
- **Radius:** Use this setting to define the radius within which neighboring objects, corners and edges should be included. Lower values result in only elements in the immediate proximity being "seen", whereas larger values will "see" elements correspondingly far away. Larger values will also result in higher precision and therefore longer render times.
- **Quality Ratio:** As mentioned, QMC Sampling will take place in critical areas. The Quality Ratio defines how many samples should be used per pixel, which in turn simply defines the graininess of the Details Enhancement. Larger values result in less grainy, softer results but require correspondingly longer render times.

Quality Ratio is an autonomous value that works independently of the rest of the IR settings (a value of 100% is equivalent to 64 samples).

- **Mode:** The options in this drop-down menu are for test purposes only so the effects of the detail enhancement, which are often very subtle, can be made more visible. You can select from the following options:
 - **Combine (Normal):** Renders accurate results.
 - **Details Only (Preview):** Renders detail enhancement without GI. Makes the detail enhancement the most visible of all the options.
 - **Global Only (Preview):** Renders indirect illumination only.

Details (Cineware Global Illumination)

These settings are available, as part of the optional Global Illumination effect, in the Detailed view of PhotoRendering Settings for the Cineware engine.



Glass/Mirror Optimization

The effect of GI on glass is barely visible and can therefore unnecessarily take up render capacity. To avoid this, use the Glass/Mirror Optimization parameter. As a result, surfaces with a Brightness value (in the Transparency or Reflectance channels) greater than the value defined in this parameter will not be illuminated by GI. This may result in somewhat darker (but faster) renderings.

This parameter has no effect on Caustics!

For example, a lot of render time can be saved when visualizing a structure containing numerous large glass surfaces.

Tip: In order for this parameter to work, the surface channel “Color” must be disabled and no texture should be present in the Transparency channel.

Refractive Caustics/Reflective Caustics

Individually enables or disables Refractive and Reflective Caustics.

What are Caustics?

In the real world, caustics occur everywhere where light is clustered or reflected by refraction (see image above) or mirroring (e.g., on a metal ring).

There are two types of caustics in Cineware:

- Photon caustics in earlier versions of Cineware: Faster than GI Caustics but must be generated using individual light sources.
- Irradiance Cache GI Caustics: In conjunction with Primary Method IR. Not very precise but affect all luminous surfaces within a scene.

To generate caustics in previous versions of Cineware, it was necessary to create light source objects from which photons were beamed throughout the scene. In contrast, GI Caustics uses any available light source (exception: Only the reflected light of regular light sources can be included; use Polygon Lights instead).

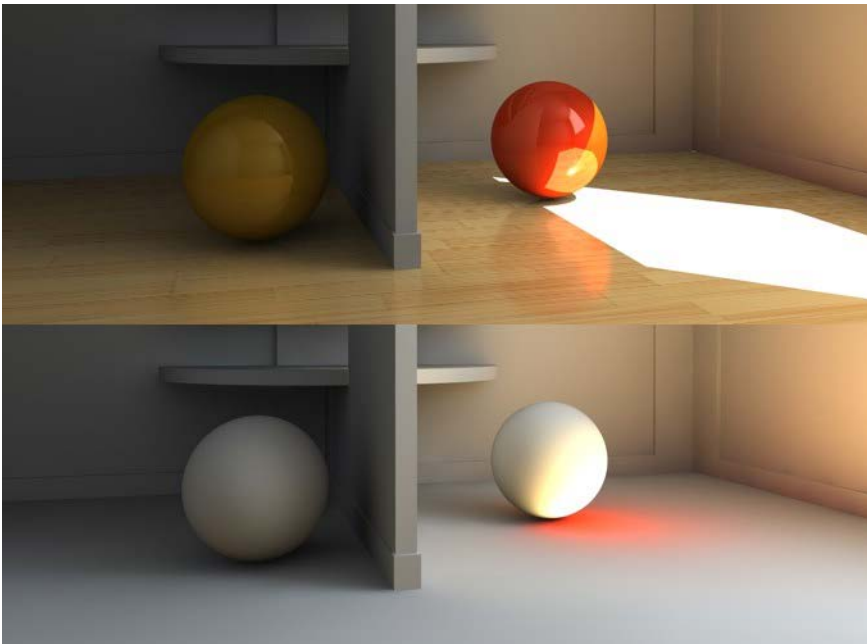
GI Caustics require bright light (e.g. HDRI images with very bright regions or Area Lights) to come to fruition. Otherwise you can increase the strength of the Generate GI parameter in the Illumination channel of the Glass surface.

Tip regarding reflected GI Caustics: Reflected GI Caustics always require an active Reflectance surface channel. Otherwise, you can achieve the best quality caustics if GI Method is set to QMC.

Diffuse Illumination Only

Activating this option deactivates textures, reflections, light from surfaces illuminated directly by light sources, etc. All bothersome elements will be deactivated and only the brightness dispersion calculated by GI will be rendered. This also lets you judge the quality of the GI.

This option should be deactivated for the final rendering.



At top: normal rendering. Bottom: Rendering with active Diffuse Illumination Only option (the sun is a real light source which means that its effects will be rendered following the first reflection).

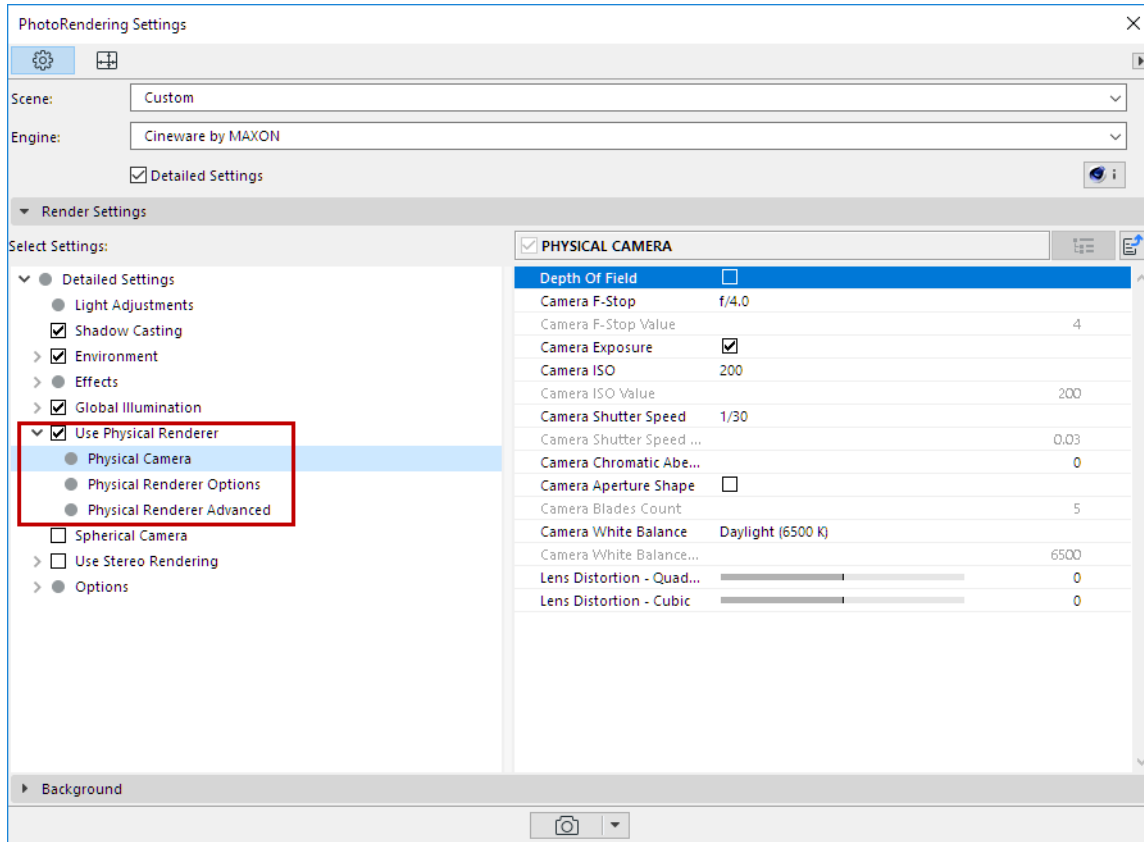
Show Samples

Shading points are displayed during the prepass phase for Irradiance Cache calculations. Sometimes these can be irritating when trying to assess the prepasses. To avoid this, disable the shading points' display. The render time will be reduced slightly and the final render result will not be affected.

Physical Renderer (Cineware)

These settings are available if you check the **Use Physical Renderer** box in the Detailed view of PhotoRendering Settings for the Cineware Engine.

For information on choosing a Rendering mode, see [Physical vs. Standard Renderer](#).



Physical Camera

Depth Of Field

Enables or disables the depth of field for the Physical Renderer.

Strictly speaking, a real camera can only focus clearly on a single distance at a time. Everything in front of and behind this plane appears increasingly blurry. The area between the nearest and farthest objects that appears focused is known as depth of field.

For more information on how [Depth of Field](#) works, see also [Depth of Field \(Cineware Effect\)](#)

Camera F-Stop

The F-Stop value is used to adjust the focal aperture. The larger the focal aperture (i.e. the smaller the F-Stop value), the smaller the depth of field will be and vice-versa. F-Stop is the most important setting for adjusting the depth of field. Small values result in correspondingly small depth of field. F-Stop will also define the amount of light that can enter the camera.

Camera Exposure

If **Exposure** is enabled, the camera will simulate the exposure process, i.e. the rendered image will vary in its brightness in correspondence with the **ISO**, **F-Stop** and **Shutter Speed** settings.

In a real camera, the amount of light to which the film is exposed determines how bright the film will be. The more light to which the film is exposed (the larger the aperture opening is larger, or the slower the shutter speed), the brighter the film will turn out and vice-versa.

Camera ISO

ISO is used to define the film's sensitivity to light. Higher ISO values make the film more sensitive and thus the images brighter.

Camera Shutter Speed

The shutter speed defines how long the shutter will remain open when a picture is taken. This setting defines the amount of light that reaches the film (only relevant if Exposure is enabled). Slower shutter speeds produce brighter images.

If you don't use Exposure, this setting has no relevance for the rendered image.

Camera Chromatic Aberration

Chromatic aberration occurs in lenses when differently colored parts of light (depending on the wavelength) are refracted with varying intensities. Chromatic aberration only occurs in blurred image regions; without blur, no chromatic aberration will take place.

The value that can be defined here defines how strong the "color offset" will be rendered. It can also be described as the degree of saturation in the blue-red regions.

A value of 0 turns chromatic aberration off.

Remember that enabling this effect also increases render times accordingly.

Camera Aperture Shape

The following settings are used to create the bokeh regions. These are regions that lie outside of the camera's area of focus and are therefore displayed as blurred. Small, bright regions are displayed on the shape of the aperture. Bokeh's can have a strong effect on the image's "impact," because blurred regions can have very different shapes depending on the shape of the aperture.

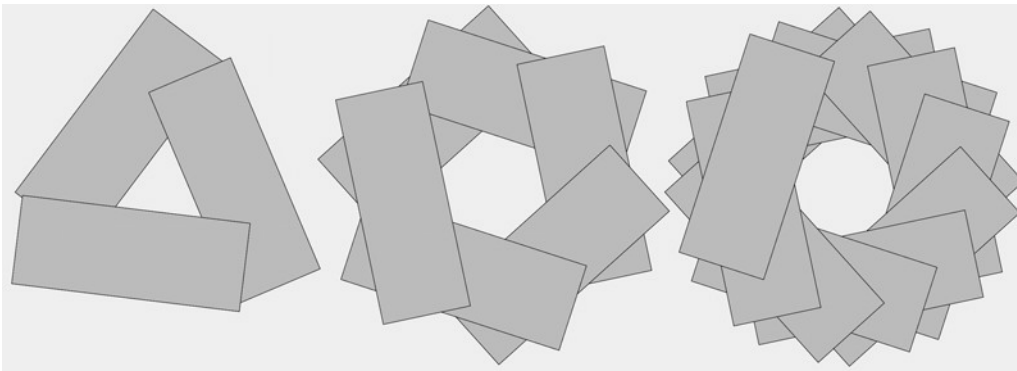
The Aperture Shape option is either circular (when disabled) or can be given a different shape by changing the **Camera Blades Count** setting ([see below](#)).

In general, the best looking bokeh (i.e. the softest and most flowing) can be achieved with a circular aperture (Aperture Shape option **disabled**).

Note that rendering high-contrast bokeh effects is a complex procedure and takes correspondingly long to render.

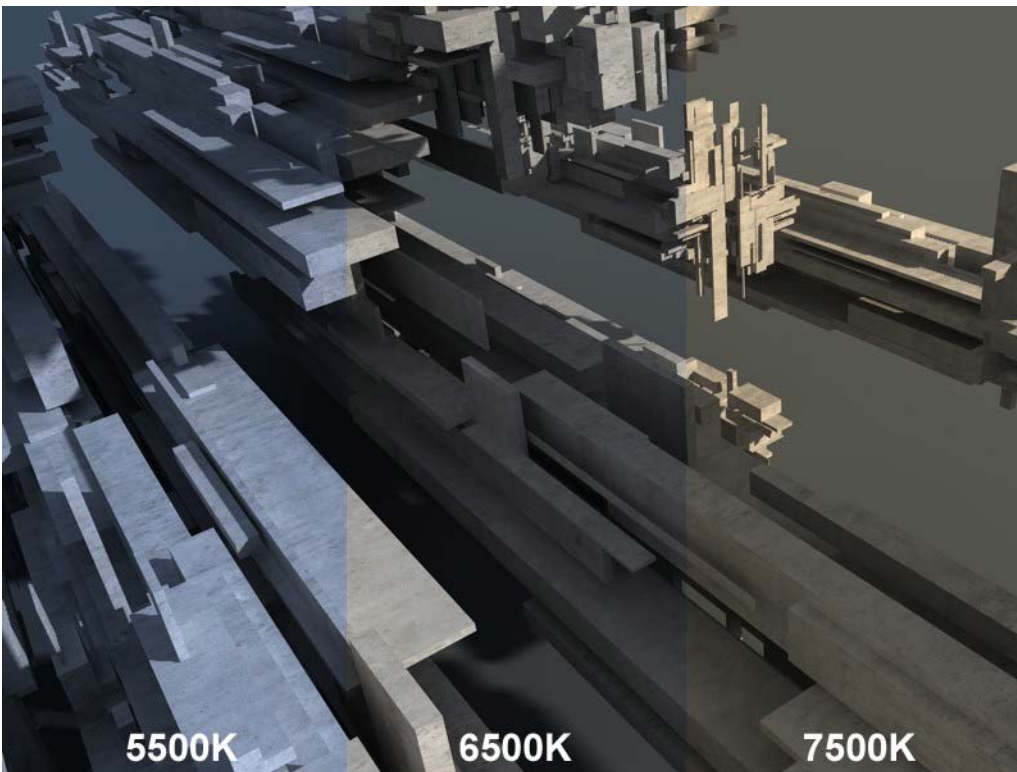
Camera Blades Count

The bokeh is defined primarily via an aperture shape. The aperture shape in turn is dependent upon the defined number of blades. The more blades are defined, the rounder your aperture shape will be. The least number that can be defined is 3.



Number of Blades from left to right: 3, 6, 12

Camera White balance (K)



An outdoor scene (with sky), with 5500K, 6500K and 7500K White Balance, respectively

White balance is used to prevent light sources (e.g., sky, sun, candles, etc.) from colorizing white surfaces. Set the **White Balance** value to that of the light source's color (*Tungsten* refers to the filament coil in a light bulb). If the presets do not quite match your needs, you can select the *Custom* option and define a custom temperature via the **Custom Temperature (K)** value. Color hues can be defined without having to change the color of the light itself.

As you can see in the image above, 5500K produces a slight blue hue and 7500K a slight yellow hue. The value 6500K (which is representative of daylight conditions), produces a grayed, cement-like hue (cement textures were applied to the objects in the image).

Color temperature is measured in Kelvin.

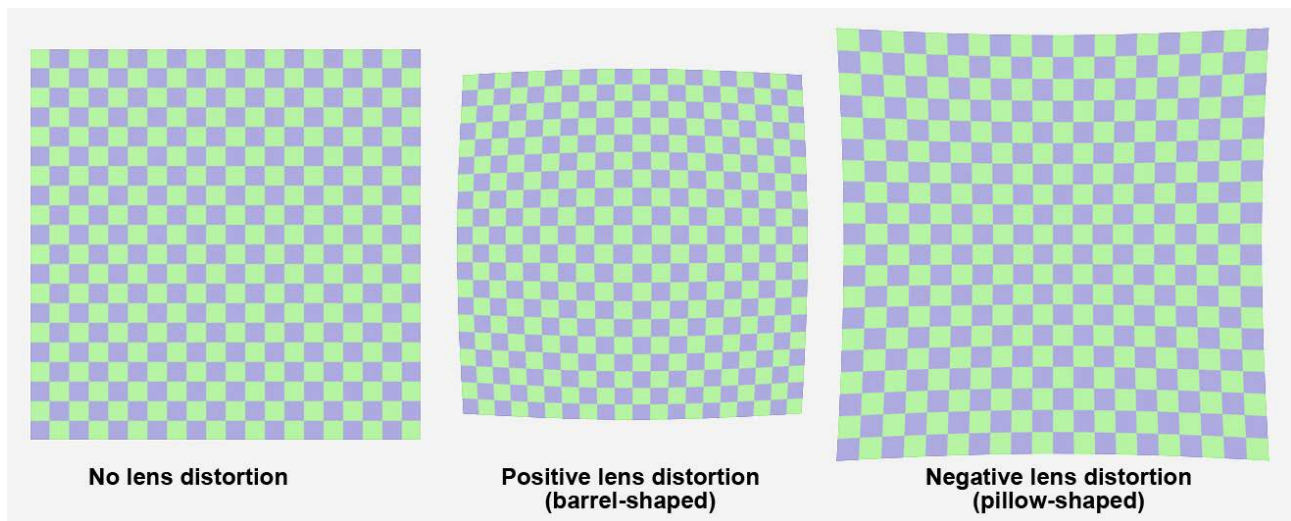
Of course white balance can be used to colorize renderings. The neutral value is *Daylight (6500K)*. Generally speaking, lower values result in hues of blue and higher values in hues of yellow (directly opposite to the depiction above; the white balance, after all, is a corrective value).

Lens Distortion - Quadratic/Cubic

Lens distortion (different magnifications with regard to distance from the lens axis) causes straight lines at the image's edge to appear curved.

In photography, wide-angle lenses (short Focal Lengths) tend to produce barrel-shaped distortions, while tele-focus lenses (long Focal Lengths) produce pillow-shaped distortions. These effects can be simulated using the Lens Distortion settings.

Note: Lens Distortion is not effective in conjunction with the Spherical camera.



The same plane is rendered using various lens distortions

Positive values result in a barrel-shaped and negative in a pillow-shaped distortion.

Use real-world tests and your camera's tracking program to determine if you should use Quadratic or Cubic, or a combination of both (these differ only slightly visually from the center of view outward).

[Watch the Video](#)

Physical Renderer Options

These settings are available if you check the **Use Physical Renderer** box in the Detailed view of PhotoRendering Settings for the Cineware Engine.

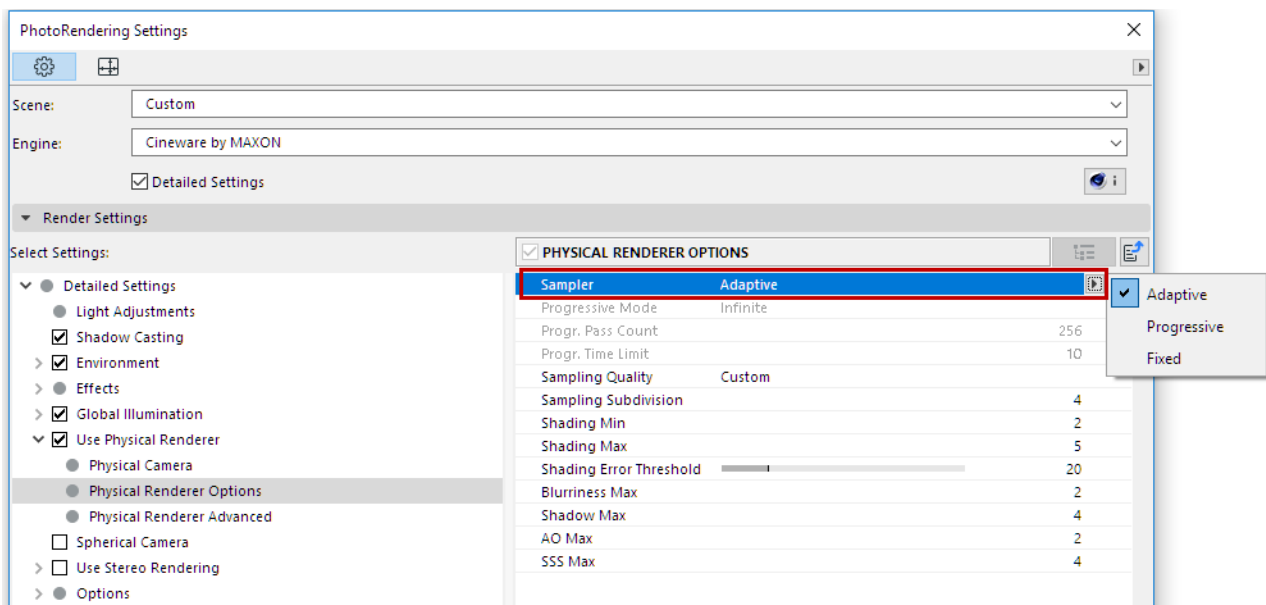
Sampler

The Sampler does just that - it samples the Project's cache from the angle of view of the camera in a pixel raster. The Sampler has to determine which regions have to be sampled again with regard to antialiasing or blur effects (depth of field, motion blur, blurriness effects) in order to achieve the desired render quality.

The samples gathered (i.e. colors), whose number far exceeds the number of pixels, will in the end be turned into pixels colors via an antialiasing filter.

The following generally applies: The more samples taken at critical regions, the better the final result will be (with correspondingly longer render times).

Sampler Settings:

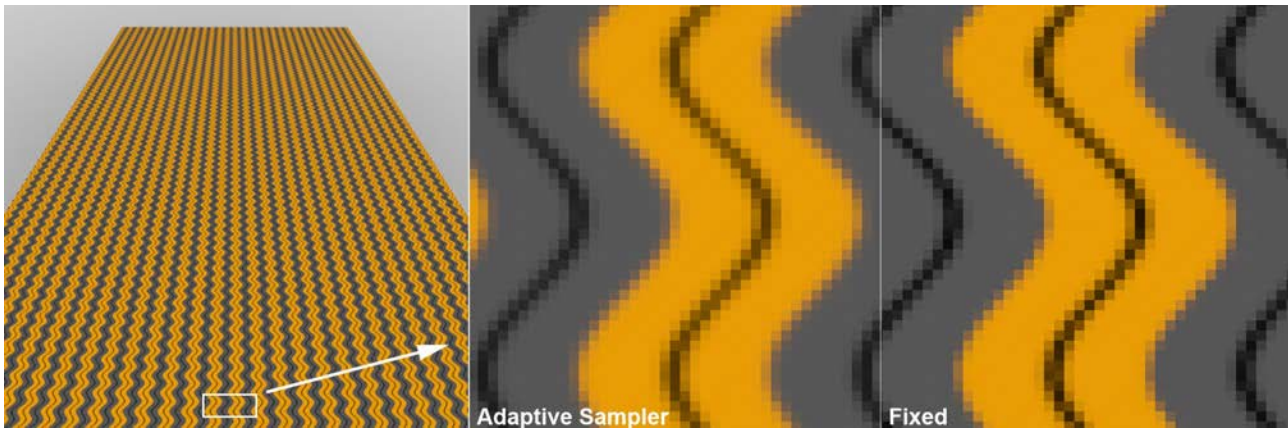


- **Adaptive:** Depending on the settings defined, an increased sampling will take place at critical or important regions. This mode is best for use in most cases because it offers a good compromise between quality and render speed (comparable with the default renderer's Best Antialiasing setting).

The Adaptive sampler is a much more intelligent sampler compared to the Fixed sampler, and often renders faster than the Fixed sampler. On the other hand, if you choose a Sampling Quality of Custom, it's more difficult to control because it calculates the following 4 values and distributes the samples accordingly:

- Sampling Subdivisions
- Shading Min
- Shading Max
- Shading Area Threshold

Special attention is paid to object edges, to which additional samples are applied to ensure that they are always antialiased optimally. The Adaptive sampler renders slightly less sharp than the Fixed mode.



- **Fixed:** A definable and unchangeable per-pixel sample number (which will, however, already be increased in blurriness effect regions) will be calculated (comparable to the default renderer's Best and Min/Max Antialiasing settings).

This sampler subdivides each pixel in a defined number of sub-pixels, according to the value entered for the Sampling Subdivisions value. For example, to make the blurred region of a motion blur effect visually appealing, you should use a relatively high value (e.g., 4 or 5). This means that 16 and 25 samples, respectively, will be calculated per pixel even in superfluous regions.

Increasing the Sampling Subdivisions level by a single value normally doubles the required render time. Here you can estimate render times quite precisely, which on the other hand is quite difficult using the Adaptive sampler.

Render time is valuable. This is why there is an Adaptive mode that recognizes at which locations more sampling must be done.

- **Progressive:** This sampler can render infinitely (samples will continuously be calculated), during which time the result will continuously improve. The initially grainy image will become increasingly less grainy, i.e. more homogeneous. The longer it renders the better the antialiasing and blur effects that will be calculated.

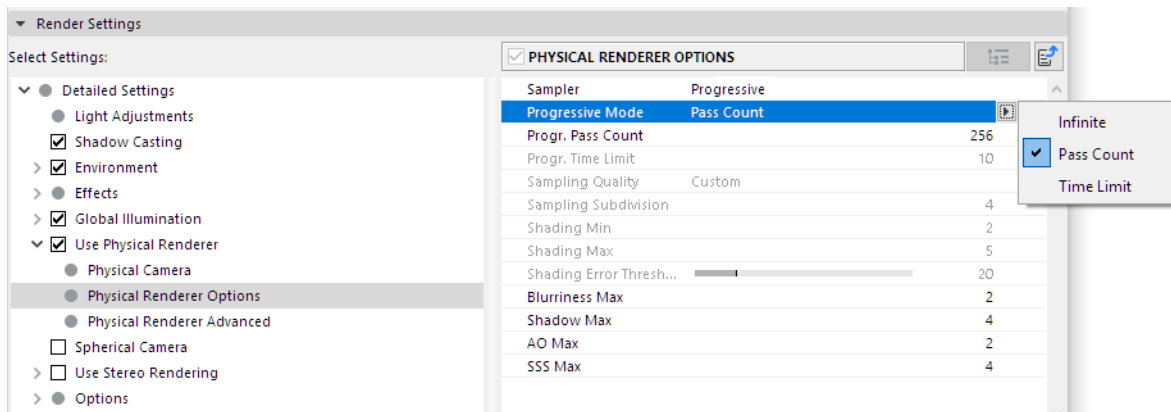


The longer the render, the better the result

This mode is primarily suited for fast preview renderings because you can very quickly determine the quality of the image as a whole. No other method offers you such a fast rendition of the entire image (with reduced quality at the start). To this end it is recommended that the Sampling Subdivision value (see below) be set to 0.

Progressive Mode/Pass Count/Time Limit

Use these settings to define whether the render should never end (**Infinite**) or if it should end after a specified number of passes (**Pass Count**) or render time (**Time Limit**, in minutes). A pass in this context is defined as a completely new rendered result over the entire surface of the given image.



Note that when using Time Limit you should allow for enough time for at least 1 pass, otherwise it can result in faulty renderings if the pass is stopped halfway through the images.

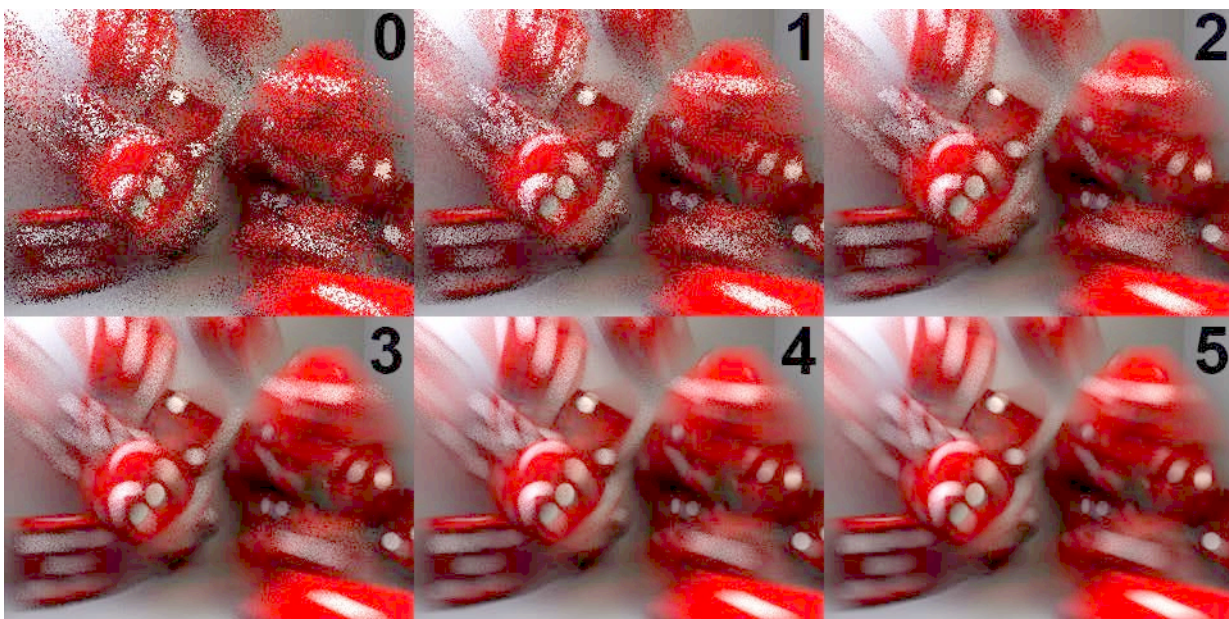
Sampling Quality

This selection menu makes your work a little easier by automatically defining parameter combinations, depending on which setting is chosen. **Low, Medium, High** refer to the quality of the rendered image - the higher the setting the less grain the image will have but the longer it will take to render. **Custom** will automatically be selected when one of the parameter values is modified manually.

Sampling Subdivisions

This setting functions somewhat differently depending on if you select Fixed or Adaptive.

- For a **fixed** number: The absolute number of samples that are calculated per pixel (depending on the subdivision values, additional samples will be calculated accordingly).
- For an **adaptive** number: A number of samples that are taken from the scene in order to gather information which will be calculated in a subsequent step (by the three shading parameters), i.e. if the values entered here are too low, greater values for the other parameters will not lead to good results.



The render results using the values shown for Sampling Subdivisions

Shading Min/Shading Max/Shading Error Threshold

The Sampling Subdivisions setting is applied and the number of shading samples required per pixel will be determined based on the shading parameters.

“Shading” is defined as the calculation of a color value based on shadows, textures, refractions, reflections, etc. This can be very complex and time-consuming process, depending on the scene.

The **Shading (Min)** value defines the minimum number of shading samples that will always be calculated (to the power of two, i.e. 1, 2, 4, 8, 16, etc.).

The **Shading (Max)** value defines the maximum number of shading samples, in correlation with the Shading Error Threshold value, that should be calculated.

The **Shading Error Threshold** value controls the number of samples that will in fact be calculated. The lower the value, the more the number of samples will lean towards Shading Subdivisions (Max) (however, only in critical regions!).

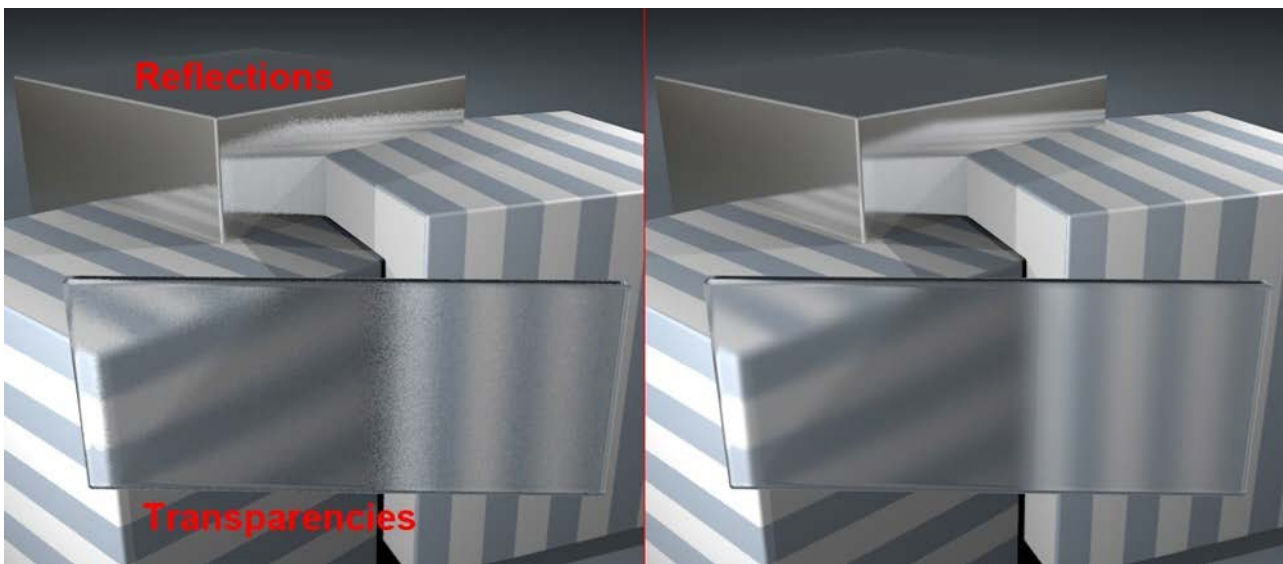
Subdivisions: Blurriness Max, Shadow Max, AO Max

The following three subdivision settings are the quality settings for certain effects (e.g. Ambient Occlusion) in the Transparency or Reflectance channels of the Surface material. The strength is defined at the surface level.

[See Ambient Occlusion \(Cineware Surface Channel\).](#)

These values will be taken into consideration when Progressive mode is used ([see Sampler](#)). Higher values will result in a slower, more gradual rendering. For preview renders in this mode it is suggested that each of these values be set to 0.

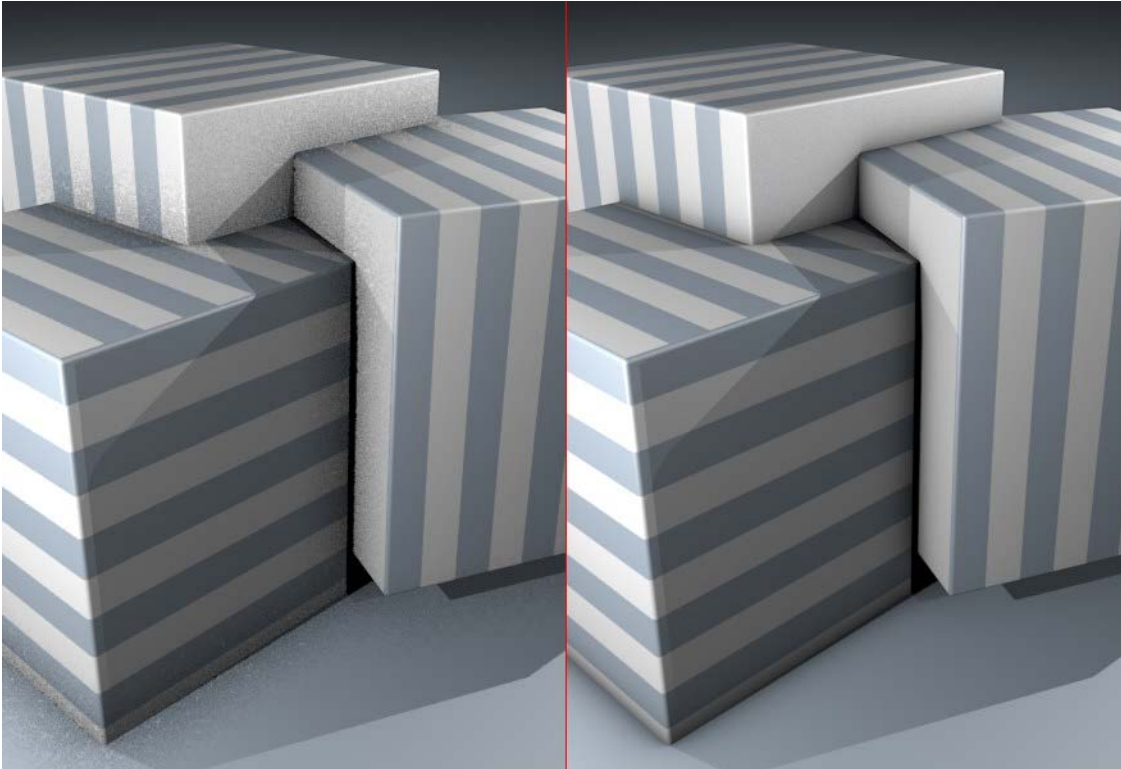
Blurriness Max: If blurriness effects are defined for surfaces (Transparency and/or Reflectance channel), use this value to adjust the quality (=graininess) of the effect. Larger values will result in better quality but will also increase render times accordingly. Render times for a given effect will double each time this value is increased by a factor of 1.



More samples (right) result in better quality in matte reflections and transparencies

Shadow Max: This parameter refers to Shadow Maps (the Physical Renderer simulates Shadow Maps because it cannot calculate them). Larger values result in better quality but also increase render times accordingly.

AO Max: AO is short for Ambient Occlusion. Use this value to adjust the quality of Ambient Occlusion effects. Larger values result in better quality but also increase render times accordingly.



More samples (right) result in improved Ambient Occlusion quality

SSS Max

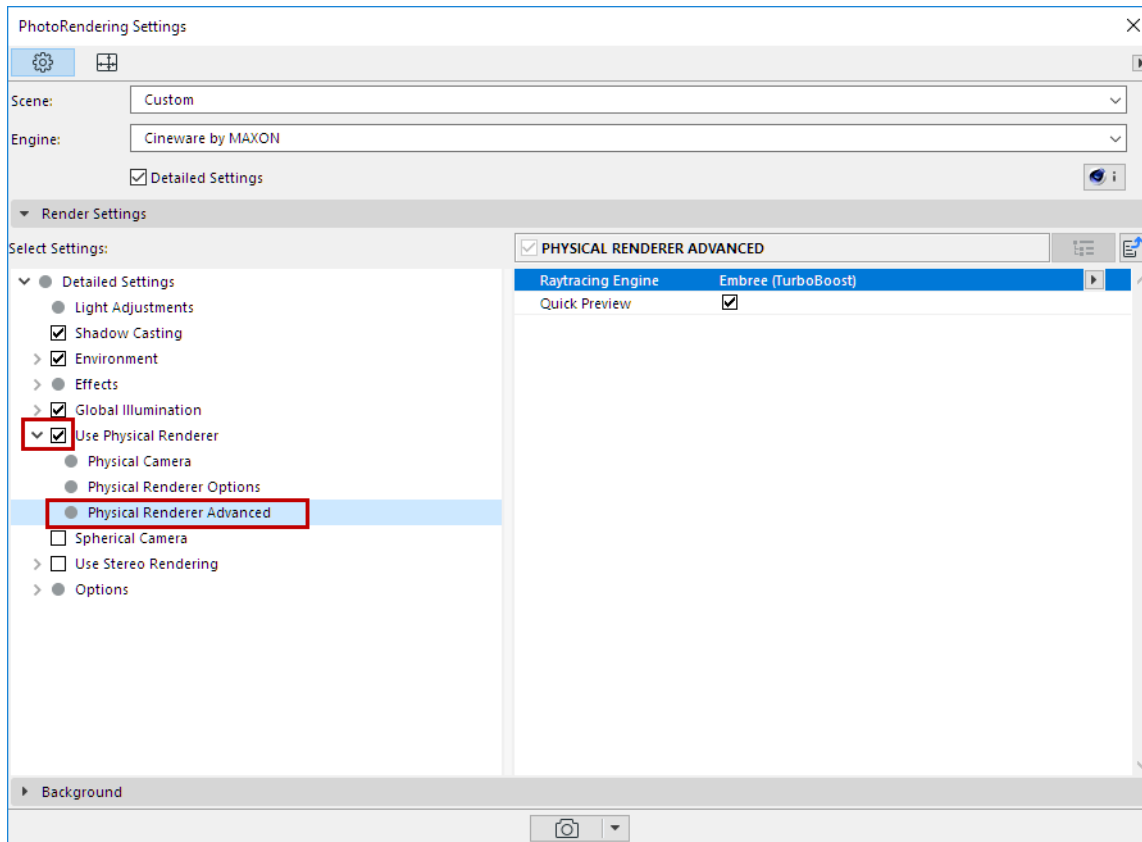
SSS stands for Subsurface Scattering Subdivision. Larger values will produce higher-quality results with correspondingly longer render times and vice versa.



This setting can be defined for surfaces via the Sampling Subdivision option ([see *Sampling Subdivisions*](#)).

Physical Renderer Advanced

These settings are available if you check the **Use Physical Renderer** box in the Detailed view of PhotoRendering Settings for the Cineware Engine.



Raytracing Engine

This setting can almost be described as a turbo booster.

The following options are available:

- Physical (Legacy): The render engine available in Archicad prior to V20. Requires the least amount of memory.
- Embree (Turboboost): The fastest Embree engine. Requires the most amount of memory.
- Embree: Only slightly slower than Embree (Turboboost) but requires less memory.

Note that the render results stay the same for the most part, but render inaccuracies can cause very slight differences between different render engines.

[Watch the Video](#)

About Embree

Embree is a technology, developed by Intel, that supports render software. Embree works with processors of the newest generation (i.e., those that can handle SSE3).

The Embree option renders much faster than Physical, with no quality loss.

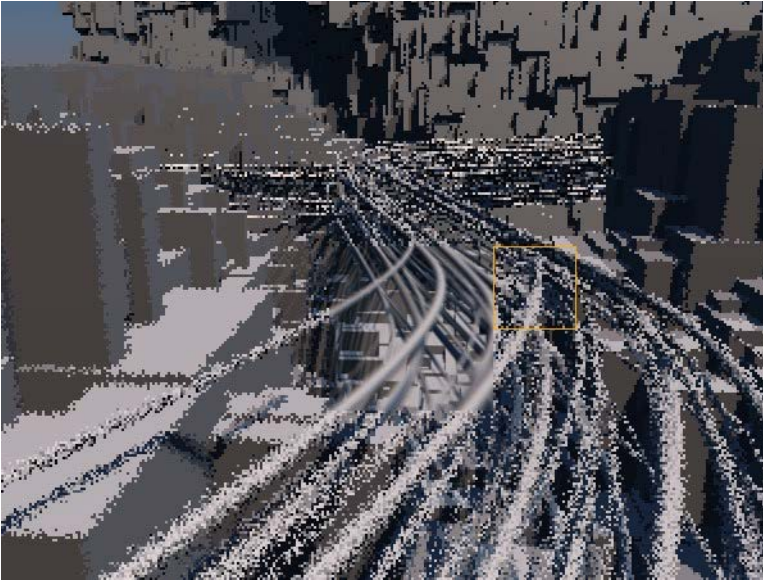
Projects that can really benefit from this new technology are those that contain a high number of polygons/objects, glass with a lot of refraction, area shadows, Ambient Occlusion (without cache). Complex textures or blurriness effects enjoy little or no benefit.

Quick Preview

If Quick Preview is enabled, a very rough preview of the entire image will be created as quickly as possible, which will immediately be replaced bucket-by-bucket as the image is rendered.

This quickly gives you a preview of the image with correct lighting, which lets you decide whether or not you should continue with rendering.

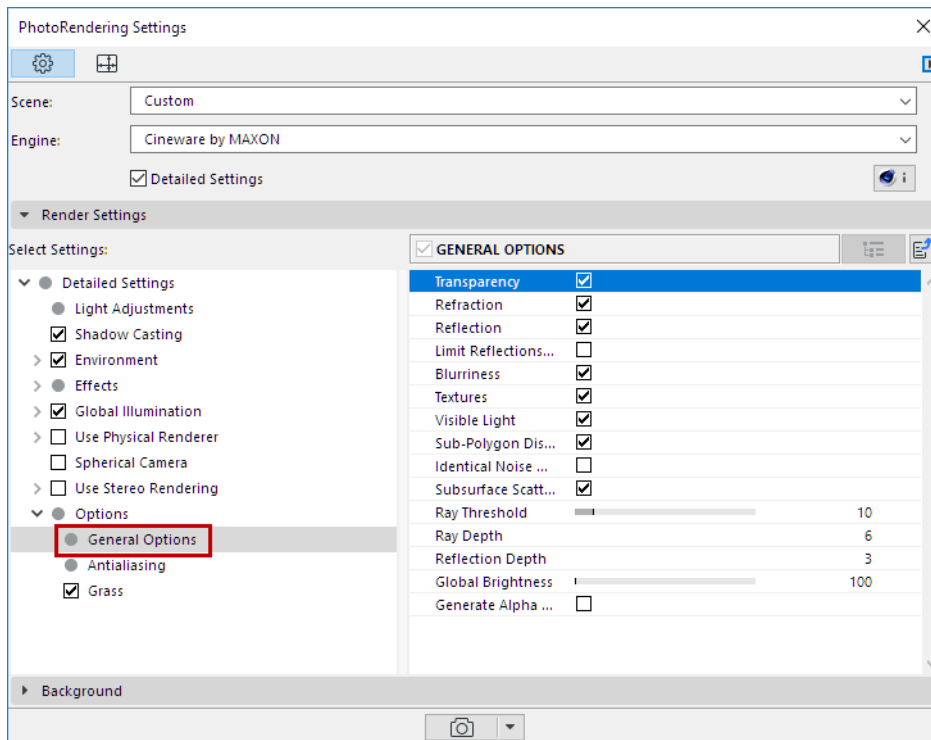
Complex textures will slow the Quick Preview function down accordingly.



A fast preview provides a rough impression of the final render

General Options (Cineware)

These settings are available in the Detailed view of PhotoRendering Settings for the Cineware engine.



This list of options affect the rendering as a whole: if you uncheck an option here, it is disabled for the whole rendering model, regardless of what else you have set elsewhere. It is worth considering which of these can be turned off so as to minimize render time. For example, you can turn off all blurriness with a single click.

Transparency

This option defines whether the Transparency or Alpha surface channels should be included in the calculation.

Refraction

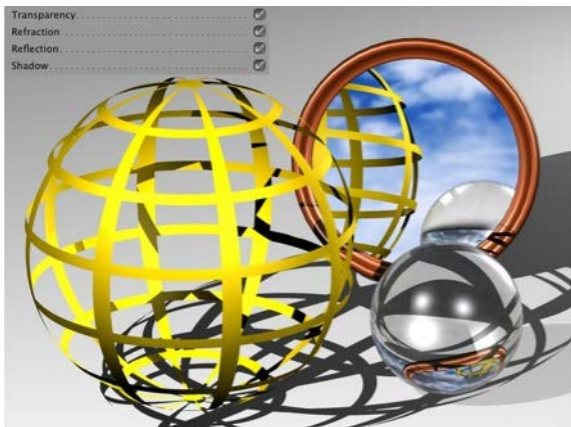
Defines whether or not refraction index values should be included when rendering transparent surfaces. Surfaces such as glass or water only look realistic if rendered using a refraction index, but the render time will increase accordingly. However, no additional render time will be added if a scene does not contain materials that refract light.

Tip: Occasionally, transparent objects that lie behind other transparent objects are rendered black. If this should occur, increase the Ray Depth value (below).

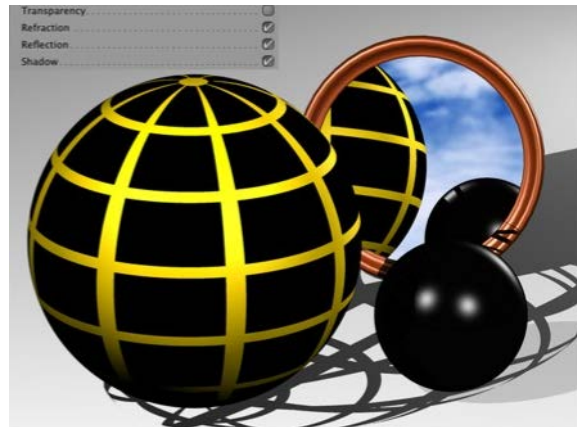
Reflection

Defines whether or not reflections should be calculated. If this option is deactivated, objects will only be rendered in their base color. This option can also be used in conjunction with the **Limit Reflections to Floor/Sky** option (below).

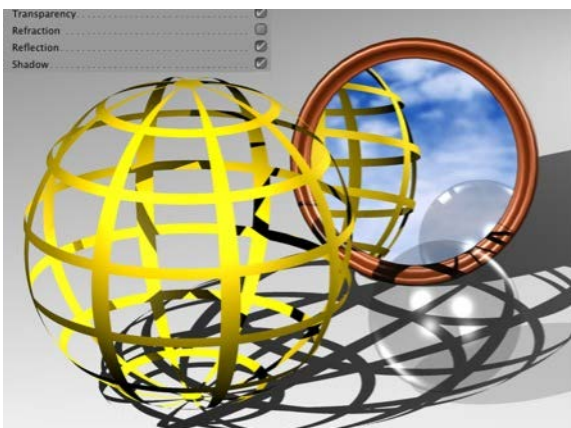
Here are some illustrations of these effects:



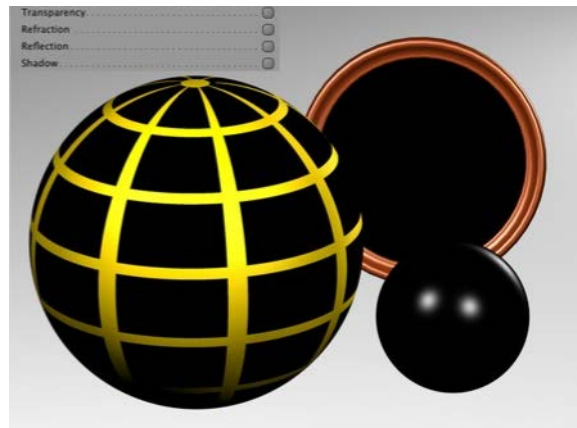
Transparency, Refraction, Reflection: All On



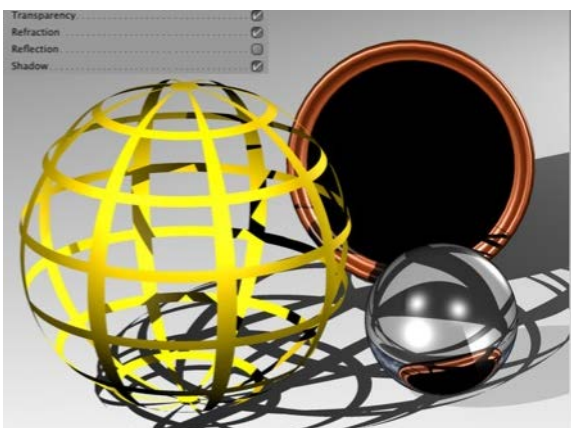
Transparency Off



Refraction Off



Transparency, Refraction, Reflection: All Off



Reflection Off

Limit Reflections to Floor/Sky

Enabling this option will cause the raytracer to only calculate the reflection of the floor and sky onto reflective surfaces (and not onto other objects!). This method is not computationally intensive and is therefore recommended for use in time-critical situations.

Blurriness

Use this option to enable/disable the blur effect for the Reflectance and Transparency surface channels.

Textures

Using this option, you can enable or disable textures when rendering; only bitmaps and shaders remain unaffected. For example, you can disable the textures for a test render or when using the cel renderer.

- If Textures is enabled, bitmap textures will be rendered.
- If Textures is disabled, bitmaps will be replaced by the color black.

Visible Light

You can use this option to deactivate visible lights globally for the entire scene, without having to do it separately for each light source (lamps, lights, Sun).

Sub-Polygon Displacement

You can use this option to deactivate SPD globally for the entire scene, without having to do it separately for each SPD surface. This option can be turned on and off quickly for test renders.

See [Sub-Polygon Displacement in Surfaces](#).

Identical Noise Distribution

Cineware uses numerous effects that (if low values are used) produce grainy results, which are often referred to as “noise”. Some examples are:

- Subsurface Scattering
- Depth of field
- Motion blur
- General Sampling in the Physical Renderer
- GI and Ambient Occlusion.

If Identical Noise Distribution is enabled, the noise distribution will be random for each rendered image, which means that successively rendered images will all have a random noise distribution. This reflects the real-world behavior of film or image sensor sampling in digital cameras, which also contain a certain amount of “noise”.

To render stills as identically to one another as possible, this option should be enabled. However, note that other effects can still produce randomness. This can, for example, be seen in GI renderings in which slightly differing results are produced despite identical noise distribution.

Subsurface Scattering

Subsurface Scattering (SSS) is an effect in which light penetrates slightly transparent surfaces, which then virtually transport, scatter and - in part or entirely - absorb and color the light. “Slightly transparent” means every material through which light can pass, however slightly. Such materials include porcelain, marble, skin, milk or various plastics, just to name a few.

SSS also works in conjunction with GI, i.e light that GI casts onto an SSS object will be included in the calculation. On the other hand, light that is generated by a SSS will only be recognized by other objects when in QMC GI Mode.

Limitation: Keep an eye on the object surface Normals. In some cases it can occur that the SSS doesn't work correctly if the Normals are oriented incorrectly (e.g. if on a sphere they all face inward).

Ray Threshold

This value helps to optimize render time. With complex scenes, particularly those containing many reflective and transparent surfaces, 90% of the processed rays contribute less than 10% to the general picture brightness and color. With a threshold value of, for instance, 15%, the rays stop their movement from the camera into the scene as soon as their brightness falls below this value.

For reflections/transparencies, this generally means that the higher the value, the greater the degree to which very small reflections/transparencies will be taken into consideration - with correspondingly longer render times.

Ray Depth

The Ray Depth determines how many transparent objects (or areas made invisible using the alpha channel) can be penetrated by the renderer. The lower you set the Ray Depth, the fewer the number of objects that can be seen through. Those areas that cannot be penetrated are rendered black.

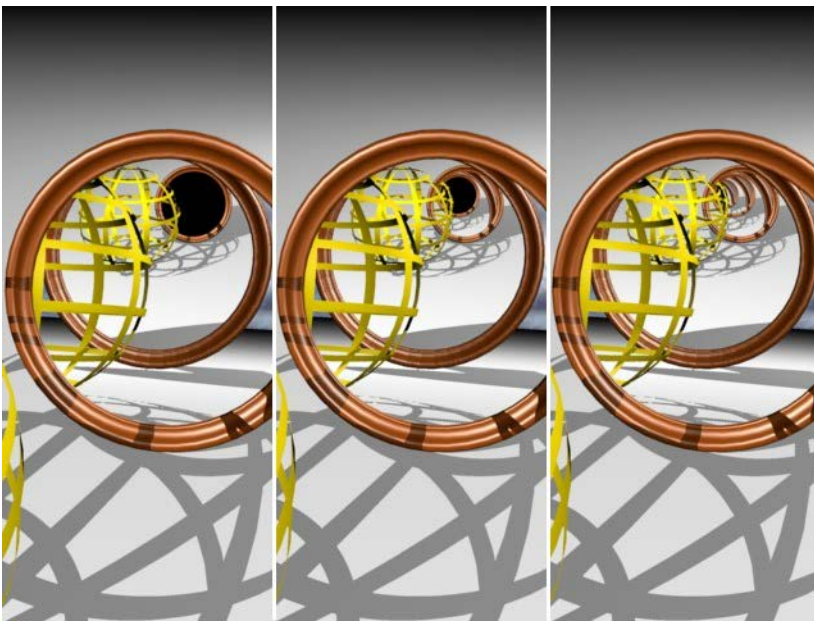
The higher the Ray Depth value, the further the rays are followed into the scene for rendering purposes.

A Ray Depth of 1 means that calculations are finished for a pixel once its ray hits something in the scene. Transparencies and alphas therefore will not be visible.

A value of 2 means that after a ray has hit a surface a second ray is calculated for the transparency.

Reflection Depth

The higher the Reflection Depth, the further rays are followed into the scene and the results rendered.



The results contain reflective objects and have been rendered with reflection depths of 2, 4 and 8

When a ray is sent into the scene, it can be reflected by reflective surfaces. With certain arrangements, e.g. two mirrors opposite each other, it is possible that a ray will be reflected forever, trapped between the mirrors, and the raytracer would never finish rendering the picture. In order to prevent this, you can set the maximum number of reflected rays.

You can also use Reflection Depth to limit the render time for the picture. Often, only the first generation of reflections is important. Further rays tend to add little to the image quality, but increase the render time considerably.

A Reflection Depth of 1 means that calculations for a pixel are finished once a ray hits something in the scene. Reflections therefore will not be visible.

A value of 2 means that after a ray has hit a surface a second ray is calculated for the reflection.

Global Brightness

This setting lets you simultaneously set the global brightness for all light sources present in your scene. A setting of 100% will utilize the intensity set in each light, 50% will decrease the intensity of each light proportionally by half, while a setting of 200% will double the intensity of each light.

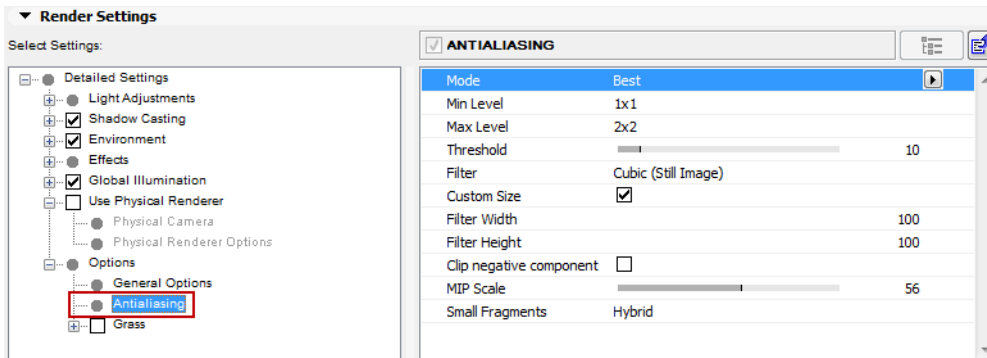
Generate Alpha Mask for Environment

If you enable this option, a pre-multiplied alpha channel will be calculated during rendering.

The alpha channel is a grayscale image of the same resolution as your color picture. Pixels in the alpha channel are either black or white. A white pixel in the alpha channel indicates the presence of an object at that position in the image while a black pixel indicates no object.

Antialiasing (Cineware)

These settings are available in the Detailed view of PhotoRendering Settings for the Cineware engine. Antialiasing is used with the Standard Renderer.



Antialiasing removes jagged edges from your images. It works by breaking down each pixel into sub-pixels; rather than calculate just one color for a pixel, several color values are calculated and averaged to produce the final color for the pixel.

Mode

Choose None, Geometry or Best.

From left to right: **None**, **Geometry** (the slightly blurred texture is a result of the antialiasing filters) and **Best** antialiasing.



- **None:** The image to be output will be calculated without antialiasing (any existing Render controls with differing settings will be ignored). Step-like structures will be recognizable at object and color edges. This mode is particularly suited for fast test renderings for which quality is not important.
- **Geometry:** This is the default setting which smooths all object edges (automatically with 16x16 sub-pixels).

- **Best:** This enables (in addition to object smoothing) adaptive antialiasing (additional sub-pixels will only be calculated at critical regions, i.e. for pixels whose color differs greatly from those of its neighboring pixels), which affects color edges (e.g. also shadows, objects behind transparencies, etc.). This in turn is controlled via several parameters available for this mode.

Higher antialiasing settings are often only noticeable in the finer details.

Min Level/Max Level/Threshold

The default combination of these three parameters will be fine in most cases.

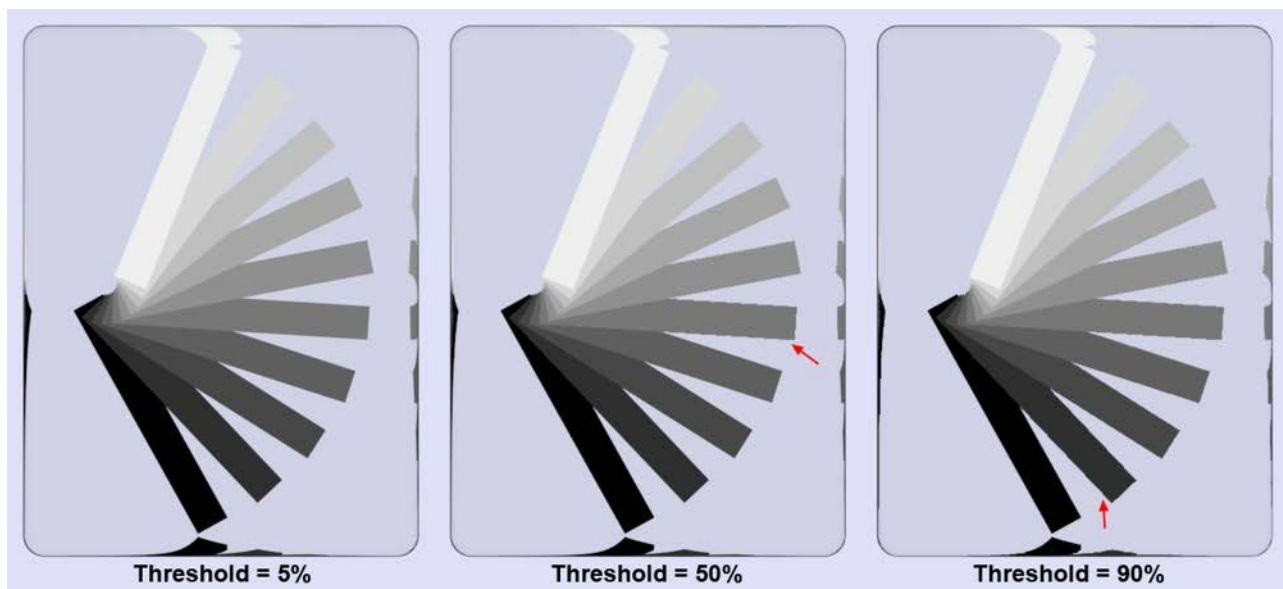
These settings control Cineware's adaptive antialiasing (set **Best** mode, above). Sub-pixels will be calculated for neighboring pixels whose color differs too greatly; for non-critical regions (large, uniformly colored regions) no sub-pixels will be calculated if at all possible.

Adaptive antialiasing only works for color edges or object edge smoothing for objects that lie behind transparent objects or in reflections.

Min Level defines the minimum number of sub-pixels that will always be rendered. The default value of 1.1 will suffice for most cases. If, however, artifacting should occur in very detailed regions, e.g. shadow elements get swallowed, higher values should be used.

Max Level is the sub-pixel dispersion that is applied to critical regions (mostly high-contrast regions, i.e. color edges or object edges behind transparencies). This value can, for example, be increased when rendering glass to ensure the rendering of finer details.

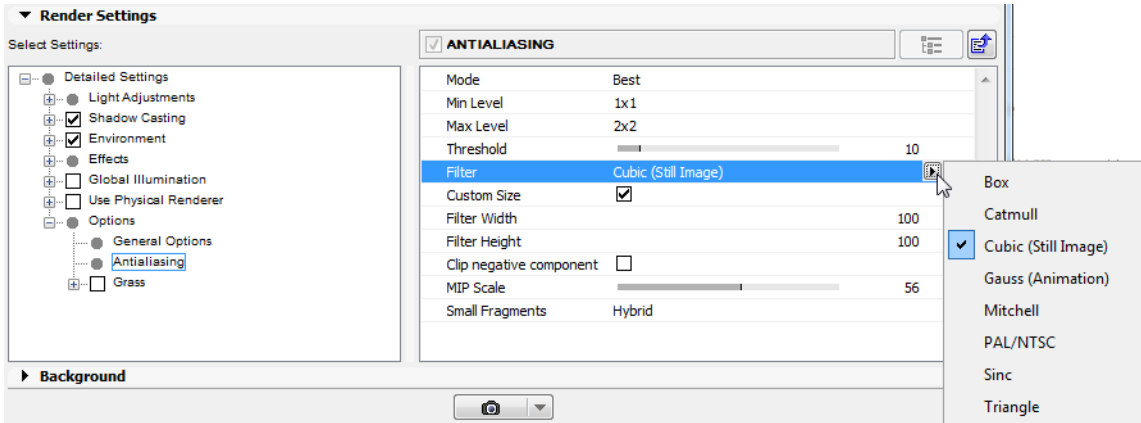
The **Threshold** value defines the degree of color divergence at which Max Level should be applied for a given pixel. Small values will allow no divergence, large values will allow greater contrasts before smoothing begins.



In an average scene with a Threshold (Color) value of 10%, about 40% of all pixels will be affected, whereas a Threshold value of 5% will result in 90% of all pixels being affected. A value of 0% will cause all pixels of a rendered image to be antialiased - including superfluous regions.

These three parameters greatly influence the render speed! Settings that are too high can easily increase render times tenfold, without much noticeable difference in render quality. The default values offer very good results in most cases, with moderate render times.

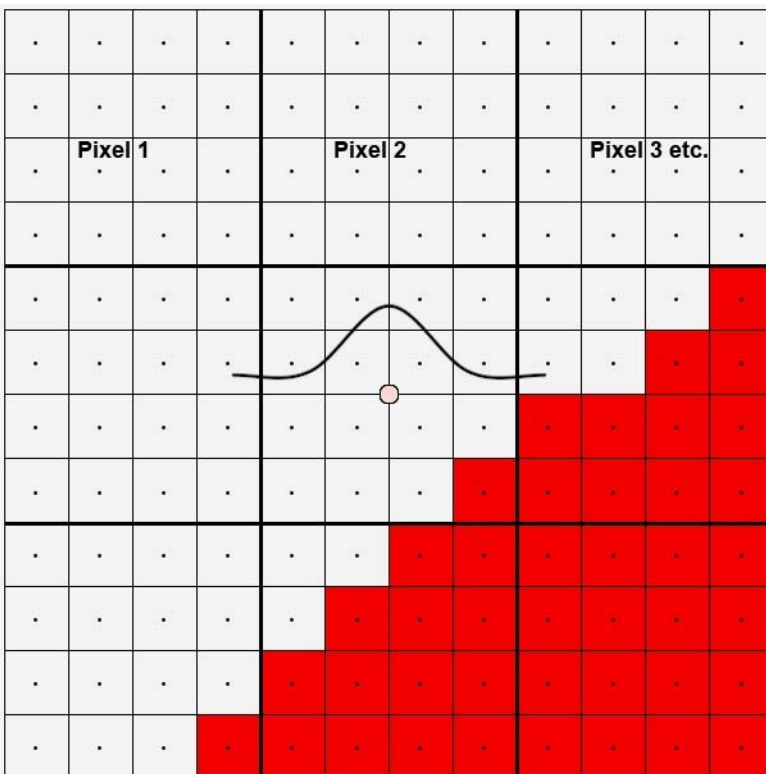
Filter



Generally speaking, the filter controls how sharp and crisp the edges will be rendered.

The filter works with both Geometry (object edge smoothing) and Best (color edge smoothing) antialiasing modes. In most cases, the filter selection has little effect on the render speed.

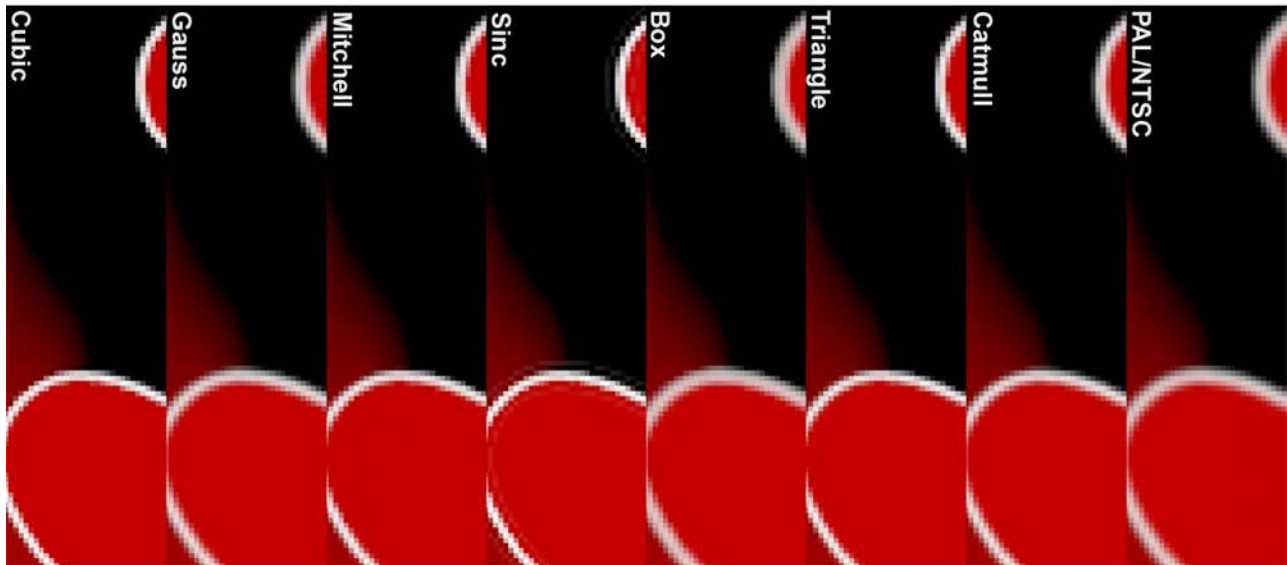
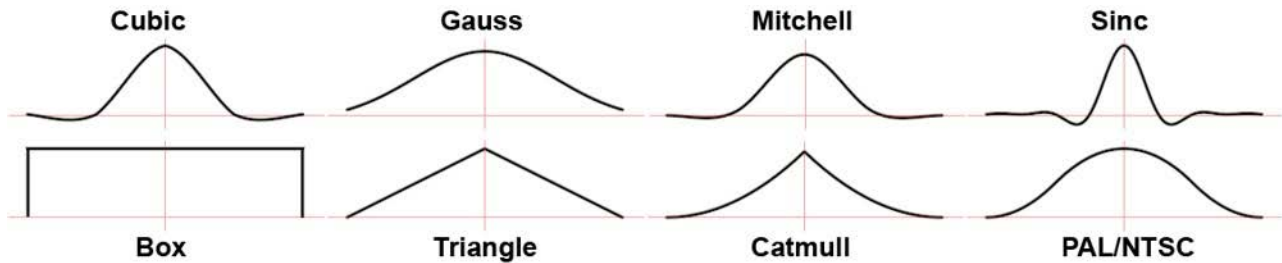
A brief explanation of how the antialiasing filter works: Depending on the antialiasing settings, a number of subpixels will be calculated per pixel whose colors are then pooled for that pixel (a pixel can only have a single color).



In the image above are 9 edge pixels that run diagonally across the image. Take a look at the center pixel. 16 sub-pixels were used to calculate this pixel. 15 are light gray, 1 red. These pixels form a curve (Mitchell in this case). Imagine this curve as flat around the center of a pixel. This curve then defines the degree of influence each sub-pixel should have. Because the filter size (Filter Width setting) and Filter Height (i.e. the spread within which sub-pixels will be taken into consideration) can be expanded to up to 4 pixels, the curve can be made to spread out quite a bit more. Conversely, this means that neighboring pixels for the most part see the

same sub-pixel and therefore have a very similar color. Of course edges can then no longer be made to look sharp and crisp.

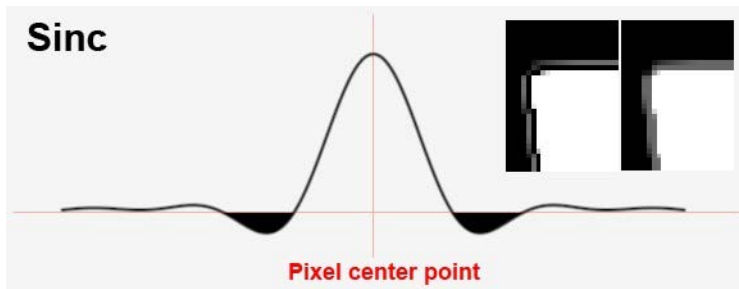
Each choice in the filter pop-up represents a different curve, defining the degree of influence each sub-pixel should have.



- **Filter Width/Filter Height:** Filter Width/Height defines how many sub-pixels (calculated from the center of the pixel outwards) will be taken into consideration when the pixel color value is calculated.
- As long as the **Custom Size** option is enabled, an ideal value for Filter Width and Filter Height will be used, which will also be displayed. The values are in relation to the pixel, i.e. a value of 0.5 for both parameters means that sub-pixels 0.5 to the left and right, and 0.5 above and below the pixel center point will be taken into consideration, i.e. all sub-pixels on the pixel surface. Higher values will extend to surrounding pixels accordingly, which will result in color edges being rendered correspondingly less sharply.
- **Clip Negative Component:** A filter function's negative regions can be clipped.

In the image above, you will notice that several curves (Mitchell, Cubic and in particular Sinc) even fall below zero. This means that at corresponding edges, contrasting color values are rendered that are not even present in the scene. This makes the edges look sharper. This can, however, lead to problems under certain circumstances. If the Clip Negative Component option is enabled, the negative regions will be clipped.

Here, a cube was rendered to which a luminous material was assigned along with the Sinc antialiasing filter. At left the Clip Negative Component was disabled, at right it was enabled.



MIP Scale

Scales the MIP strength globally for surfaces. For example, a MIP Scale of 200% doubles the MIP strength for each surface.

See [Sampling Methods in Surfaces](#).

This value should be increased when using highly detailed textures that are directly facing the camera. This will result in a better display of continuous lines - without interruption - by the antialiasing.

These values should be increased if you have detailed textures that lie perpendicular to the angle of view of the camera. The tendency for antialiasing to display continuous lines as dashed lines will be reduced.

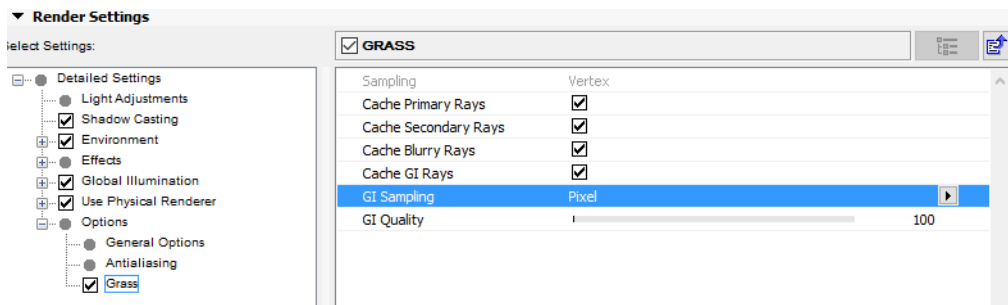
For crisp, sharp stills, keep in mind that you may need to change the Sampling setting of surfaces. Although the default Sampling setting, MIP, is a good choice for materials on objects that extend towards the horizon — such as a floor — for things like bottle labels you will get a much sharper picture by changing Sampling to Square, Alias 1, Alias 2 or Alias 3.

Small Fragments

Let's say you have a complete house consisting of thousands of polygons located at your scene's horizon, which is so far away from the camera that it would only be about one pixel in size when rendered. The Small Fragments functionality is designed to render such regions more effectively (as well as regions with high levels of Subsurface Scattering).

Grass (PhotoRendering Option)

These settings are available in the Detailed view of PhotoRendering Settings for the Cineware engine.



Use these controls to define the rendering parameters for the Grass surface in the scene, if the Grass surface channel is being used. [See *Grass \(Cineware Surface Channel\)*](#).

Sampling

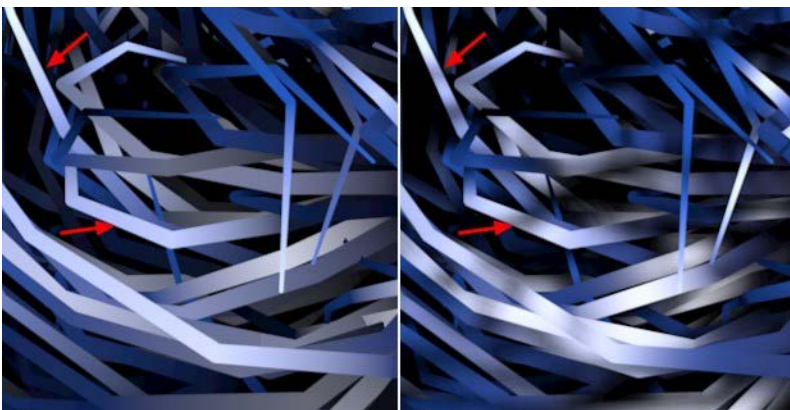
When grass is rendered, light shadow and color are calculated at different locations on the grass and subsequently interpolated:

- **Pixel:** No interpolation will take place - each pixel will be calculated individually. This method delivers the better results but also take the longest to render.

Otherwise the Pixel option should be selected in the following instances:

- If a texture in the Color channel is projected onto each individual hair.
- If the Specular Lights are too blurred, regardless of changes made to the Reflectance channel.
- In general, render quality will increase when very few grass segments are used.

- **Vertex:** The start and endpoints of segments are sampled and interpolated. This setting offers a good compromise between quality and render time and will lead to good render results in most cases.

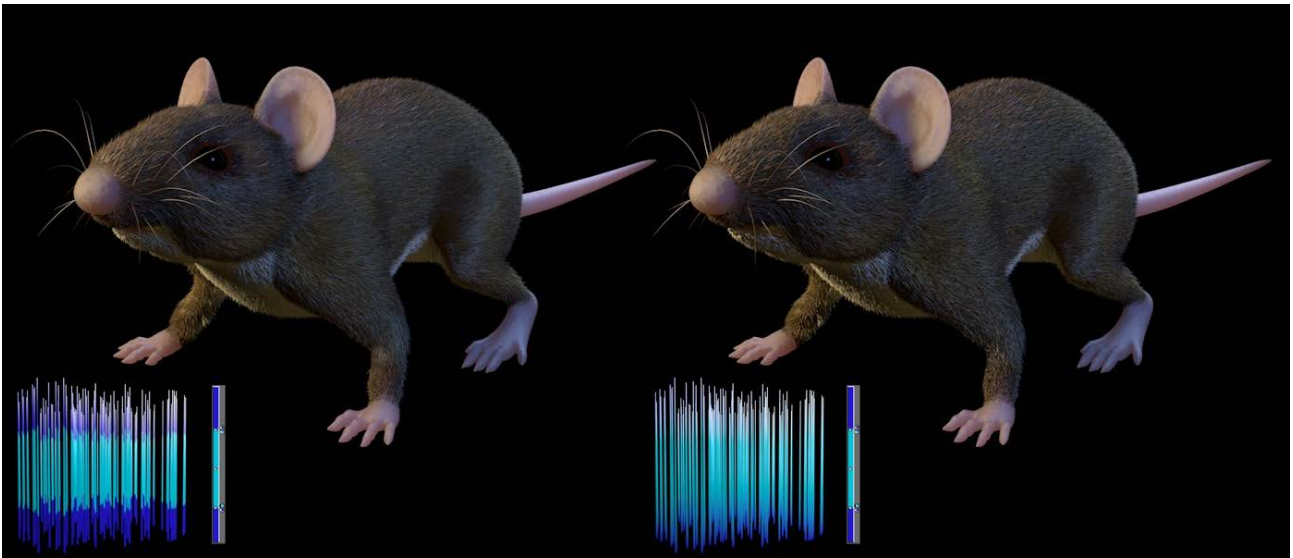


Left Vertex, right Pixel. Note how individual grass shadows are taken into consideration at the right, which are ignored on the left.

Cache

These options are displayed when the Physical Renderer is used. They are designed to deliver faster rendering but will result in corresponding loss of quality. Generally speaking, you should simply activate the options, see if the results meet your standards and enjoy the faster rendering.

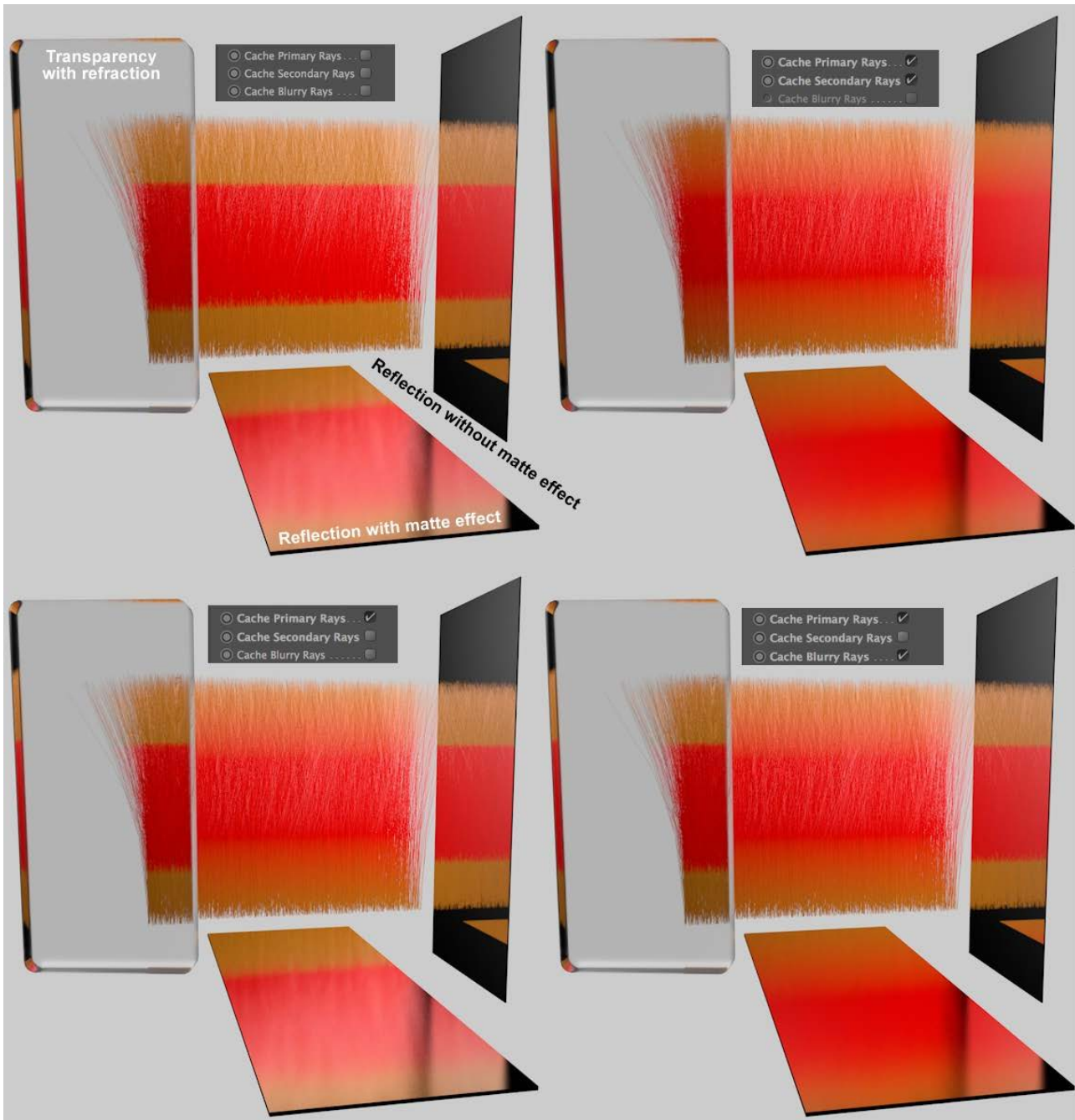
The Cache options temporarily save the grass illumination surface properties - diffusion, specular and indirect lighting (GI) - to the RAM (primarily at the grass vertices). These one-time calculated values will then be used multiple times during rendering, which saves time. However, the calculation is simplified, which can have negative effects on render quality. These effects are very minimal and barely noticeable.



Left: Cache Primary Rays disabled; right: enabled.

Note how the fur on the mouse at the right appears more coarse. The grass consists of only 3 segments and the respective Grass surface has the color range to the right assigned to it. You can see how the colors of the grass on the right blend. Here, the color is calculated on each of the 4 grass vertices and interpolated. Now imagine this “blurriness” applied not only to the color but also to other grass properties such as specular, etc., and you have an impression of how the Cache options work.

The following image shows various Cache option combinations and their effect:



Note how the color range for the grass at the top left looks best with no caching but requires the most render time, and how the other examples have poorer quality but shorter render times. The reduction in quality has been exaggerated in these examples by using only 3 grass segments. Normally you will have a far greater number of segments, which means that the loss in quality will be much less.

Cache Primary Rays

Effects of the rays as seen from the angle of view of the camera will be cached. This includes shading and shadow effects as well as the grass's specularity (see image above).

Cache Secondary Rays

Secondary rays are used to calculate reflections, refractions, blurriness effects and GI. If this option is enabled, these elements will be cached. Secondary rays are a superset of matte and GI rays, which means that the following two options can be disabled separately if this option is enabled. Secondary rays cannot read/use the primary rays' cached specularities. This will then come after transparency and reflection.

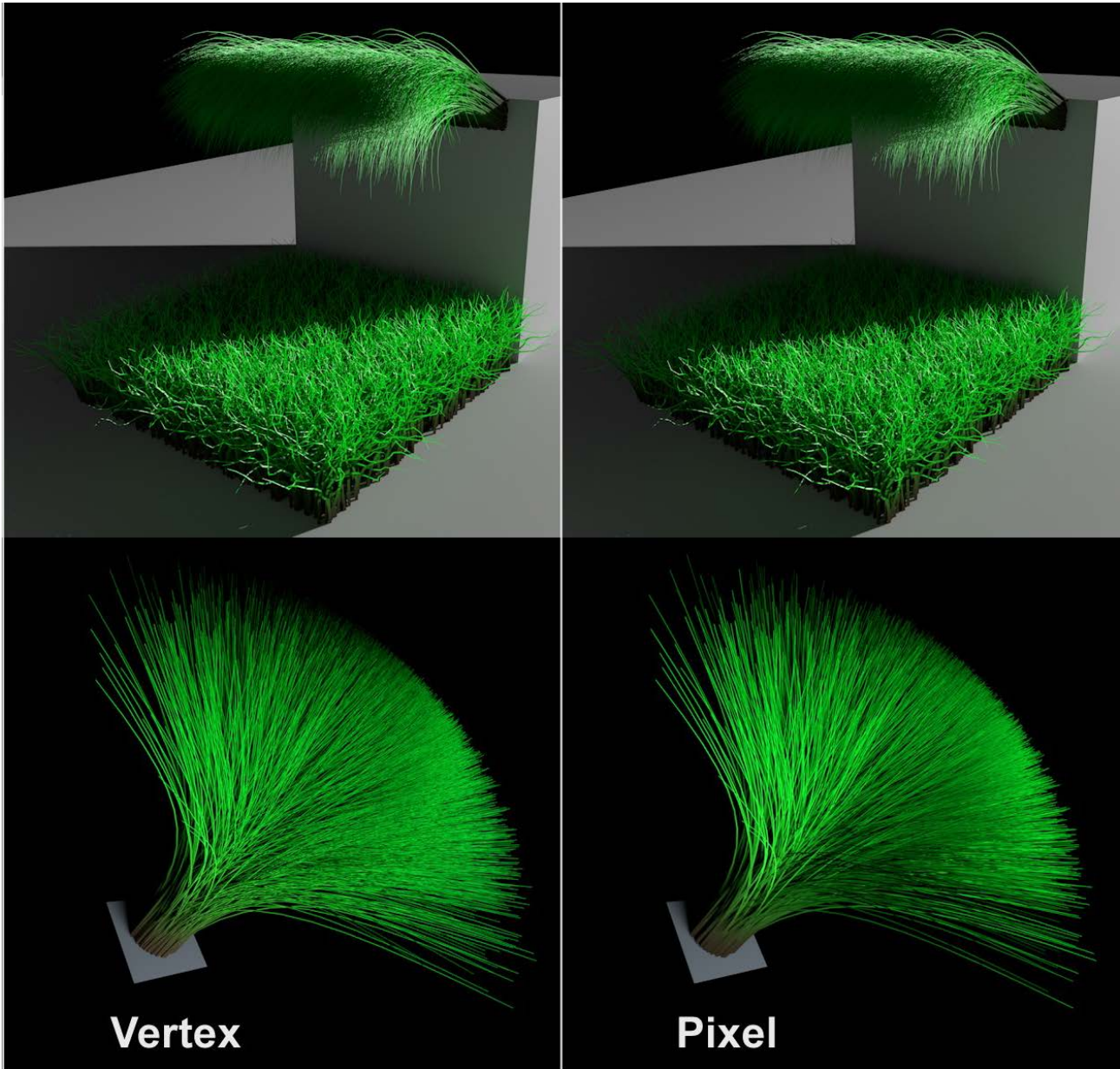
Cache Blurry Rays

Blurry rays are needed in order to calculate Blurriness. These will then be cached if this option is enabled.

Cache GI Rays

This option caches the GI sampler's effect and is only effective if used in conjunction with Global Illumination. The visible effect of the cached GI rays is very minimal, which means that this option can actually remain enabled. This in turn results in much faster rendering in conjunction with QMC.

GI Sampling



The top row is illuminated using a light source, the bottom row using a polygon light

You can define where GI should be rendered or cached on grass:

- **Pixel:** The entire grass will be rendered with GI. This is the slowest and most precise method.
- **Vertex:** Only the grass vertices will be rendered with GI. The result is very similar to that of the Pixel mode but is generally rendered brighter because the grass normals are, internally, always oriented towards the camera.

GI Quality

This value represents a multiplier for the Samples parameter. It is designed for use with QMC, which requires more samples in conjunction with grass. When the aforementioned Cache options are used, this setting will be taken into consideration. However, if the Cache GI Rays option is enabled, this setting will have no effect.

Spherical Camera (Cineware)

Use this feature to create a rendering with a 360 view. The resulting image can be viewed as a navigable panorama picture in various media, such as YouTube, Facebook or Twitter.

Introduction

The increasing number of virtual reality applications (VR) require new methods of creating image material for this purpose. The Spherical Camera setting renders the scene with a 360° view, which can be viewed with a VR headset or via 360° YouTube videos.

Note: Stereoscopic rendering is also possible. See also [Stereo Rendering \(Cineware\)](#).

The Spherical Camera is not restricted to VR. It can also be used to render HDR images, which are simply an all-round view of the scenery.

Since Cineware renders two-dimensional images, the all-round view has to be distorted accordingly so it fits into a rectangular bitmap. For this purpose, Cineware offers the most important projection types.

The resulting view provides a spherical perspective, similar to HDRI. The rendered picture is assigned a field-of-view (FOV) tag: either Equirectangular or Dome. These tags are recognized by certain imaging engines (e.g. built into YouTube, Facebook, QuickTime Viewer, etc.) which then produce the panorama.

Note: For information on uploading to YouTube, see: <https://support.google.com/youtube/answer/6178631>.

Camera Setup

The camera should be positioned at or very near eye level. It's often helpful to position the camera level in the direction of view and to set up the view without rotation (Roll Angle should be 0), to have a neutral angle of view from which to start.

Note that the projection only takes effect for rendering and will not be displayed in the 3D window.

Render Output

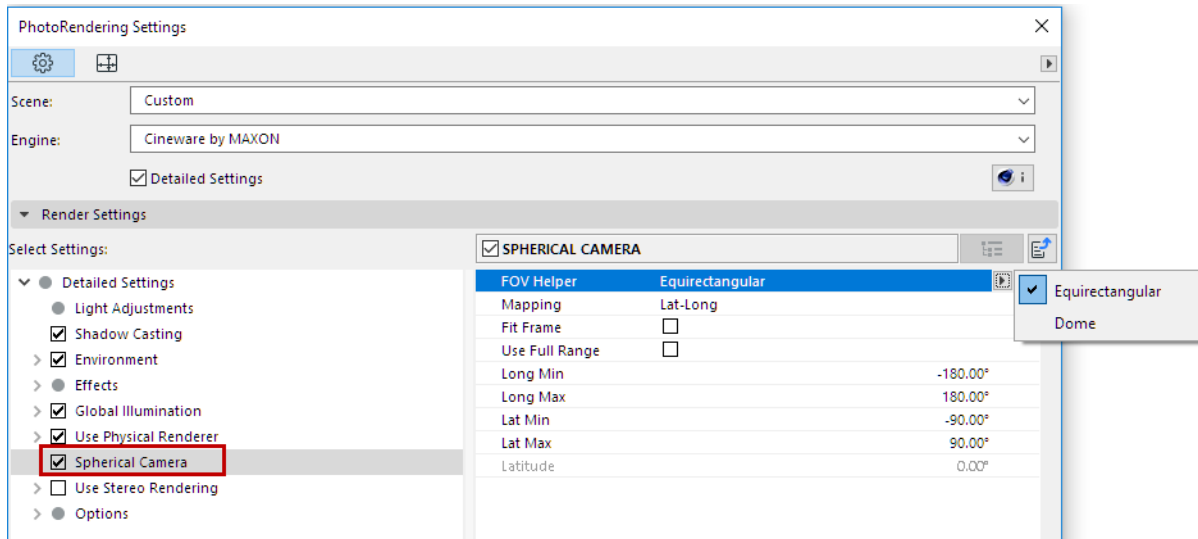
Make sure that the image sizes are correct for output. Since the entire scenery (not just the normal camera view) is used to create a bitmap, the image resolution should go beyond what you usually use. The following formats should be used as reference for mono images. (When using Stereo Rendering, the respective side should be doubled, depending on the Stereo Layout being used):

- Lat-Long: 2:1, e.g. 2000x1000, 4000x2000
- Cube Map (Cross): 4:3, e.g., 2000:2667, 4000x3000
- Cube Map (3x2): 3:2, e.g., 2000x1333, 4000x2667
- Cube Map (String): 6:1, e.g., 2000x333, 4000x667

Spherical Camera Settings

Enable Spherical Camera

Check the **Spherical Camera** box at PhotoRendering Settings.



FOV Helper

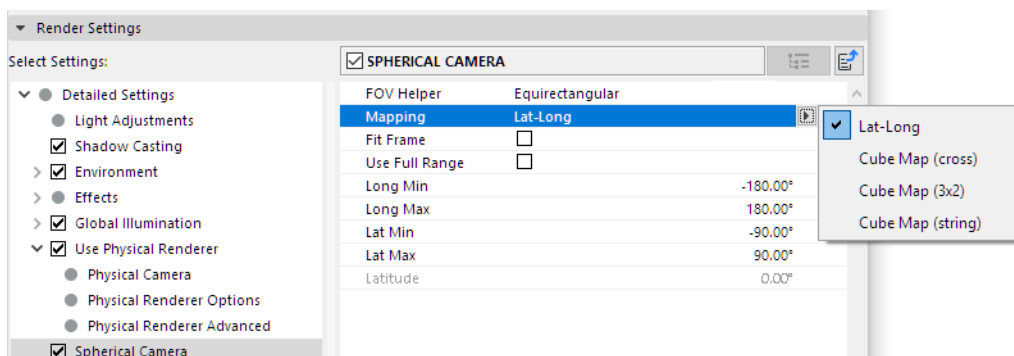
Use this setting to define which part of the scene the camera sees:

- **Equirectangular:** The entire spherical environment around the camera.

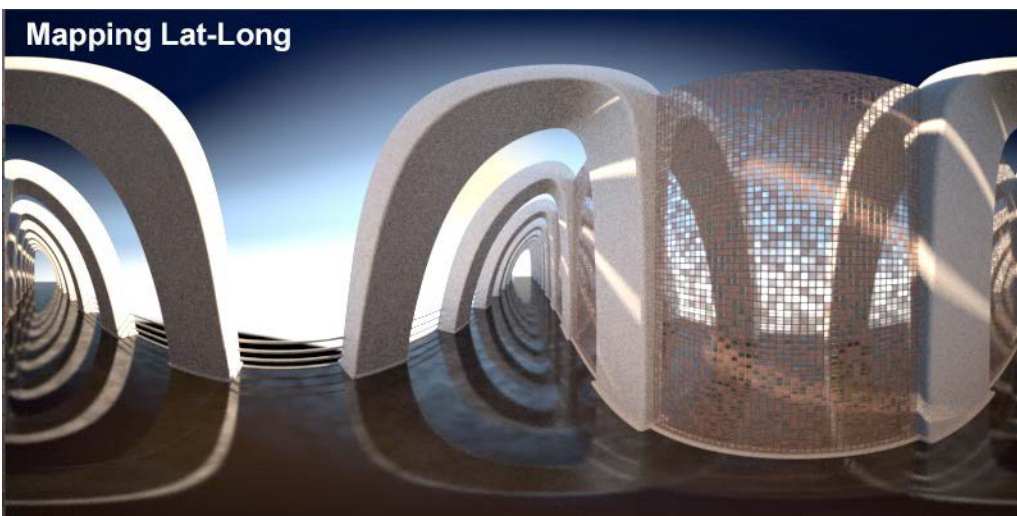
Note: For YouTube, the views have to be rendered with FOV set to Equirectangular, and with Mapping set to Lat-Long.
- **Dome:** Only the top hemisphere of the camera’s environment will be included (the bottom half will be rendered black). If you restrict the field of view further using the Min/Max settings, this will be displayed by the FOV.

Mapping

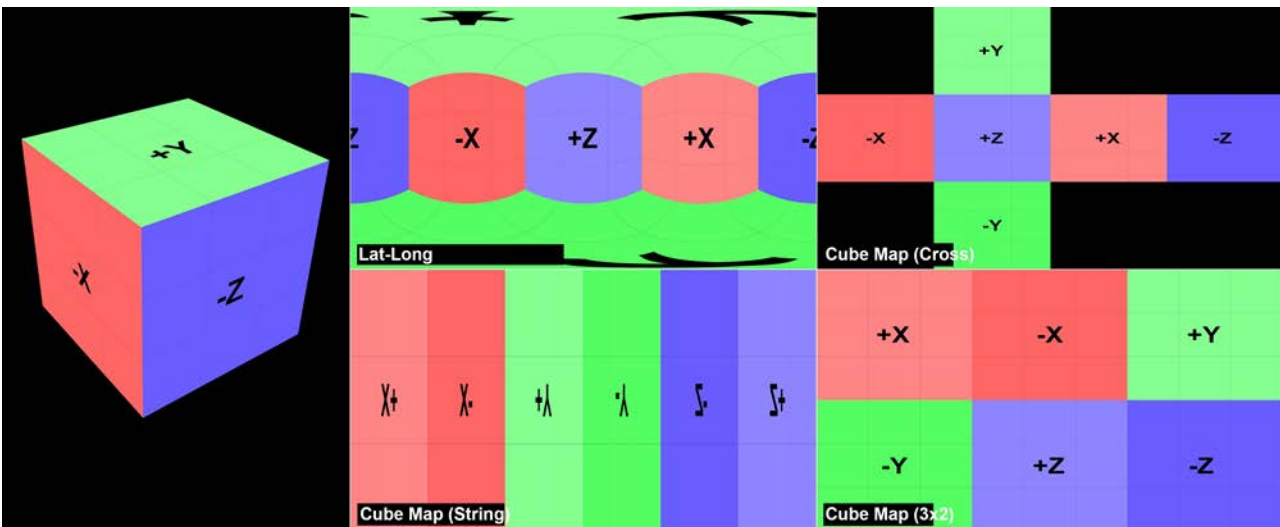
If you imagine the scene around the camera, the entire environment must be projected onto a rectangular bitmap. Define the method by choosing from the following Mapping options:



- **Lat-Long:** Also known as equirectangular or latitude/longitude. The “texture” will be peeled off cylindrically, similar to how a world map is designed with a distortion-free “equator” and increasing distortion towards the poles.



- **Various Cube projection types:** The environment will be depicted through the camera along all object axes, as illustrated in the image below.



The Spherical camera lies at the center of the cube. It was then rendered using the respective Mapping type.

The Mapping type that is best suited for your needs depends on how the rendering will subsequently be used. Make your selection depending on the software with which the image will be edited. The most commonly used method is Lat-Long, which can also be used if you create an HDRI or a depiction of the scene for a spherical projection.

Fit Frame

If the FOV is restricted further using Lat-Long Min/Max (see options below), the region to be rendered will be stretched so the entire bitmap is filled out. If this option is disabled, the FOV will only take up part of the bitmap and the rest will be depicted as monochromatic.

Use Full Range

If FOV is set to Equirectangular, check **Use Full Range** to use the entire all-around view. If you leave it unchecked, then only a portion will be used, based on the following settings:

- Long Min [-180..180°]
- Long Max [-180..180°]
- Lat Min [-90..90°]
- Lat Max [-90..90°]

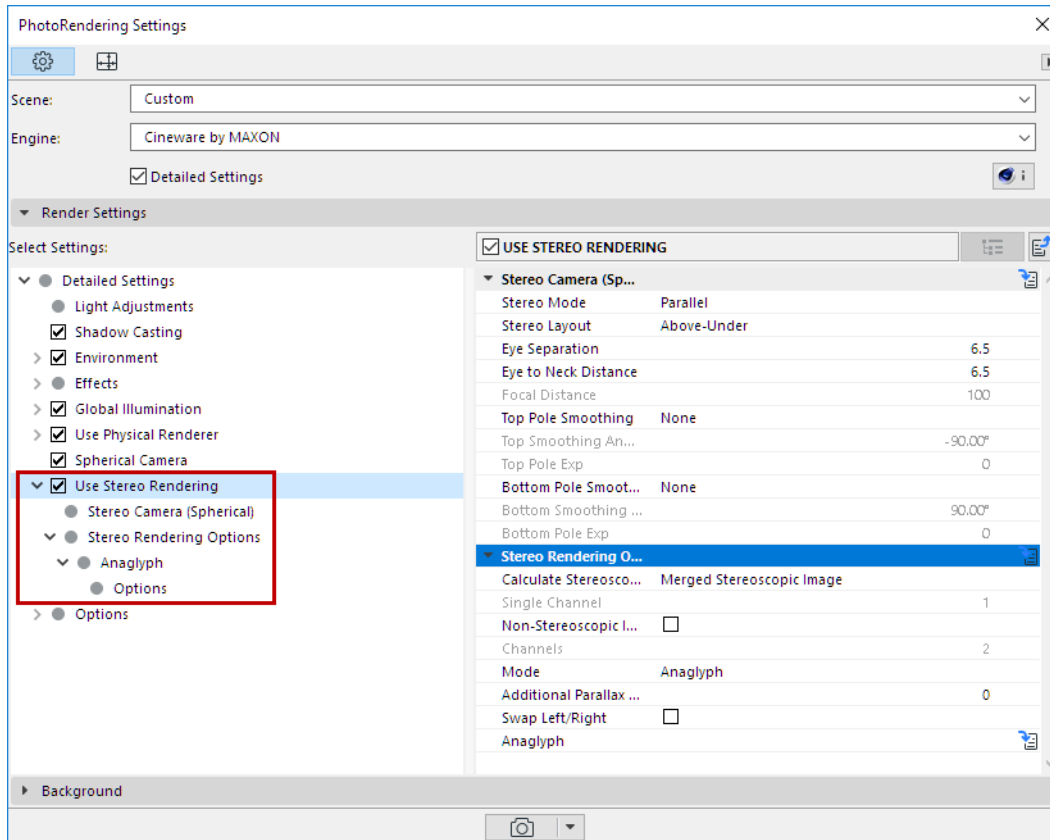
Use these settings to restrict the FOV starting from the X axis (Lat Min/Max) or Equator (Long Min/Max).

Latitude

If FOV is set to Dome, the Latitude setting can be used to define the dome section as a spherical element.

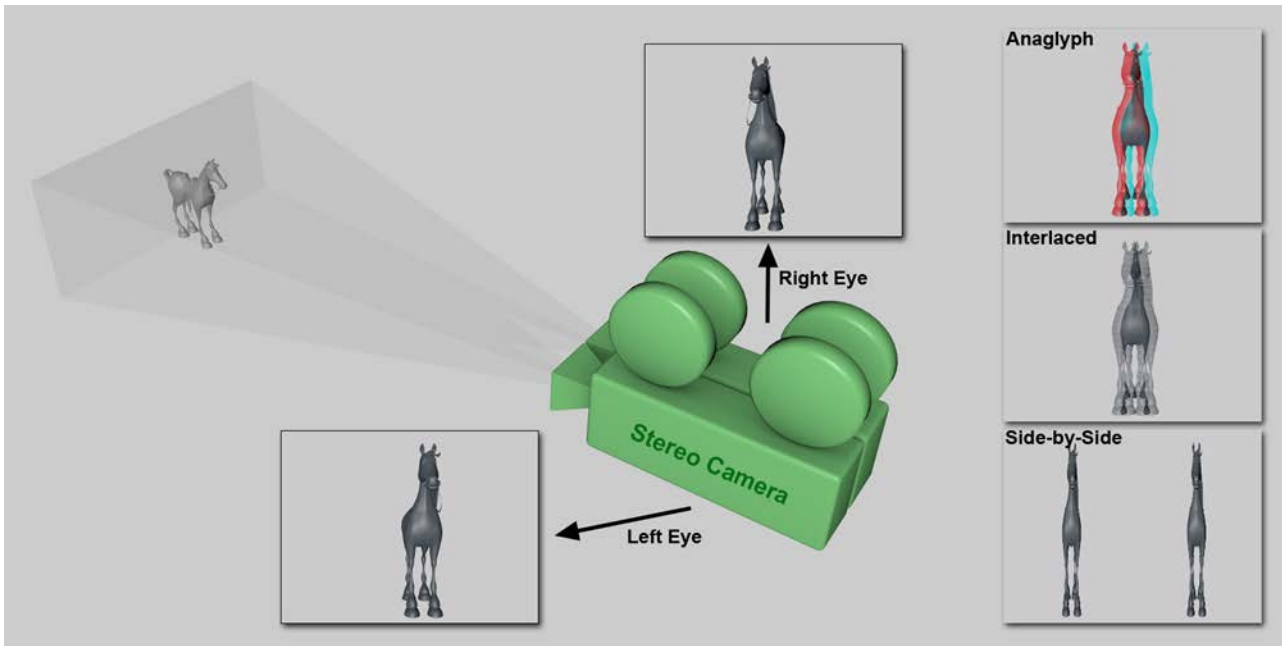
Stereo Rendering (Cineware)

Use **Stereo Rendering** settings to define all properties having to do with the creation of two or more images whose perspectives are slightly offset.



Introduction

In recent years, advances in technology have made it possible to display 3D images and movies true to color and free of fatigue for the human eye. Most techniques record two images of the same scene from slightly offset perspective views (those of the left and right “eyes”). These two images must then be displayed to the viewer in a way that the left eye only perceives the left image and the right eye the right image (which is done using specially designed glasses). The rest is done more or less automatically: the brain combines these two images into a single image.



Two images with different perspectives are combined to create a stereoscopic image.

Topics in this Section

[Options with a Spherical Camera](#)

[Options with a Non-Spherical Camera](#)

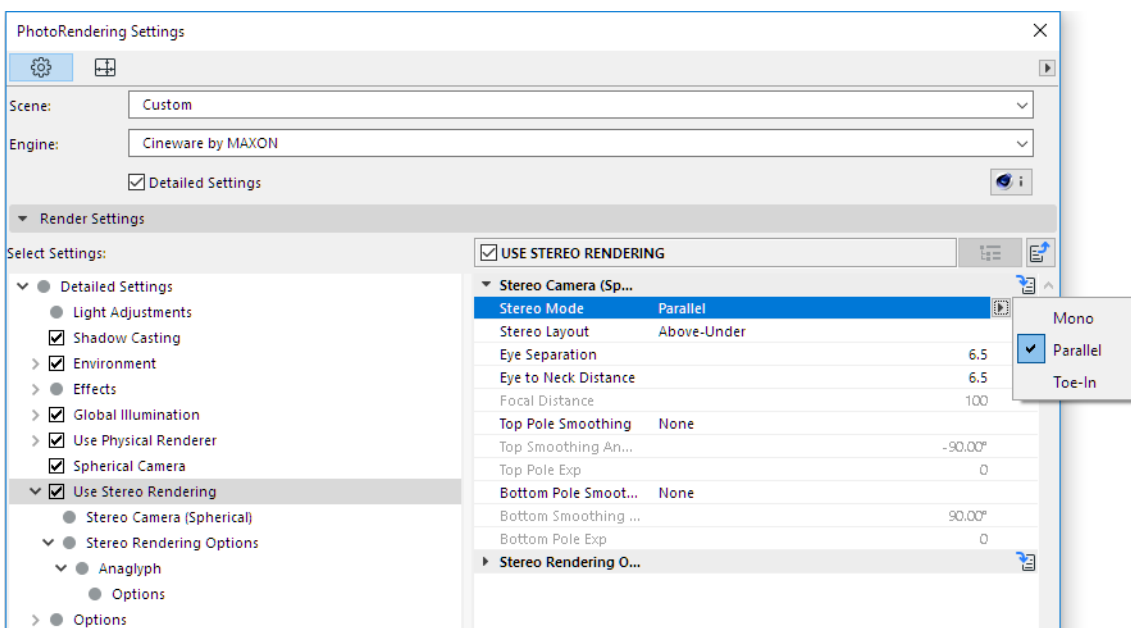
[Stereo Rendering Options](#)

Options with a Spherical Camera

Special settings are available if you want to make the 360° view of a Spherical Camera into a stereoscopic view. The left and right eye views will be combined to a single image (left = top; right = bottom).

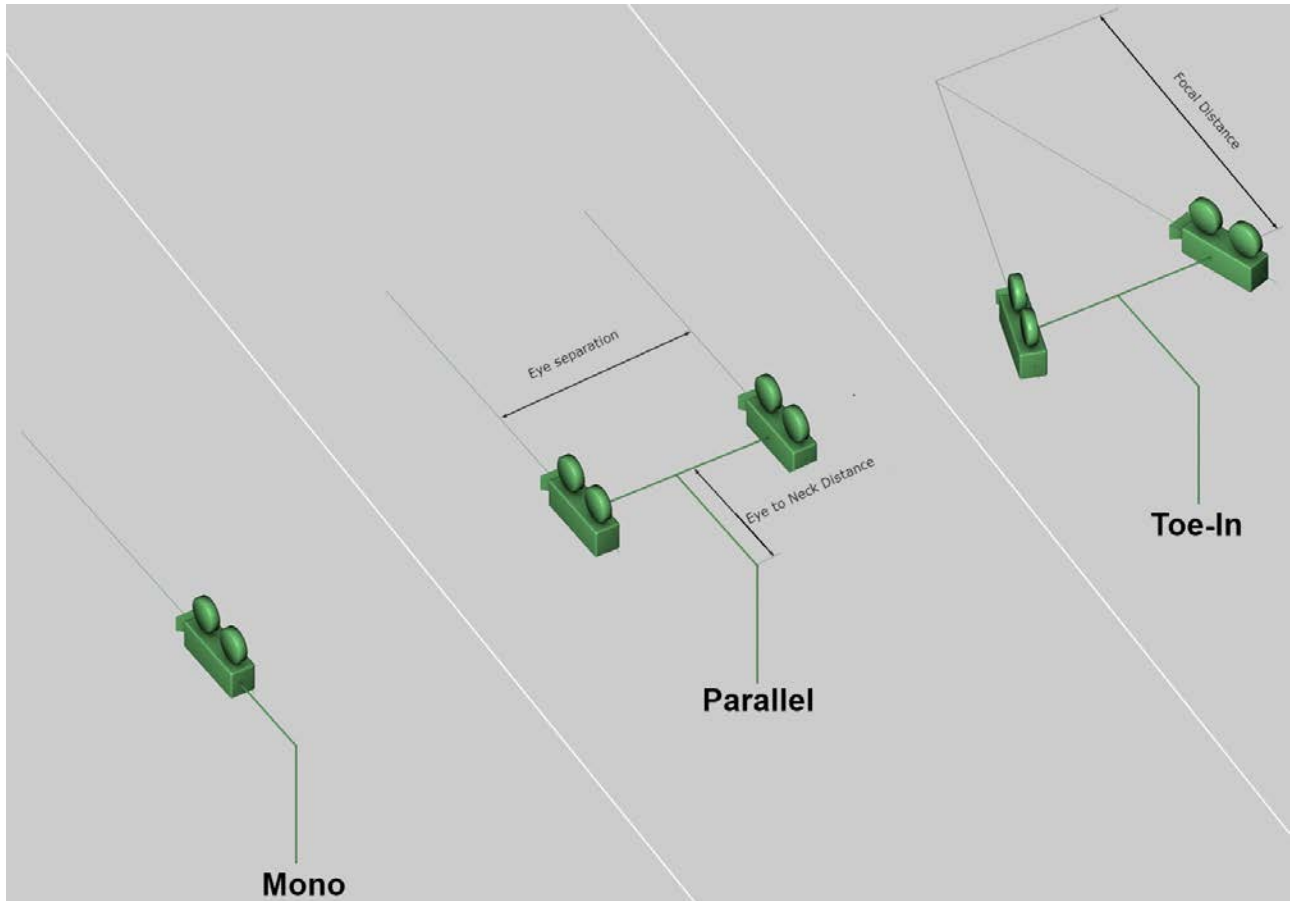
Check the Spherical Camera option to access the special Stereo Rendering options for a spherical camera.

If you are not using Spherical Camera: See [Options with a Non-Spherical Camera](#).



Stereo Mode

- **Mono:** No stereoscopic images will be rendered.
- **Parallel:** Both cameras will be arranged with parallel axes of view.
- **Toe-in:** Both cameras' axes of view will cross. The point of intersection can be defined using the Focal Distance.



The various Stereo modes with the camera axes (the cameras are rotated internally for rendering panoramas)

The primary difference between the two stereo modes lies in the definition of the null parallax, i.e., the distance from the camera at which no parallax is generated.

Note: Parallax is the apparent displacement of an object when seen from two different points.

- If Toe-In is selected, the null parallax will be defined using the Focal Distance setting and cannot be subsequently modified.
- If Parallel is selected, the null parallax is infinite and can be subsequently modified (by moving the left and right perspectives).

Stereo Layout

Define how the two stereo images should be arranged, or which one (left or right) should be rendered alone.

Eye Separation

This value defines the distance between both cameras/eyes. The default distance of 6.5 cm represents the average distance between human eyes.

Eye to Neck Distance

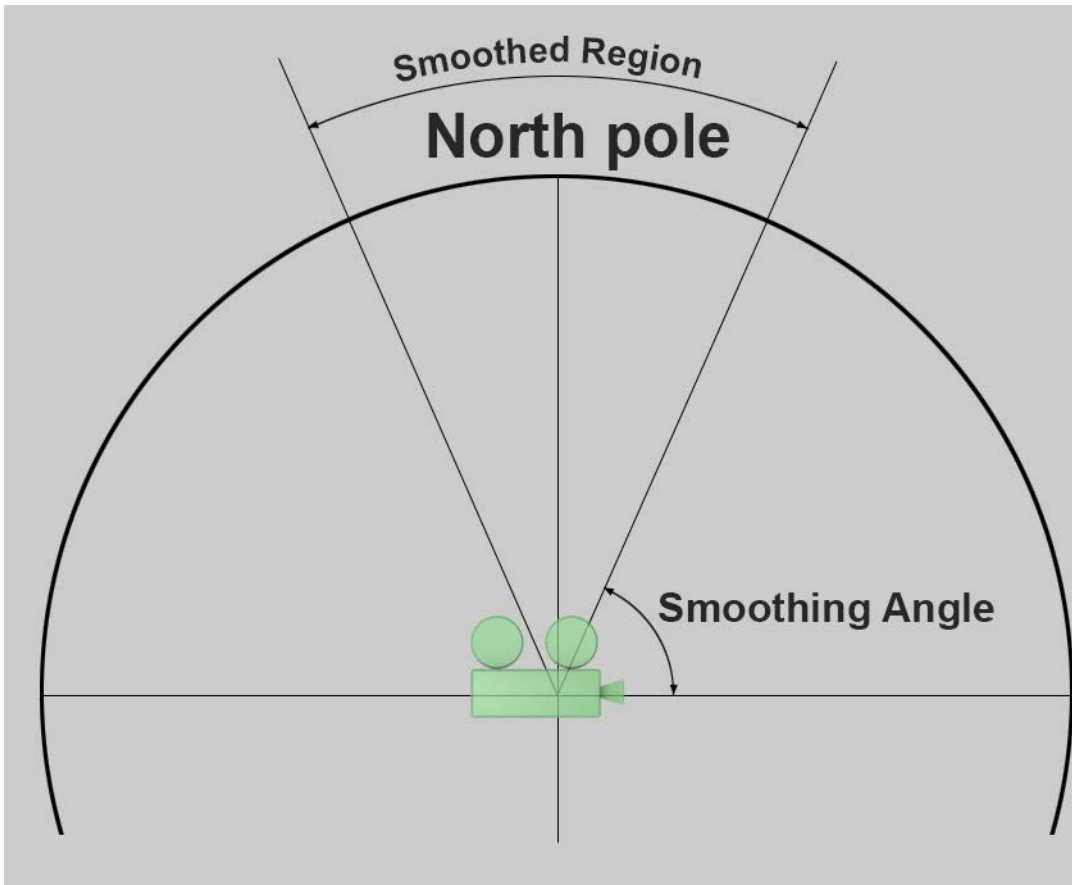
Depending on which target model is rendered, you can define the horizontal distance from the neck to the eyes (see image above). If the Eye to Neck Distance is set to 0, the camera's rotational point will no longer lie at the center of both cameras.

Focal Distance

If Stereo Mode is set to Toe-In, you can use this setting to define where the null parallax should lie. Objects that lie in front of this point and face the camera's direction of view will protrude out of the monitor towards the viewer; objects that lie behind this point will more or less "sink" into the monitor.

Top Pole Smoothing/Bottom Pole Smoothing

For technical reasons, stereoscopy cannot be rendered correctly around the poles. To prevent unwanted artifacting, the stereo effect can be gradually faded in these regions. In most cases, this can be done with no problem because the visually important elements mostly occur horizontally around the camera's location (and not vertically above or below it).



The smoothing can be defined separately for top and bottom poles. In the region in which the smoothing takes place, the views will be equalized with the left and right cameras until they are identical with the maximum possible smoothing.

Smoothing Types

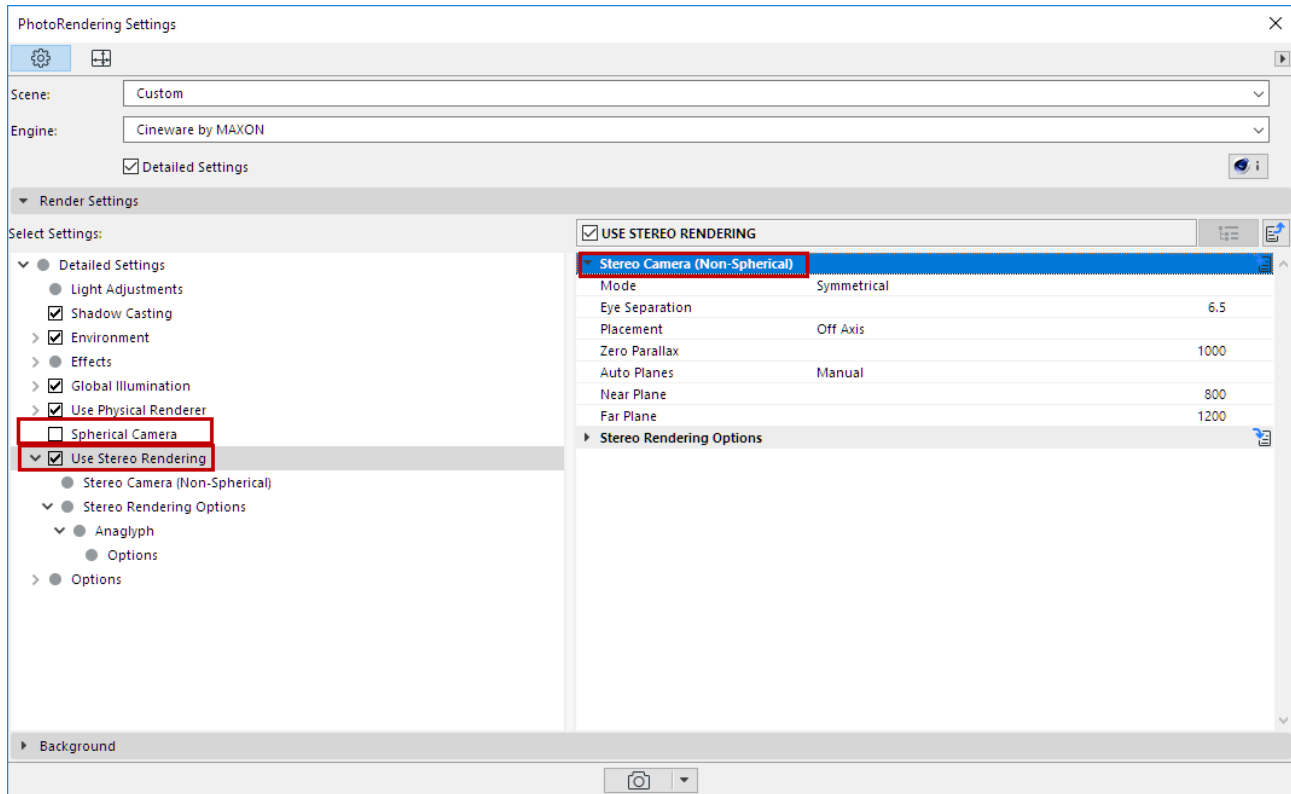
Smoothing, using the defined **Smoothing Angle**, will take place from the start to the pole using one of two methods:

- **Linear:** Smoothing occurs with linearly increasing strength, i.e., abrupt deployment

- **Exponential:** Smoothing occurs with exponentially increasing strength, i.e., gradual deployment.
 - Use **Top/Bottom Pole Exp** settings to affect the Exponential option.

Options with a Non-Spherical Camera

If you are not using the Spherical Camera option, the following Stereo Rendering options are available.



Mode

- **Mono:** This is the normal camera mode without stereoscopy.
- **Symmetrical:** Use this mode to render normal stereoscopic images. A double camera will be assumed and each camera will be assigned half of the defined **Eye Separation** value, left and right, respectively.

Depending on the parameters defined, both cameras will be positioned as follows on the Camera object's X axis:

- **Left:** left camera is set to 0, right camera is set to **+ distance between eyes**
- **Right:** left camera is set to **-distance between eyes**, right camera is set to 0

Eye Separation

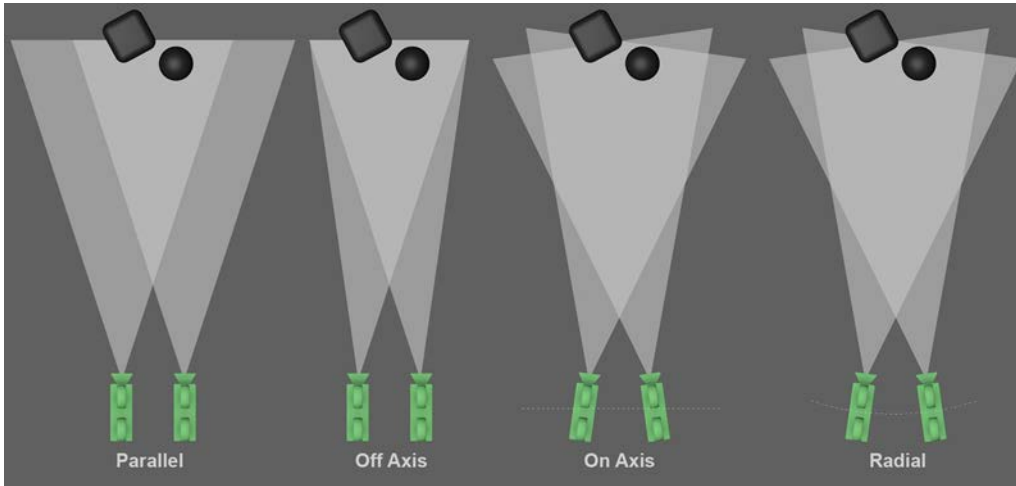
This value defines the distance between the left and right eye. The default value of 6.5cm reflects the average distance between a human's eyes. Normally, this value should be as small as possible. Greater values will produce a correspondingly larger spatial view but it will also become more strenuous for the viewer to view the scene.

Exception: if a scene, e.g., a landscape, is depicted that lies far away from the camera.

Placement

Even though several options are available, most of them are only there for compatibility reasons. For stereoscopic double images, you should always (with the exception of special circumstances) set Placement to **Off Axis**.

Each example in the image below uses a double camera only.

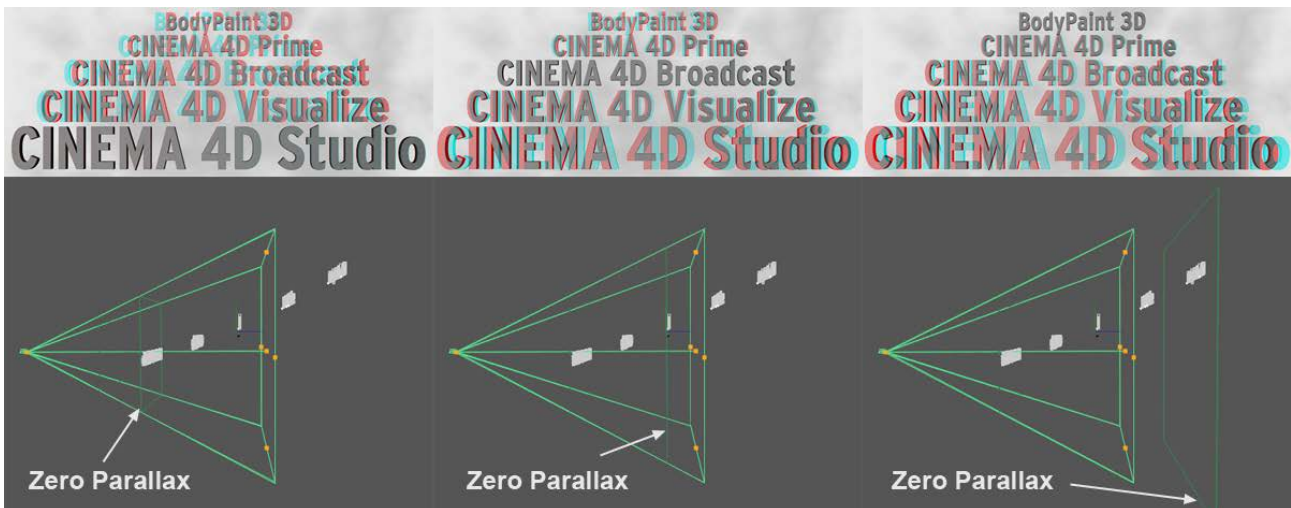


Depending on placement, both (or several) cameras will be oriented differently.

- **Parallel:** This is the simplest stereoscopic camera arrangement. Both cameras are positioned parallel to each other (the image axes are also parallel). Stereoscopic images with this type of arrangement only depict objects that lie in front of the projection plane. Hence, the null plane CANNOT be moved by modifying the Zero Parallax value. This is in contrast to other following three modes.
- **Off Axis:** Basically the same camera arrangement as Parallel, but with an offset, which means that the image axes are no longer parallel; they intersect. The zero parallax lies at this point of intersection (see next setting, [Zero Parallax](#)). Objects can be depicted to lie spatially either in front of or behind the projection plane (i.e. in or in front of the monitor).
Tip: This mode is recommended for stereoscopic double images because it delivers the best results for the widest range of applications.
- **On Axis:** When this mode is selected, both cameras are rotated so their Z axes intersect with the zero parallax. This reflects approximately how the human eye works but is not recommended for the creation of stereoscopic images because a vertical parallax can result. This mode is also referred to as “toe-in”.
- **Radial:** This mode is similar to On Axis, except that both cameras don’t lie on the Z axis but rather on an arc (whose center point lies on the point of intersection of both cameras’ zero parallaxes).

Zero Parallax

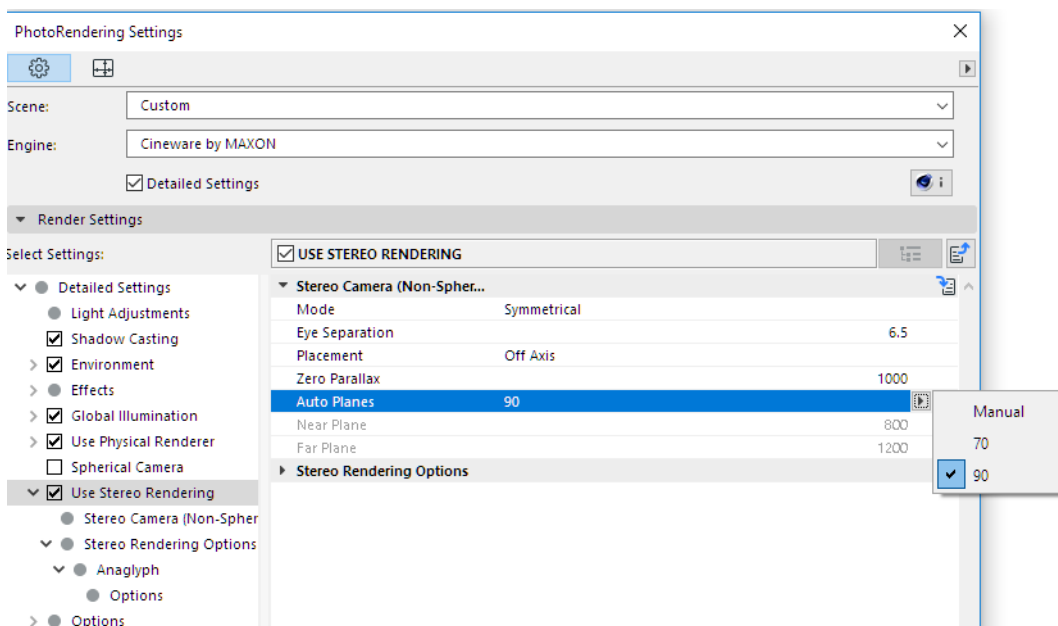
The zero parallax is a virtual plane that lies vertically to the camera’s angle of view and defines where the projection plane lies, i.e., the plane that represents the monitor screen at its depth. Objects that lie in front of this plane in the direction of the camera protrude spatially out of the monitor in the direction of the viewer; objects that lie behind this plane lie “within” the monitor.



Auto Planes

Choose either an Auto value of 70 or 90, or choose **Manual** to enter separate Left Plane and Right Plane values.

If you want to play it safe, select 90 and place all visible objects in the direction of the camera and behind this plane.

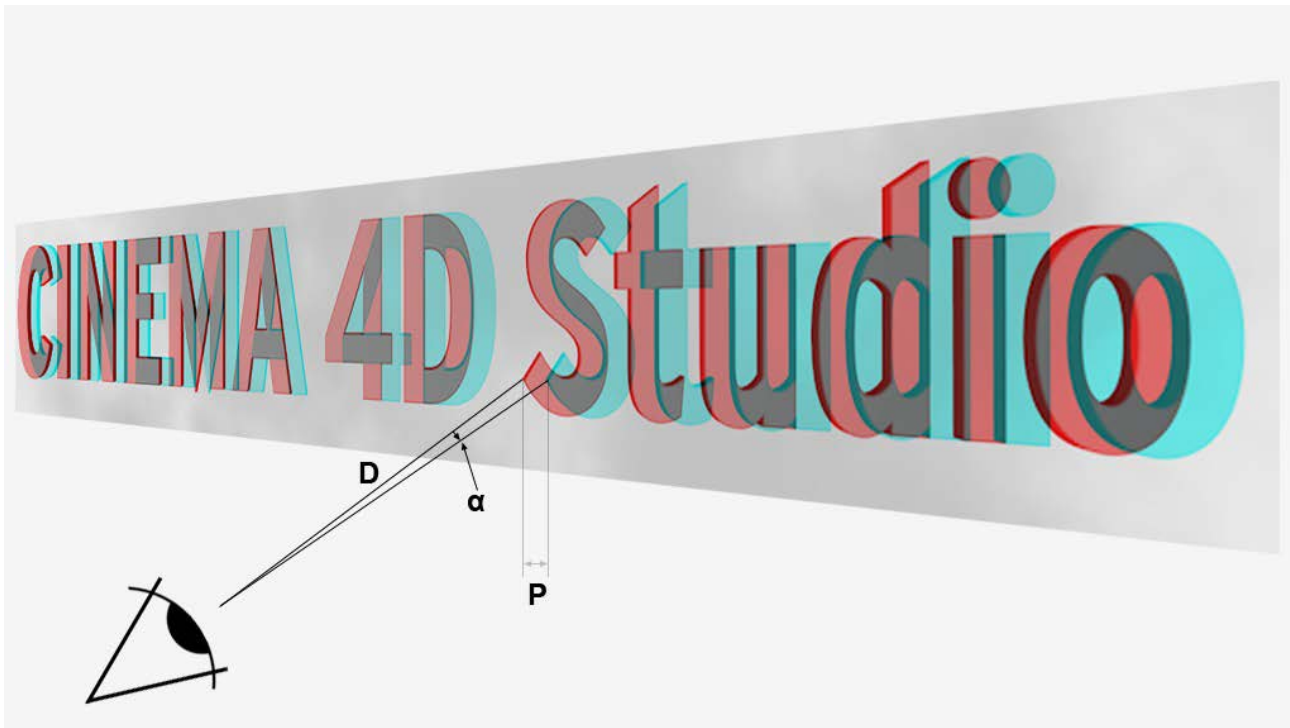


The Auto Planes do not affect rendering! They only serve as visual references that you can adjust manually. If you determine via test renderings that the optimal spatial impression within a given distance from the camera has been achieved (objects that, for example, lie too close to the camera – while the zero parallax is positioned far away – can only be seen with great effort by the human eye), then you can define these planes accordingly and can place objects in the view correctly, within the restricted space that results.

The options 70 and 90 represent a parallax of 70 and 90 arc seconds, respectively, for the near plane. These values are described in technical literature as values at which the human eye can perceive spatially with normal effort. Therefore, objects should lie behind this near plane.

A formula exists that defines the maximum parallax (distance between red and cyan (anaglyph)):

$$P = \tan a * D$$



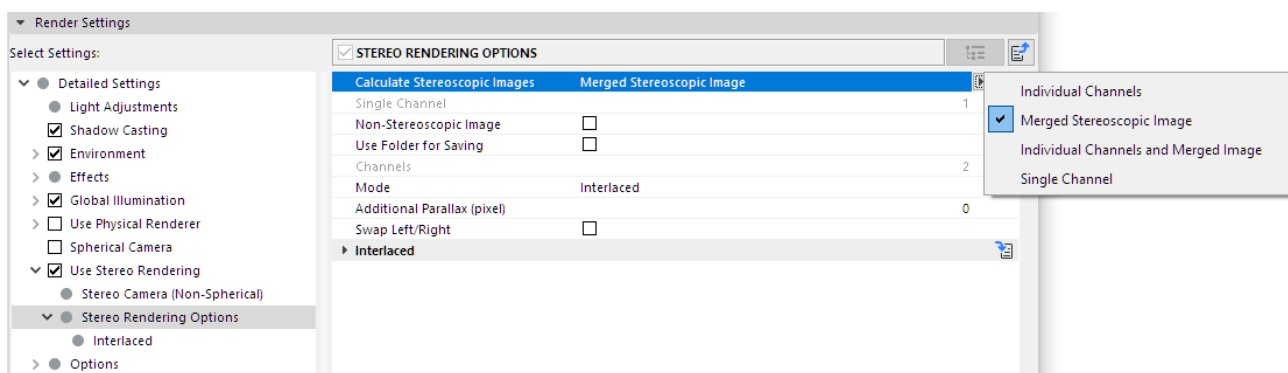
where P = Parallax, D = Distance of the viewer from the projection plane (e.g., the monitor), a = Angle between 2 points that the eye can comfortably perceive (this should be max. 1.5° or somewhat less).

For an average distance between eyes and monitor of 50 cm, an average reference value of 13 mm results.

Stereo Rendering Options

Calculate Stereoscopic Images

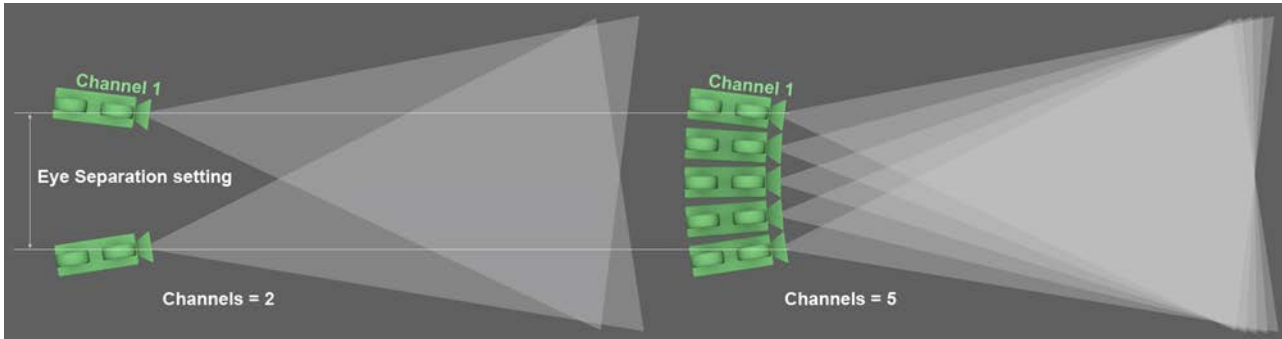
Define how the stereoscopic images should be rendered and saved.



- **Merged Stereoscopic Image:** A stereoscopic image will be rendered using the left and right eye views only and no other channels. This is the mode with which you will normally render.

In contrast to stereoscopic techniques that use 2 channels, multi-channel techniques let you render multiple camera views, which can then be edited as channels (or streams) using external applications. Auto-stereoscopic playback devices can then be used to view these images with two matching channels (which can change depending on the angle of view).

- **Individual Channels:** Several camera views can be rendered, depending on the number of channels defined. Channel 1 will always be the left eye perspective and channel X will always be the right eye perspective. If Channels is set to greater than 2 additional views will be rendered between these views. Select this mode if you want to subsequently create a stereoscopic image, either in the Picture Viewer or using an external application.



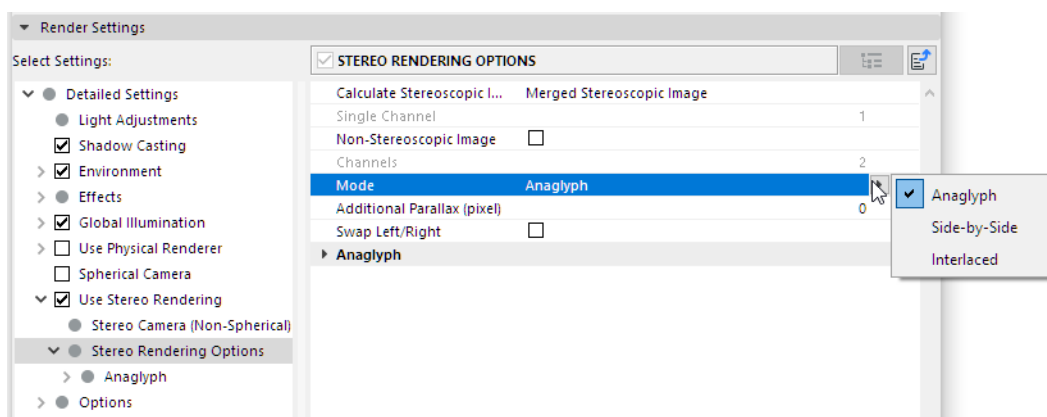
At left a stereoscopic setup with 2 channels, at right a stereoscopic setup with 5 channels.

- **Individual Channels and Merged Image:** In addition to the left and right eye views (or any number of additional intermediate camera views), a combined, stereoscopic image will be created out of a combination of these views.
- **Single Channel:** Only the channel defined by the Single Channel value will be calculated. This mode is recommended if you – for whatever reason – only want to render a single camera view. Define which channel should be rendered. 1 will always render the left eye view. The value defined for Channels will be the right eye view. in-between values will represent the intermediate camera views, as described here: [see Placement](#).

Non-Stereoscopic Image

Enable this option if the normal camera view should be calculated in addition to the stereoscopic views.

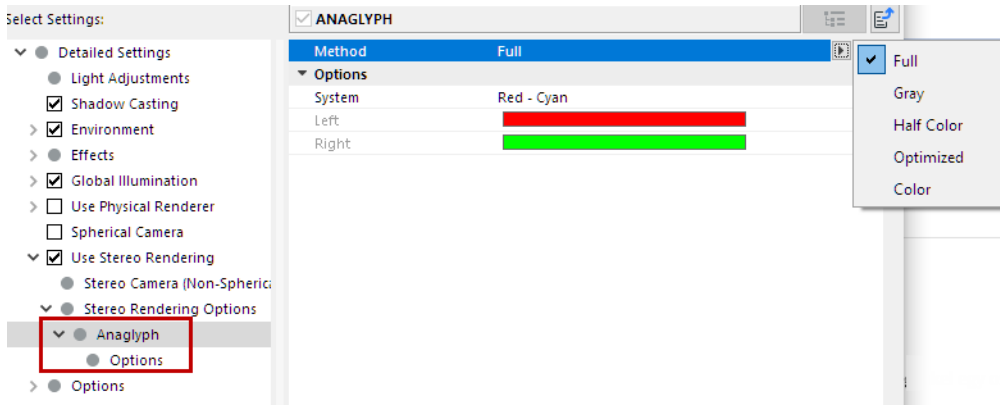
Stereo Rendering Mode: Anaglyph, Side-by-Side, Interlaced



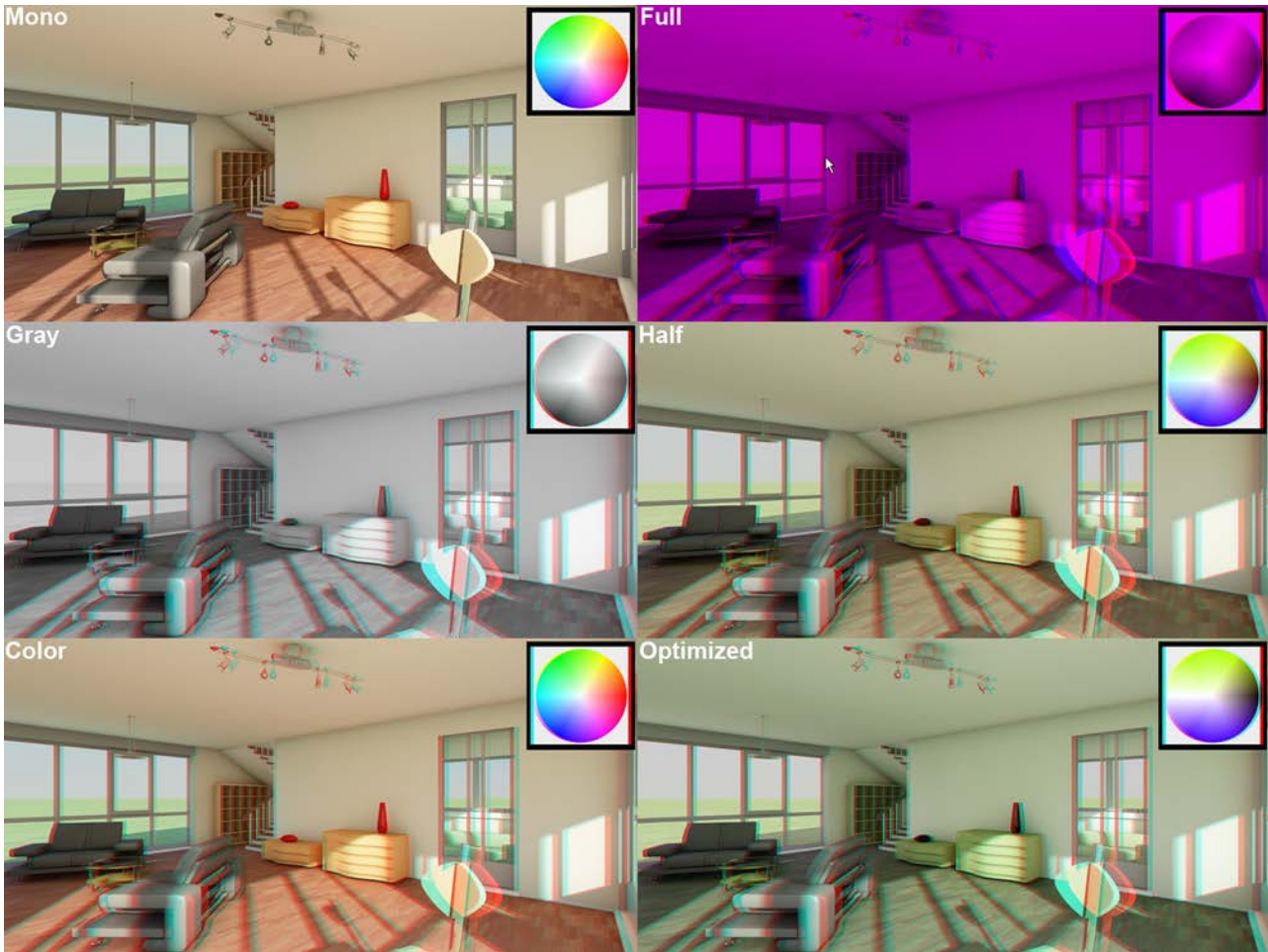
Anaglyph

This is the most well known method, which has been used in movie theaters since the 50s. An image's color information is separated using 2-color glasses (previously red-green, today most commonly red-cyan).

Advantage: simple, affordable glasses. Disadvantage: color range is in part very limited.



- Method:** Set the color of the stereoscopic image. A problem with the anaglyph technique is that some colors cannot be displayed without straining the viewer’s eyes (red when using red-cyan coding). Recommendation: Use **Optimized**, because it offers the least strenuous “viewing experience”.

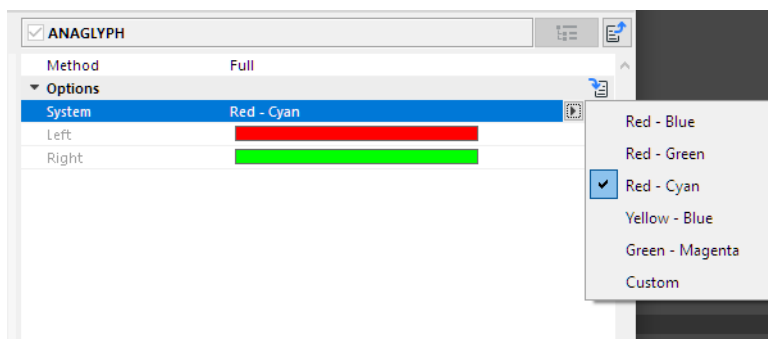


The different available methods (Full with anaglyph colors red-blue). “Mono” = Stereo Rendering OFF. Models by DOSCH Design.

The following list is arranged according to the quality that can be expected from anaglyph images, from worst to best:

- Full:** The oldest (and lowest quality) method of anaglyph display; the display is dark and monotone. This mode is designed for use with the Red-Blue or Red-Green anaglyph techniques.

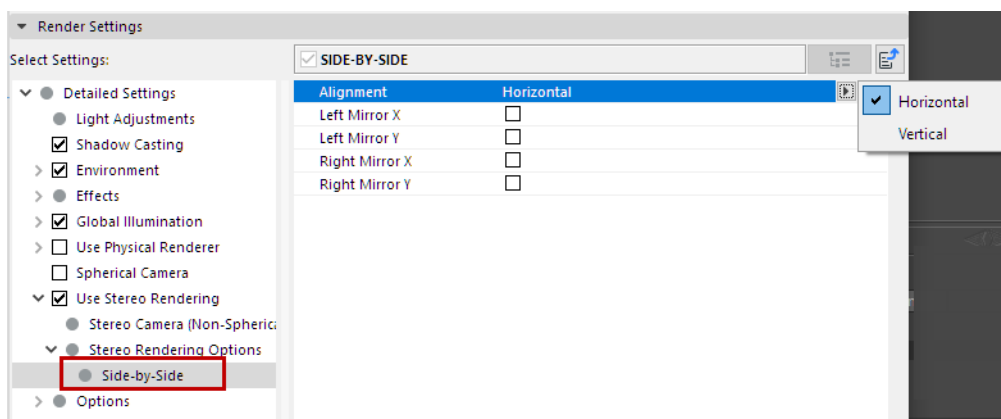
- **Gray:** The anaglyph image will appear as a gray-scale image through the glasses (not designed for use with red-blue or red-green). For brighter images use Full.
- **Half Color** and **Color:** These modes allow only a limited color reproduction compared to the other options. Blue, green and yellow tones can be reproduced very well when the common red-cyan code is applied. If Color is selected, “retinal rivalry” can occur, i.e., red surfaces (red-cyan) will cause the left eye to pass on maximum color intensity to the brain and the right eye will only see “black”. This is irritating and strenuous for the eyes. This effect can be minimized by selecting **Half Color**. However, red will then be darkened to such a degree that it can no longer be recognized as such.
- **Optimized:** This mode is similar to Half Color, but offers better color reproduction and minimizes the retinal rivalry effect.
- **System:** Define the color of the stereoscopic color coding here when using Anaglyph mode. Both colors should be the same as the lenses of the 3D anaglyph glasses you will be using. If your client does not supply you with color information, use Red-Cyan.



- Use **Custom** option to define the individual colors for the stereoscopic color coding (however, it will then be difficult to find a matching pair of glasses...). If Method is set to anything but Full, you will only be able to define the left eye color. The left color must be the same as the color of the left lens of the glasses. The right eye color will automatically be set to the left eye’s complimentary color.

Side-by-Side

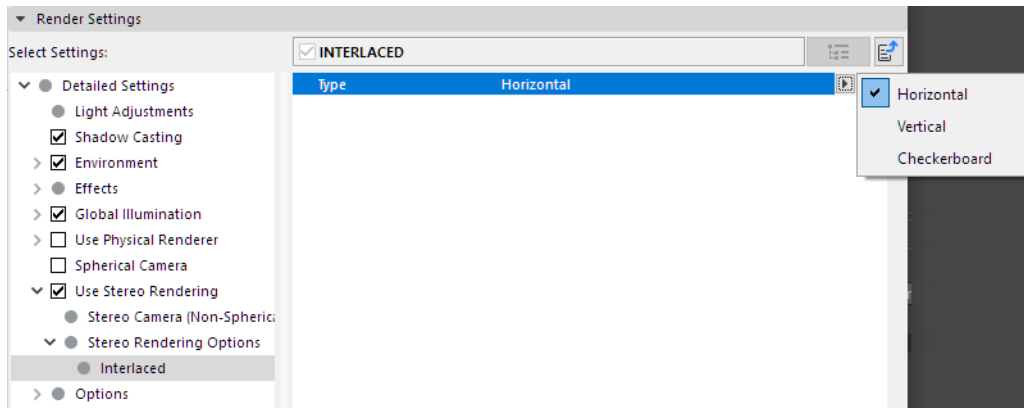
Left and right images are switched and squeezed into a normal image size. Some televisions use this technique for HD 3D because the transmission band width is the same as the HD band width. The terminal device must be able to decode these double images and display them sequentially (most commonly in combination with shutter glasses). Disadvantage: reduced resolution, expensive technology (special hardware required).



- **Alignment:** This mode defines whether or not both image parts should be arranged next to each other (horizontally or vertically).
- **Left Mirror X/Left Mirror Y/Right Mirror X/Right Mirror Y:** Use these checkbox options to mirror the image halves along the X or Y axes.

Interlaced

This method requires a monitor with a polarization filter and glasses with polarized glass. Since both images are coded into a single image (e.g., left eye all even, right eye all uneven lines), the resolution is halved. Advantage: affordable glasses, good color reproduction. Disadvantage: special monitor required, reduced resolution.



Define whether the coding should take place via offset lines (**Horizontal**) or columns (**Vertical**).

Checkerboard is a combination of both modes.

Additional Parallax (pixel)

Moves the image halves by the defined value in pixels. This can be used to increase the stereoscopic effect.

Swap Left/Right

If enabled, the left and right image halves will be swapped.

A few guidelines for achieving good stereoscopic images

There are several rules that should be followed when creating stereoscopic images. This is necessary so they can be easily viewed without any unnatural effects or fatigue for the viewer's eyes. Hence, the following guidelines should be followed.

- **Depth of Field:** It is generally suggested that a larger depth of field (i.e. little blur) be used. A common 2D technique is to take the background out of focus using a slight depth of field. A blurred wall behind a sharp object makes this surface look flat. Such techniques are in contrast to stereoscopic techniques.
- **Distance from object(s):** The 3D effect depends a lot on how far the viewer is from the projection plane (monitor, screen, paper, etc.). The farther away the viewer is from the projection plane, the stronger the 3D effect will be (the perceived impression of depth between an object that is near and one that is far away)! This should be taken into consideration when creating a stereoscopic zone.
- **Ghosting** is when one eye perceives the image information of the other eye (leads to irritation). This is especially noticeable in images with high contrast (anaglyph images in particular are prone to this). Therefore, you should avoid such contrasts in your images if at all possible. Very small parallax values can also reduce ghosting.

- It is often irritating for the eye when objects are cut off at the edge of an image (if they do not lie on the projection plane). However, because there are always objects that lie on the edge of an image, it shouldn't necessarily be the most important object in the image, the one that is currently the center of attention, that is being cut off.
- For videos it is important to give the eye time to adjust to strong parallax changes (between different settings). Short, fast cuts should then be avoided.
- Avoid exaggeration: Chain saws constantly flying towards the viewer or similar scenes can be fatiguing for the eye. Such visual gimmicks should be used sparingly or be well thought out.

Using Redshift Renderer

The new Redshift renderer (GPU based) from Maxon, based on Maxon's R23 engine, is included in Archicad as a Technology Preview, with limitations.

As a GPU-based renderer, Redshift brings significant benefits in terms of performance compared to Cinerender.

At the current stage of development, separate Redshift parameters have not been implemented in Archicad. Instead, Redshift makes use of the corresponding Cinerender parameters where applicable.

Requirements for using Redshift

As of Archicad 25 (Update 2) and Archicad 26: Redshift is a Technology Preview feature available for users with an SSA/Forward agreement.

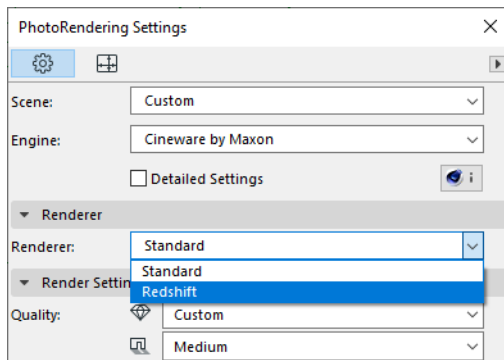
For system requirements, see this link:

<https://www.maxon.net/en/requirements/redshift-requirements>

A high-end graphics card with 8GB VRAM or more is required.

Activate Redshift in Archicad

1. Go to **Document > Creative Imaging > Photorendering Settings**.
2. Set the Engine to "Cineware by Maxon"
3. Set the Renderer to "Redshift"



Related Topics:

[Redshift Renderer: Current limitations](#)

Redshift Renderer: Current limitations

At the current stage of development, you will experience certain limitations when using Redshift to render in Archicad (version 25 and 26), as compared with Cineware. Please check the following notes and recommendations.

Surface channels

Notes on surfaces: General issues

Notes on surfaces: Default catalog

Shaders

Light sources and objects

Basic Rendering settings

Detailed Rendering settings

Other known limitations

Surface channels

Blending among layers

When using multiple layers, try to set them up as simply as possible. More complicated layer combinations - even with blending mode set - might result in unexpected texture display.

Bump, Reflectance and Transparency

For many surfaces, the Bump, Reflectance and Transparency channels should be turned off or used with low values (1-15)

Alpha

On Mac only: You may encounter problems with alpha and alpha transparency

Other surface channels

Redshift renderer does not support the following surface channels:

- Diffusion
- Environment
- Fog
- Glow
- Displacement
- Grass
- Illumination

Other channels only work in a simplified way (not all additional parameters work)

Notes on surfaces: General issues

Brick

- Rendering is slower
- Bump channel should be turned off or used with low values (1-15)

Concrete

- Diffusion
- Bump channel should be turned off or used with low values (1-15)

- Layering/foldering issues

Glass

- Reflectance and transparency issues
- Bump overrides transparency

Metal

- Bump channel should be turned off
- Reflectance should be turned off or minimized
- On Mac: You may encounter problems with alpha and alpha transparency

Water

- Bump channel should be turned off
- Reflectance should be turned off or minimized

Notes on surfaces: Default catalog

For the default basic textures, be aware of the following:

Metal

Reflectance should be turned off or minimized

Roof

Reflectance color layer overrides color channel texture

Brick and Mortar

Bump, Parallax and Displacement issues

Air

Color layer in the alpha channel doesn't work; the texture is completely invisible in the rendering.

Foliage - Leaves Tree Small and Foliage - Wild grapes

The sun highlights the texture, so the Reflectance channel in Surface Settings should be turned off.

Concrete 02

Rendering with the Low quality preset will result in inferior textures as compared to Cineware. (Higher quality gives better texture backing.)

Insulation - Fiberglass

Alpha channel in Surface Settings should be turned off.

Metal - Nickel

The texture becomes black in the render. To avoid this, Reflectance channel should be turned off. (Other metals may also produce odd results due to Alpha and Reflectance layers).

Pavement - Brick Moss

Texture can be fine-tuned by turning off the first Noise layer in the Color channel or by modifying the Noise layer's Blend mode and Blend strength values.

Water - Wavy and Water - Pond

Reflectance channel should be turned off.

Glass - Lamp

Does not illuminate nearby surfaces.

Shaders

Keep in mind the following shader-specific limitations.

Fresnel

Will override the surface texture

Gradient

With 3D types, the texture is misaligned

Colorizer, Filter, Fusion, Layer, Posterizer

These texture options build on other textures, so there may be inconsistencies

Shader effects and Sketch

- Most of them work, but with inconsistencies
- Variation effect does not work at all

Procedural shaders

Textures may be misaligned

Light sources and objects

Limitations related to light objects in renderings.

Area Light

Works only with Rectangle and Disc shapes in Redshift

Parallel Light

There is no shadow behind the light source in Redshift

Window Light

- The light source does not illuminate nearby surfaces without Global Illumination
- Light is very bright

Sun Object

Differs from Cineware (can be too bright)

Light Effects

Not supported

Basic Rendering settings

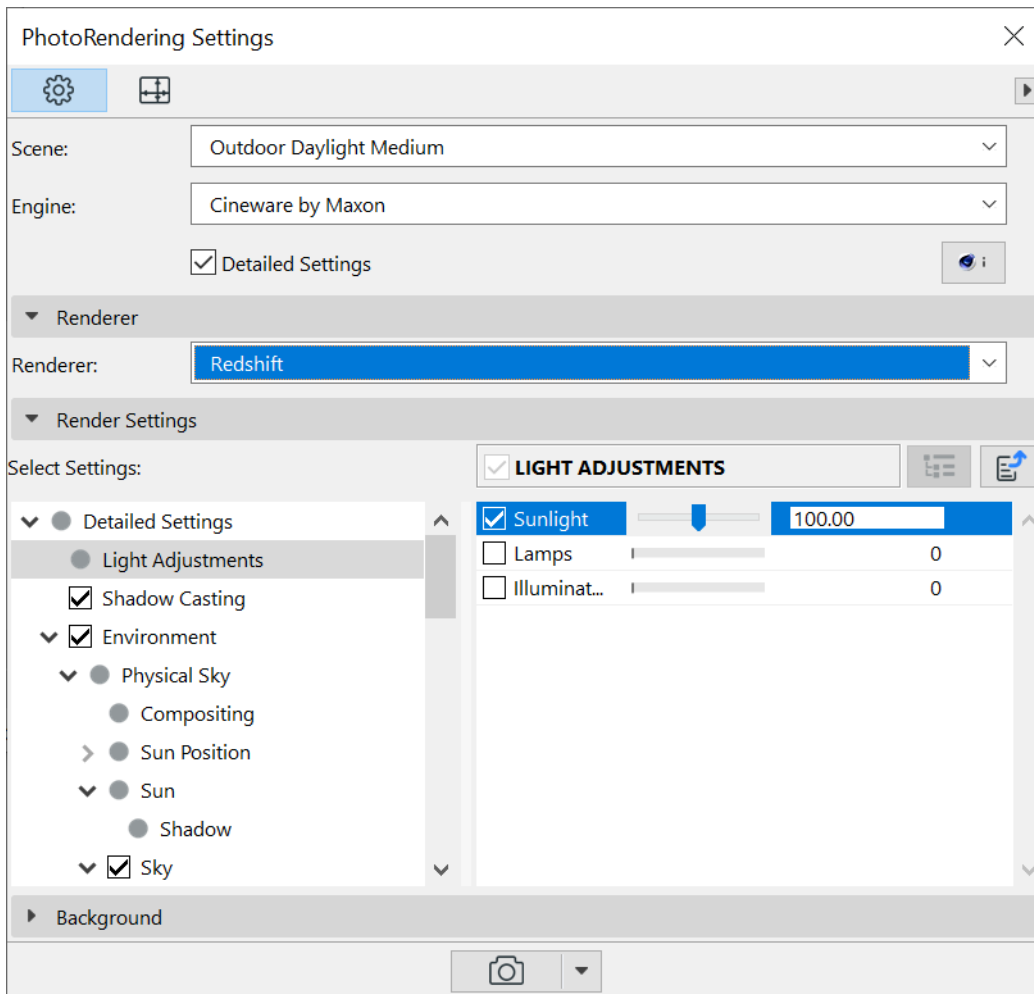
Shadow quality

Doesn't work due to lack of Shadow Casting function.

Detailed Rendering settings

By design: several "Detailed Settings" parameters do not appear on the Archicad interface, because they are not supported. See notes below.

Defaults have been set to be as useful as possible.



Photographic Exposure

Use these camera settings to easily make interior/exterior renders darker or lighter.

Shader Baking

Adjusting this setting can improve texture quality if needed. Shader Baking affects textures even when using Sky.

Denosing

Denosing effects may differ depending on Win vs. Mac, as well as the video card used. Optix is available only for Nvidia (Win)

Global Illumination

The default setting is Irradiance Cache and Irradiance Point Cloud. Leave these as-is for faster renders. For final-quality renders, Brute Force is preferable, but be prepared for long rendering time.

Options - removed

Grass and Antialiasing removed

General Options removed

Shadow Casting - removed

Environment - some parameters removed

The following parameters are removed:

- Sun Shadow: Color, Transparency, Accuracy;
- Sky: Intensity, Ozone;
- Atmosphere
- Fog
- Rainbow
- Sunbeams

Effects

The following effects or parameters are removed:

- Caustics
- Cel Renderer
- Color Mapping
- Color Correction
- Depth of Field
- Vignetting
- Monochrome model (works only with White Model)
- Distance Fog
- Lens and Filters
- Spherical Camera
- Use Stereo Rendering

Other known limitations

- Parallel rendering does not work: Redshift cannot render two or more images at the same time
- HDRi sky is dark. Additional sunlight should be used in this case

Tool Settings Dialog Boxes

This section of the User Interface Reference describes each of the Tool Settings dialog boxes, for most tools found in the Archicad Toolbox.

The descriptions of individual tools are prefaced by the first part (**Working in Tool Settings Dialog Boxes**), which contains general information, plus information on controls that are common to every Tool Settings dialog box.

Working in Tool Settings Dialog Boxes

Wall Tool Settings

Roof Tool Settings

Shell Tool Settings

Slab Tool Settings

Mesh Tool Settings

Zone Tool Settings

Morph Tool Settings

Settings of Library Part Elements

Object Tool Settings

Special Lamp Parameters

Door/Window Tool Settings

Corner Windows

Custom Settings Panel for a Door/Window

Skylight Tool Settings

Dimension Tool Settings

Dimension Text Settings

Text Content

Convert Dimensions Dialog Box (Add-on)

Text Tool Settings

Text Style

Label Tool Settings

Drawing Settings

Fill Tool Settings

Line-type Tool Settings

Section and Elevation Settings

Interior Elevation (IE) Tool Settings

Worksheet and Detail Tool Settings

Change Tool Settings

Grid Tool Settings

Wall End Tool Settings

Spline Tool Settings

Hotspot Tool Settings

Figure Tool Settings

Camera Tool Settings

Settings of other tools are described elsewhere in this Reference Guide:

Columns

Beams

Curtain Wall

Stairs

Railings

Openings

Working in Tool Settings Dialog Boxes

Tool settings dialog boxes can be accessed from the Toolbox, the Info Box palette, or from context menus. They contain the settings that define the appearance and parameters of the elements drawn using the tools.

To open a Tool Settings dialog box, do one of the following:

- Double-click the tool icon in the **Toolbox**.
- Click the tool icon in the **Info Box**.
- With an element selected, choose **Edit > Element Settings > (Tool) Settings**.
- Click an element with the right mouse button (**Mac: Ctrl key**), and choose the **(Tool) Selection Settings** command from the appearing Context menu.

The title bar of the dialog box displays whether you are viewing **Default** settings or those of **Selected** elements.

The information at right of a tool settings dialog box alerts you to the status of the current settings in the box:

- **Default:** These settings will be used as the default for new elements.
- **Selected:** These settings are applied to the currently selected elements (the number of selected elements is displayed).
- **Editable:** Displays the number of selected elements which are also editable.

Favorites Button in Tool Settings

The **Favorites** (star) button in the top left corner of all Tool Settings dialog boxes (except Camera) opens a list of the Favorites stored for that tool only.

Archicad comes with predefined Favorite settings for each tool, and you can define your own Favorites.

For more information, see [Favorites](#).

See also [Search Favorites](#).

Panel Visibility in Tool Settings

The settings for each tool are organized into several panels that can be opened or closed individually by clicking the caption of the panel. You can also choose to show or hide any of the panels, if you prefer, at:

Options > Work Environment > Tool Settings Dialog Boxes.

See [Tool Settings Dialog Box Customization](#).

When you open new panels, some visible panels may shut automatically, and/or the dialog box will move upward on the screen to prevent an overcrowded or too-large dialog box. To turn off these automatic functions, set the checkboxes in **Options > Work Environment > Dialog Boxes and Palettes**.

For details, see [Dialog Boxes and Palettes](#).

Navigating Within Tool Settings Panels

A few keyboard shortcuts make it easy to navigate in any of the Tool Settings dialog boxes:

- To open and close the unfoldable settings groups within a particular panel, use the right- and left-arrow keys.
- To move from one setting field to the next within a particular panel, use the up- and down-arrow keys.
- As you move onto the next field, the edit field is already highlighted, so you can type in values directly.
- To activate/deactivate a checkbox in the current field, use the Spacebar.

- To open the pop-up menu indicated by an arrow icon (such as a Fill Type or Pen Color chooser), press Alt + Down Arrow. Press ESC to close.
- To exit a settings list within a particular panel, press TAB.
- To jump to the next panel or control, press TAB. To jump to the preview panel/control, use Shift+TAB.

Layer Definition in Tool Settings

At the bottom of each Tool Settings dialog box, the current **Layer** choice is displayed in a drop-down list. To change the current element's layer, choose a different layer in this pop-up menu.

For more information, see [Layers](#).

Listing Panels

This panel controls the handling of the current Tool in calculations. It is not visible by default. Use the Work Environment's Tool Settings Dialog Boxes section to activate this panel for any of the design and documenting Tools.

For more information on the Calculation function, see [Calculation \(Legacy\) and the Archicad Calculation Guide.pdf](#).

- **Link Property Objects by Criteria:** Use this control to link a Property Object to the element for calculations.
- **Individually:** Check this box to assign additional information, such as strengthened reinforcement, to selected items, or to create an assignment outside of the norm of the criteria; the **Choose** button becomes active. Use this to link a single Property Object to this element. (To link multiple Property Objects, use the **Link Property Objects by Criteria** option.)

Classifications and Properties Panel

See [Properties and Classification Systems](#).

ID and Categories

- **ID:** Type an ID in this box for tracking this element in Quantity Calculations.

The ID field serves to identify and group elements in list views. Any character can be used.

If a number is included anywhere in the ID, Archicad will add one to this number for each new element drawn, provided that the **Assign New Element ID to Each New Element** checkbox is enabled in the **Options > Work Environment > More Options** dialog box. Each new element will have a unique ID.

If elements are duplicated or multiplied, the IDs of the replicas will remain the same as those of the originals.

If you paste elements into a Project, you may have elements with conflicting IDs. Archicad does not automatically exclude ID conflicts. It is up to the user to designate different IDs for elements that may be in conflict.

IDs can be customized both before creating elements and after selecting a number of existing ones, by using **Document > Listing Extras > Element ID Manager**.

For more information, see [Element ID Manager](#).

Note: All construction elements also have a globally unique, automatically generated ID which is conserved throughout the life of the Project. You can use this identifier for labeling or in lists.

- **Structural Function:** Optionally, define the element as either a "Load-Bearing Element" or a "Non-Load-Bearing Element" part of the building.

If you have classified model elements as “Load-Bearing”, then you can use the Partial Structure Display “Core of Load-Bearing Elements Only” option to display those elements only.

See [Partial Structure Display](#).

If you collaborate with partners who are using structural programs and export your Archicad model to IFC format, the Structural Function classification automatically adds the “load-bearing” property to structural elements in the export file.

See [Structural Function Category for IFC](#).

- **Position:** Optionally, define the element as either an “Exterior” or an “Interior” part of the building. This is useful to streamline coordination with energy analysis applications.

You can search for elements by their Position classification, using the criteria of the Find & Select dialog box

See [Position Category for IFC](#).

- **Classification by DIN:** Set this value in the Classification and Properties section of Morph Settings. Available only in Morph Settings, and only if you are using the Flächenberechnung (Area List) Add-on (used primarily in German-speaking markets).

Renovation Controls

Renovation Status: This control is available only for a selected element. (In Default Settings, the note “Use palette to set default” indicates that the Renovation Palette can be used to set a default Renovation Status for all elements.)

For a selected element, use this pop-up to assign a Renovation Status: Existing, New, or To be Demolished.

Show on Renovation Filter: By default, this option, for a selected element, is set to “All Relevant Filters.” That is, this element will be shown, hidden, or shown with an override, depending on its Renovation Status and on the Renovation Filter in effect. However, you can restrict the display of any selected construction element to one Renovation Filter only: use this Show on Renovation Filter control to choose one particular Renovation Filter; this element will exist in that Renovation Filter only.

For more information, see [Renovation](#).

Element Properties

For more information, see [Properties and Classification Systems](#).

IFC Properties

This part of the Element Settings dialog box displays IFC Properties and Element Type Properties assigned to the selected Archicad element. These property and attribute assignments help to optimize your Archicad model for data exchange with other applications.

For details on Interoperability using IFC, see [Working with IFC](#).

Wall Tool Settings

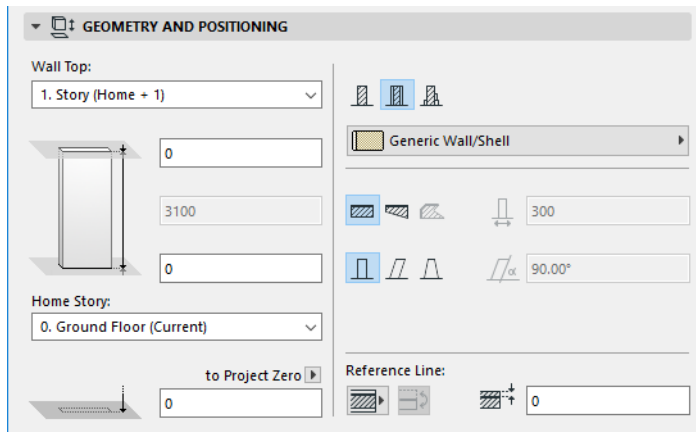
For a description of generic settings common to all tools in the Toolbox, see [Working in Tool Settings Dialog Boxes](#).

For general information, see [Walls](#).

For information on the Structural Analytical Parameters Panel of Wall Tool Settings:

See [2D Members \(Slab, Wall, Roof\)](#).

Wall Geometry and Positioning Panel



By default:

- Wall top is linked to one story up
- Home Story is the current story.

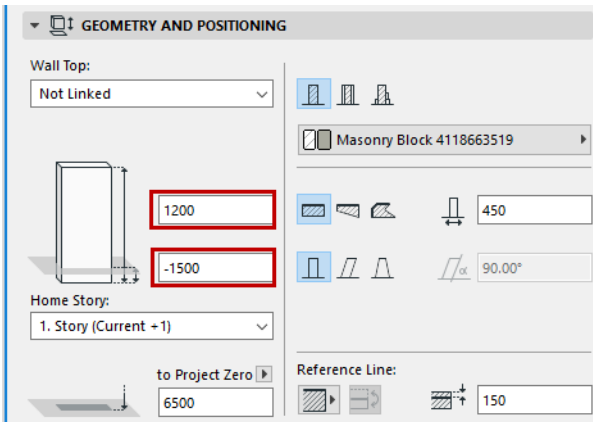
The story to which the Wall bottom is linked is called the Home Story, but you can offset the bottom in either direction.

Use the controls in this panel to change any of these values, and to define the geometry and complexity of the Wall element.

Wall top: Use this control to either top link the Wall relative to its Home Story (Home+1, Home+2, etc.), or to make it a fixed-height wall (choose “Not linked”).

If you later modify the positions and heights of stories in your projects, the heights of any linked Walls will follow automatically.

- Optionally, define an **offset** for the Wall top from its top linked story (the Wall height changes accordingly). This offset value can be positive, negative or zero.
The offset field is not available if the Wall has no top link.



Wall height: Enter a value for the total height of the Wall. (For a top linked Wall, this field is not editable.)

- The Wall height value is affected by the top and bottom offsets, if any.

Home Story

See [Home Story](#).

Bottom Elevation [to Reference Level]

Calculates the current elevation of the Wall's reference line, as measured from the Reference level (by default, this Reference level is Project Zero).

Click the pop-up arrow to change the Reference level, if needed.

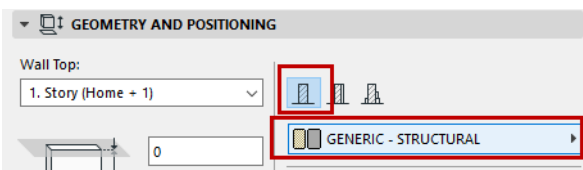
Notes:

- Reference levels are defined at **Options > Project Preferences > Reference Levels**.
- This is a calculated value, not a link. If you change the Reference level, the bottom of the Wall will not change its position.

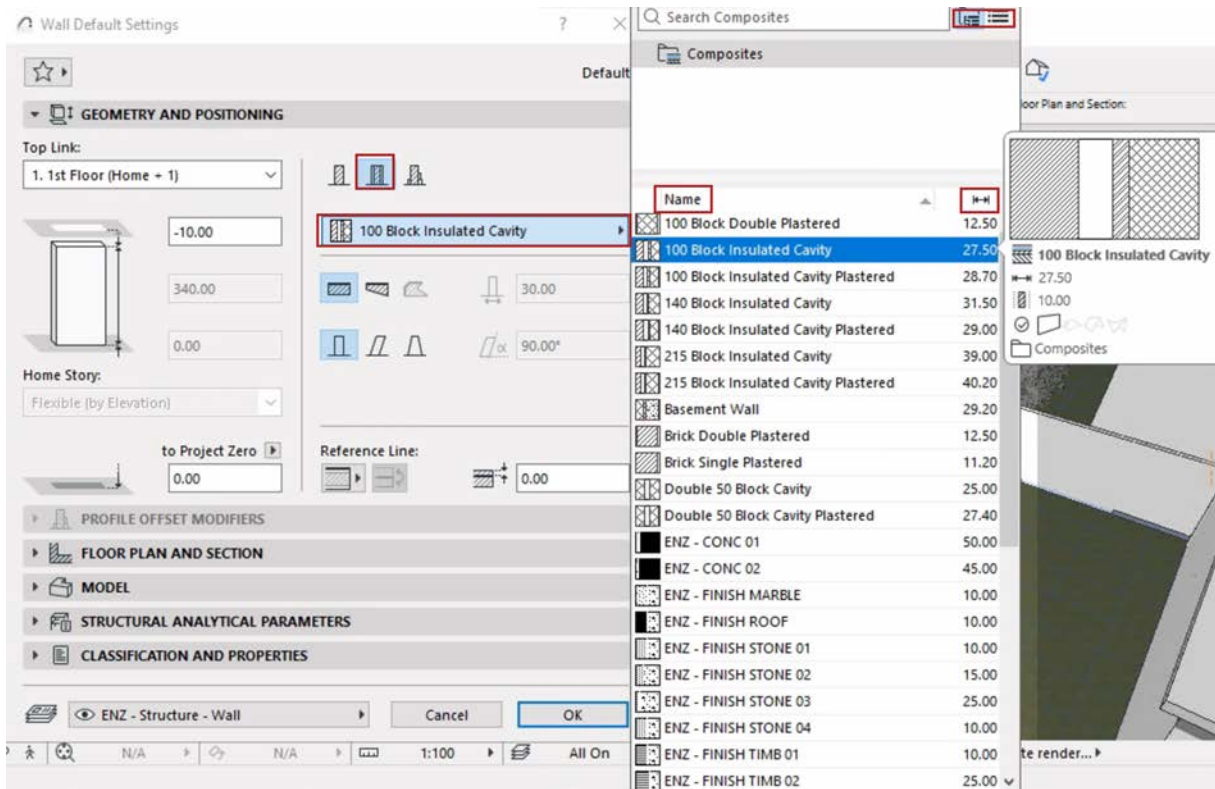
Wall Structure

Click one of these icons to define the Wall as **Basic**, **Composite**, or **Complex Profile**, and then choose an attribute that reflects its structural properties:

- For a **Basic** Wall: use the pop-up to choose its Building Material.

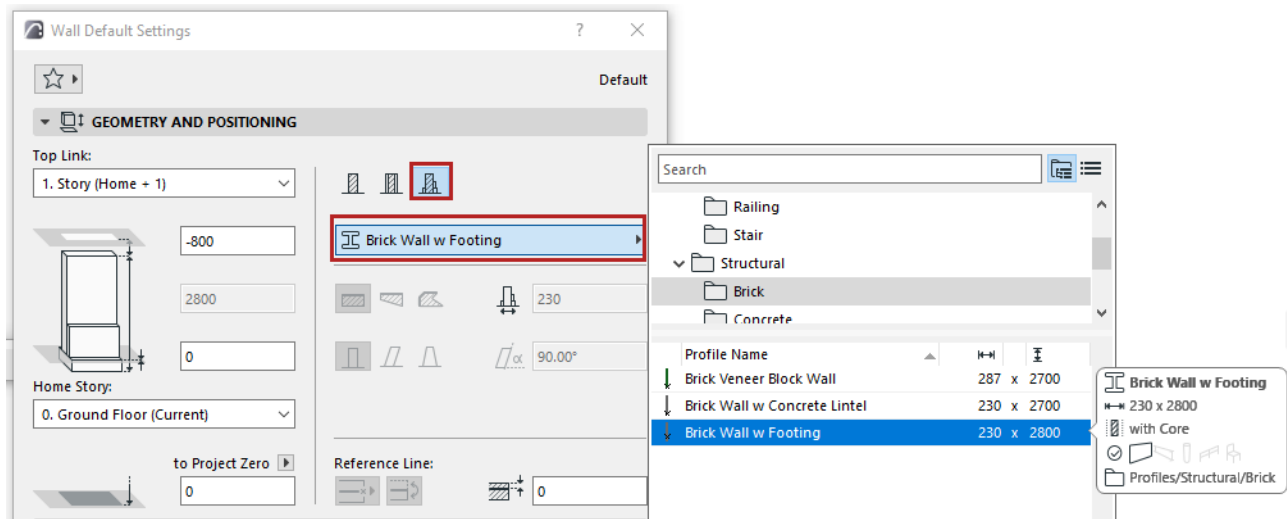


- For a **Composite** Wall: use the pop-up to choose its Composite structure.



Note: This pop-up lists composites whose “use with” setting includes the wall element: see [“Use With” - define composite availability](#)

- For a **Complex Profile** Wall: use the pop-up to choose its Profile.

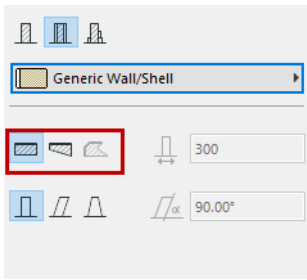


Note: This pop-up shows only those Profiles whose “Use With” control includes Walls. Profiles are defined at Options > Element Attributes > Profile Manager.

Geometry Method

Click one of these three icons to draw the Wall as either straight, trapezoidal or polygonal.

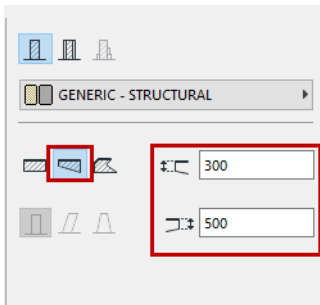
- These options are not available for Complex Profiles.
- The polygonal option is not available for composite Walls.



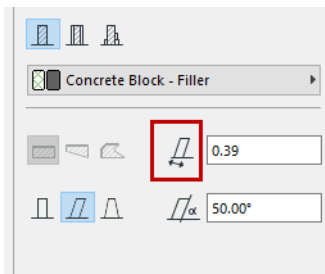
Wall Thickness

Enter a value for the wall thickness.

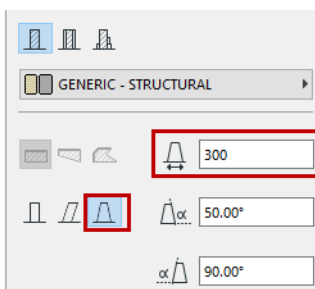
- If you are creating a trapezoidal Wall, two thickness fields are active: enter a thickness value for each end of the Wall.



- The thickness of a composite Wall is the total value of the individual skins as set in **Options > Element Attributes > Composites**. If you are creating a composite Wall, the thickness field is consequently not editable here.
- Thickness does not apply to polygonal Walls; the field cannot be edited.
- The thickness of a slanted Wall is its thickness perpendicular to its slant vector:



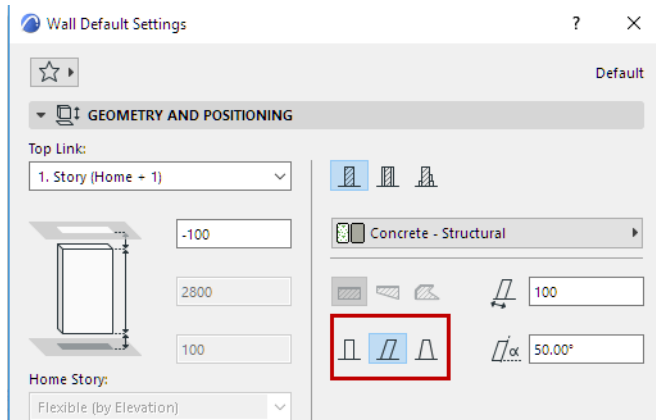
- The thickness of a double-slanted Wall equals the width of the Wall base.



Wall Complexity

The Wall Complexity icons give you a choice of Wall geometry: Straight, Slanted, or Double-Slanted. (*Wall Complexity controls are not applicable and not available for Trapezoid Walls and Polygonal Walls.*)

- For a slanted or double-slanted Wall, enter the value for the slant(s) in degrees.



For more information, see [Create a Slanted or Double-Slanted Wall](#).

PolyWall Corners Can Change

This checkbox is only visible when the chosen geometry method is Polygonal Wall. It controls the display of wall contours when connecting such a wall to other, plain walls.

For more information, see [Create a Polygon Wall \(PolyWall\)](#).

Reference Line Location

Click an option to define the Wall's reference line.

For more information, see [Wall Reference Line](#).

Wall Floor Plan and Section Panel

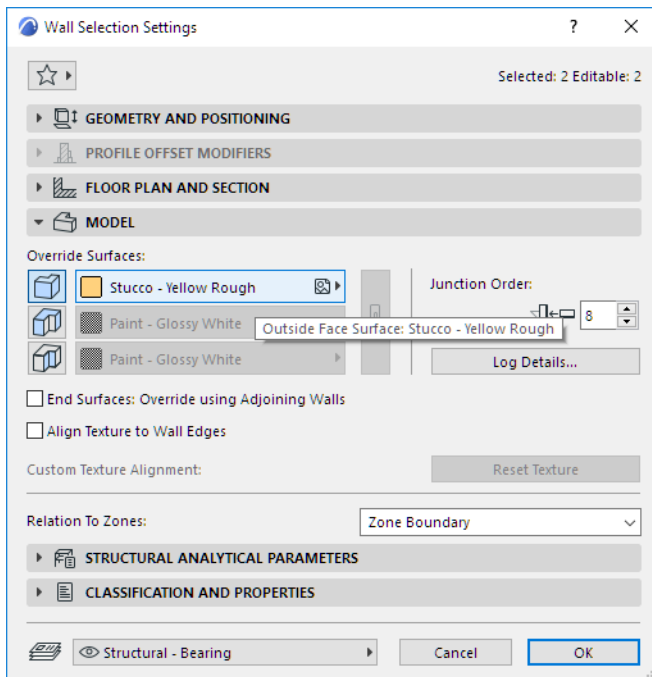
See [Element Display in Floor Plan and Section](#)

Wall End Lines: This pop-up gives you four ways to display a freestanding wall on the Floor Plan. The options let you show or hide one or both end lines of the wall. (The end line is the contour line perpendicular to the longitudinal axis of the wall.)

Wall Model Panel

This panel affects the 3D display of the wall.

Override Surfaces: Click the toggle button of any or all three pop-ups if you wish to override the surface assigned to this element (in its Building Material). You can override the surfaces of the Wall's faces (outside and inside), and/or all Wall edges.



See also [Override Surfaces With Surface Painter](#).

For information on assigning surfaces to a Polygon wall, see [Create a Polygon Wall \(PolyWall\)](#).

- For a composite Wall, the “outside” surface is the first listed skin of the composite, in Composite Settings.
- For a profile Wall, there are two surface override possibilities: Extrusion edge and Ends surface.

Note: You can also apply a Custom surface to any edge(s) of the profile element.

See [Apply Custom Surface or Line to Profile Edge](#).

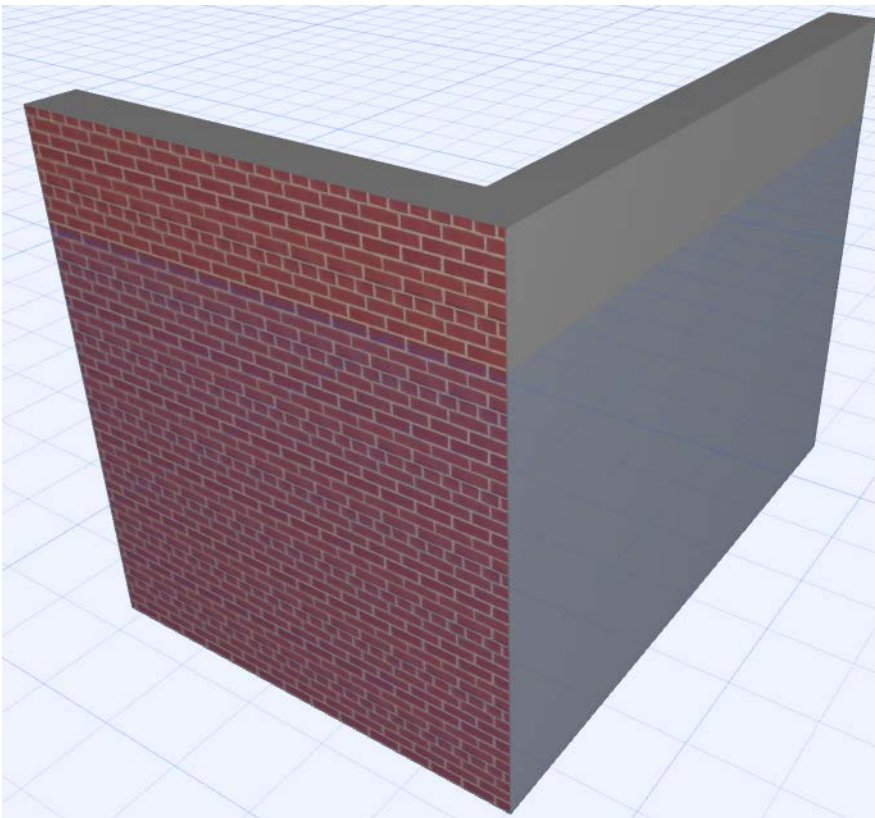
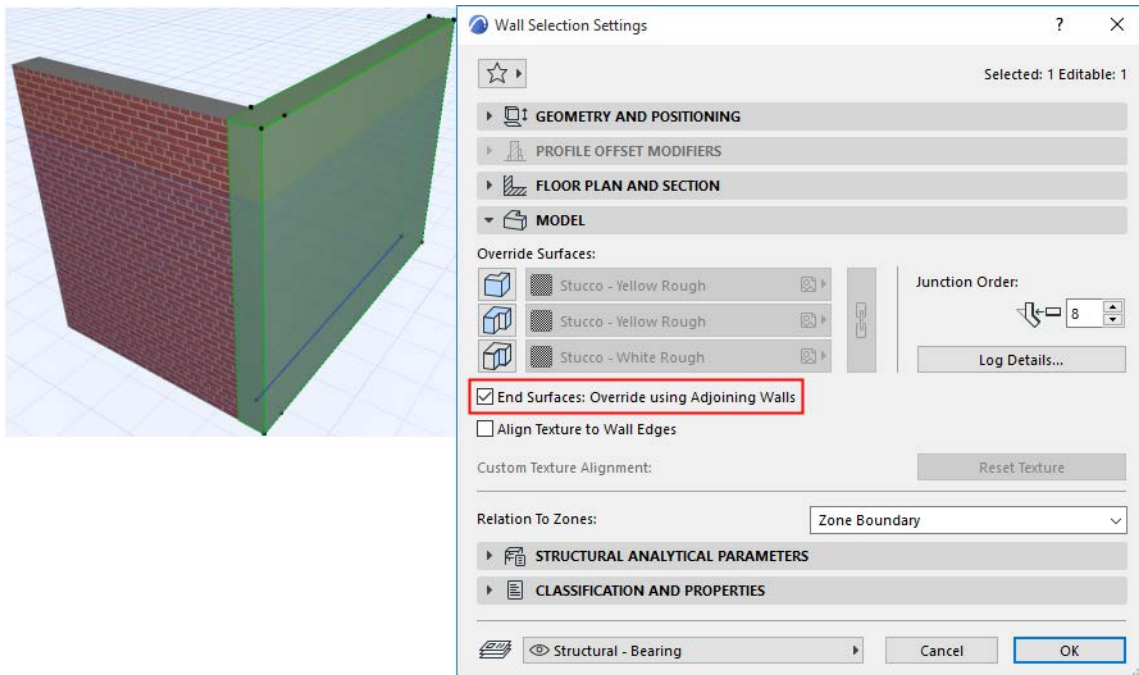
Click the **chain** icon to assign the last surface chosen to all faces and edges. Deselecting the chain icon will restore the surfaces that you originally set for each wall face/edge.

Align Texture to Wall Edges: Check this box to place the texture origin at the bottom corner of the wall. This can be useful, for example, on freestanding walls to avoid a fragmented texture on the edges.

See also [Align 3D Texture](#).

End Surfaces: Override using Adjoining Walls: Check this box if you want the selected Wall’s end surface to take on the surface of the adjoining Wall.

Note: Any adjustments to the surface texture alignment will be correctly reflected on both walls.



Custom Texture Alignment: If this message is activated, it indicates that the currently selected wall (already placed in the project) has been assigned a custom 3D texture.

- In this case, the **Reset Texture** button is also activated. Click to restore the origin of the Texture of the selected wall.

Note: The **Custom Texture Alignment** text serves as feedback on the condition of the selected wall. If it is greyed out, no custom 3D texture has been assigned yet.

For more information, see [Align 3D Texture](#).

Junction Order

Junction Order is relevant when

- two Walls meet whose Building Materials have the same priority
- 3 or more Walls meet in a junction

A higher Junction Order number takes priority over the lower.

When 3 or more Walls meet in a junction, the relative Junction Order of each Wall determines which two Walls are joined first.

Log Details Dialog Box

The **Log Details** button opens a dialog box for defining the display of walls constructed from logs:

- **Use Log Wall:** Check this box to use a Log Wall.
Note: Archicad calculates the number of logs from wall and log heights.
- **Start with half log:** Check to start the wall with a half-height log.
- Log Shape:
 - **Square logs**
 - **Rounded on Outside Face**
 - **Rounded on Inside Face: Use logs that are cambered on the inside. The outside wall face will remain flat.**
 - **Rounded on Both Sides.**
- **Log Radius:** choose an option for measuring log radius: either from the log centerpoint, or from the middle point of the opposite log.
- Surface of Horizontal Edges:
 - **As in Wall Settings:** Click this radio button to apply the Building Material's surface.
 - **Override with Outside/Inside Face Surface**

Relation to Zones

See these topics:

[Calculating Zone Area and Zone Volume](#)

[Relation to Zones](#)

Roof Tool Settings

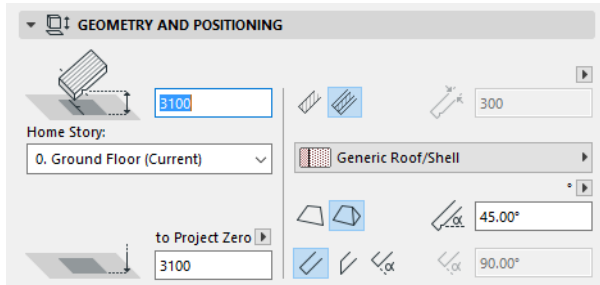
For a description of generic settings common to all tools in the Toolbox, see [Working in Tool Settings Dialog Boxes](#).

For general information on how to create and edit roofs, see [Roofs](#).

For information on the Structural Analytical Parameters Panel of Roof Tool Settings:

See [2D Members \(Slab, Wall, Roof\)](#).

Roof Geometry and Positioning Panel



Pivot Offset: Enter a value here for the offset of the pivot line from the Roof's Home Story.

Home Story

See [Home Story](#).

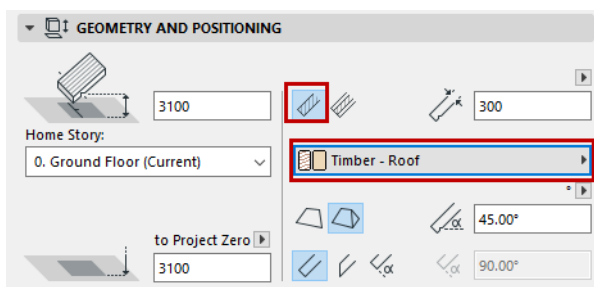
The Roof pivot height is physically linked to its Home Story. If you later modify the Home Story's position (e.g. redefine the floor level), the Roof will change its position.

Absolute Pivot Elevation [to Project Zero]: Enter a value here for the elevation of the pivot line, as measured from the Reference level (by default, this Reference level is Project Zero). Click the pop-up arrow to change the Reference level, if needed.

Note: Reference levels are defined at **Options > Project Preferences > Reference Levels**.

Roof Structure: Click one of these icons to define the Roof as either Basic or Composite, then choose an attribute that reflects its structural properties:

- For a **Basic** Roof: use the pop-up to choose its Building Material.

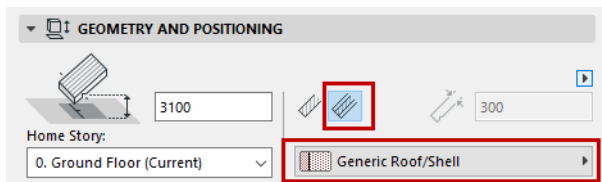


- **Roof Thickness:** Enter a value for the thickness of the roof pane, measured either perpendicularly or vertically to the face of the roof. Click the pop-up arrow to choose a way to measure the roof thickness.



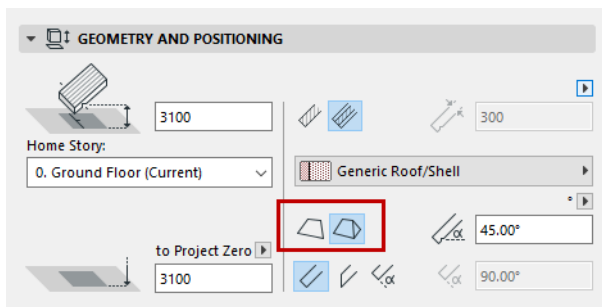
Note: The thickness of a composite Roof is the total value of the individual skins as set in **Options > Element Attributes > Composites**. If you are creating a composite Roof, the thickness field is consequently not editable here.

- For a **Composite** Roof: use the pop-up to choose its Composite structure.



Note: This pop-up lists composites whose “use with” setting includes the roof element: see [“Use With” - define composite availability](#)

Roof Geometry Method: When defining default settings before creating a Roof, choose either **Single-plane** or **Multi-plane Roof** geometry.



If Multi-plane is selected here, the Multi-plane Geometry panel of Roof Settings is activated.

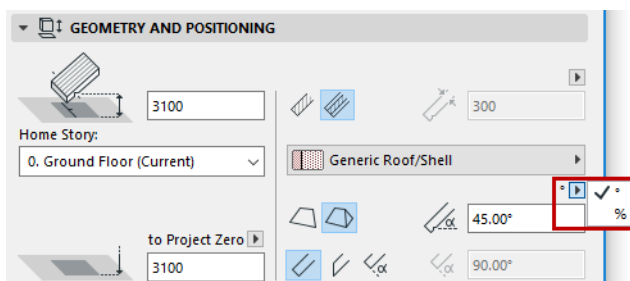
[See Roof Multi-plane Geometry Panel.](#)

Roof Pitch: Enter a value for the pitch (slant angle) of the roof.

Note: The pitch can be measured in degrees, in percentages, or (if you are using imperial units) in the amount of roof rise per 12 feet or 12 inches of horizontal run. Click the arrow icon next to the Roof Pitch field to display these unit choices.

If your Roof Settings refer to a Multi-plane Roof, then the pitch unit set here is also used when setting the pitch of individual Roof levels.

[See Roof Multi-plane Geometry Panel.](#)



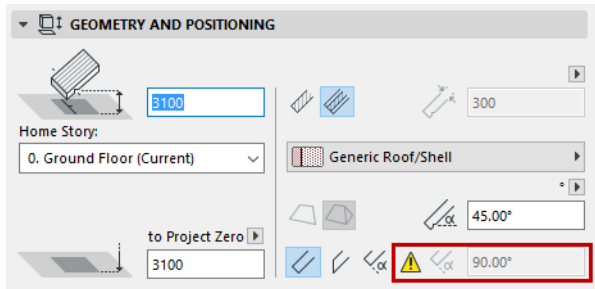
The Roof pitch is typically positive, but it can also be a negative value, in which case the resulting Roof is “reversed”: its roof plane will slope downward.

Edge Angle: Choose an angle (perpendicular, vertical, custom) for all roof edges.

- If you choose custom, enter the desired angle value in the field at right.

Note: You can set a different custom edge angle for any single edge(s). If you have done so, a yellow “custom” icon appears alongside the edge type setting control in Roof Settings.

See [Customize Edge of Roof or Roof Hole](#).



Roof Floor Plan and Section Panel

See [Element Display in Floor Plan and Section](#)

See also [Use Symbolic Display in Floor Plan for Single-plane Roofs and their Skylights \(as in Archicad 14 and before\)](#)

Cover Fills

See [Assign a Cover Fill](#).

Roof Model Panel

This panel controls the 3D display of the roof.

Override Surfaces: Click the toggle button of any or all three pop-ups if you wish to override the surface assigned to this element (in its Building Material). You can override the surfaces of the Roof’s top and bottom, and/or all Roof edges.

See also [Override Surfaces With Surface Painter](#).

Note: The top surface override, if it is a vectorial hatching, can also be applied as a cover fill for the Roof (by checking the “Use Fill from Surface” box in Floor Plan and Section panel of Roof Settings).

Moreover, if the cover fill has a customized fill origin (via Link to Fill Origin option), the 3D Model will reflect that fill orientation too.

For a composite Roof, the “top” surface is the first listed skin of the composite, in Composite Settings.

Click the **chain** icon to assign the last surface chosen to all parts of the element. Deselecting the chain icon will restore the surfaces that you originally set for each part of the element.

Note: The chain icon is available when all the surfaces use overrides.

If you have customized any of the Roof’s **Edge surfaces** or **Top/Bottom surface** (in the Custom Edge Settings or Custom Plane Settings dialog boxes), a yellow “custom” icon appears alongside the surface setting control in Roof Settings, indicating that one or more edge surfaces have been changed from the general setting.

See [Customize Edge of Roof or Roof Hole](#). See also [Customize Roof Plane](#)

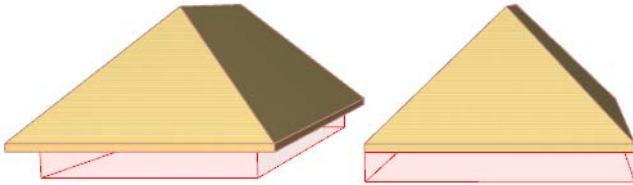
Custom Texture Alignment: If this message is activated, it indicates that the currently selected roof (already placed in the project) has been assigned a custom 3D texture. In this case, the **Reset Texture** button is also activated. Click to restore the origin of the Texture of the selected roof.

For more information, see [Align 3D Texture](#).

Trimming body: Use this control to define the geometry of the Roof's trimming body. The trimming body will cut other elements in accordance with this geometry, when you trim another element to this Roof.

- Pivot lines down
- Contours down

Note: To see these trimming bodies while you work (in the 3D window only), turn on **Trimming Bodies** in the **View > On-Screen View Options** menu.



For more information, see [Trim Elements to Roof or Shell](#).

Roof Multi-plane Geometry Panel

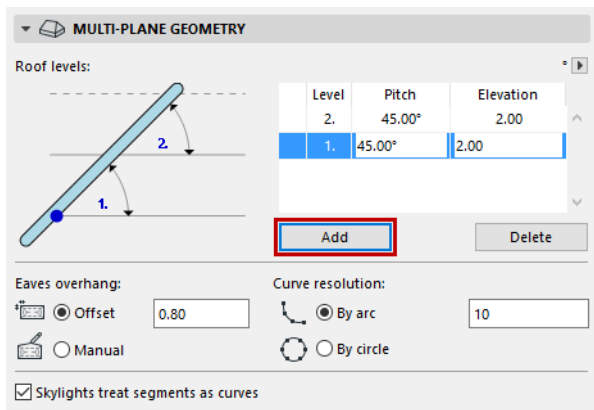
This panel is active only if you have selected a Multi-plane Roof, or if the Multi-plane Geometry Method is selected (Geometry and Positioning Panel) of Roof Default Settings.

Roof levels: Use these controls to add one or more levels to a Multi-plane Roof.

By default, a Multi-plane roof has one level.

Click **Add** to add another Roof level.

- In the table, adjust the **pitch** of any roof level as needed.
- **Elevation** refers to the point at which the new level begins:



Eaves overhang:

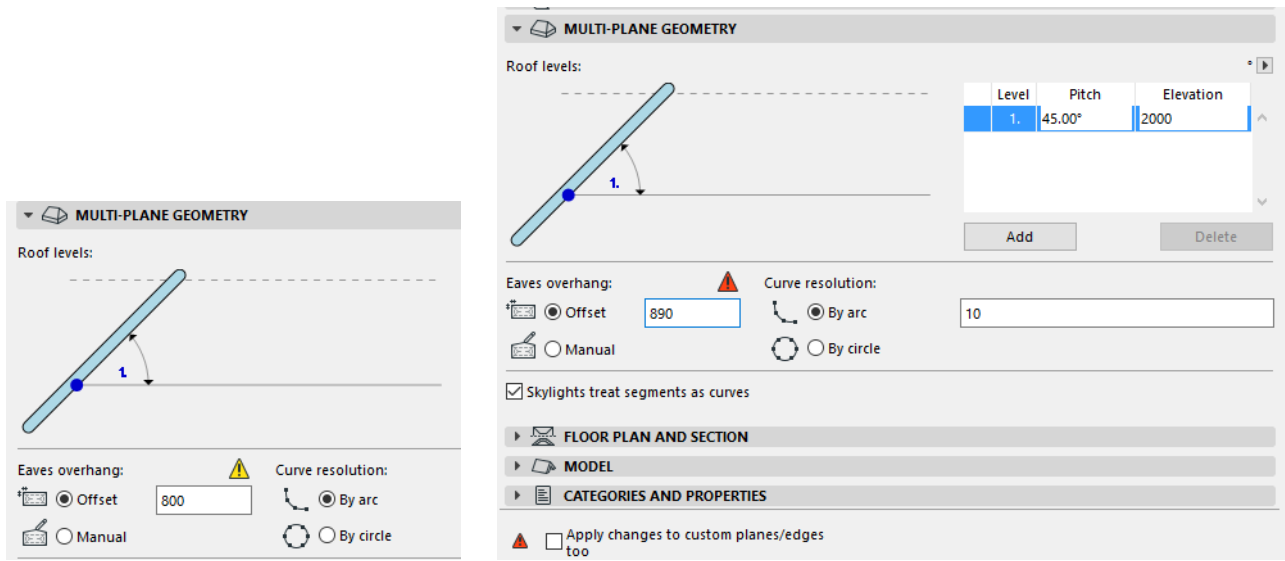
- **Offset:** Enter a value for the distance of the eaves overhang from the pivot polygon. This value is applied to the Roof as a whole.

You can adjust this value for any particular roof plane in the Customize Roof Plane dialog box.

See [Customize Roof Plane](#).

If you do so, the value in Roof Settings (applied to the roof as a whole) does not change, but a yellow “custom” icon appears alongside the eaves overhang control in Roof Settings, indicating that the eaves overhang value for one or more roof planes has been changed from the general setting.

If you later change the Eaves overhang “offset” value in Roof Settings, the yellow “custom” icon changes to red. This means that the Roof’s eaves overhang is going to change: do you want to apply this change to the custom-overhang planes too? If so, check the box at the bottom of Roof Settings (Apply changes to custom planes and/or edges too). If you do not check the box, your custom eaves offsets remain the way you set them.



- **Manual:** The overhang value control changes to “Manual” if you edit the roof contour graphically.

See [Edit Roof Contour](#).

At any time, you can undo these manual edits by clicking “Offset” and entering a fixed value for the eaves overhang.

Curve resolution

Choose an option for dividing the segments of curved planes in this Roof.

- **By arc:** Divides the curved plane(s) into the given number of segments.
- **By circle:** Divides a full circle into the given number of segments, and applies that geometry to the curved plane(s) of the Roof.

You can customize the curve resolution setting for any individual Roof plane.

See also [Customize Roof Plane](#).

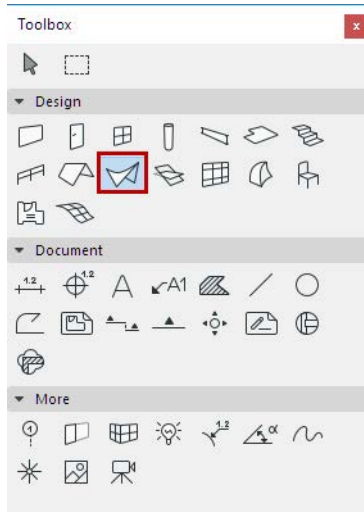
See [Add a Tower to the Roof](#).

Skylights treat segments as curves: When a Roof plane with a curved pivot line changes its geometry, Archicad can adjust any skylights in the plane in one of two ways:

- Position skylight as onto a curved surface: this is the default setting, with the checkbox enabled in the Multi-plane Geometry panel of Roof Settings. Skylight geometry is adjusted so that it smoothly adjusts to a curved surface.
- Position skylight as onto a flat plane: if you uncheck the checkbox, Archicad will adjust the Skylight geometry so that it fits flat onto the Roof plane.

See also [Position Skylight to Curved Roof Plane](#).

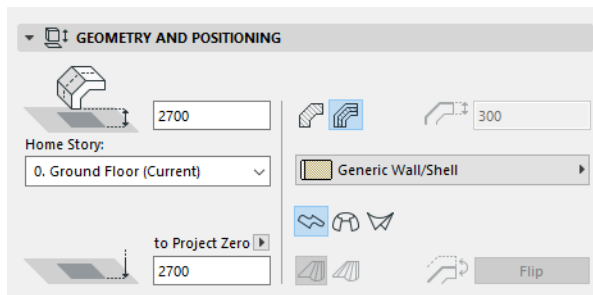
Shell Tool Settings



For a description of generic settings common to all tools in the Toolbox, see [Working in Tool Settings Dialog Boxes](#).

For general information on how to create and edit shells, see [Shells](#).

Shell Geometry and Positioning Panel



Reference Line Offset to Home Story: Enter a value here for the elevation, measured to the Home Story, of the initial point of

- the Shell's extrusion vector (for an extruded Shell), or
- the Shell axis (for a rotated Shell), or
- the Shell's first profile (for a ruled Shell).

These reference points are distinguished by a highlighted hotspot on each selected Shell.

Home Story

[See Home Story](#).

Reference Line Elevation [to Reference level]

Elevation of the Shell's starting point (reference point) as measured from the Reference level (by default, this Reference level is Project Zero). Click the pop-up arrow to change the Reference level, if needed.

Notes:

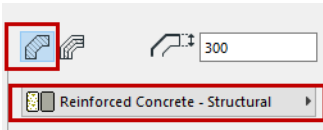
- Reference levels are defined at **Options > Project Preferences > Reference Levels**.

- This is a calculated value, not a link. If you change the Reference level, the bottom of the Shell will not change its position.

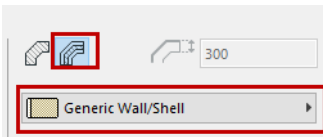
Shell Structure

Click one of these icons to define the Wall as **Basic** or **Composite**, then choose an attribute that reflects its structural properties:

- For a **Basic** Shell: use the pop-up to choose its Building Material.



- For a **Composite** Shell: use the pop-up to choose its Composite structure.



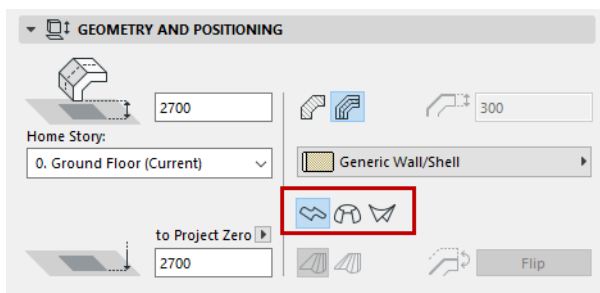
Note: This pop-up lists composites whose “use with” setting includes the shell element: see [“Use With” - define composite availability](#)

Shell Thickness: Enter a value for the thickness of the Shell.

If a composite structure is used for the Shell, this field is gray and cannot be modified, as the Shell’s thickness is then determined by the thickness of the Composite as defined in **Options > Element Attributes > Composites**.

Geometry Method

In Shell Default Settings, click one of the three icons to define the Geometry Method with which to place the new Shell:



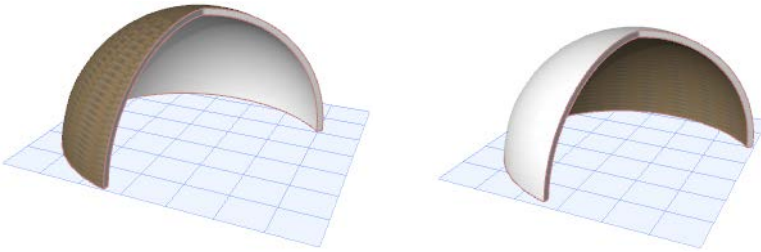
- Extruded
- Revolved
- Ruled

See [About Shells](#) for more information.

Morphing Rule (for Ruled Shell only): Click on either the “Smooth” or “Paired” method for defining the geometry of a Ruled Shell, based on how the respective nodes of the two profiles should be connected.

For more information, see [Ruled Shell Morphing](#).

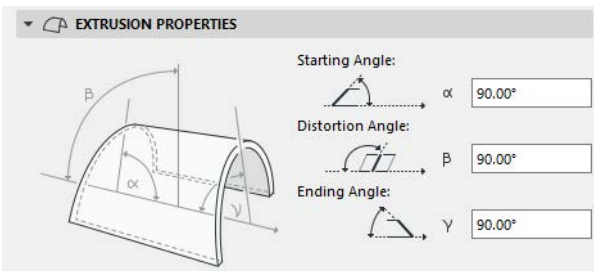
Flip: (*Editable in Shell Selection Settings.*) The Shell is composed of a membrane and a Shell body attached to one side of the membrane. Use Flip to change sides: the Shell body moves to the other side of Shell membrane.



Shell Special Properties Panel (Extrusion Properties, Revolution Properties)

If you are using either the **Extrusion** method or **Revolved** method, this panel contains special properties related to the Shell geometry.

Extrusion Properties



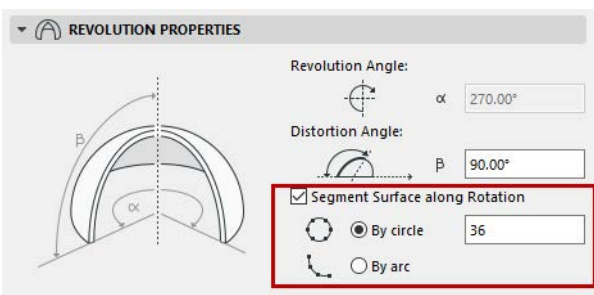
Starting angle: The angle between the plane of the Shell and the Shell edge at the initial point of the extrusion vector.

Distortion angle: Angle of the Shell profile, measured to the Shell plane. By default: 90 degrees, but changing it enables you to extrude the Shell profile at a angle, creating a distorted Shell, provided that either its starting or ending angle is other than 90 degrees.

Ending angle: The angle between the plane of the Shell and the Shell edge at the endpoint of the extrusion vector.

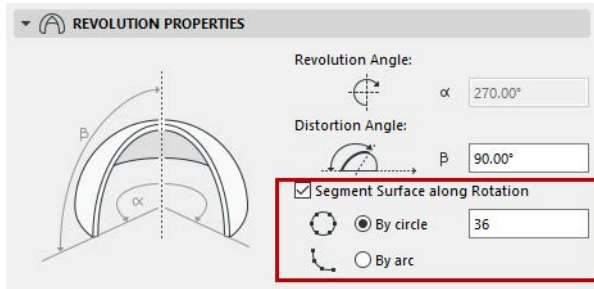
Revolution Properties

For a Revolved Shell:



- **Revolution angle:** Angle of revolution of the Shell around its axis.
- **Distortion angle:** Angle of the axis of the revolved Shell, measured to the Shell plane angle. Modifying the distortion angle will affect the Shell profile. (Changing the slant angle alone, which is available through graphical editing, will not change the Shell profile, only “tip” it over.)

- Segment Surface Along Rotation

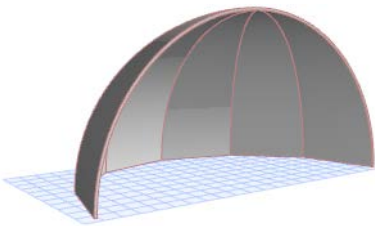


By default, the surface of Revolved Shells is smooth. You have the option of dividing it into segments as needed.

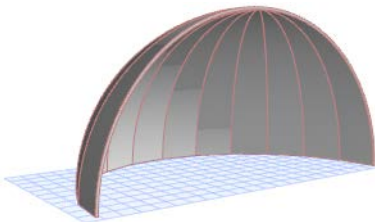
Check the box if you would like to divide the rotated Shell into segmented planes. Next, enter a number for the number of segments.

The given number of segments can be applied to the existing Shell in one of two ways:

- By circle:** Divides a full circle into the given number of segments, and applying that geometry to the existing arc of the Shell.



- By arc:** Divides the existing arc of the Revolved Shell into the given number of segments.



Shell Floor Plan and Section Panel

[See *Element Display in Floor Plan and Section*](#)

Cover Fills

[See *Assign a Cover Fill*](#).

Shell Model Panel

This panel affects the 3D display of the Shell.

[See also *Override Surfaces With Surface Painter*](#).

For a composite Shell, the “reference side” surface is the first listed skin of the composite, in Composite Settings.

Note: The Reference Side surface override, if it is a vectorial hatching, can also be applied as a cover fill for the Shell (by checking the “Use Fill from Surface” box in Floor Plan and Section panel of Shell

Settings). Moreover, if the cover fill has a customized fill origin (via Link to Fill Origin option), the 3D Model will reflect that fill orientation too.

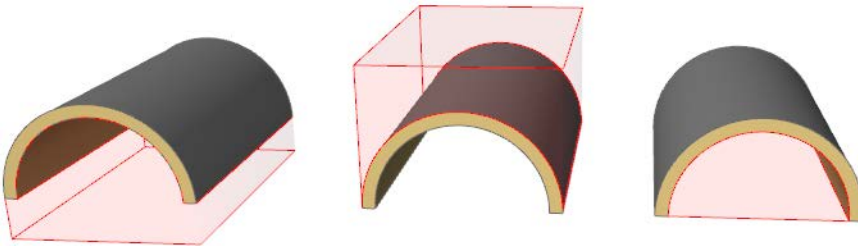
Note: You can apply custom surface settings to any one or more edges of the Shell, using Custom Edge Settings from the Pet Palette. If you do so, the Model panel of Shell Settings will indicate that one or more edges uses a custom surface, using a yellow patch.

See [Customize Edge of Shell or Edge of Shell Hole](#).

Trimming body: Use this control to define the geometry of the Shell's trimming body. The trimming body will cut other elements in accordance with this geometry, when you trim another element to this Shell.

- Downward Extrusion
- Upward Extrusion
- Editable

Note: To see these trimming bodies while you work (in the 3D window only), turn on **Trimming Bodies** in the **View > On-Screen View Options** menu.



For more information, see [Trim Elements to Roof or Shell](#).

Slab Tool Settings



For a description of generic settings common to all tools in the Toolbox, see [Working in Tool Settings Dialog Boxes](#).

The Slab tool is used for drawing and editing slabs, the basic horizontal building blocks of the Virtual Building.

For more information, see [Slabs](#).

For information on the Structural Analytical Panel of Slab Tool Settings:

See [2D Members \(Slab, Wall, Roof\)](#)

Slab Geometry and Positioning Panel

Slab Thickness: Enter a value for the slab's thickness.

Note: If the structure of the slab is a composite, then the slab's thickness is defined in **Options > Element Attributes > Composites**, and equals the sum of the skins' thicknesses. A composite slab's thickness cannot be edited in Slab Settings; you must go to **Options > Element Attributes > Composites**.

If you want to edit the slab thickness in Slab Settings, choose a non-composite (basic) slab structure.

Offset to Home Story: Define an offset for the Slab's reference plane from its Home Story. The reference plane has several possible settings. (See [Reference Plane](#).)

Home Story

See [Home Story](#).

Offset to Reference Level: Define an **offset** for the Slab's reference plane from the given Reference level (by default, this Reference level is Project Zero).

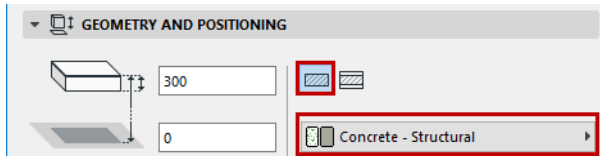
- The Slab reference plane has several possible settings.

[See Reference Plane.](#)

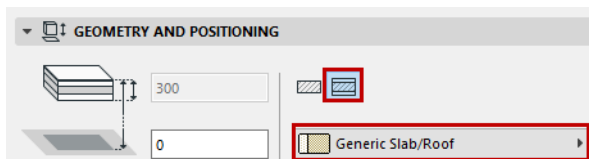
Slab Structure

Click one of these icons to define the Slab as **Basic** or **Composite**, and then choose an attribute that reflects its structural properties:

- For a **Basic** Slab: use the pop-up to choose its Building Material.



- For a **Composite** Slab: use the pop-up to choose its Composite structure.



Note: This pop-up lists composites whose “use with” setting includes the slab element: see [“Use With” - define composite availability](#)

Slab Edge Angle

By default, the Slab's edge is perpendicular to its surface (90 degrees). If you wish to use a custom edge angle, click the corresponding icon and enter the angle in degrees in the field to the right.

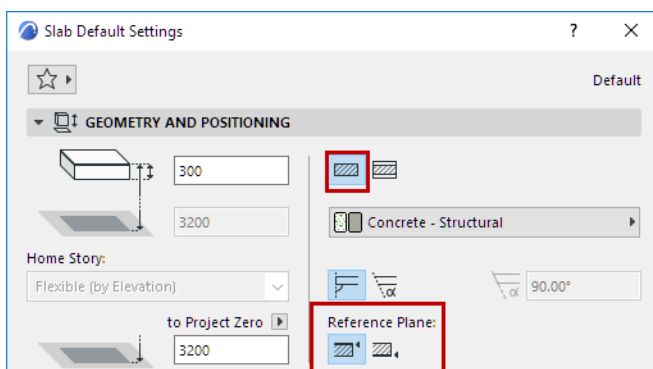
Note: You can set a different custom edge angle for any single edge(s). If you have done so, a yellow “custom” icon appears alongside the edge type setting control in Slab Settings.

[See Set Custom Slab Edge Angle and Edge Surface.](#)

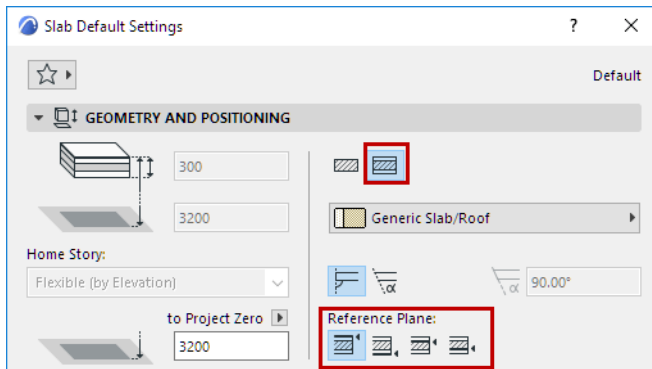
Reference Plane

Depending on the structure you choose for the Slab (either a basic or a composite slab), you have varying options for the Slab Reference Plane:

- A basic Slab's reference plane is either the top or the bottom of the Slab.



- A composite Slab's reference plane can be one of four options: top of slab, bottom of slab; top of core, bottom of core.



Note: If the chosen composite structure has no core, then the core-related reference plane choices are grayed out.

Use the Info Box or Geometry and Positioning Panel of Slab Settings to choose a reference plane.

See also [Slab Reference Plane](#).

Slab Floor Plan and Section Panel

See [Element Display in Floor Plan and Section](#)

Note: The line type of slabs displayed on remote stories is set in **Options > Project Preferences > Legacy**.

Cover Fills

See [Assign a Cover Fill](#).

Slab Model Panel

This panel affects the 3D display of the slab.

See also [Override Surfaces With Surface Painter](#).

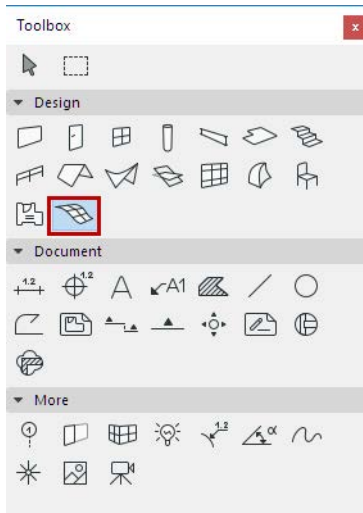
See [Set Custom Slab Edge Angle and Edge Surface](#).

For more information, see [Align 3D Texture](#).

Mesh Tool Settings

For a description of generic settings common to all tools in the Toolbox, see [Working in Tool Settings Dialog Boxes](#).

The Mesh tool is for creating surfaces, skirted surfaces and solid bodies by defining the elevation of its relevant nodes and interpolating between them.



A mesh is created on the foundation of the Mesh Reference Plane and the ridges of the Mesh. You draw the main contours of the mesh projected to Mesh Reference Plane. You can then raise the characteristic points of the mesh's top surface out of this plane.

For more information, see [Meshes](#).

Mesh Geometry and Positioning Panel

Mesh Height: Set the depth (if any) to which the Mesh will extend below the Mesh Reference Plane.

- The resulting base elevation will be displayed as a negative value in the Info Box.

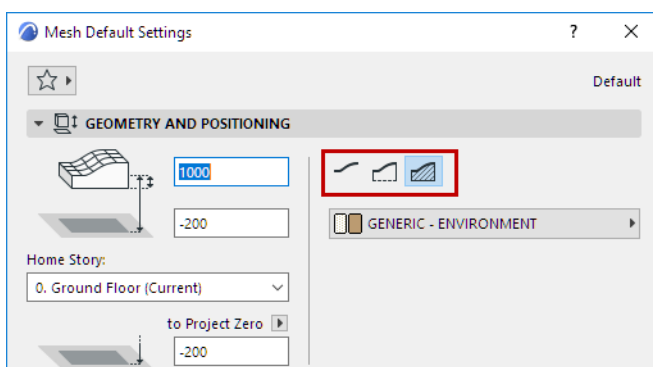
Offset to Home Story: Set the height of the Mesh Reference Plane as measured from its Home Story.

Home Story

See [Home Story](#).

Mesh Construction Method

Choose a method for constructing the Mesh:



- Surface Only
- Surface with Skirt
- Solid Body

Building Material: Use the pop-up to choose a Building Material for the Mesh.

Mesh Floor Plan and Section Panel

See [Element Display in Floor Plan and Section](#)

Note: These settings only apply to solid bodies, not to surface-only or skirted Meshes.

Note: The line type of meshes displayed on remote stories is set in **Options > Project Preferences > Legacy**.

- **Ridge Selection:** Define whether to **Show all Ridges** or **Show User-Defined Ridges** (i.e., only those that you have defined when drawing the Mesh.)
- **Ridge Pen:** Use the pop-up to assign a pencolor to both user-defined and generated ridges.

Cover Fills

See [Assign a Cover Fill](#).

Mesh Model Panel

This panel affects the 3D display of the mesh.

For more information, see [Align 3D Texture](#).

3D Appearance

Control the appearance of the generated ridges (either sharp or smooth) in the 3D model.

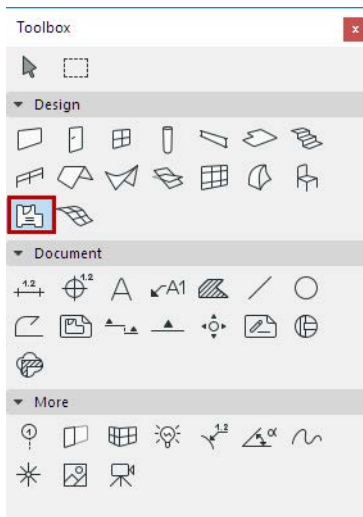
- **All Ridges Sharp:** Show the mesh in the 3D window with sharply connecting triangles.
- **User Defined Sharp:** Show the mesh surfaces with sharply connecting User Defined Ridges, and with smoothly connecting surfaces everywhere else.
- **All Ridges Smooth:** Show the mesh with smoothly connecting surfaces.

If you choose **All Ridges Smooth**, only user-defined ridges will appear in 3D view.

Zone Tool Settings

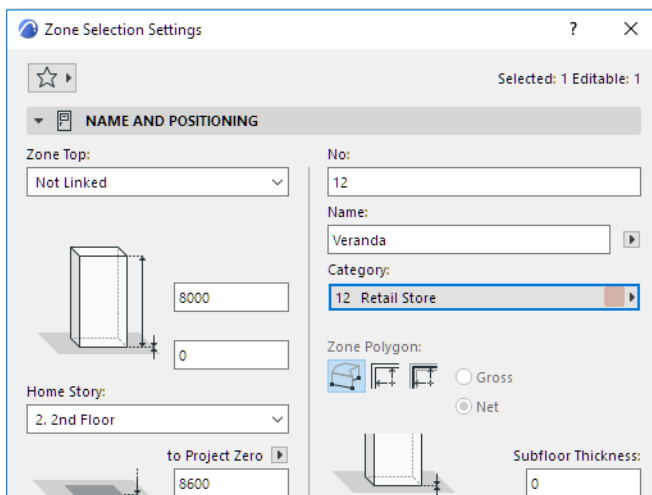
For a description of generic settings common to all tools in the Toolbox, see [Working in Tool Settings Dialog Boxes](#).

Zones are 3D elements, whose floor plan representation consists of zone fills and zone stamps.



For more information, see [Zones](#).

Zone Name and Positioning Panel



Zone Category: Choose a Zone category defined in **Options > Element Attributes > Zone Categories**. Their definition includes the Category name, code, color and zone stamp, and a set of parameters.

For a detailed description, see [Zone Categories](#).

Zone Name and Number (No.): Enter name and number values in these fields for each Zone individually.

By default:

- The Zone top is linked to one story up
- The Home Story is the current story.

The story to which the Zone bottom is linked is called the Home Story, but you can offset the bottom in either direction.

Use the controls in this panel to change any of these values, and to define the geometry and complexity of the Zone element.

Zone Top: Use this control to either top link the Zone relative to its Home Story (Home+1, Home+2, etc.), or to make it a fixed-height Zone (choose “Not linked”).

If you later modify the positions and heights of stories in your projects, the heights of any linked Zones will follow automatically.

- Optionally, define an **offset** for the Zone top from its top linked story (the Zone height changes accordingly). This offset value can be positive, negative or zero.

The offset field is not available if the Zone has no top link.

Zone height: Enter a value for the total height of the Zone. (For a top linked Zone, this field is not editable.)

- The Zone height value is affected by the top and bottom offsets, if any. If you change the top offset of a top linked Zone, or its bottom offset from the Home Story, this change will be reflected in the height value.

Subfloor Thickness: Optionally, enter a Subfloor Thickness value (positive or negative). The bottom of the Zone will be offset. You may wish to use this value in a Zone Stamp marker.

- If the Zone has a top link, the bottom of the Zone body will vary depending on the Subfloor thickness value plus the Home Story offset value (the top of the Zone body remains in place).
- If the Zone top is set to Not linked, the entire Zone body will move up or down depending on the Subfloor thickness value.

Home Story

[See Home Story.](#)

- Optionally, define an **offset** for the Zone bottom from its Home Story (When top linked, the Zone height changes accordingly.) The subfloor thickness remains constant.

The bottom of the Zone is physically linked to its Home Story. If you later modify this story’s position (e.g. redefine the floor level), the Zone will change its position (while respecting the defined offset, if any).

Bottom Elevation [to Reference level]

Height of the Zone, as measured from the Reference level (by default, this Reference level is Project Zero).

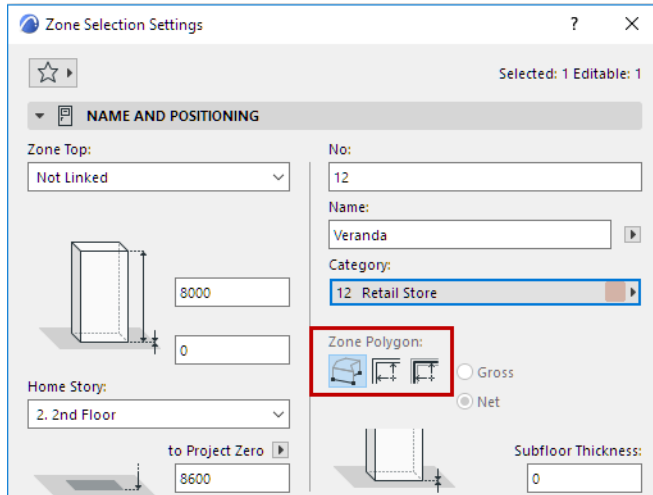
Click the pop-up arrow to change the Reference level, if needed.

Notes:

- Reference levels are defined at **Options > Project Preferences > Reference Levels**.
- This is a calculated value, not a link. If you change the Reference level, the bottom of the Zone will not change its position.

Zone Polygon

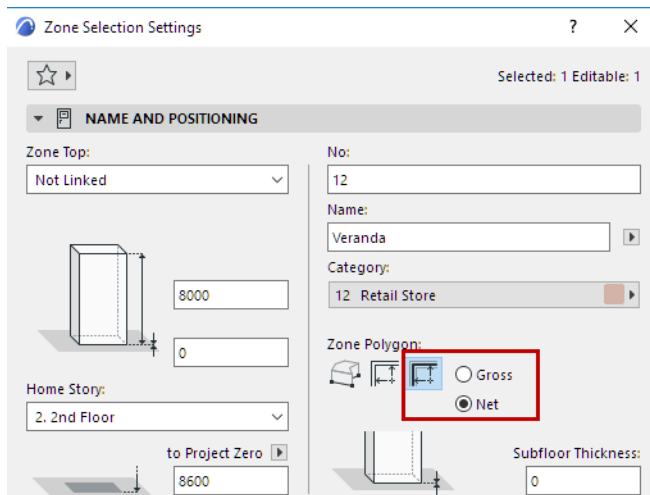
Click one of the icons to choose the **Manual**, **Inner Edge** or **Reference Line** method for creating a zone.



Use the Info Box icon to access the three options of the manual method: Polyline, Rectangular or Rotated Rectangle.

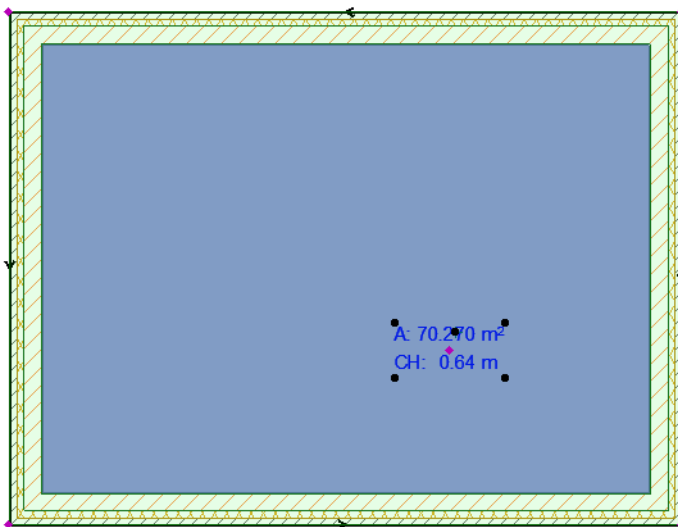
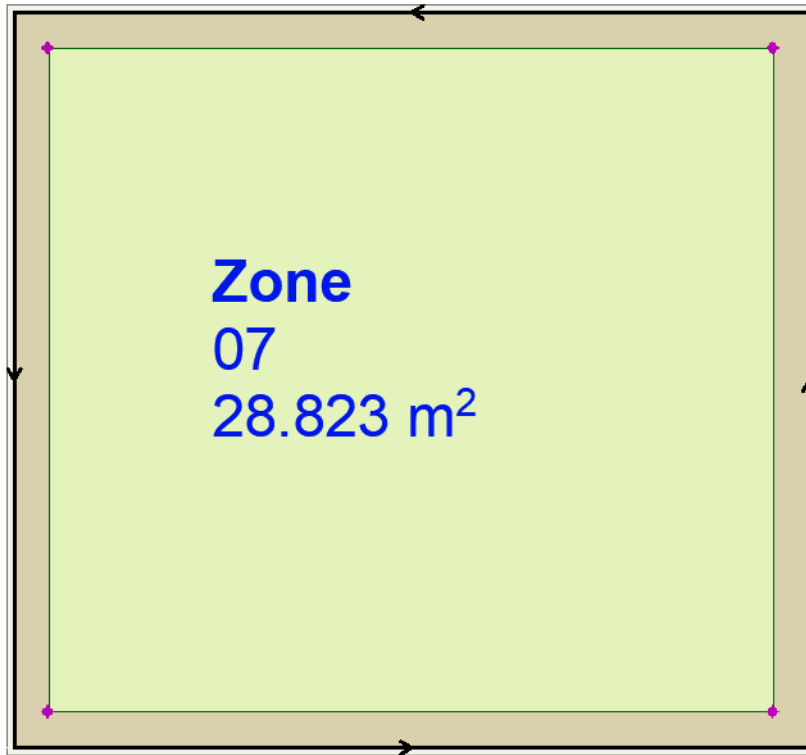
For more information, see [Creating Zones](#).

Zone Polygon Display (Gross/Net): Use these options to set the Floor Plan display of a Zone created with the Reference Line method.



- **Net:** Zone polygon will not be displayed beyond the inner edge of the boundary walls.
- **Gross:** Full zone polygon is displayed.

Regardless of the option, the Zone area remains unchanged, only the polygon is displayed differently.



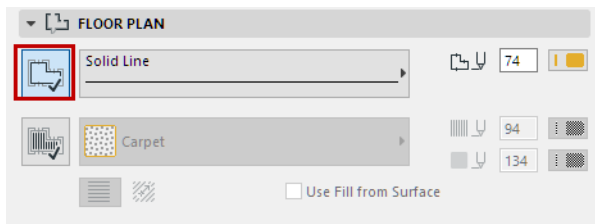
Zone Polygon Net vs. Zone Polygon Gross

Zone Floor Plan Panel

The Floor Plan panel defines the appearance of the Zone's fill on the Floor Plan.

Zone Fill Outline:

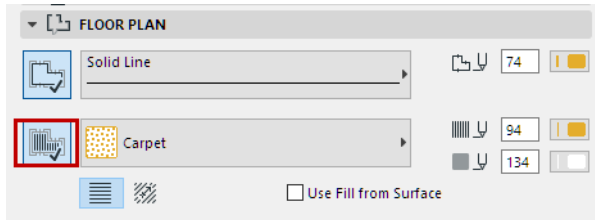
- Click the button if you want your Zone fill to have an outline.



- Choose a Line Type and an outline color from the Line Type and Pencil pop-up palettes.

Zone (Cover) Fill Pattern:

- Click the button to choose a cover Fill pattern and background from the Fill pop-up.

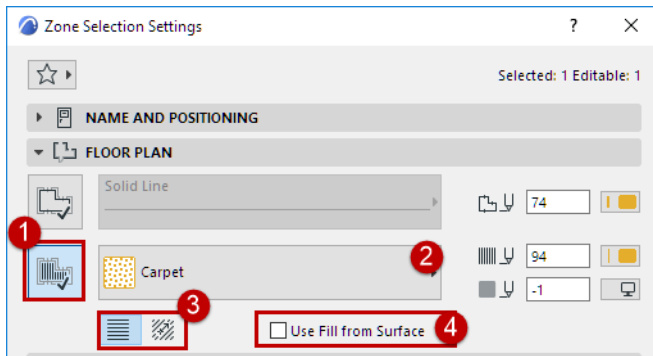


Note: Only those Fills defined as Cover Fills in **Options > Element Attributes > Fills** dialog box are available here.

For more information, see [Assign a Cover Fill](#).

- **Use Fill from Surface** checkbox: Zone fill on the Floor Plan will display the same top-side hatching as you set for the Zone’s 3D display in the Zone’s Model panel.

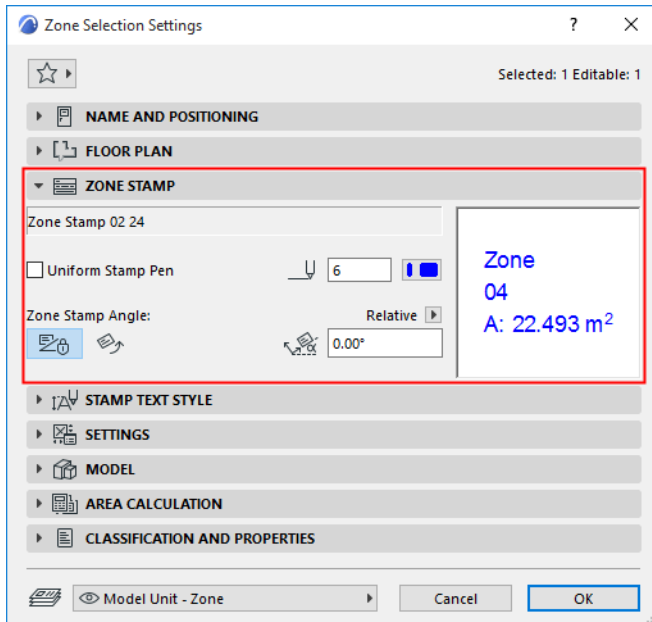
Link Fill Orientation: Choose a method for displaying the vectorial cover fill:



For more information, see [Set Orientation of Vector or Symbol Fill Patterns](#).

Zone Stamp Panel

Use these controls to set the Zone Stamp's appearance on the Floor Plan.



The first field displays the name of the Zone Stamp. You cannot assign a different zone stamp here; the zone stamp shown is the one associated with the Zone Category you chose in the **Name and Positioning Panel**.

Note: The Zone Stamp assigned to each Zone Category is determined by the settings in **Options > Element Attributes > Zone Categories**.

Uniform Stamp Pen

Check this box to use this pencolor for all parts of the marker symbol, regardless of any custom colors set elsewhere for this marker.

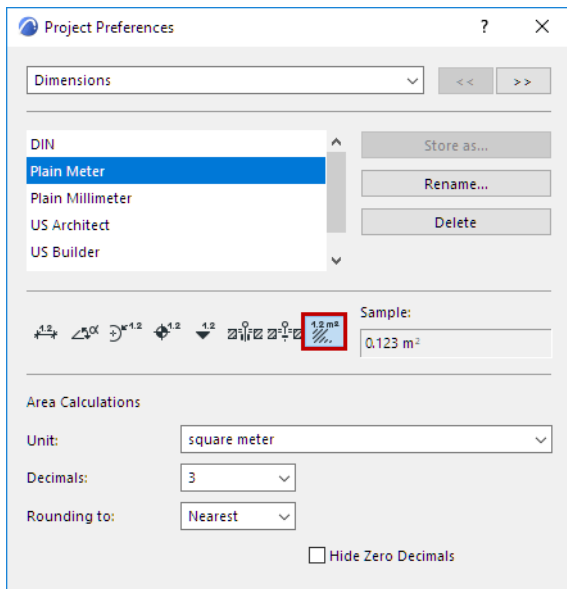
Zone Stamp Angle

- Enter a rotation angle for the Zone Stamp.
- Use two icons to define how this angle value should be interpreted when the element is placed:
 - **Fixed Angle to Screen**
 - **Fixed Angle to Model:** Zone Stamp will be rotated along with the view.

See [Set Orientation](#).

For more information on Zone Stamps, see [Zone Stamp Panel](#).

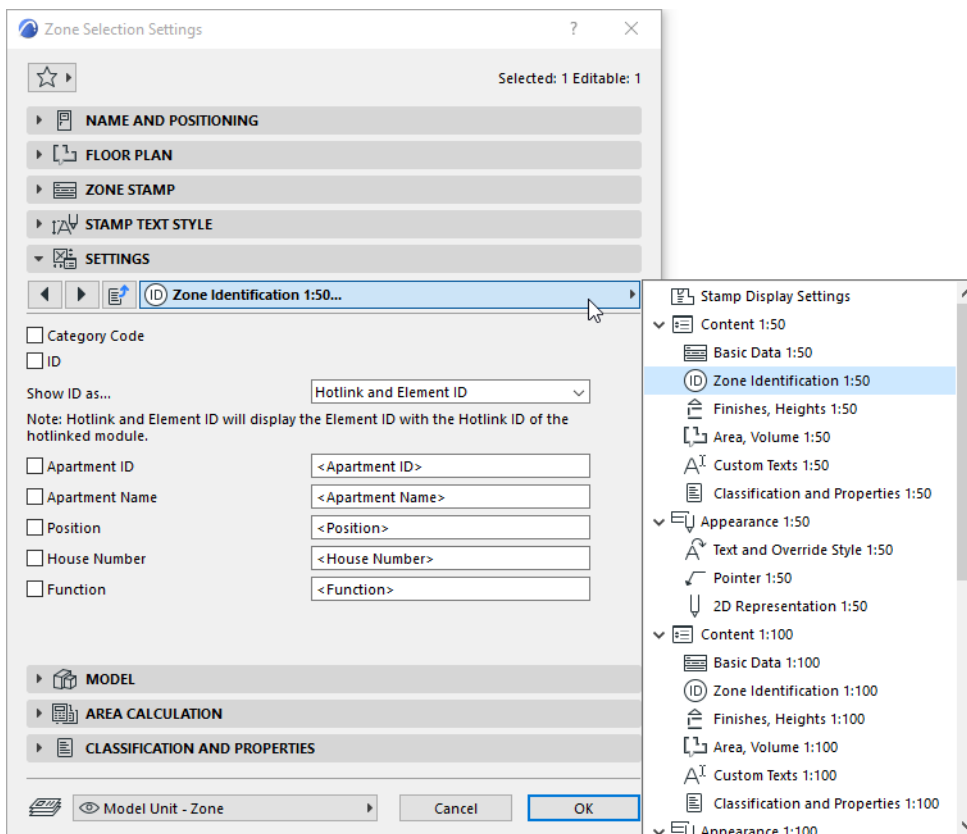
Note: To set the units for the Zone Area display on this Zone Stamp, use **Options > Project Preferences > Dimensions**, and select Area Calculation units.



Note: Customization of the Zone Stamp parameters will be lost if you change the Category of the Zone. Parameters will revert to the values stored with the Zone Category attributes.

Zone Custom Settings Panel

This panel contains additional choices linked to the Zone Stamp.



Zone Model Panel

The **Model** panel controls the 3D appearance of Zones, including Surface choices and crop options. Use the pop-up to choose a surface to display this Zone.

- **Use this Surface on All Zone Faces:** Click this option to display all zone faces of the 3D zone shape using the single surface you selected with the pop-up above.
- **Inherit Boundary Walls and Cropping Elements Surfaces:** Click this option to have individual surface polygons of the zone inherit the corresponding surfaces of boundary walls and any elements that you cropped the Zone with.

Note: If the Surface has a texture, the texture is automatically aligned to the longest edge of the Zone.

Related Topics:

[Crop Zone to Another Element](#)

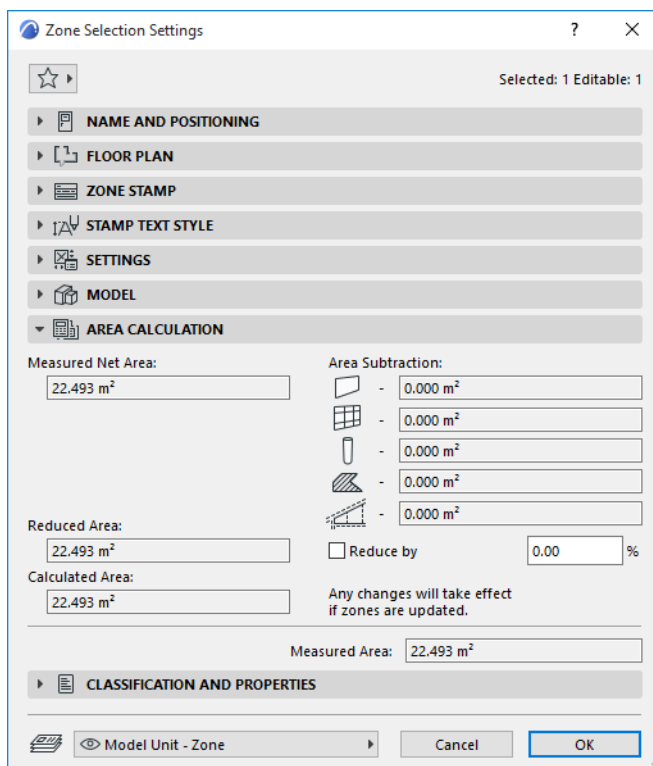
[3D Display of Zone Spaces](#)

Zone Area Calculation Panel

Use the controls in this panel to view the measured area of the Zone and to optionally refine the calculated values.

Note: To see the effect of any changes in Zone calculation settings, you must execute the **Design > Update Zones** command.

See [Updating Zones](#).



Measured Net Area

The **Measured Net Area** is the Zone Polygon only up to the *inside face* of the Bounding Walls. (In contrast, the Measured Area includes the area of bounding Walls all the way to their reference lines.)

Compared to the Measured Area, Measured Net Area is further reduced by Walls, Columns, and/or Curtain Walls that are inside the polygon IF their Relation to Zones is set to „Zone Boundary.”

Reduced Area

The **Reduced Area** results from subtracting the area of certain elements, subject to Zones Preferences, from Measured Net Area.

- Subtract the area of Walls, Columns, and/or Curtain Walls that are inside the polygon IF their Relation to Zones is set to „Reduce Zone Area only.”
 - Reduction of these Wall, Curtain Wall and Column areas is further subject to Zones Preferences.
- Subtract the area of Area of Fills if their „reduce Fill area” option is enabled.
- Subtract the area of Area of low ceilings and wall recesses as defined in Zones Preferences

Calculated Area

Click the **Reduce by** checkbox if you want to further reduce the Reduced Area by any given percentage (enter the desired percentage value in the input field). The result is the Calculated Area.

Area Subtraction

These read-only fields display how much of the zone’s original area is subtracted, depending on elements within or intersecting the zone.

- **Walls, Curtain Walls and Columns**
Displays the area subtracted for these elements as a result of settings at **Options > Project Preferences > Zones**.
- **Fills**
Area subtraction value displays the area subtracted as a result of the Fill’s Subtract from Zones option. (Turn it on or off in Fill Settings - General Settings panel).
- **Low ceilings**
If the zone is cut by a roof or Shell, displays the area subtracted as a result of settings at **Options > Project Preferences > Zones**.

Related Topic

[Calculating Zone Area and Zone Volume](#)

Morph Tool Settings

For a description of generic settings common to all tools in the Toolbox, see [Working in Tool Settings Dialog Boxes](#).

For general information on how to create and edit Morphs, see [Morphs](#).

Morph Default Settings enable you to set parameters for Morphs to be created.

Morph Selection Settings vary depending on whether you have selected

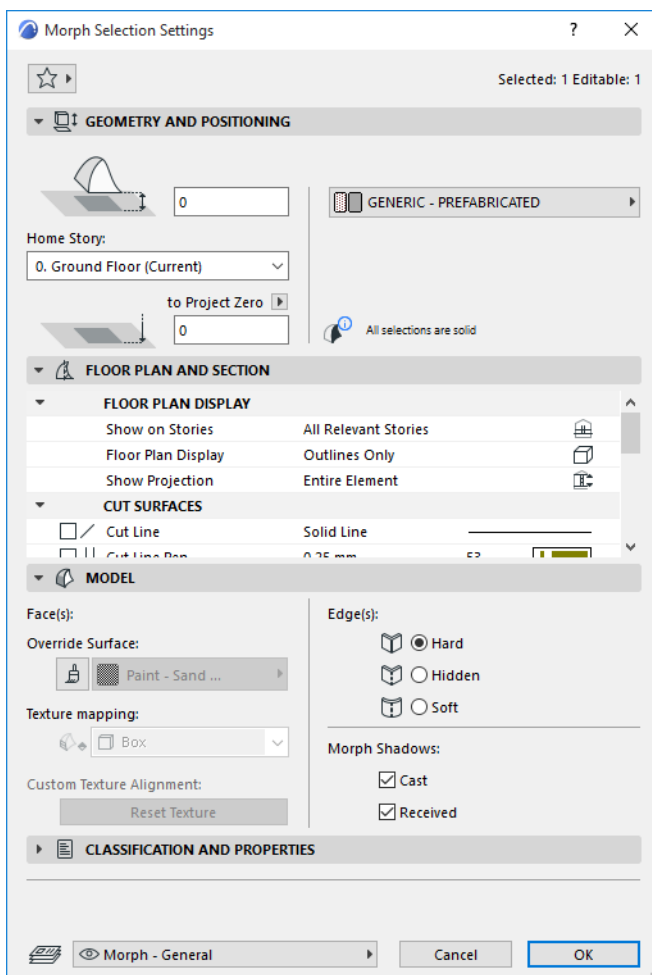
- the entire Morph, or
- one or more of its sub-elements (faces and/or edges),
- or a combination thereof.

See also [Morph Selection](#).

Settings are shown for the *last-selected face or edge*, but when you change the options, they are applied to *all selected faces and/or edges*.

Morph Selection Settings: Entire Morphs

If you have selected the entire Morph, all controls of Morph Settings are available:



Exception: Texture Mapping (either Box or Planar) can only be modified for selected individual faces. For a Morph as a whole, the texture mapping is always “Box.”

Morph Selection Settings: Faces and/or Edges

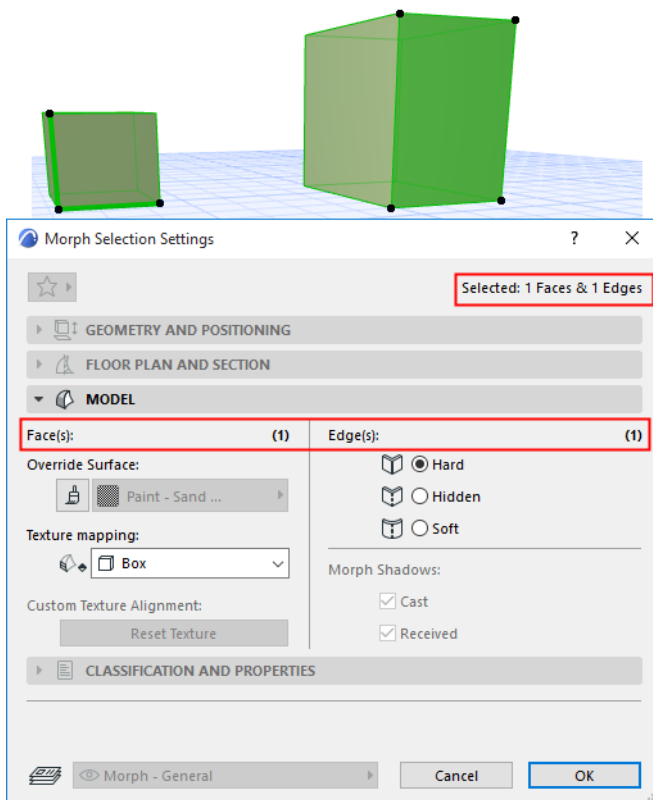
Use sub-element selection to select any combination of particular edges and faces from any number of Morphs to edit them.

In this case, only the Model panel is available, and the only available controls are Override Surface and Texture mapping (if any faces are selected) and Edge Type (if any edges are selected).

The rest of the controls (Morph Shadows) are gray, because these options can be applied here only to the Morph as a whole, not to its selected sub-elements.

Settings are shown *for the last-selected face or edge*, but when you change the options, they are applied to *all selected faces and/or edges*.

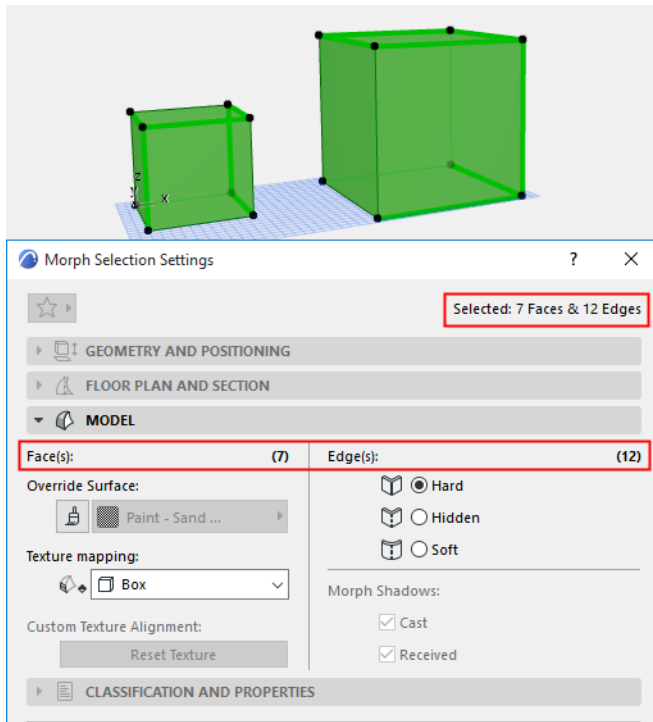
The numbers of editable faces and editable edges, if any, are shown in the Model Panel as well as in the Info Box.



Morph Selection Settings: Whole Morph plus Faces and/or Edges

If you have selected an entire Morph plus any additional faces or edges, you will be able to edit sub-element options only: all the sub-elements of the entire Morph, plus the additional selected sub-elements.

Again, only the Model Panel of Morph Selection Settings is available, and only the Face Surface, Texture Mapping, and Edge Type can be edited. The total number of editable items is shown in the Model Panel as well as in the Info Box.



Morph Geometry and Positioning Panel

Note: The controls of this panel are available only in Default Morph Settings, or if you have selected one or more entire Morph elements (no sub-elements).

Bottom Offset to Home Story: Enter a value here for the Morph’s elevation from the Home Story. This point is distinguished by a highlighted hotspot on the selected Morph; this is the Morph’s bottommost point.

Home Story

[See Home Story.](#)

The Morph’s bottommost point is physically linked to its Home Story.

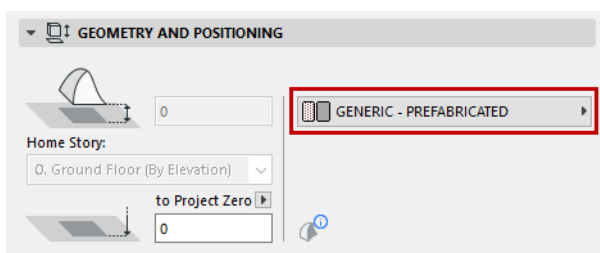
Bottom Elevation [to Reference level]

Height of the Morph’s lowest point as measured from the Reference level (by default, this Reference level is Project Zero). Click the pop-up arrow to change the Reference level, if needed.

Notes:

- Reference levels are defined at **Options > Project Preferences > Reference Levels.**
- This is a calculated value, not a link. If you change the Reference level, the bottom of the Morph will not change its position.

Building Material: Use the pop-up to choose a Building Material for the Morph.



Morph Floor Plan and Section Panel

See [Element Display in Floor Plan and Section](#)

Note: The controls of this panel are available only in Default Morph Settings, or if you have selected one or more entire Morph elements (no sub-elements).

Outlines:

- **Outline Contour Display:** Adds an outline contour around the morph element in Floor Plan/Section/3DD even if edges are hidden or soft.

Cover Fills

See [Assign a Cover Fill](#).

Morph Model Panel

This panel affects the 3D display of the Morph.

Note: If you have selected one or more Morph sub-elements (faces and/or edges), then the only available controls are Face Surface, Texture Mapping and Edge Type.

You can apply custom settings to one or more Morph faces or edges by using the Custom Face Settings or Custom Edge Settings commands of the pet palette.

See [Transforming Morphs: Custom Face Settings](#).

If you have customized any of the Morph's surfaces (in the Custom Face Settings or Custom Edge Settings dialog boxes), a yellow "custom" icon appears alongside the surface/edge control in Morph Settings, indicating that one or more edge surfaces or types have been changed from the general setting.

See [Custom Morph Surface: Feedback in Morph Settings](#), and [Change Edge Type](#).

- **Texture Mapping:** This control (either Box or Planar) can only be modified for selected individual faces. For a Morph as a whole, the texture mapping is always "Box" and cannot be edited.
- **Edges:** Choose an option for displaying all Edges of the Morph.
- **Morph Shadows:** Check one or both boxes to display shadows either **cast** by the Morph and/or **received** on the Morph (shadows cast by other elements). Effects are visible whenever shadows are used (e.g. Rendering, 3D window).

Related topics

[Override Surfaces With Surface Painter](#).

[Transforming Morphs: Custom Face Settings](#)

[Morph Edge Type: Hard, Hidden, Soft](#)

[Morph Texture Mapping and Alignment](#)

[Change Edge Type](#)

[Watch the Video](#)

Morph Classification and Properties Panel

Note: The controls of this panel are available only in Default Morph Settings, or if you have selected one or more entire Morph elements (no sub-elements).

See [Classifications and Properties Panel](#).

Settings of Library Part Elements

The following Archicad tools are used to place Library Part elements:

- Object
- Lamp
- Door
- Window
- Skylight
- Corner Window
- Wall End
- MEP tools

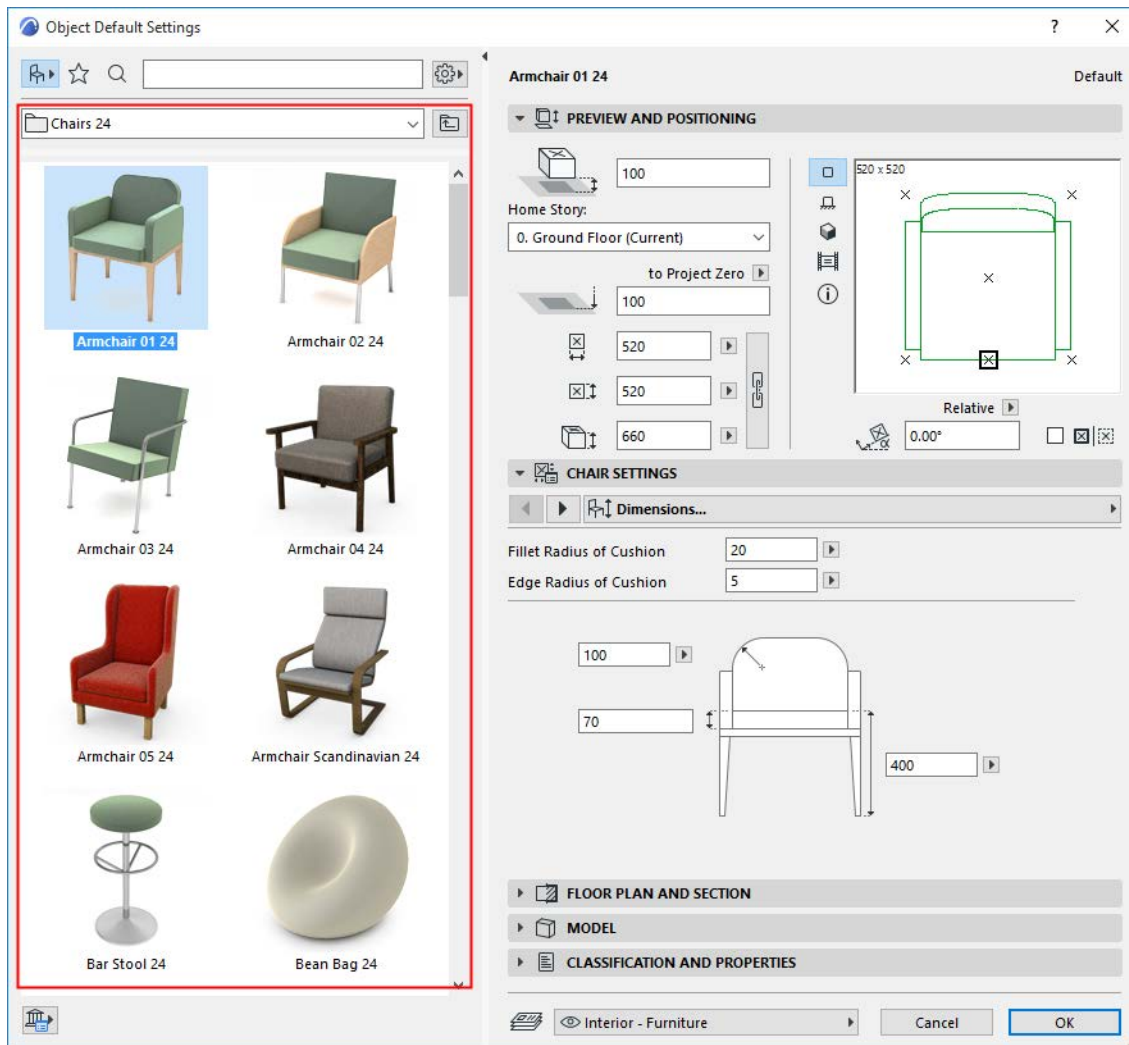
For specifics on the settings of library parts, see the following sections:

Object Tool Settings

Door/Window Tool Settings

Skylight Tool Settings

For all Library Part-type elements, the browser section on the left side of the Settings dialog box contains identical functions. These functions are described below.



Customize Dialog Box Panels

Library View/Favorites View

Preferred Display (List vs. Icons)

View and Apply Settings

Search for Library Parts/Favorites/BIM Components

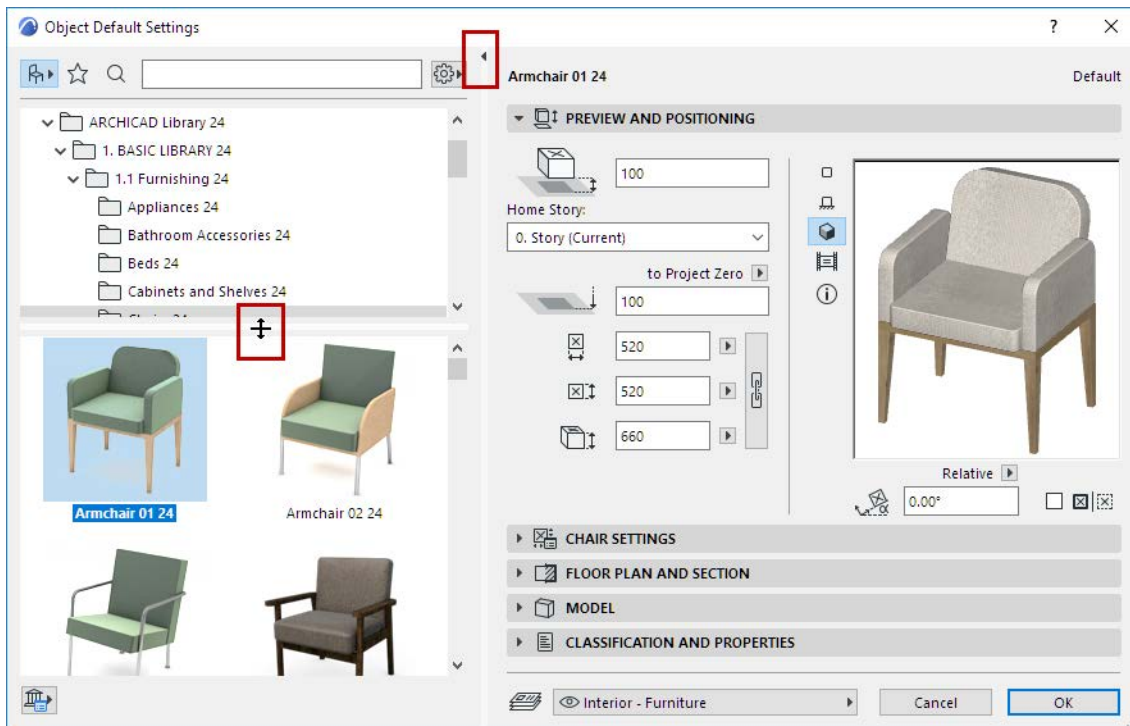
Settings in Library View

Settings in Favorites View

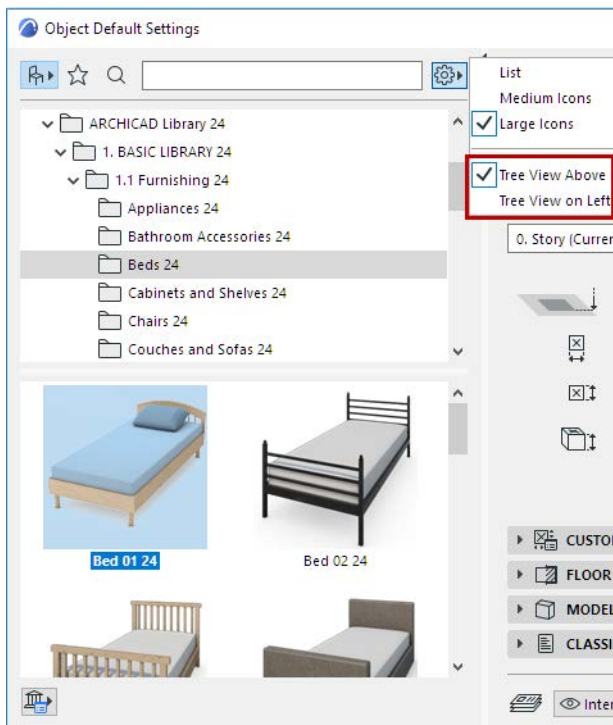
Browse Folders and Subfolders (Library View)

Customize Dialog Box Panels

- Double-click the vertical splitter bar, or click the black arrow, to open/close the browser area
- Drag or double-click the splitter to resize or close the folder tree area

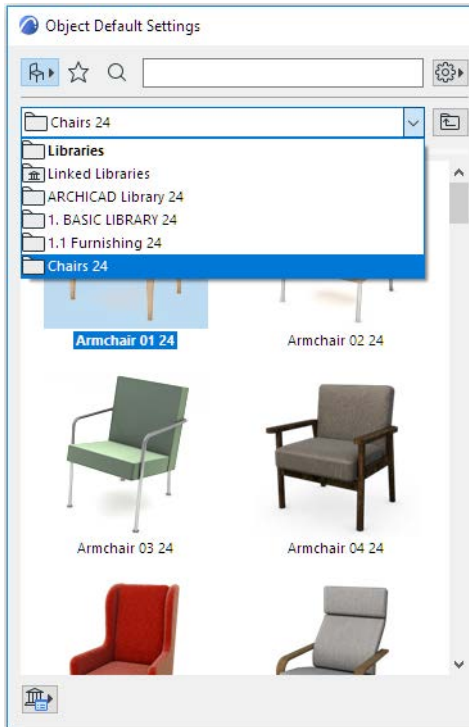


- From the browser Settings pop-up, use **Tree View Above** or **Tree View on Left** to change this preference



If the tree view is visible, the folder chooser is automatically hidden.

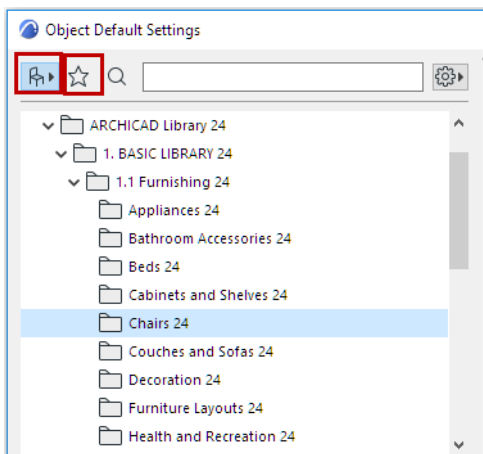
To use the folder chooser, hide the tree view:



Folder chooser is shown; Tree view is hidden

Library View/Favorites View

The browser area has two views:

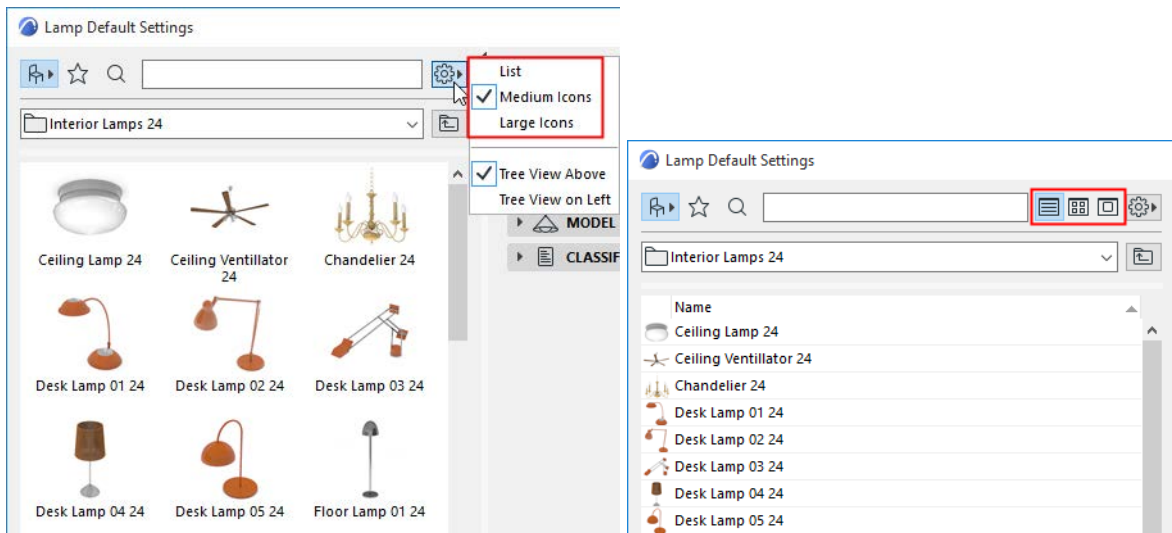


- **Library View** (chair icon). Displays Library Parts in a hierarchical structure. (See [Settings in Library View, below.](#))
- **Favorite View** (star icon). Displays only the Favorites for this tool. (See [Settings in Favorites View, below.](#))

Preferred Display (List vs. Icons)

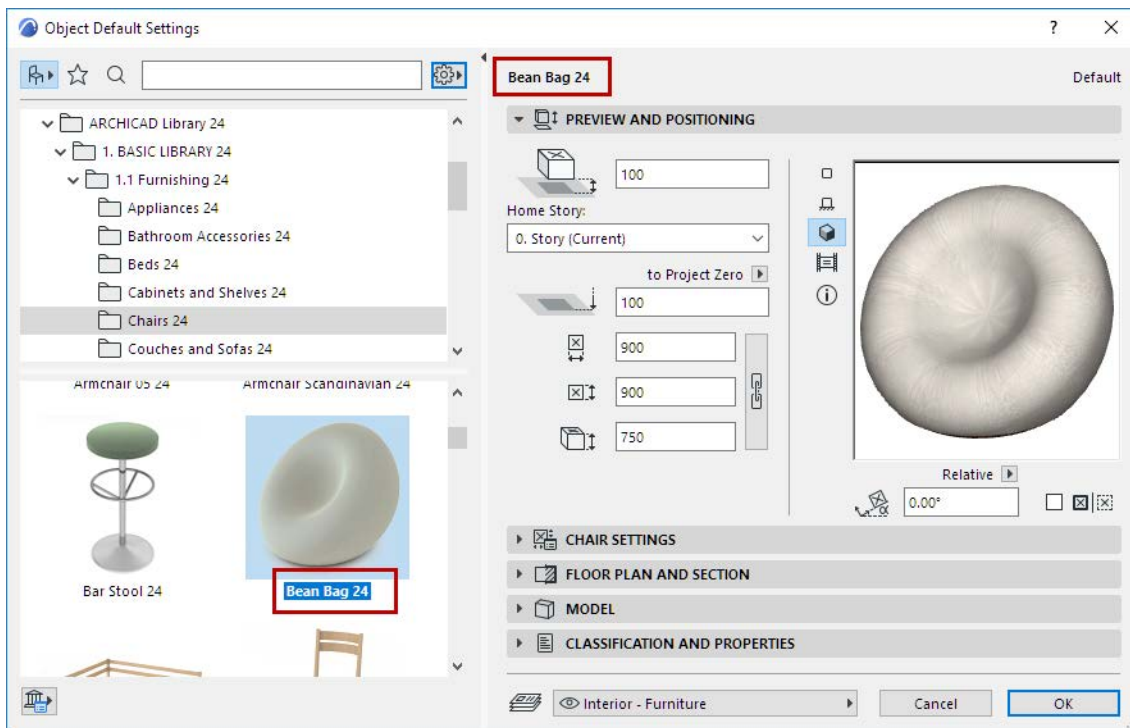
From the Settings pop-up, choose how to display the Library Parts: **List View**, or **Medium** or **Large Icons**. Stretch-and- release the Palette horizontally to also access these options as buttons.

Note: In List view, you can sort the contents alphabetically by Name, by clicking the Name header.



View and Apply Settings

Click any library part in the browser area to see its settings on the right side of the dialog box. The name of the library whose settings you see is shown in bold.



Reset to Default Settings

If you customize any Library Part's GDL settings, you can reset to its default using the Reset command from the Info Box:



Search for Library Parts/Favorites/BIM Components

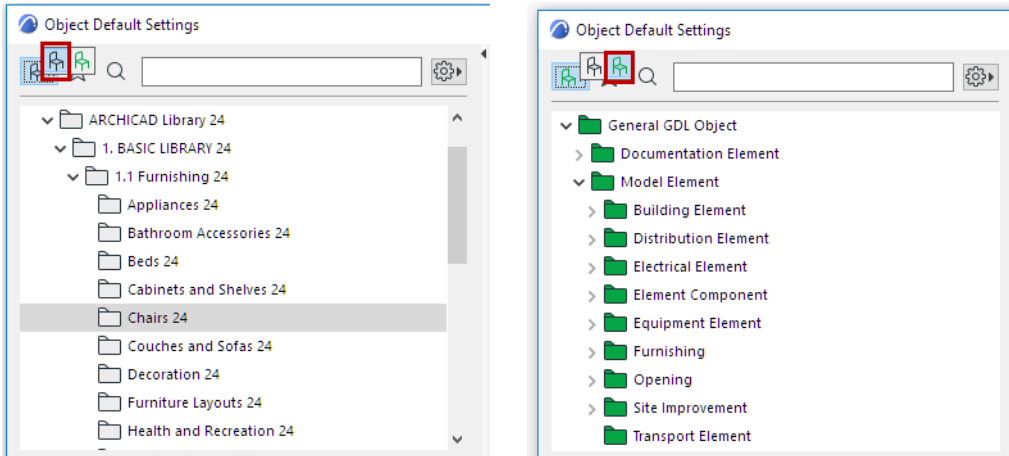
See [Search for Library Part](#).

See [Search Favorites](#).

Settings in Library View

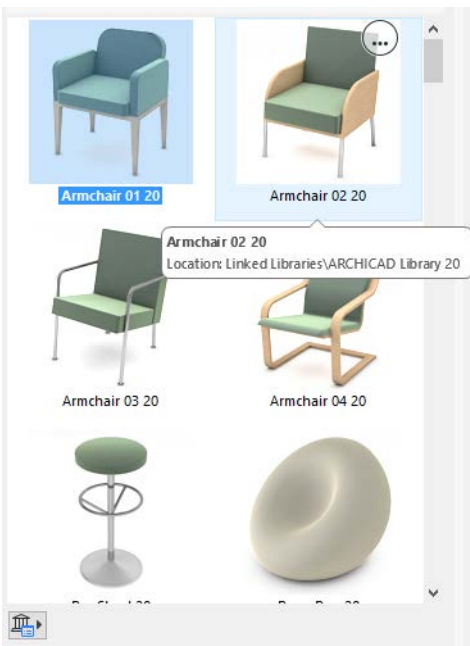
The Library View displays Library Parts in a hierarchical structure.

Click and hold to switch between folder view (the default) and subtype view:

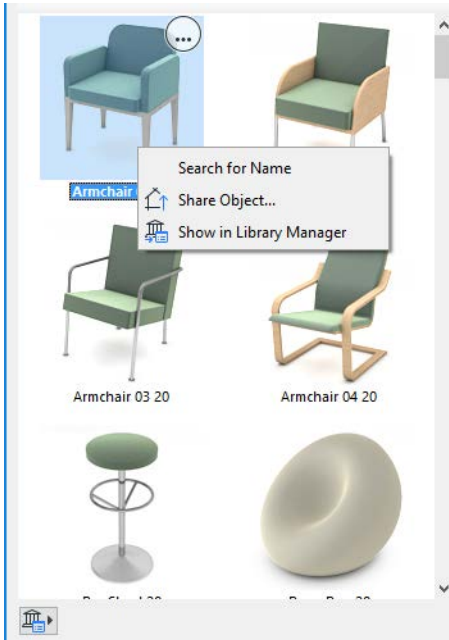


For information on subtypes, see [About GDL Object Subtypes](#).

Hover your cursor over a library part to see its name and location in the library.



A library part in a loaded library has the following context menu commands:



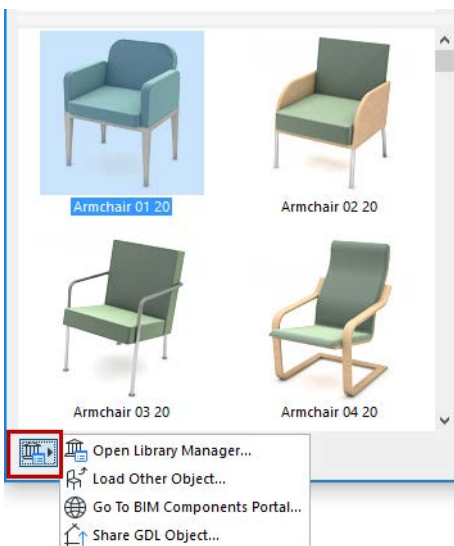
- Click **Search for Name** to start a search using the Library Part name
- Use **Share Object** to share the object on BIM Components

[See *Share Object on BIM Components*.](#)

- Use **Show in Library Manager** to see this library part in the context of its library hierarchy in Library Manager

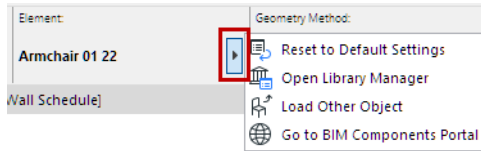
[See also *Library Manager*.](#)

Use the icons at bottom left to access the following commands:



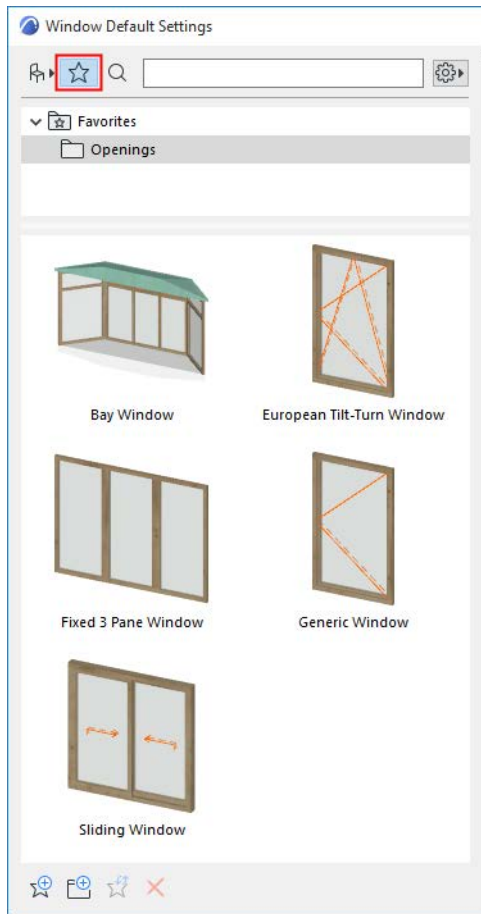
- Open Library Manager
- Load Other Object
- Go to BIM Components Portal
- Share GDL Object (if a library part is selected)

Note: Some of these commands are also available from the pop-up in the Library Part's Info Box.



Settings in Favorites View

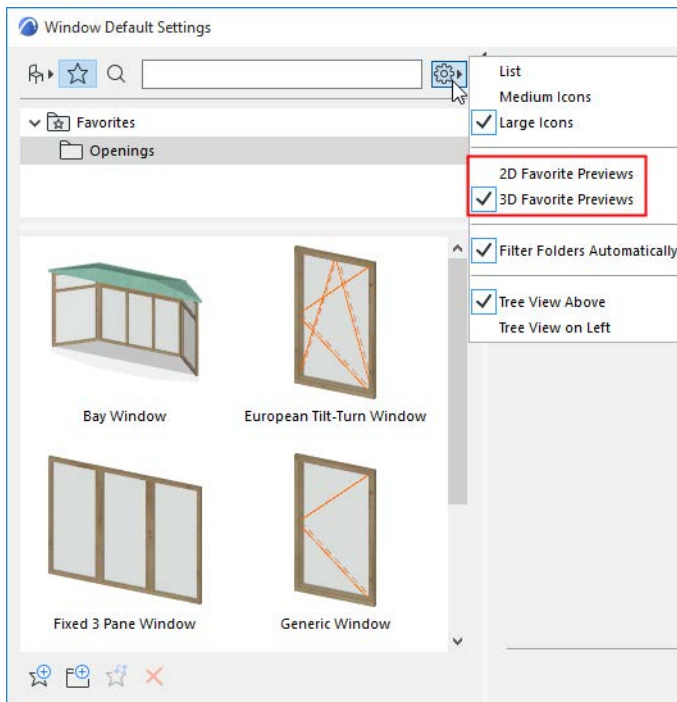
In the Favorites view of Library Part Settings, only the Favorites for this tool are shown.



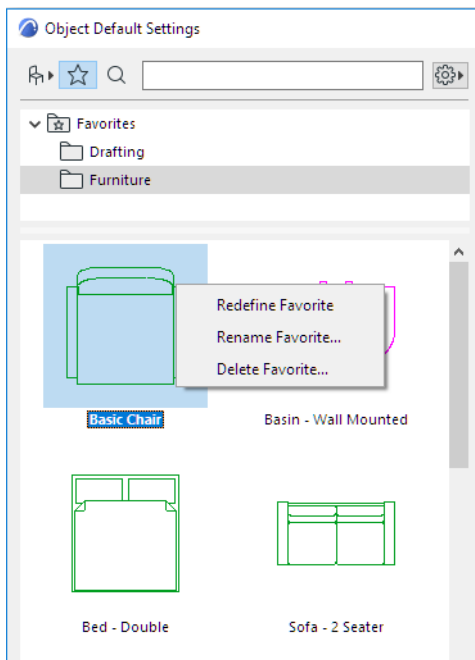
For general information, see [Favorites](#).

- Use the Additional Settings pop-up to switch between 2D and 3D Favorite previews

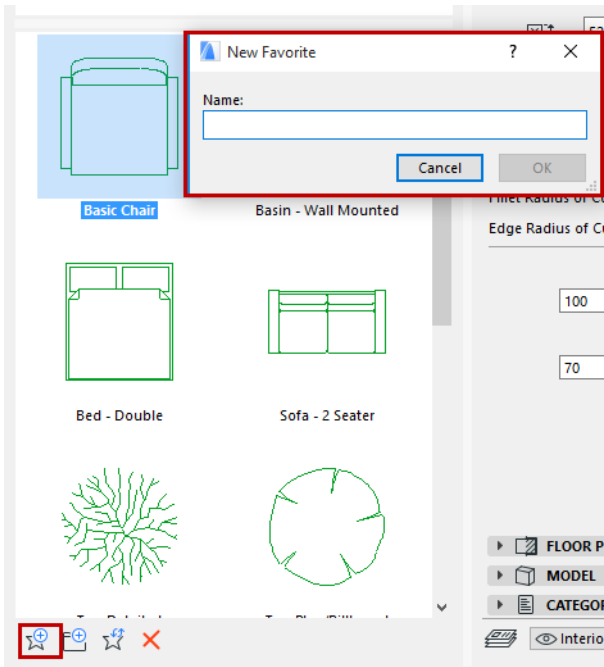
Note: To switch on the fly between the 3D and 2D preview of any Favorite, hover your cursor over the preview and click **Shift**.



- Hover the cursor over a Favorite to see certain details about it
- Click a Favorite to view/edit its settings in the right-hand panels.
- Available context menu commands:
 - **Redefine Favorite:** Click to redefine this Favorite based on the currently active settings (those in the right-hand side of the Object Settings dialog box)
 - Rename Favorite
 - Delete Favorite: A warning appears to confirm the delete.



Use the **New Favorite** command at the bottom of Library Part Settings (Favorites view) to store a new Favorite, using the current default Library Part Settings:



Optionally, create a Folder for the new Favorite.

For further Favorites management functions, use the Favorites Palette.

[See Favorites Palette and Edit Settings of One or More Favorites.](#)

Object Tool Settings

For a description of generic settings common to all tools in the Toolbox, see [Working in Tool Settings Dialog Boxes](#).

The **Object** tool is used to place object type library parts into the project.



The left-hand browser area of the **Object Settings** dialog box is similar to those of other Library Part elements.

See [Settings of Library Part Elements](#).

Browse Folders and Subfolders (Library View)

In **Object Tool settings**, you can access a wide variety of objects that are not part of a dedicated subtype. These objects are divided into three main folders: Basic Library; Visualization Objects; and Add-on Library, and several sub-folders. As you can see by the folder names, the Basic Library contains all kinds of Furnishings (Beds; Chairs; Office Equipment, etc.); Decorations (such as clocks and vases); Health/Recreation items (such as a piano, a billiard table, a TV). Additional folders contain Building Structures (such as fences and moldings); Special Constructions (such as fireplaces and shutters); Mechanical Elements (such as air conditioners and elevators); and 2D Elements (such as electric and graphic symbols.)

The Visualization folder contains Site improvements (e.g. trees) and objects depicting People and Vehicles.

For information on searching for library parts, see [Search for Library Part](#).

Panels of the Object Settings Dialog Box

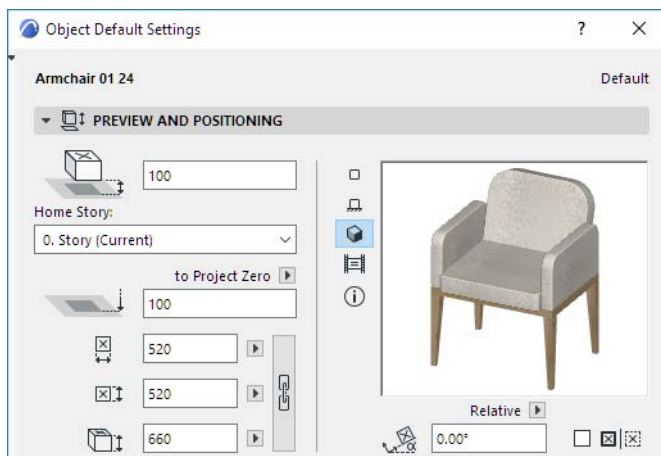
[Object Preview and Positioning Panel](#)

[Object Custom Settings Panel](#)

[Object Floor Plan and Section Panel](#)

[Object Model Panel \(Not Visible by Default\)](#)

Object Preview and Positioning Panel



Bottom Offset to Home Story: Enter a value to define the object's base elevation relative to its Home Story.

Bottom Elevation [to Reference level]: Shows the object's absolute base elevation, measured to Project Zero or to any other custom-defined Reference Level.

[See also Reference Levels Preferences.](#)

Notes:

- Reference levels are defined at **Options > Project Preferences > Reference Levels**.
- This is a calculated value, not a link. If you change the Reference level, the bottom of the object will not change its position.

Dimensions 1 and 2: enter the two orthogonal size values (e.g. length and width) of the object.

Click the chain icon to keep these values proportionally constant.

Height: Enter the object's height.

Enter a rotation angle for the object, if any.

Use the pop-up to define how this angle value should be interpreted when the element is placed:

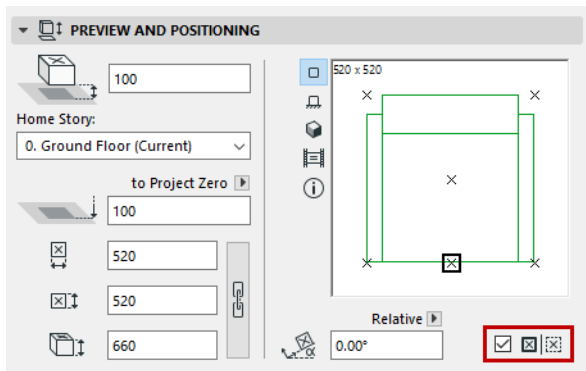
- **Relative to Orientation** (the angle will be measured relative to the Oriented View);

[See also Set Orientation.](#)

or

- **Absolute to Coordinate System** (the angle will be measured from the (0,0) point of the project coordinates).

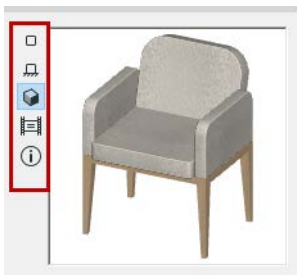
Check the **Mirror Library Part** box to mirror the chosen Library Part when placing it, or to mirror a selected object. (View the preview after checking this box to see how the object will be placed.)



Preview Options

The **Preview Area** at the right shows you the object in different displays, depending on which of the icons (at left) that you choose. As you set different parameters for the object, keep an eye on this preview to see the effects of your edits.

Click among the Preview Options to vary the preview picture:



Move the cursor inside the **Preview Area** and the cursor will change to a “flip” arrow, allowing you (in the Plan or 3D previews) to alternately view each side of your Library Part.



The preview options, from top to bottom:

- Plan view
- Front view
- 3D view
- predefined preview picture

Note: To modify the predefined preview picture, use the Preview Picture controls in the GDL Object Editor.

[See Preview Picture.](#)

Move the cursor inside the **Preview Area** and the cursor will change to a “rotate” arrow, allowing you (with successive clicks) to rotate the Plan or 3D preview.

Information

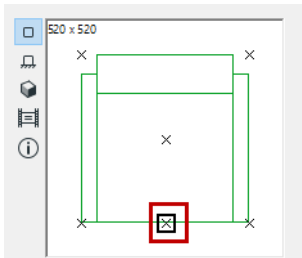
Click the information icon to view a brief description of the object.

Click the **Info** button to see further information about the library part (e.g. author, description). If you share your own library part to the BIM Components Portal, you have the opportunity to enter this information during the process.

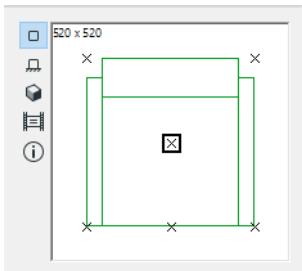
[See *Share Object on BIM Components*.](#)

Placement Anchor

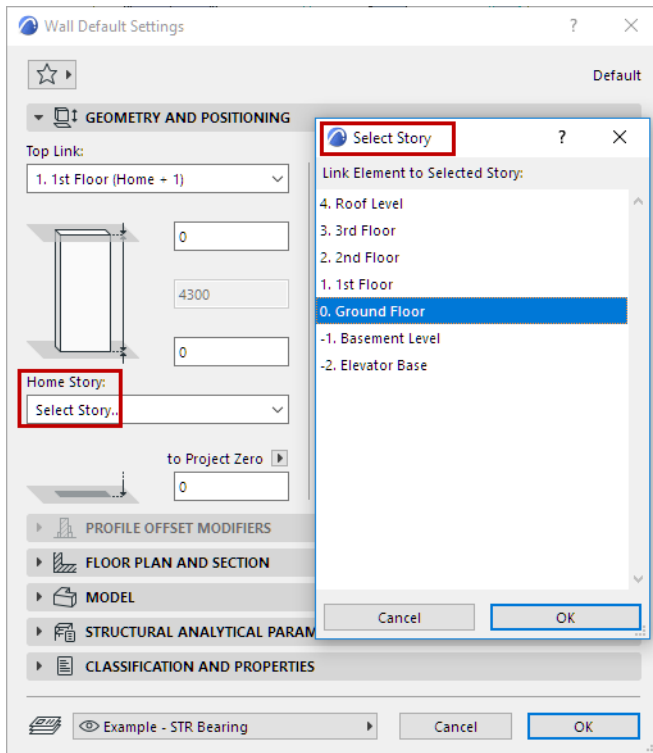
Objects can be positioned by their hotspots, defined in the 2D symbol of the Library Part. One Hotspot is initially defined as the primary hotspot. This hotspot is marked with a highlighted rectangle, and it will act as the default insertion point and anchor point for the Object.



The other hotspots are displayed as X's. Click any of them if you wish to use it as an insertion/anchor point instead of the default primary hotspot.



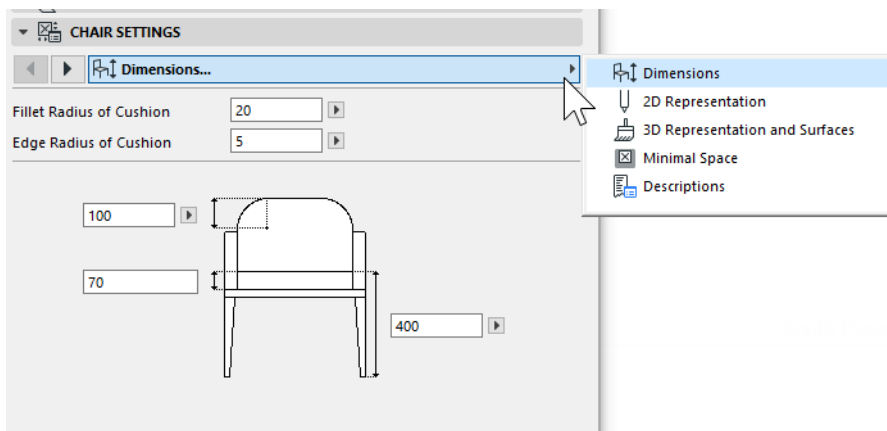
Home Story: Choose a Home Story for the Object: by default, it is the Current Story, but you can click Select Story instead and choose any other Story as the Home Story.



For more information, see [Home Story](#).

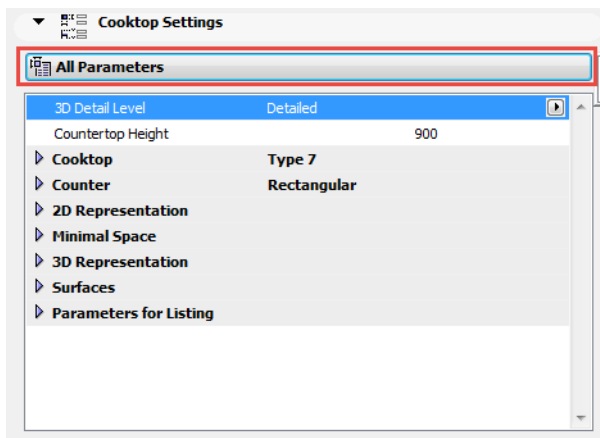
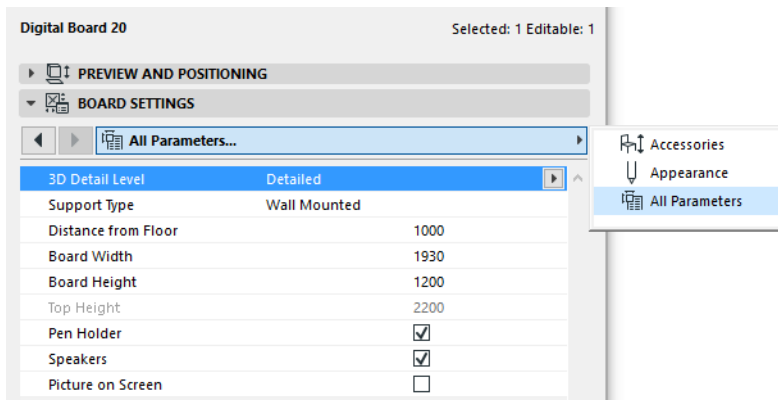
Object Custom Settings Panel

The available parameters vary depending on the object. Where applicable, click the pop-up menu to access different parameter categories.



For information on transferring modified parameters from one object to another, see [Parameter Transfer Between Objects](#).

- All Parameters: Some objects have Parameter lists as an alternative interface for the graphical Custom Settings. If you prefer the Parameter list, choose this from the Custom Settings Panel pop-up:

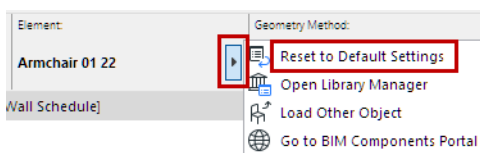


The **Minimal Space** parameter (included with many Objects from the default Archicad Library) defines the area surrounding the placed object that you want to keep free. You can opt to display the minimal space of all objects in 2D views, depending on the global options set at Model View Options.

See [Miscellaneous Settings for Library Parts](#).

Reset to Default Settings

If you customize any Library Part's GDL settings, you can reset to its default using the Reset command from the Info Box:



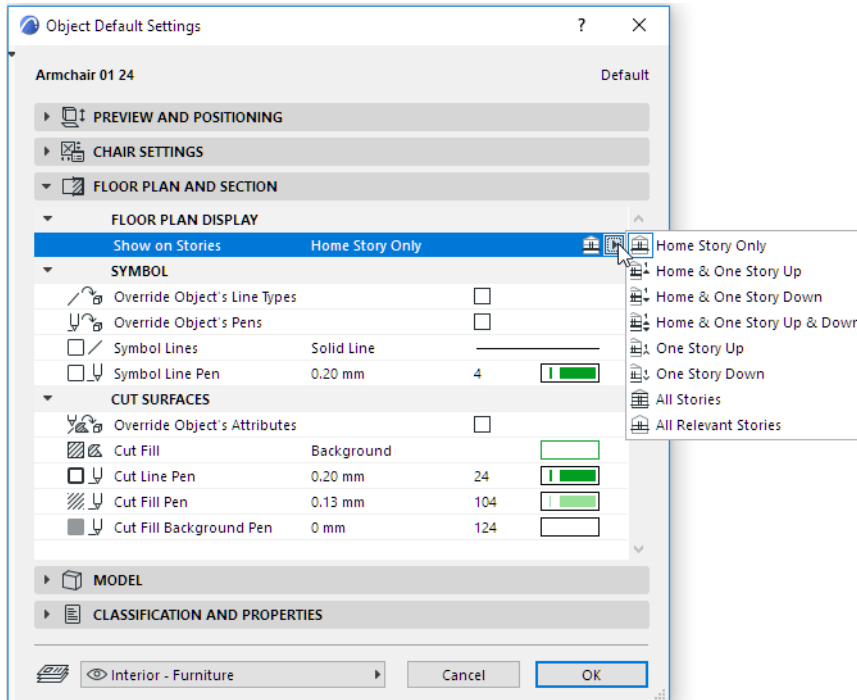
Object Floor Plan and Section Panel

Use this panel (or the equivalent controls in the Info Box) to do the following:

- Define the Stories on which the Object should be displayed
- Optionally override the 2D representation parameters that have been defined for the object in its GDL script (see [Override Object Attributes, below](#)).

Show on Stories: Use this pop-up (where available) to control the stories on which objects will be displayed: on its Home Story only, or in outline form on other stories as well.

The Object/Lamp can also be shown One Story Up or One Story Down only.



For more information, see [Show On Stories](#).

Override Object Attributes

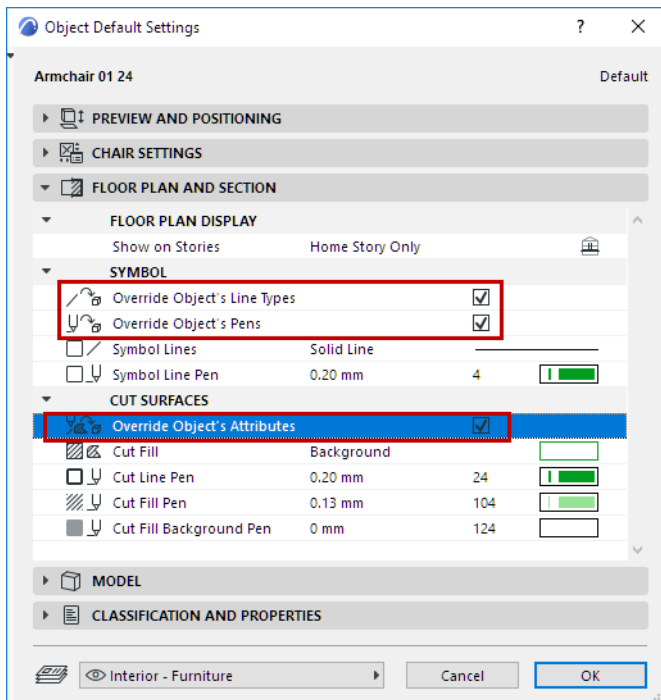
Typically, you will define the 2D attributes (line type, pen, fill) for an object using the 2D Representation settings in the Custom Settings panel, described above.

See [Object Custom Settings Panel](#), above.

However, you may wish to override these attributes, using the simplified settings of the Floor Plan and Section Panel of Object Settings.

Define the attributes for cut, uncut and overhead lines, pens and fills as needed.

For these settings to take effect, make sure that the applicable **Override Object's** /Line Types/Pens/Attributes checkbox(es) are *on*.

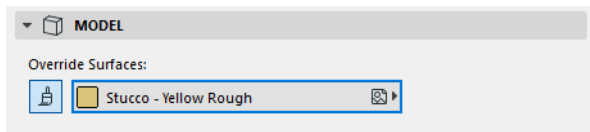


Note: For some objects, no GDL Object parameter is defined. In this case, the object will be displayed using the attributes defined in the Floor Plan and Section panel (regardless of the Override checkboxes).

Object Model Panel (Not Visible by Default)

This panel is not visible by default. If you need it, make the panel visible in Door/Window Settings, using **Options > Work Environment > Tool Settings Dialog Boxes**.

Use this panel to if you want to define one single Surface for the Object.



Activate the Surface button, then choose a Surface from the pop-up.

Special Lamp Parameters



The left-hand browser area of the **Lamp Settings** dialog box is similar to those of other Library Part elements.

See [Settings of Library Part Elements](#).

The right-hand Settings panels of the Lamp Tool dialog box controls are similar to those of Object Settings.

See [Object Tool Settings for a description of the common Settings panels](#).

Basic Light Parameters

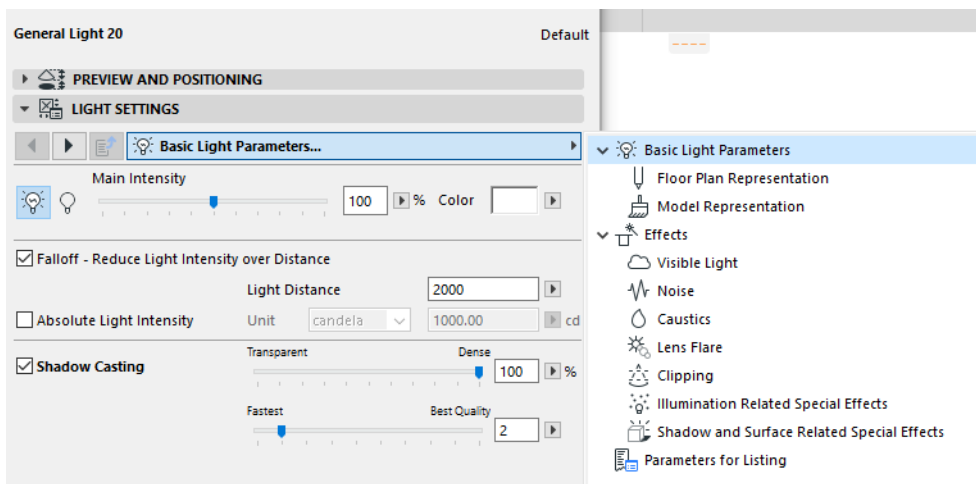
All lights and lamps have certain parameters in common, on the Light Settings Panel of Lamp Tool Settings.

Some of these are described below:

See [Basic Light Parameters](#).

General Light Sources and Effects

The General Light Sources folder contains light sources that serve to illuminate the project, and have many sophisticated optional effects. The effects are advanced-level settings that are separated from the basic parameters in the Settings dialog box.



These effects parameters are introduced below, in this section:

See [General Light Sources and Effects](#).

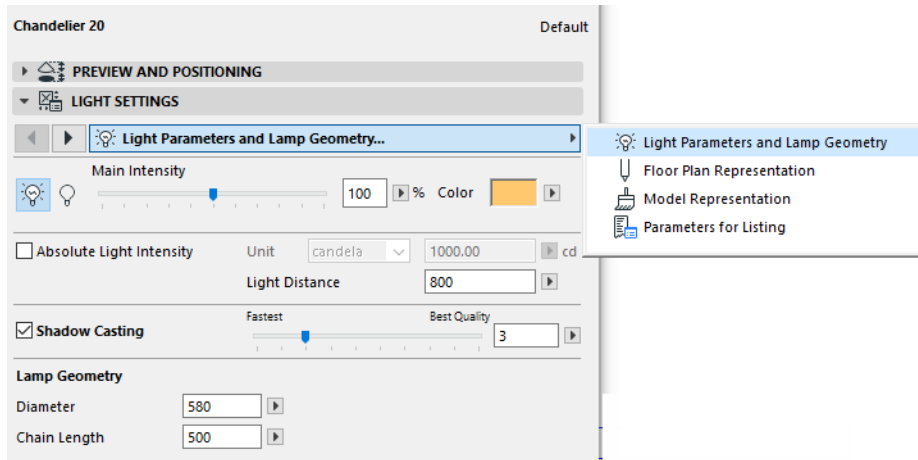
[Watch the Video](#)

The effects you set for your lamps and light sources are taken into consideration for Photorenderings. As a rule, the global effects are set in PhotoRendering Settings (turn on GI, Lens effect, Intensity). You can then fine-tune these in your individual lamp/light objects.

Note that certain individual Lamp parameter settings will take effect only if the corresponding global parameter in PhotoRendering Settings is switched on. These situations are noted in the descriptions of light effects, which follow.

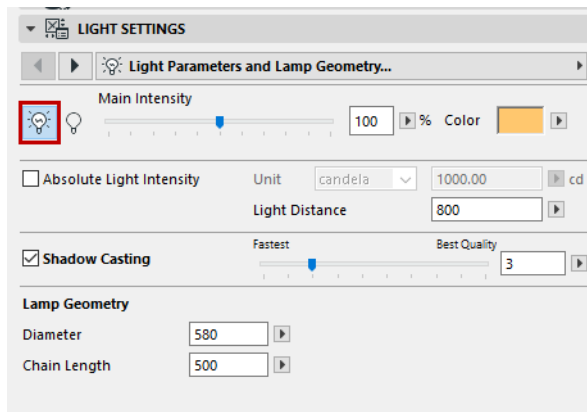
Basic Light Parameters

All lights and lamps have basic light parameters in common, on the Light Settings Panel of Lamp Tool Settings. Some of these are described below.



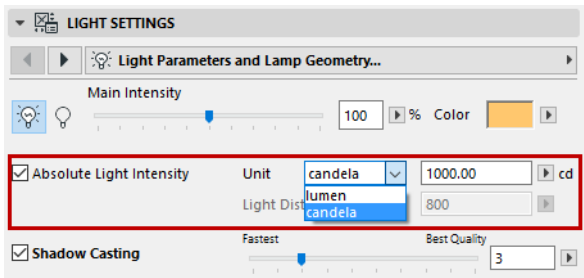
Turn Light On/Off in PhotoRendering

- Click the light bulb on the Light Parameters panel to turn the Light ON or OFF. The Light or Lamp must be turned on here in order for Lamp-related PhotoRendering Settings to have an effect.



Light Color and Intensity

- Set the light source's color by clicking the Color field.
- The **Main Intensity** of the light can be adjusted with this slider. By default, it is set to the middle of the slider (at 100 percent), which corresponds exactly to the light intensity defined for this lamp with the controls below. Adjust the Main Intensity slider to either strengthen or weaken this defined light intensity.
 - **Absolute Light Intensity:** Define the light source intensity in candelas or lumens.

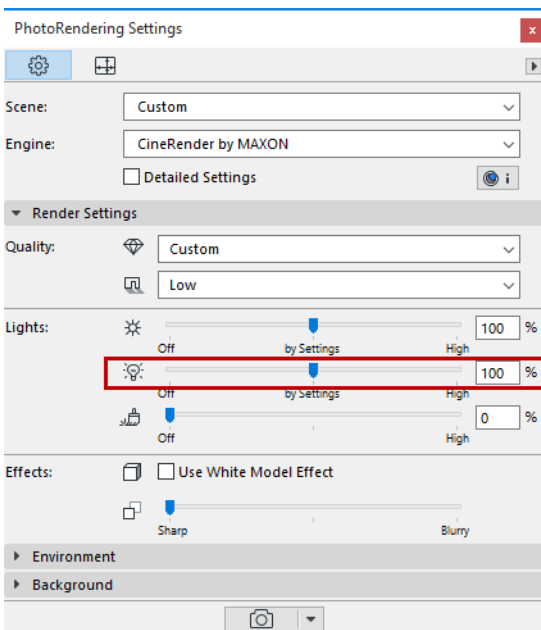


- **Light Distance:** (Available for lamps only if Absolute Light Intensity is unchecked.) This is an alternative way of defining the light source intensity. Light Distance represents the distance over which the source light intensity changes from 100% to 0%.
- **Falloff - Reduce Light Intensity over Distance:** (Available for some general light sources). Check this box to reduce intensity over distance. Falloff type is always Inverse Square.

Effect of Lamp Intensity Settings in PhotoRendering

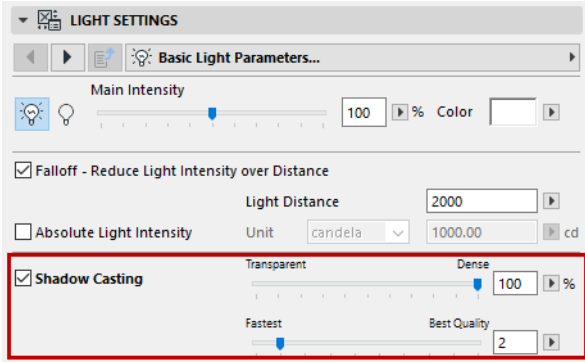
By default, the individually set intensity of each placed light object will be in effect in photorenderings with Cineware. This is shown in the Basic Render Settings “Lamp Intensity” slider, which is set to “By Settings” (100 percent) by default.

Adjusting this slider in PhotoRendering Settings will override individual lamp settings and either increase or decrease them accordingly.



Shadow Casting

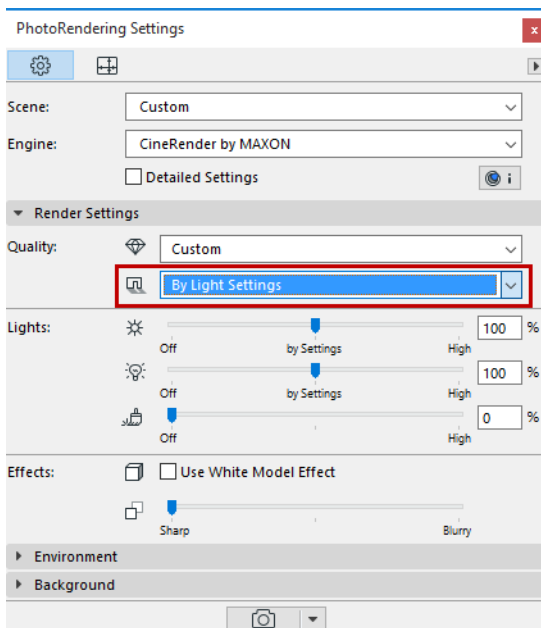
Check this box in the Light Settings dialog box to have the light cast a shadow.



The slider position defines shadow quality; the higher the quality, the longer it will take to render. The default value of 2 enables fast rendering and is usually adequate. You should adjust this slider only if the light effect seems too grainy.

Effect of Shadow Casting Settings in PhotoRendering

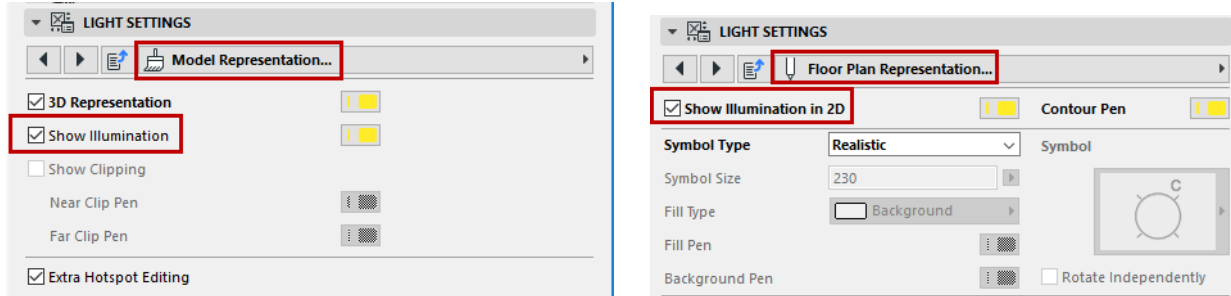
Individual Lamp settings will take effect in Cineware PhotoRenderings only if you have set the Shadow Quality option in the PhotoRendering Palette's Cineware Basic Settings to "By Light Settings."



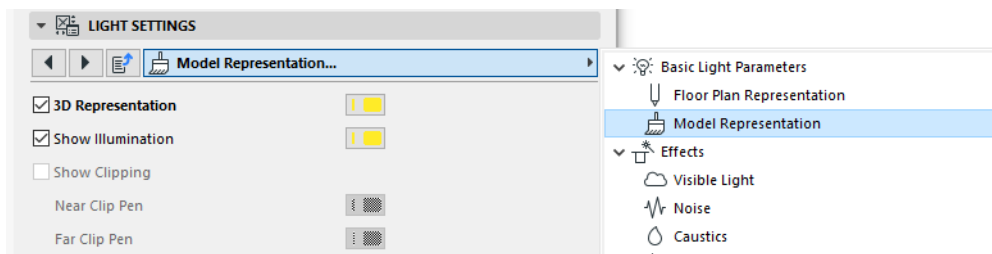
Light Object: Show Illumination in 2D and 3D

You can depict a light object’s arc of illumination in 3D windows (and, for General Light Objects, on the Floor Plan as well.)

Use the Floor Plan and Model Representation page of Light Settings panel. The arc shown corresponds to the Light Distance value (set on the Light Parameters page).

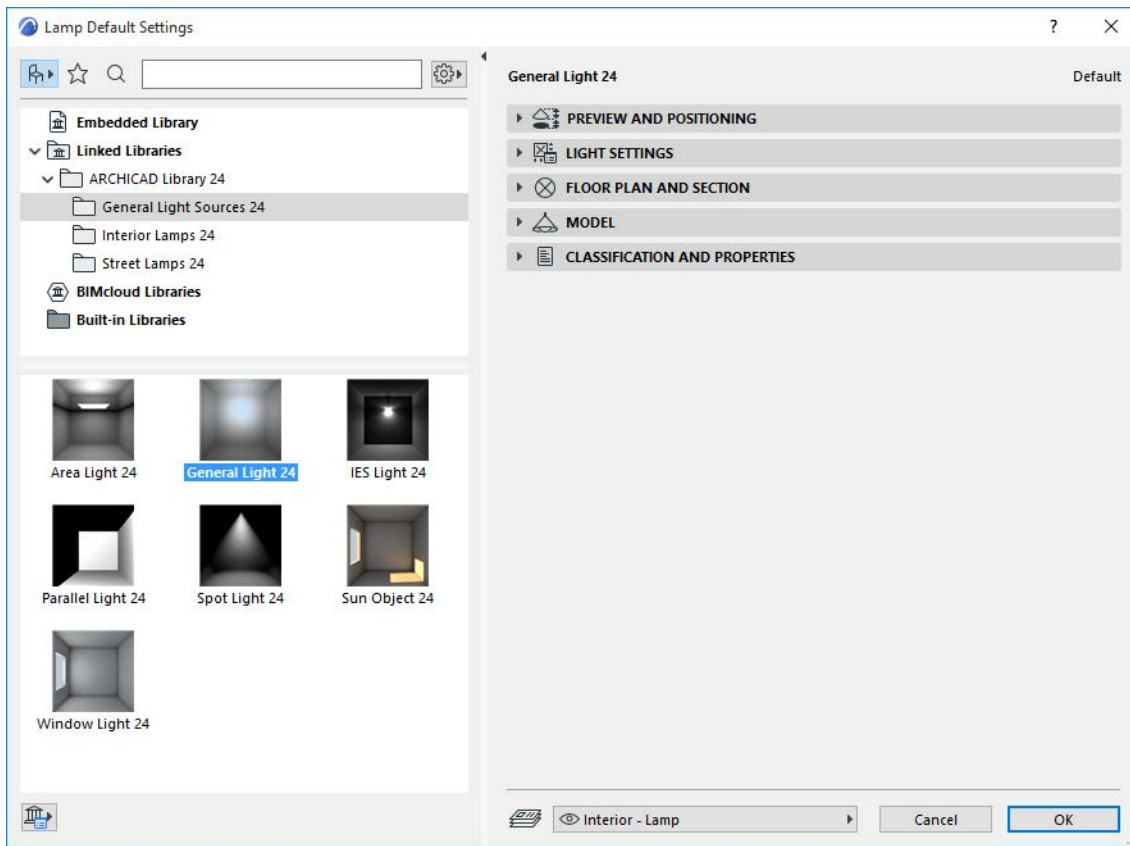


General Light sources have a separate Model Representation settings page for 3D:



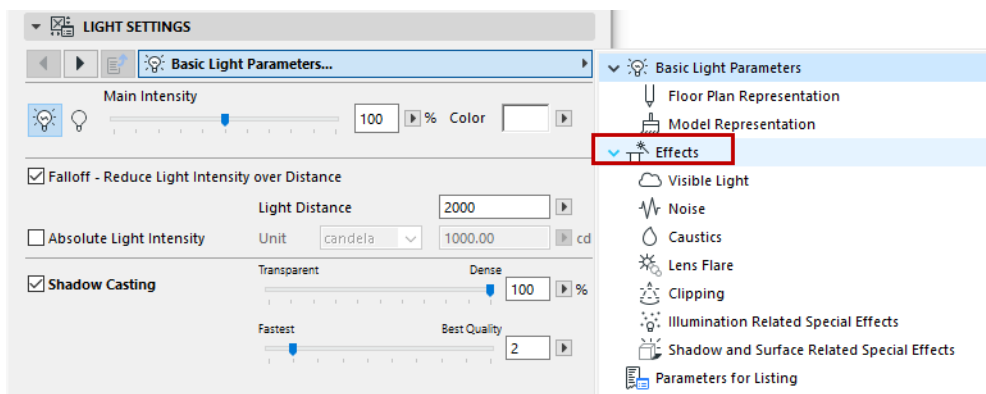
General Light Sources and Effects

Choose from among seven light objects in this category to provide a general source of light.

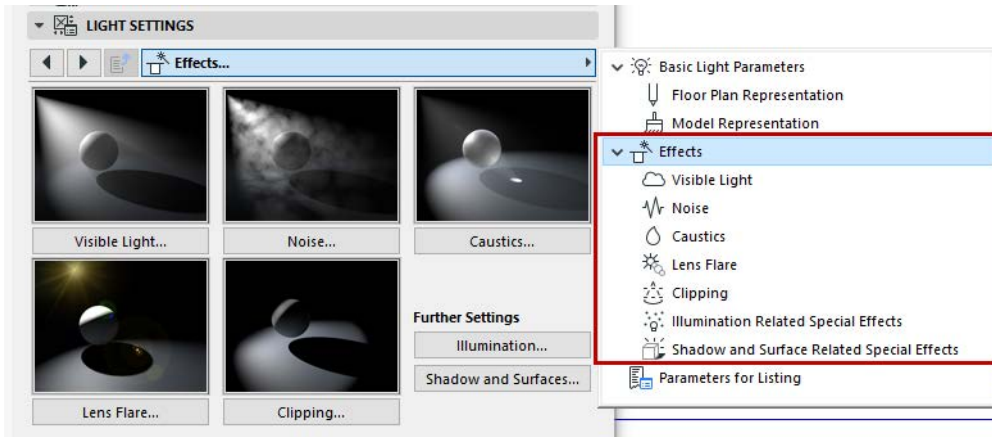


[Watch the Video](#)

General lights, unlike lamps in the Interior or Street Lamp categories, include special effects such as Visible Light and Noise parameters. Click the parameters pop-up to access these Effects parameters.



The main Effects page contains graphical links to the sub-pages:



On each page, one or more sample images helps you understand the effect of the parameters.

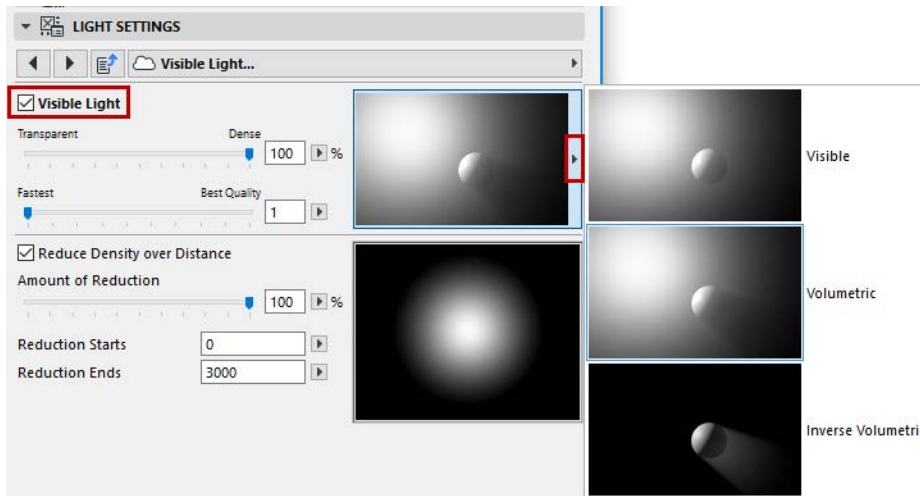
Note: Not all effects are available for all General Light objects.

Visible Light

This effect controls properties of the light beam in the scene.



Use the pop-up to choose a Visible Light type: Visible, Volumetric, or Inverse Volumetric. The images on the pop-up help you understand the options.



- **Visible:** This type of effect will produce visible light that passes through all objects and does not affect them; this effect casts no shadow in the beam.
- **Volumetric:** Use Volumetric if you want your Visible light effect to cast a shadow.
- **Inverse Volumetric:** This is the opposite effect of the volumetric light: the light is visible where the light cone would normally be in shadow. This can be useful if you want to give the impression that an object is radiating light (e.g. a neon company logo on the side of a building).



Density/Quality: Use the two sliders to set these parameters of the light beam.

- **Transparent/Dense slider:** defines how dense this light effect is.
- **Fastest/Best Quality slider:** The “Fastest” setting means few samples are taken, and the PhotoRendering effect is generated relatively quickly. The Best Quality will take more samples and take longer.

Reduce Density over Distance: Check this to have the light density decrease toward the edges.

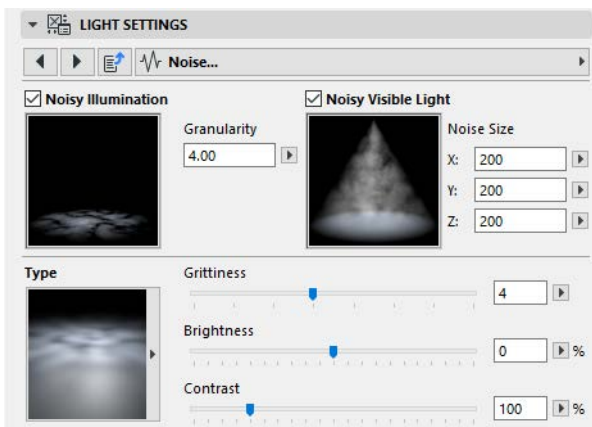
- **Amount of Reduction:** 100 percent means the light fades entirely.
- Use the Reduction Starts and Reduction Ends fields to define where to begin and end this reduction.

Noise

Noise adds irregularities to the light source in two ways, to make the effect more realistic.



The images in the Light Settings Noise controls help illustrate what the parameters mean.



- **Noisy Illumination:** Check this box to add noise to the illuminated surface.
- **Noisy Visible Light:** Check this box to add noise to the Visible light beam itself.

Use the pop-up to choose a Noise type: plain Noise, or Soft/Hard/Wavy Turbulence (which produce a cloud-like effect). The images on the pop-up help you understand the options.

Use the sliders to further adjust the Noise effect you have chosen:

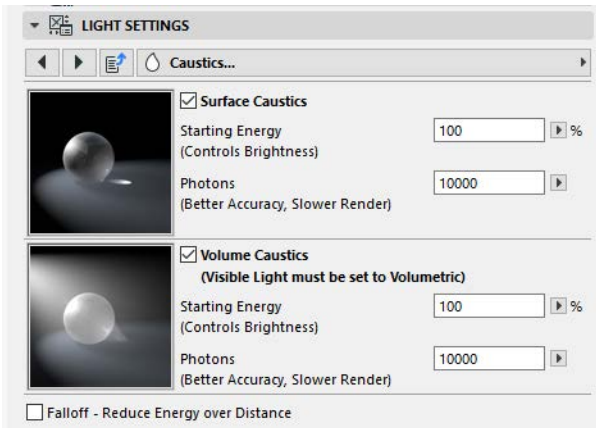
- **Grittiness:** A higher value will create a larger number of irregularities for a grainier effect.
- **Brightness:** Affects the overall brightness of the white “blotches” of the noise effect.
- **Contrast:** Affects the black/white contrast of the noise effect.

Caustics

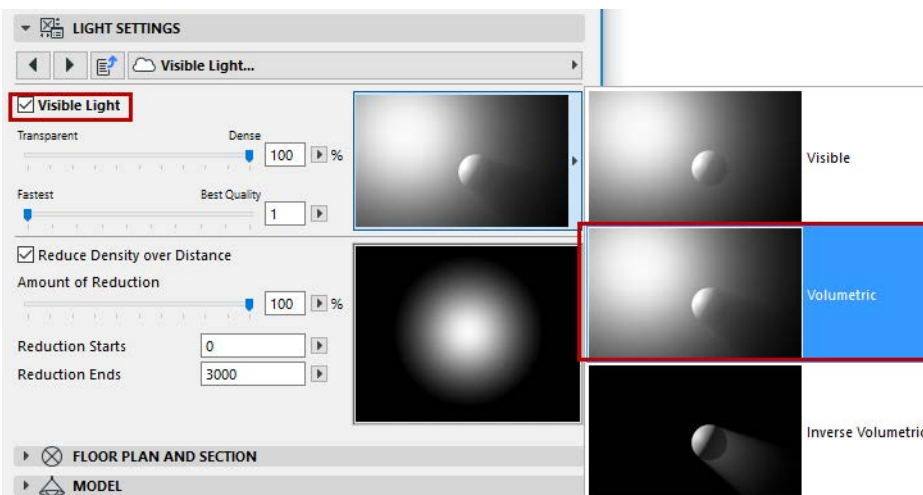
Caustics are patterns of focused light created on surfaces and within a visible light beam.

Check the box of the types of Caustics you wish to create (you can use both effects at once):

- **Surface Caustics:** Achieves a Caustic effect on the illuminated surface.
- **Volume Caustics.** Achieves a Caustic effect within the light beam itself.



Important: The Volume checkbox will have an effect only if you have chosen enabled the Volumetric option for the Visible Light parameter (Visible Light page of Light Settings):



For both types of Caustics, you can set the following parameters:

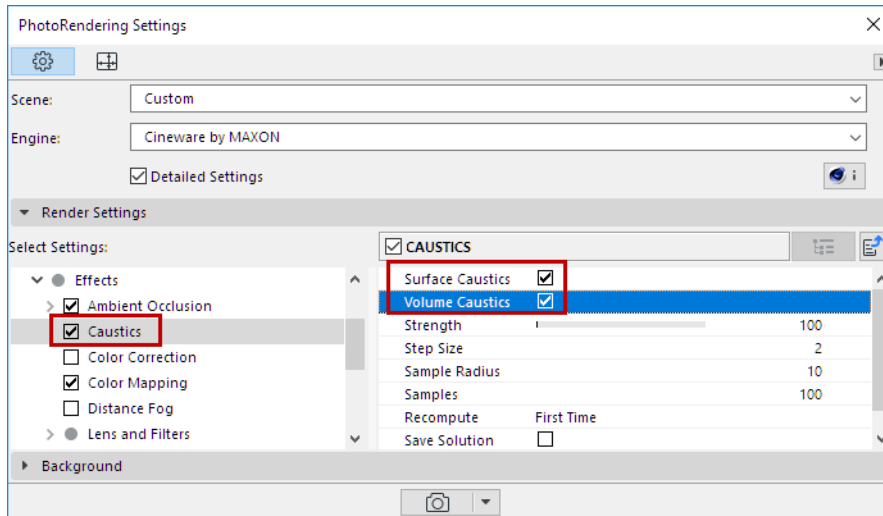
- Starting Energy: Affects the brightness
- Photons: Affects the accuracy and quality of the Caustic effect; higher values increase rendering time.

Falloff - Reduce Energy over Distance: This is unchecked by default. If checked, you may not see the Caustic effect, if the light source is far enough away from the surface.

Using Caustic Effect in PhotoRendering

The Caustics settings of your Light Objects will have the desired effect in PhotoRenderings only if

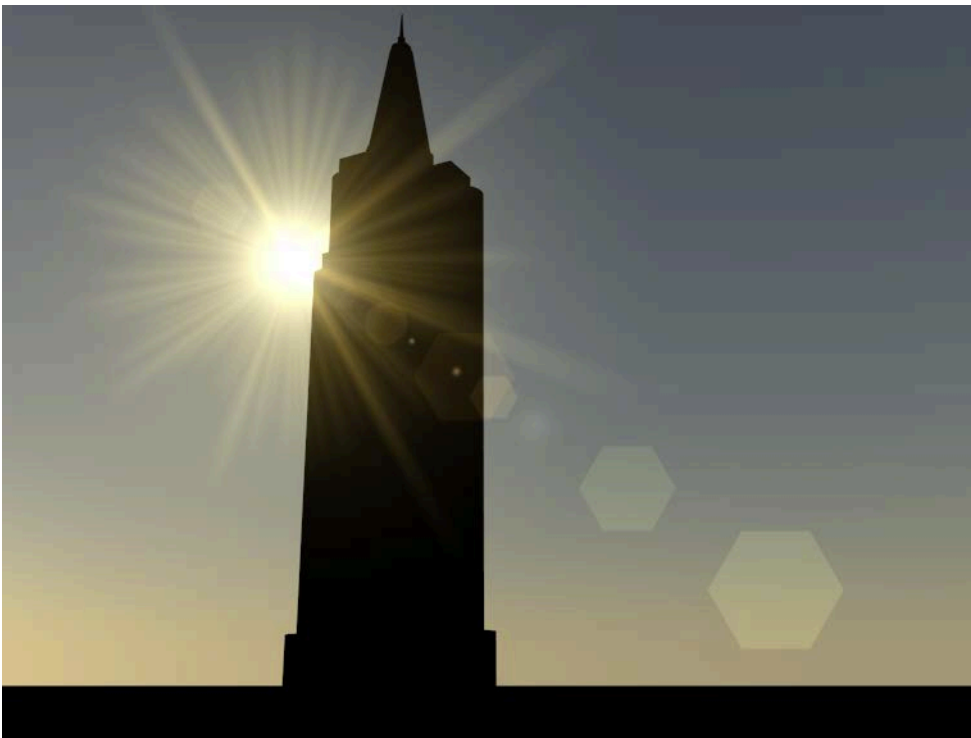
1. the corresponding Caustics effects are enabled in PhotoRendering Settings ([see *Caustics \(Cineware Effect\)*](#)); and



2. the Caustic channel of the affected Surface(s) are also turned on.

Lens Flare

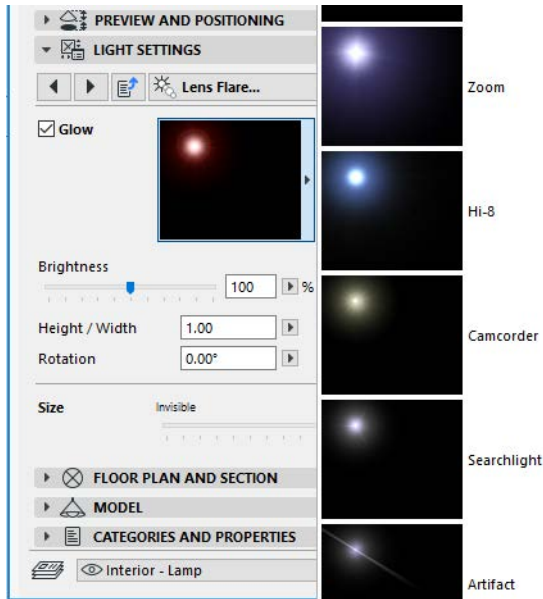
Lens Flare simulates the aberrations of real-world camera lens systems and film material. Its two main components are Glow and Reflexes.



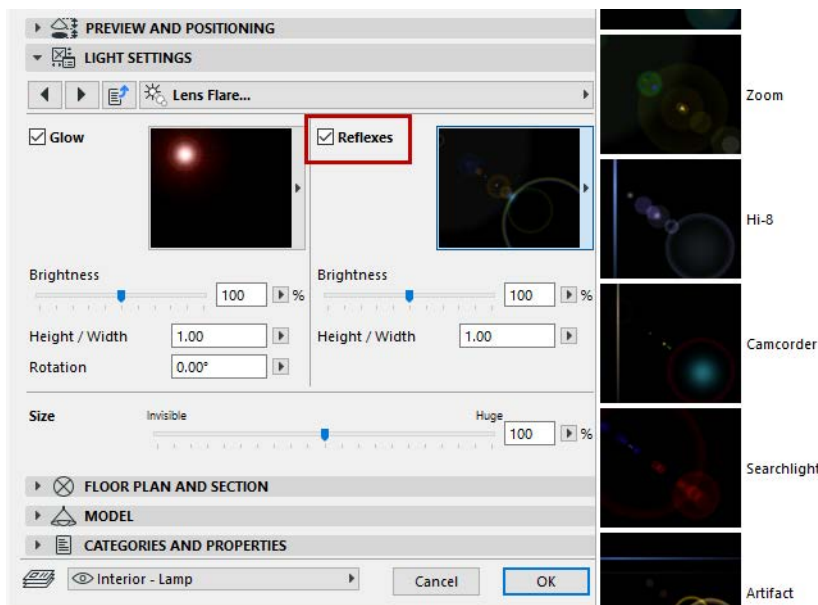
- **Glows** are a type of over-exposure to light
- **Reflexes** are lens reflections

You can choose from a large number of predefined glow and reflex effects: click the pop-ups to see the images for each.

Glow: Check this box and use the pop-up to choose a Glow style for your Lens Flare.



Reflexes: Check this box and use the pop-up to choose a Reflex style for your Lens Flare.



Use the **Brightness** slider to adjust the brightness of the Glow and/or Reflexes in the lens flare.

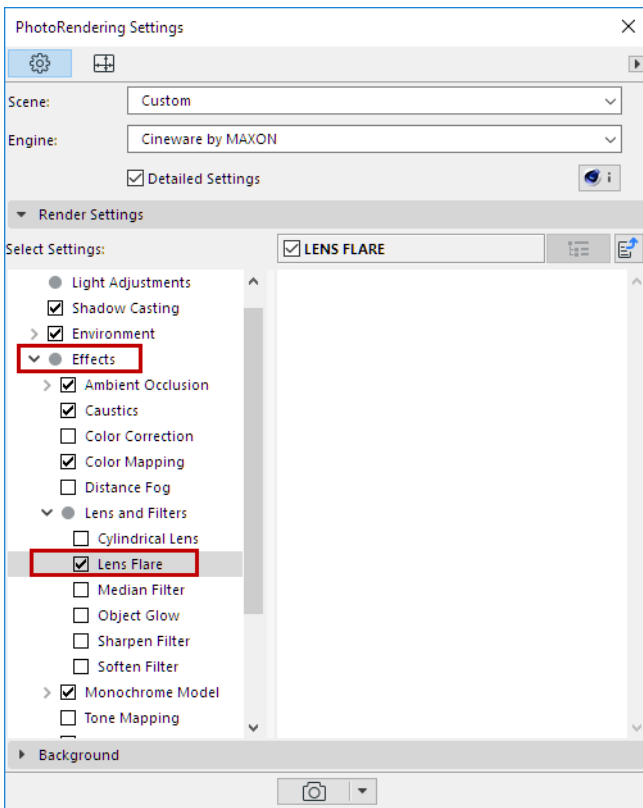
The **Height/Width** value express a proportion; changing this value will distort the shape of the effect (glow or reflex).

Use the **Rotation parameter (enter a value in degrees)** to rotate the Glow element.

Using the Lens Flare Effect in PhotoRendering

Lens flare is a so-called “post effect” which is applied after the rendering calculation is completed.

The Lens Flare settings of your Light Objects will have the desired effect in PhotoRenderings only if you turn on Lens Effects in PhotoRendering Settings (Cineware engine, Detailed view):

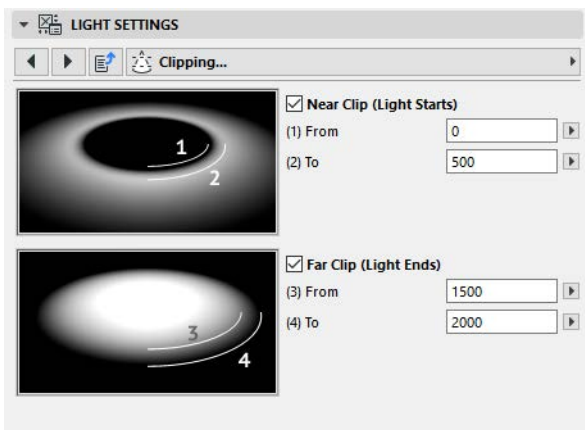


Clipping

This effect clips the light effect between two points.

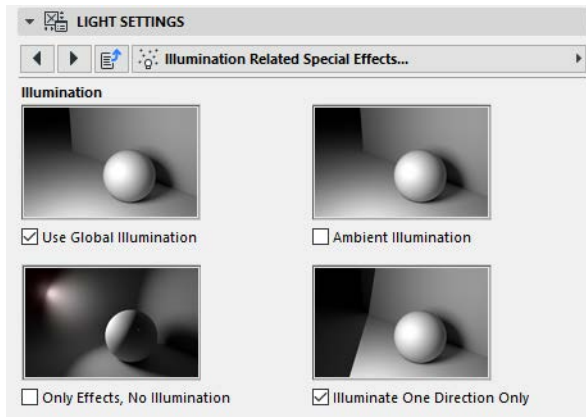
Check the boxes to create a Near Clip and/or Far Clip.

For each Clip, define where the clip effect should start and end.



Illumination-Related Special Effects

These effects are important general parameters for your light effects. Try out the checkboxes to see how the sample image changes, to help you understand the effect.



Use Global Illumination: This is on by default. Global Illumination (GI) is a sophisticated method for achieving realistic lighting effects in your model. The algorithm simulates the effects of not just direct light sources, but indirect light effects - that is, inter-reflections caused by light rays bouncing off surfaces.

Using Global Illumination can greatly increase render time.

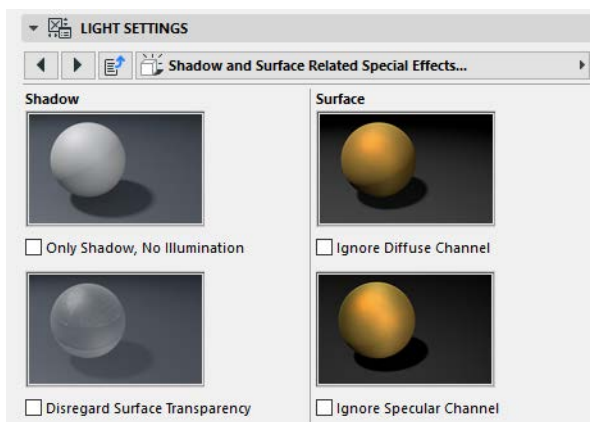
[See also Consider Global Illumination.](#)

Ambient Illumination: Normally the brightness of a surface is determined by the angle at which a ray of light hits it. When Ambient Illumination is switched on, however, the angle does not matter. All surfaces are lit with the same intensity; this results in a much flatter look. Only the material color is considered in the lighting calculations.

Only Effects, No Illumination: With this option, just the visible light effect itself, plus any lens effect, will be shown. No objects are illuminated by the light source.

Illuminate One Direction Only: Check this if your rendering doesn't need the light to illuminate the space in all directions. (Use the hostpots on the **Show Illumination** feedback lines in 2D and 3D to graphically adjust the direction.)

Shadow and Surface Related Special Effects



- **Only Shadow, No Illumination:** Use this if you want the light object to only cast a shadow, without producing any illumination. This can be useful if your scene contains multiple lights, and you don't need all the illumination for the scene.

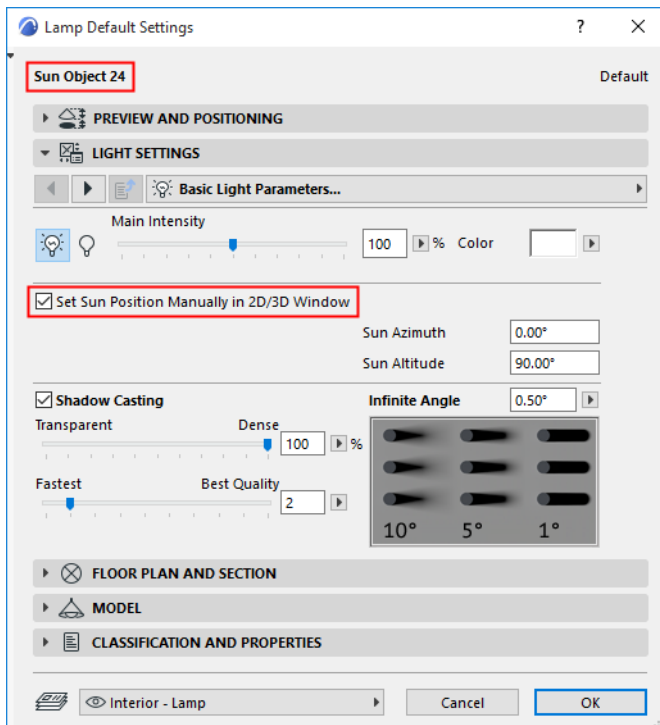
- **Disregard Surface Transparency:** Check this option to help minimize render times (e.g. for a test render).
- **Ignore Diffuse Channel:** With this option, the color properties of an object are ignored by the light source; only specular surfacing is produced by the light. This can be useful for objects such as a golden signature, where you would like specular glints, but no lightening of the color properties. This also produces a more metallic effect.
- **Ignore Specular Channel:** With this option, the light source produces no specular highlights on your scene's objects. This can be useful if, for example, an object is lit from both sides by two light sources, producing too many shiny highlights. You can retain the lighting but reduce the highlights by using Ignore Specular Channel for one of the lights.

Sun Object

One of the General Light Sources is the Sun object. It is a simpler alternative to the sun defined in the Cineware's Physical Sky settings; and it takes better advantage of Cineware capabilities than the Archicad Sun.

The sun object provides lighting based on the Time and Location settings of the Archicad Sun ([see Sunlight Settings.](#))

Alternatively, the Sun Object can be positioned manually: In Sun Object settings, check the Set Sun Position Manually checkbox. The Sun Azimuth/Altitude values are now editable, and they will be updated automatically as you move the object in the project.



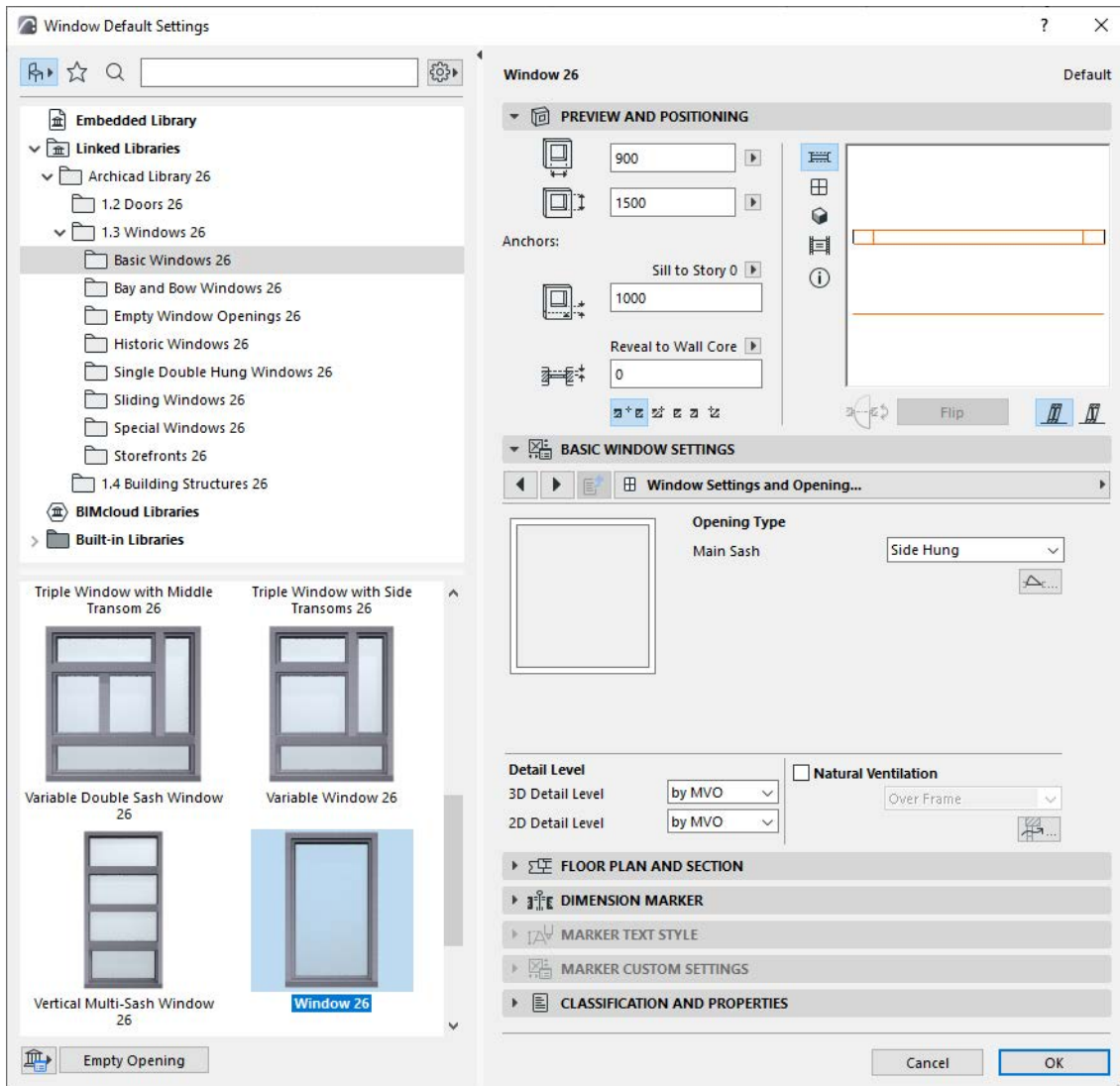
Note that the PhotoRendering Palette's Environment settings are not reflected in the 3D model, so in order to gauge the effect of the Sun Object, you must watch its effect in the Rendering Palette Preview, while moving it in the 3D window.

Door/Window Tool Settings

Door/Window libraries and their library part settings may vary considerably depending on the library that you are using.

The following description corresponds to the “International” library that is the basis for many of the language versions of Archicad.

For information on settings of a specialized Door/Window that are not described here, please contact your distributor.



For information on browsing and searching in this dialog box, see:

[Settings of Library Part Elements](#)

[Search for Library Part](#)

Empty Opening: Click this button to clear the current Library Part choice and automatically use an Empty Hole for the Door or Window opening.

[See also Creating an Empty Opening.](#)

Settings Panels

The Door and Window Settings dialog box contains the following panels:

- Preview and Positioning

For details, see [Door/Window Preview and Positioning Panel](#).

- Custom Settings (the name of the panel varies depending on the type of Door/Window)

For a description of controls here, see: [Custom Settings Panel for a Door/Window](#)

- Floor Plan and Section
- Dimension Marker
- Marker Custom Settings (this panel's availability depends on the type of marker chosen in the Dimension Marker Panel above. If no marker is chosen, this panel is gray.)
- Classification and Properties

Topics in this section:

[Door/Window Preview and Positioning Panel](#)

[Where to Set Door/Window Parameters](#)

[Floor Plan Display of Door/Window: Symbolic vs. Projected](#)

[Other Panels in Door/Window Settings](#)

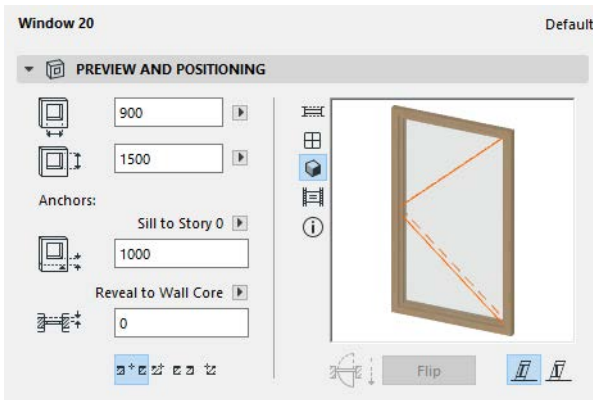
[Door/Window Model Panel \(Not Visible by Default\)](#)

[Door/Window Reveal Controls \(for Non-Standard or Legacy Openings only\)](#)

Related topics:

[Custom Settings Panel for a Door/Window](#)

Door/Window Preview and Positioning Panel

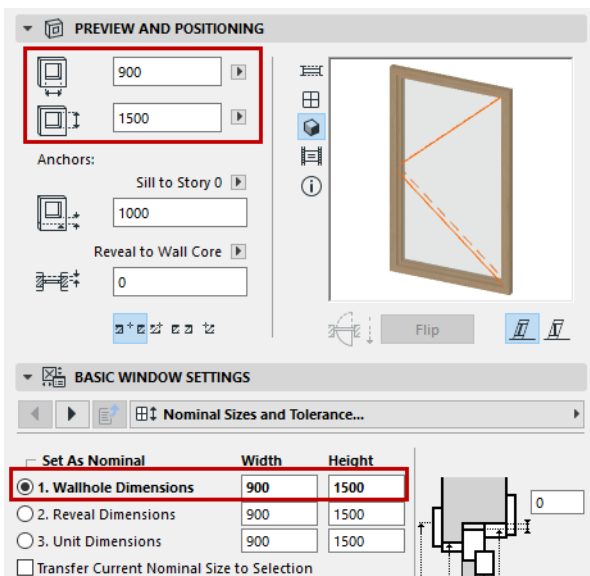


Use the fields to enter the following values for the Door/Window:

Width: Enter Width of the opening here.

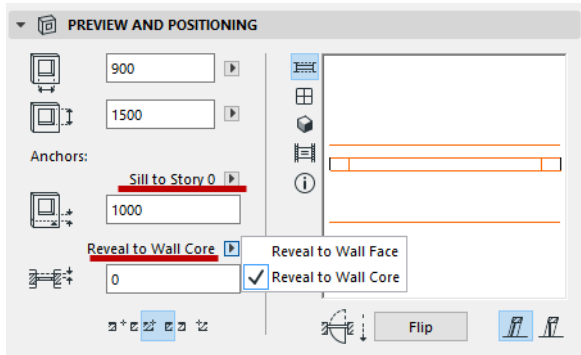
Height: Enter Height of the opening here.

Note: These Width/Height values are the window's **Nominal Size** values, which can also be entered in the Graphical Interface Panel's Nominal Sizes and Tolerance panel.



Sill/Header Value: Enter the height of the Door or Window Sill or Header from the chosen reference point.

Define the reference point for the sill/header by choosing from the pop-up above the value field:



See also [Sill or Header Heights](#).

Reveal: Enter the depth of the Door/Window Reveal, if any.

Anchor the Reveal to either the Wall face or the Wall core (the closest core skin, if there are several core skins). This is relevant if you are using a composite Wall.

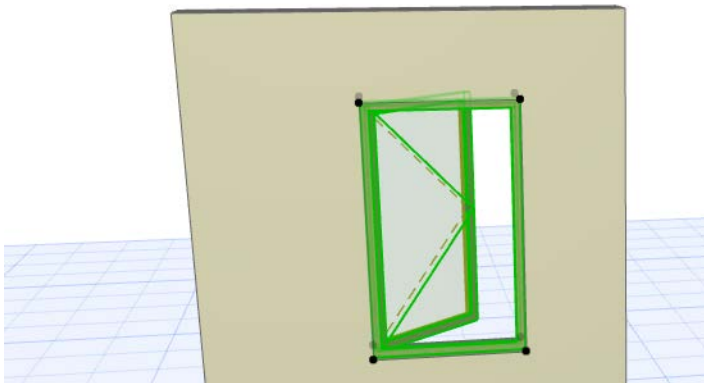
See also [Anchor Reveal](#).

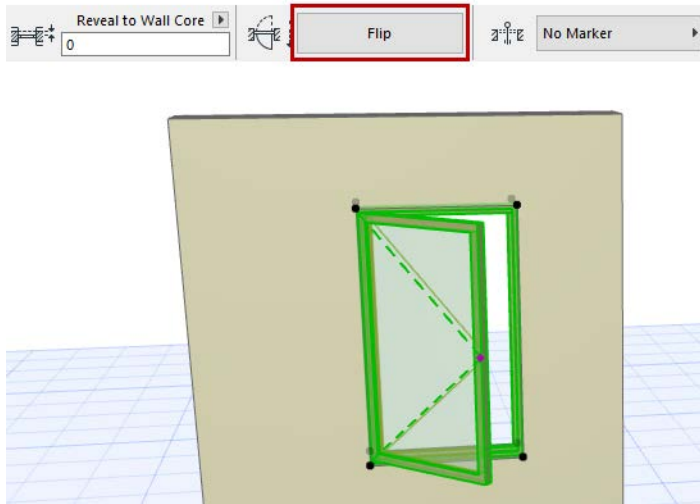
Note: The Reveal value, on the Preview and Positioning Panel, is the *same* as the Reveal Depth value on the **Reveal** tab page of the Custom Settings panel of Door/Window Settings.

Preview Area

See also [Preview Options](#).

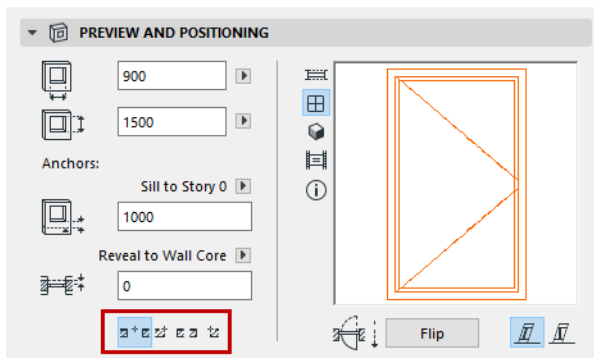
The **Flip** button switches the opening direction of a selected door/window from one side to the other, while keeping the frame in place. (This button is also available in the Info Box).





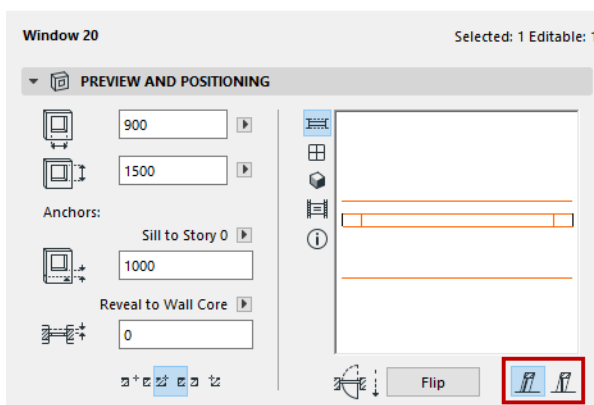
Anchor Point: This control affects the method used to place a new Window/Door. Choose either side or the Center as your preferred geometry method for placing the Door/Window.

As you place the opening into a Wall, you can switch Geometry Methods on the fly (use the Anchor Point control in the Info Box, or its shortcut.)



[See also Placing Doors or Windows.](#)

Opening Plane: This control is relevant if your door/window is placed into a slanted wall.



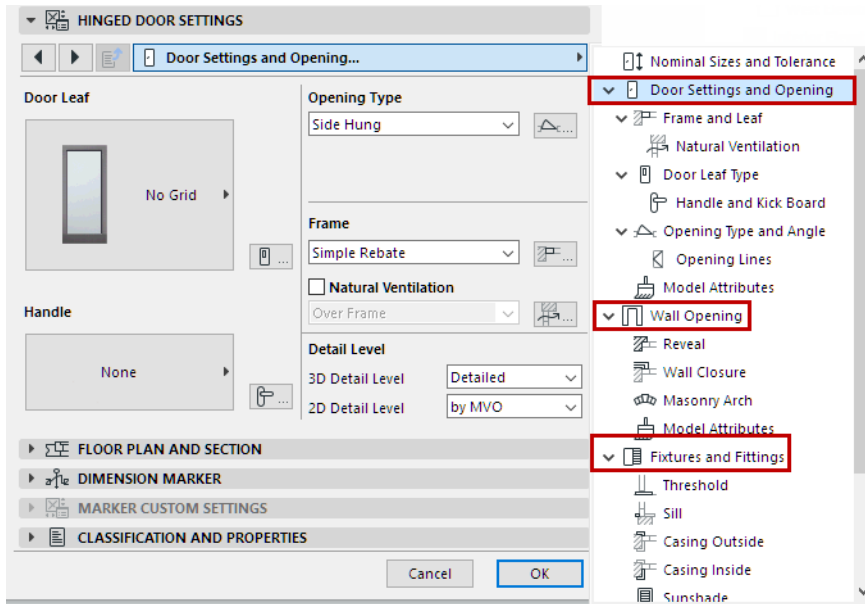
- Associated to Wall: The plane of the door/window will follow the plane of the wall.
- Vertical: The door/window will be placed vertically regardless of the plane of the wall it is placed into.

[For more information, see Setting the Window/Door Plane in Slanted or Complex Walls.](#)

Where to Set Door/Window Parameters

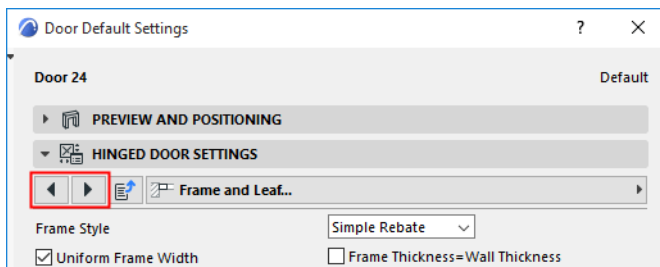
For every Door/Window in the INT library, you can set parameters using the Custom Settings panel, which has a graphical interface. The name of the panel varies depending on the Door/Window type that is selected. Parameters are grouped into a hierarchy of tab pages.

For fast, simple setup of doors and windows, you only need the three main tab pages: Settings and Opening, Wall Opening, Fixtures and Fittings.



To set additional details, go deeper into the tab page hierarchy, or access them with shortcuts from the main pages

Another way to switch tab pages is to use the arrows at the top right of the panel.

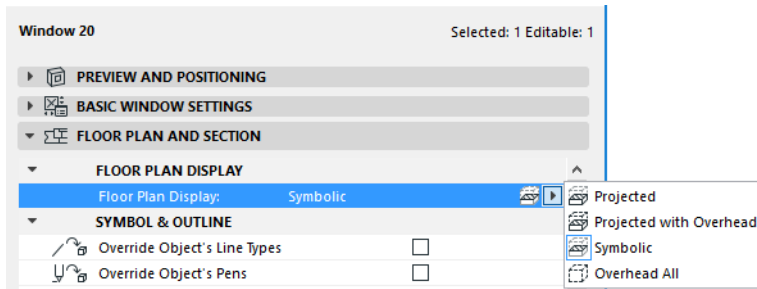


For a description of these controls, see: [Custom Settings Panel for a Door/Window](#)

For more information on Door/Window attributes, see [Override GDL Object Attributes of Doors/Windows](#).

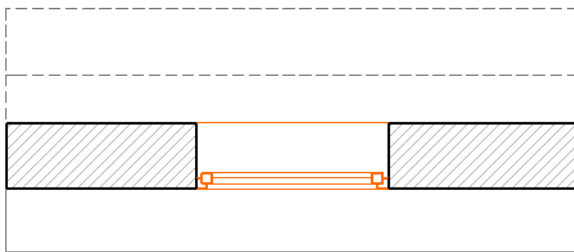
Floor Plan Display of Door/Window: Symbolic vs. Projected

Use the Floor Plan Display control in Door/Window Settings (Floor Plan and Section Panel) to define how to display the opening:

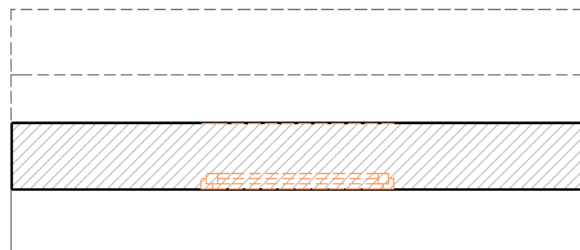


- **Symbolic** (default setting): This is the object's standard symbol, which is defined independently of the element's 3D model appearance.
- **Overhead All**: the element's symbolic 2D view is shown with its overhead attributes.

Note: Define Overhead attributes (line type, pen) in the Floor Plan and Section panel of Door/Window Settings.



Symbolic

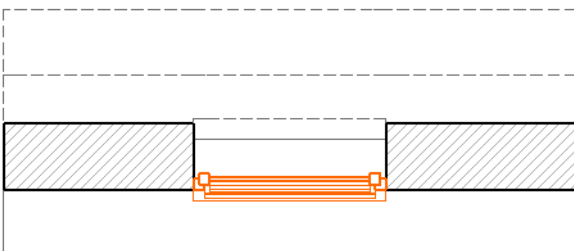


Overhead All

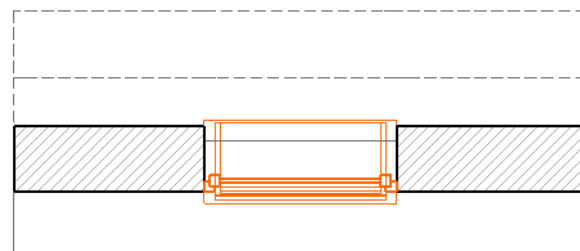
[See *Override GDL Object Attributes of Doors/Windows*.](#)

In case of a slanted wall, however, you might prefer a more realistic display of the cut door/window: choose either Projected or Projected with Overhead to see all parts of the window in a slanted wall.

- **Projected**: shows cut part of element's 3D model, plus its uncut (downward) part.
- **Projected with Overhead**: shows cut part of element's 3D model, plus the element's overhead part (i.e. the part of the element that is above the Floor Plan Cut Plane, plus its uncut (downward) part).



Projected



Projected with Overhead

Other Panels in Door/Window Settings

Door/Window Model Panel (Not Visible by Default)

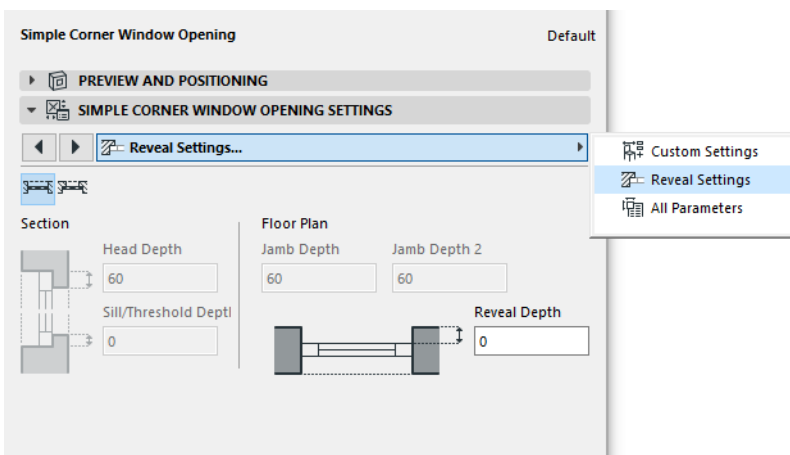
This panel is not visible by default. If you need it, make the panel visible in Door/Window Settings, using **Options > Work Environment > Tool Settings Dialog Boxes**.

Use this panel to if you want to define one single Surface for the Door or Window.



Activate the Surface button, then choose a Surface from the pop-up.

Door/Window Reveal Controls (for Non-Standard or Legacy Openings only)



A separate tab page for Reveal Settings are available only for certain Doors/Windows that are not part of the standard Archicad library.

Use these controls to customize the various elements of the reveal, such as the Head Depth, Sill/Threshold Depth, Reveal or Sill construction, left and right Jamb Depth and Reveal Depth.

Door/Window Dimension Marker

From the pop-up list, choose a predefined Marker or browse for a different marker object.

To show or hide door/window markers on the plan, use the Door/Window Options in **Document > Model View > Model View Options**.

For more information, see [Model View Options for Construction Elements](#).

Dimension Markers are essentially parametric GDL objects permanently linked to the opening.

The **dimension units** of Door/Window markers, and the dimension units of sill heights displayed in Door/Window markers, can be customized in **Options > Project Preferences > Dimensions**.

For more information, see [Dimensions Preferences](#).

The preview window gives you feedback on the selected Marker's appearance.

Subfloor Thickness: Use this control if you want the Door/Window marker to indicate a sill height value which takes into account the height difference (if any) between the floor level and the bottom of the wall (for example, due to carpeting or parquet flooring). This value will then be calculated as part of the sill value shown in the Door/Window marker.

This value will not affect the positioning of the Door/Window.

See also [Sill Height Using Subfloor Thickness](#).

Door/Window Marker Text Style

See [Text Style](#).

Door/Window Marker Symbol and Text

See [Marker Symbol and Text Panel](#).

Skylight Tool Settings

The left-hand browser area of the **Skylight Settings** dialog box is similar to those of other Library Part elements.

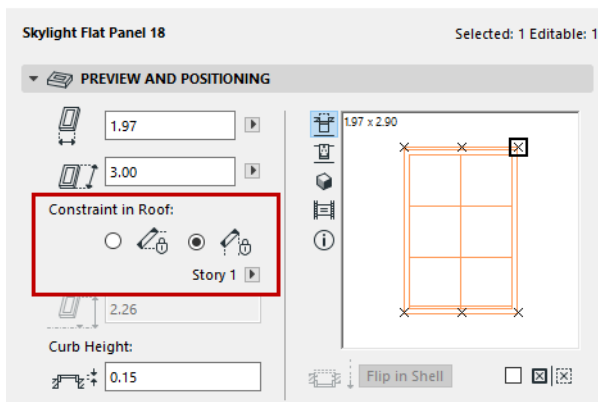
[See Settings of Library Part Elements.](#)

[For general information, see Skylights.](#)

Skylight Preview and Positioning Panel

Width: Enter Skylight width.

Height: Enter Skylight height.



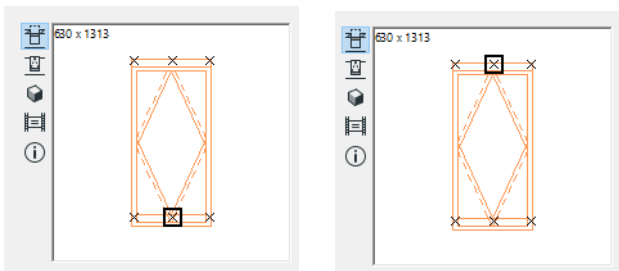
Constraint in Roof: Only available for selected Skylights placed in Roofs (not Shells). Use this control to decide how to constrain the Skylight position in a modified Roof plane:

- either **Horizontally** (the Skylight will retain its position as seen on the Floor Plan)
- or **Vertically** (the Skylight will retain its vertical elevation regardless of its Floor Plan position.)

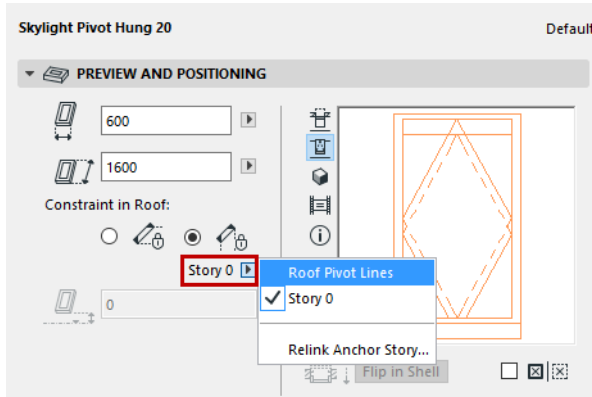
[For more information, see Skylight Constraint Relative to Roof.](#)

Sill or header height: This field that expresses the height of the Skylight anchor point (either its sill or header) as measured from either a particular story, or from the Roof Pivot line. (If you have constrained the Skylight position horizontally, this field is read-only.)

- Click on a hotspot the Preview window to define the Skylight sill/header anchor point (at the top or bottom of the Skylight).



- From the pop-up, choose whether to measure this height from a particular story, or the Roof pivot line.



The **Flip in Shell** button is only available for selected Skylights placed in Shells (not Roofs). Click Flip in Shell to change the orientation of the Skylight to the opposite side (e.g. from “inside” to “outside”).

Check the **Mirror Library Part** box to mirror the Skylight when placing it, or to mirror a selected Skylight.

Preview Area

See [Preview Options](#).

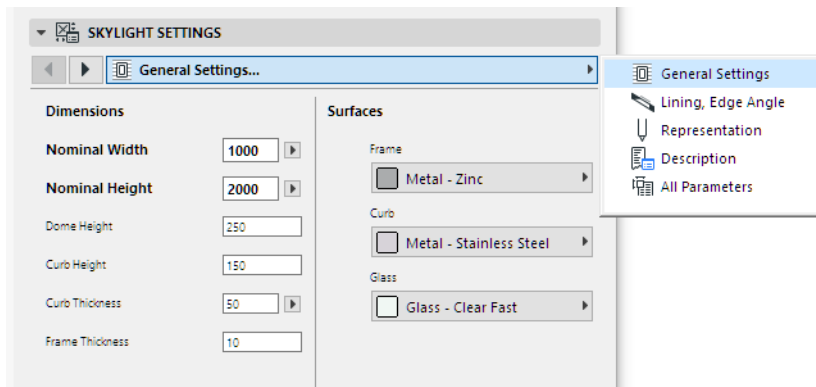
A Skylight is positioned by its hotspot, defined in the 2D symbol of the Library Part. This hotspot is marked with a highlighted rectangle, and it will act as the insertion point and anchor point for the Object.

The other hotspots (if any) are displayed as X's. Click any of them if you wish to use it as an insertion/anchor point instead of the default primary hotspot.

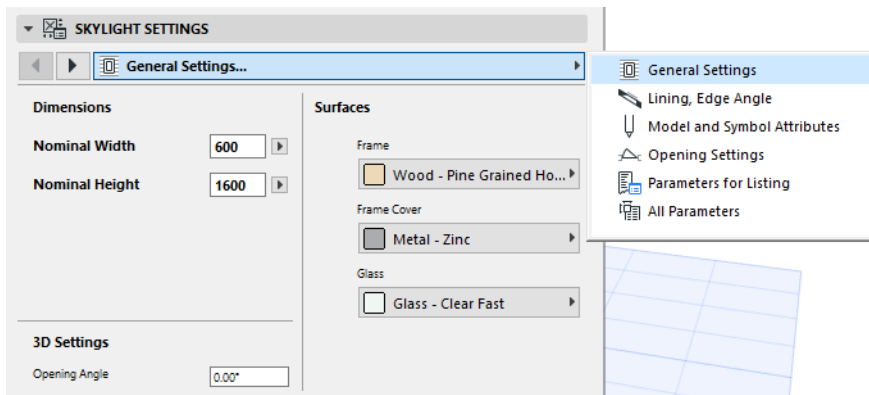
Skylight Settings

Use this interface to set parameters for the Skylight.

Note: Fill and pencolors for Floor Plan display can be defined either in this panel (using the Representation controls), or else overwritten by entering custom fills and pens in the Floor Plan and Section panel below.



Click any of the parameters to select it and to modify the value of the variable.



Skylight Floor Plan and Section Panel

On the Floor Plan, Skylights are, by default, based on a 3D projection, not a symbolic depiction.

Note: Skylights in Single-plane Roofs that were migrated from Archicad 14 or older version projects can be displayed, if you prefer, using a Symbolic depiction.

See [Legacy Preferences](#).

For information on overriding the Skylight's defined GDL attributes:

See [Override GDL Object Attributes of Doors/Windows](#).

Skylight Dimension Marker Panel

The **Dimension Marker** panel features the customizable options for the Skylight's Dimension Marker.

To show or hide Skylight markers on the plan, use the Skylight Options in **Document > Model View > Model View Options**.

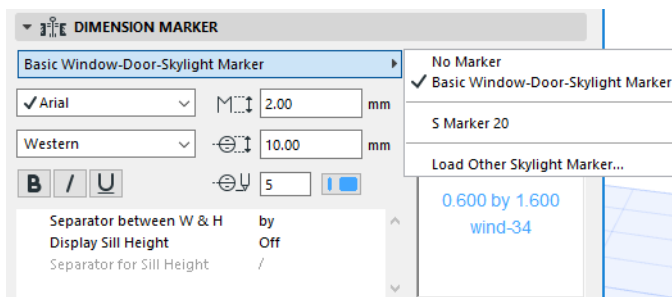
For more information, see [Model View Options for Construction Elements](#).

Dimension Markers are essentially parametric GDL objects permanently linked to the opening.

The **dimension units** of Skylight markers can be customized in **Options > Project Preferences > Dimensions**.

For more information, see [Dimensions Preferences](#).

From the pop-up list, choose a predefined Marker or browse for a different marker object.



The preview window gives you feedback on the selected Marker's appearance.

Define Marker size and pencolor.

Uniform Marker Pen: Check this box to use this pencolor for all parts of the marker symbol, regardless of any custom colors set elsewhere for this marker.

Skylight Marker Settings Panel

The **Marker Settings** panel contains options for the Library Part type dimension marker. The panel is active only if an eligible Marker has been selected in the Dimension Marker panel.

Sill Height: Use this control if you want the Skylight marker to indicate a sill height value which takes into account the height difference (if any) between the story level and the sill of the Skylight (e.g., to account for carpeting). This value will then be calculated as part of the sill value shown in the Skylight marker.

See also [Sill or Header Heights](#).

This value will not affect the positioning of the Skylight.

Dimension Tool Settings

For a description of generic settings common to all tools in the Toolbox, see [Working in Tool Settings Dialog Boxes](#).

There are five variations of the Dimension Tool. Each has its own dedicated tool in the Toolbox.

Linear Dimension Tool Settings

Elevation Dimensions

Radial Dimensions

Level Dimensions

Angle Dimensions

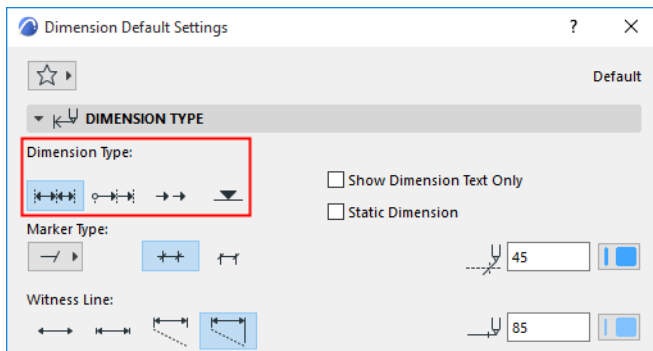
Linear Dimension Tool Settings



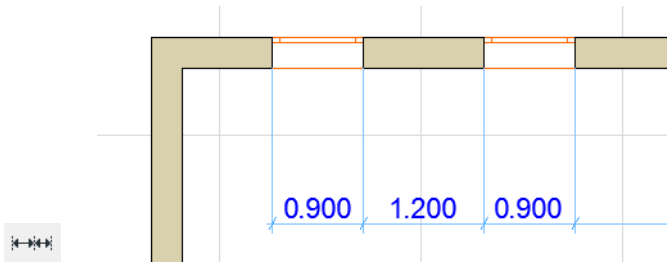
Dimension Type Panel

Dimension Type

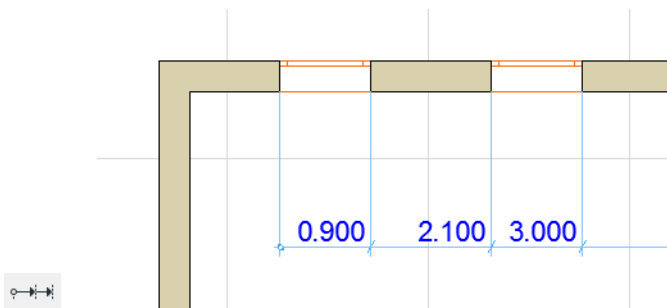
Define how you want to calculate your linear dimension.



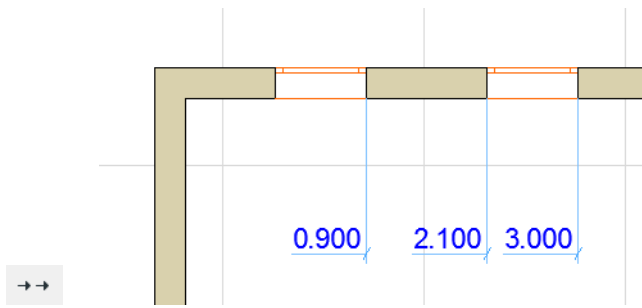
- With the **Linear** method, distances between two adjacent Reference Points are measured and displayed.



- With the **Cumulative** method, the first Reference Point is considered the zero point of the dimension chain. All dimension values of the chain will give you the distance between any Reference Point and the zero point.



- With the **Base-line** method, measuring the dimensioning distances is the same as with the Cumulative method, but the zero point is not marked.



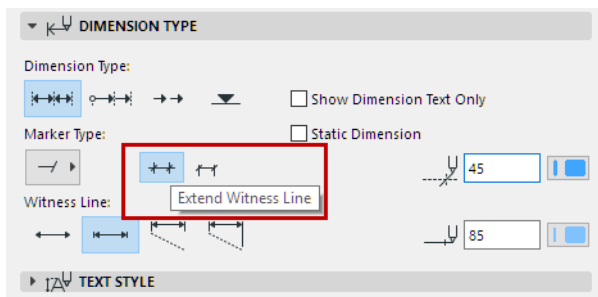
- With the fourth icon, you create **Elevation Dimensions**.



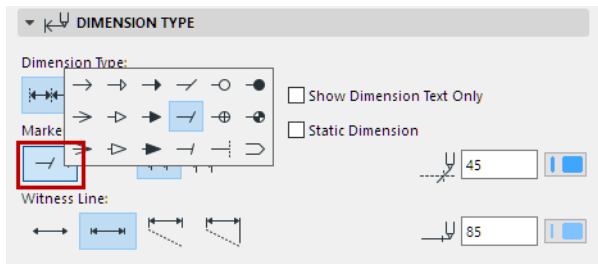
See specific settings at [Elevation Dimensions](#).

Marker Type

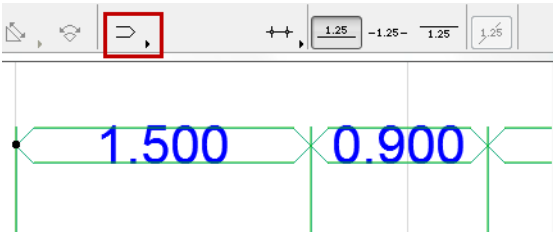
Extend Witness Line: Choose whether the witness line should extend past the node.



Marker Selection: Choose a Marker type from the pop-up.



- The last arrowhead type results in a double Dimension Line enclosing the values. With this marker, value position settings are not effective.



Static Dimension

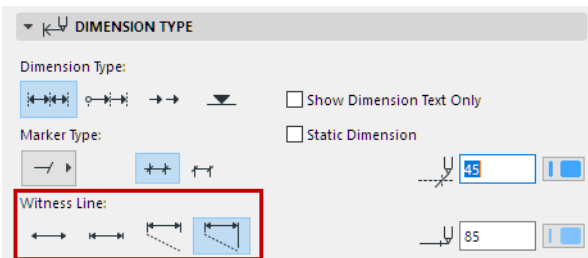
Enable this checkbox to create static dimensions that are not associated with any of the elements in the Project.

Show Dimension Text Only

Check this box to display the dimension text only, without any lines.

You can display a reference line (on-screen only) for this type of dimension as an aid in editing: turn it on at **View > On-Screen View Options > Dimension Reference Lines**.

Witness Line Format



Click an icon to define the Witness Line format. From left:

- None

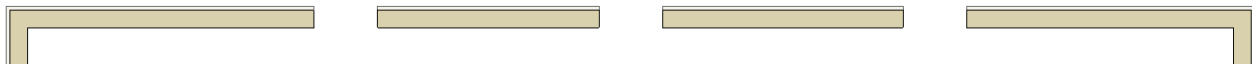
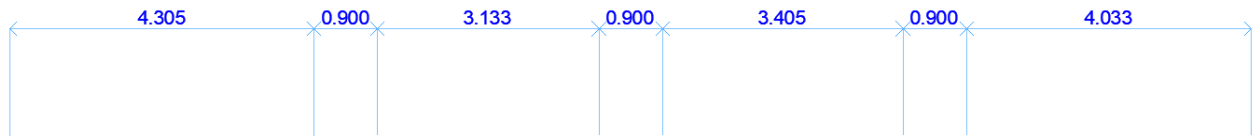


- Sized height



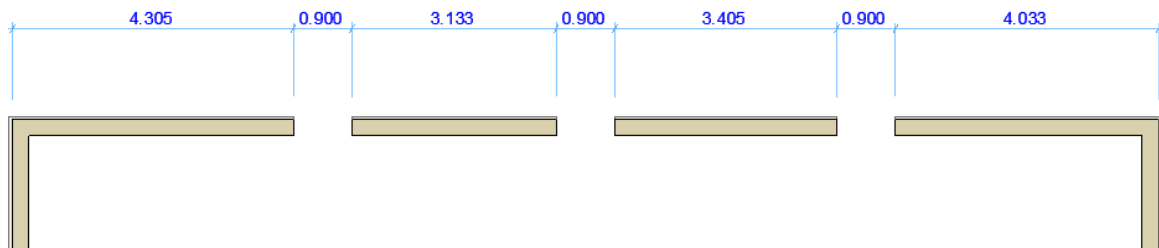
Note: Set Marker Size in the Marker and Witness Lines Options panel, below.

- Custom height



Note: Set length of custom witness lines in the Marker and Witness Lines Options panel, below.

- Dynamic height (associated to the dimensioned element).



Note: Set the gap between the Witness Line and the dimensioned element in the Marker and Witness Lines Options panel, below.

Pens for Dimension Marker and Line: Choose Pencolors/Penweights for the Dimension Marker and for the Dimension Line.

Text Style

Pointer Line

Dimension Marker and Witness Line Options Panel

These options set the Witness Line options, depending on the type of Witness Line you chose in the Type panel, above. ([See Dimension Type Panel.](#))

Marker Size: Enter the size of the marker here.

Dynamic Witness Line Gap: Enter the element-Witness Line gap value here for the Dynamic Witness Line.

Custom Witness Line Length: Enter the length of the Custom Witness Line here.

Dimension Details Panel

The **Dimension Details** panel contains options for the dimensioning of Windows and Doors and composite Walls.

Note: These controls are in addition to the options available through the Dimension Marker panel of the Door/Window Settings dialog boxes, and works for both smaller and larger sizes for German type reveals.

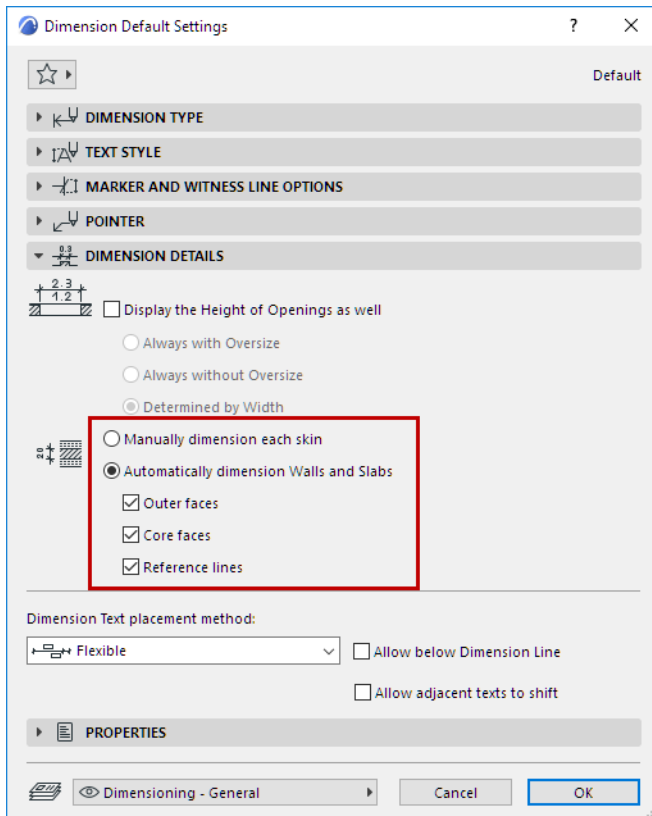
Display the Height of Openings as well: Check this box if you want the dimension marker to display the opening's height value along with the width. If you check this box, choose one of the three radio buttons at right to define how to display the Height value:

- Always with Oversize: Include the opening's vertical oversize, if any, in the Height value.
- Always without Oversize: Do not include any oversize when calculating the displayed Height value.

- Determined by Width: If the dimensioned Width includes an oversize, show Height with its Oversize. If the dimensioned Width does not include an oversize, do not include any oversize in the displayed Height value.

Dimension Components of Composite and Profile Elements

- With the **Manual** method, click the endpoints of each skin in turn before placing the dimension.



- With the **Automatic** method, dimension points are placed at the faces and/or reference lines depending on the options checked below (at least one category must be checked)
 - Outer faces
 - Core faces
 - Reference lines

The Automatic method works on Walls on the Floor Plan, and on Walls and Slabs in Section.

[See also *Dimension Wall or Slab Thickness*.](#)

Notes:

- On the Floor Plan, the core(s) of a profile element will be dimensioned along its cut part.
- In Section, the core(s) of a profile element will be dimensioned along its bounding box.
- If you edit the composite or profile structure so that it no longer has a core, the core-only dimension will automatically change to dimension the full thickness of the Wall/Slab.

Dimension Text Placement Method

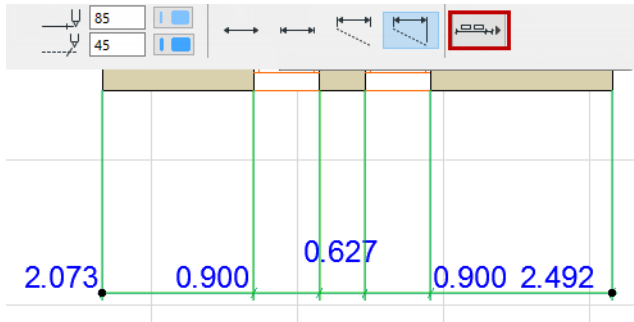
Use these options to control how Dimension Texts should be adjusted if they become too crowded.

Note: These options are available if the dimension type is set to “linear” (not cumulative, baseline, or elevation). ([See *Dimension Type Panel*](#).)

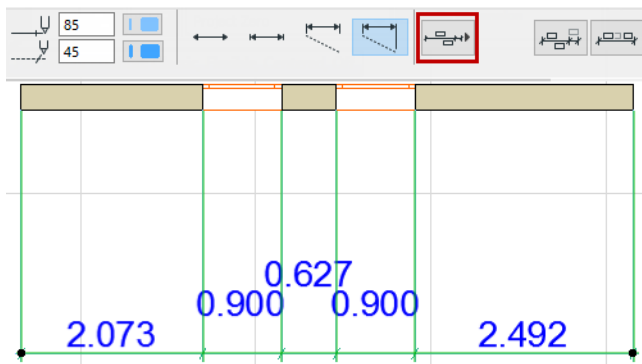
- Classic: The algorithm used in older versions of Archicad.

Dimension Texts are placed horizontally and in the middle of the measured segment.

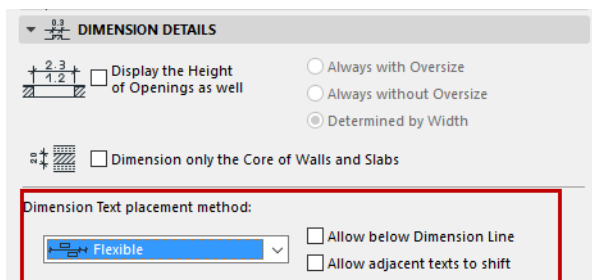
- If there is not enough room, Dimension Texts are shifted sideways so as to be readable. Adjacent Dimension texts also shift sideways.
- If there's no space to move sideways, they are adjusted vertically upwards, by rows.



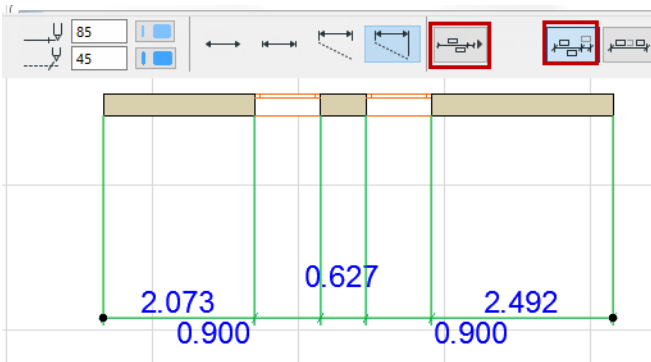
- Flexible (default): This method is more sophisticated than the Classic: it makes automatic adjustments even if a dimension text has been manually moved, or consists of multiple lines.
- Dimension Texts are placed horizontally and in the middle of the measured segment.
- If the space is too narrow, Dimension Texts will shift sideways. (Adjacent texts stay put.)
- If there is not enough space for a sideways shift, texts may be adjusted vertically.



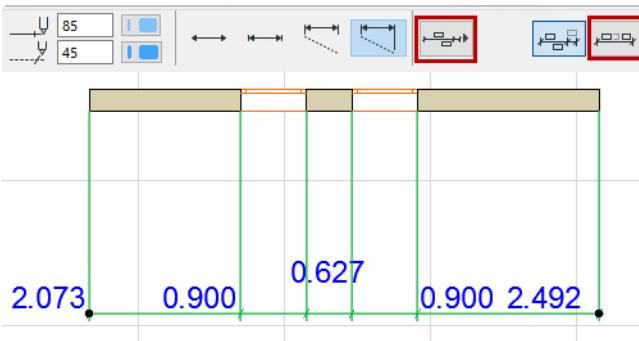
Two additional checkmarks options are available with this method:



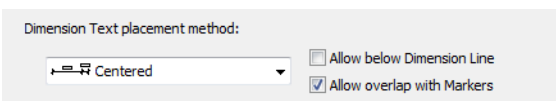
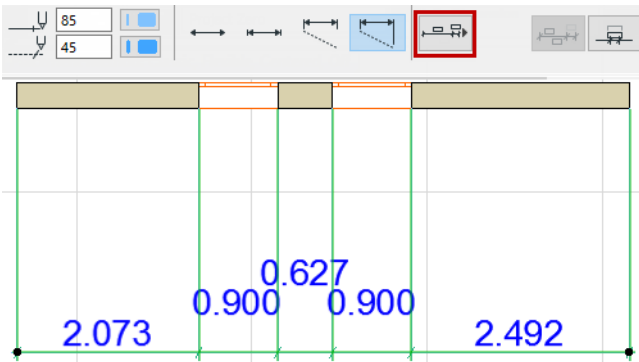
- **Allow below dimension line:** As part of the adjustment, texts may be displayed below the dimension line as well as above.



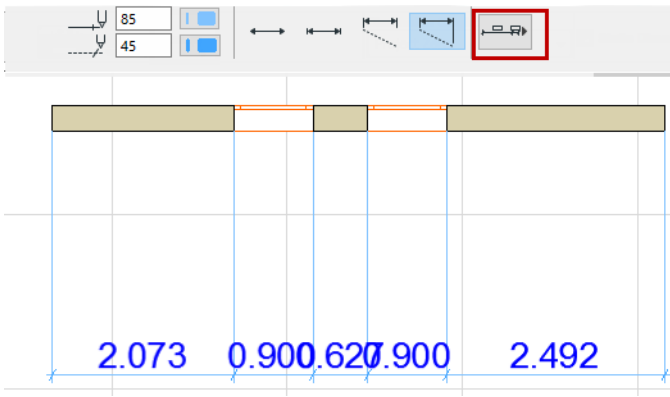
- **Allow adjacent texts to shift:** When a dimension text is shifted sideways, its adjacent dimension texts may also shift sideways (to the middle of the space remaining) as needed.



- Centered:
 - All dimension texts are placed right in the middle of the measured segment. Nothing is adjusted sideways.
 - If there is not enough room, they are adjusted vertically.



- **Allow below dimension line:** As part of the adjustment, texts may be displayed below the dimension line as well as above.
- **Allow overlap with Markers:** Texts may overlap with the dimension markers. (If this is off, then texts will be adjusted vertically to avoid overlap with Markers.)
- Off: No automatic adjustment takes place to enhance readability. All the texts are simply placed in a row, directly above the dimension line without regard to readability.



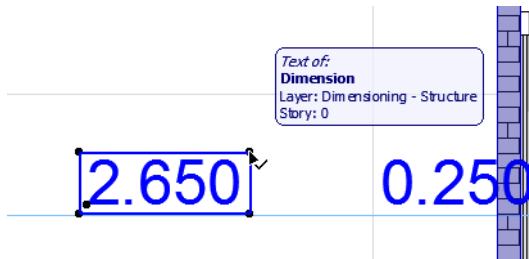
Dimension Text Settings

To open the **Dimension Text Settings** dialog box, you must first select the Dimension Text item (not the entire Dimension line).

Select Dimension Text

To select a Dimension text, do one of the following:

- Use the Quick Selection cursor
- Click on a corner of the text with the Checkmark with Arrow cursor



Note: To select a Fill Area text, make sure you have enabled Textbox handles at View > On-Screen View Options.

Open Dimension Text Settings

Select a dimension text (see above), then do one of the following:

- Click the **Text** icon in the **Info Box**
- Double-click the Dimension Tool icon in the Toolbox
- Ctrl+t
- Use **Edit > Dimension Text Settings** (available when a Dimension Text is selected)

Copy-Paste Dimension Text

Only simple text items can be copied; superscripts will not be copied.

1. Select the dimension text only (not the dimension line itself)
2. Use the Copy command (**Edit > Copy** or **Ctrl/Cmd + C**)
3. Paste the dimension text (**Edit > Paste** or **Ctrl/Cmd + V**) - for example, to a text box or onto a Layout.

Related Topics:

[Text Content](#)

[Text Style](#)

[Pointer Line](#)

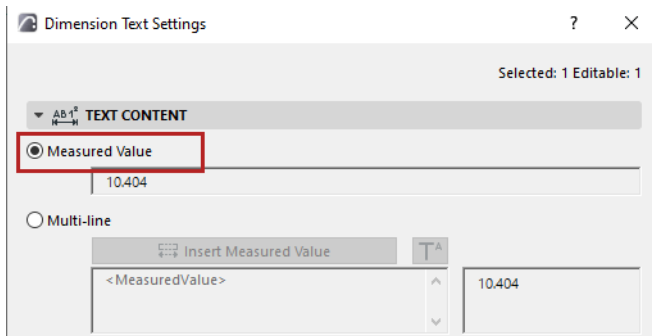
Text Content

This panel is available in

- Dimension Text Settings
- Level Dimension Settings

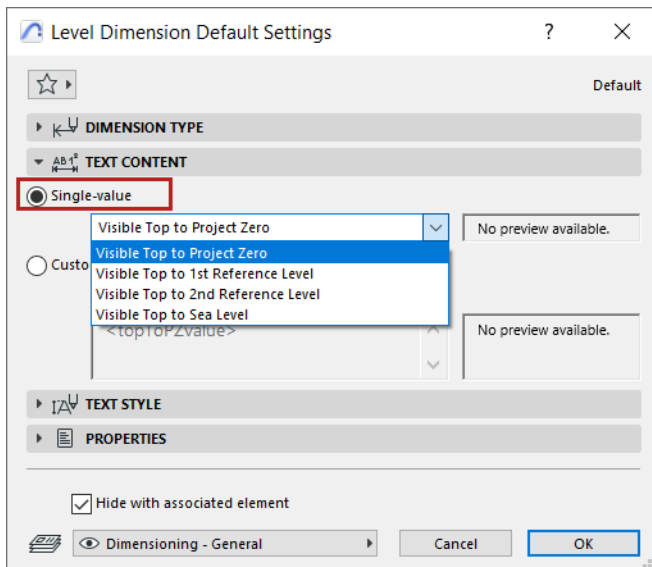
Measured Value (Dimension Text)

Displays the measured value of the selected dimension.



Single-value (Level Dimension)

Basic option to display the elevation of the element's visible top.



Note: “Visible Top” means the currently *visible* top of the element (which depends on Partial Structure Display).

Choose a reference level from which to measure the element's visible top:

- Project Zero
- Sea Level
- Other reference level

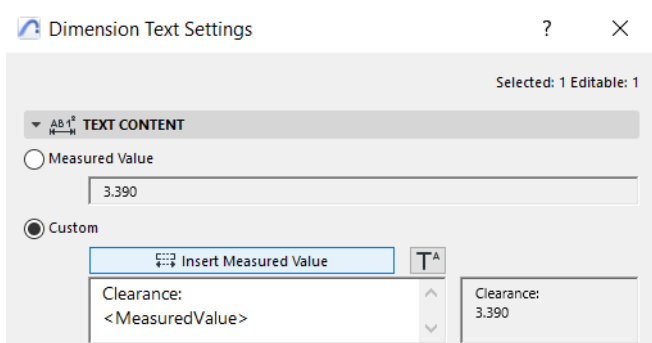
Define reference levels at **Options > Project Preferences > Reference Levels)**

Custom

Additional options for measured values and reference levels. You can also add custom text and superscript, and format multi-line text.

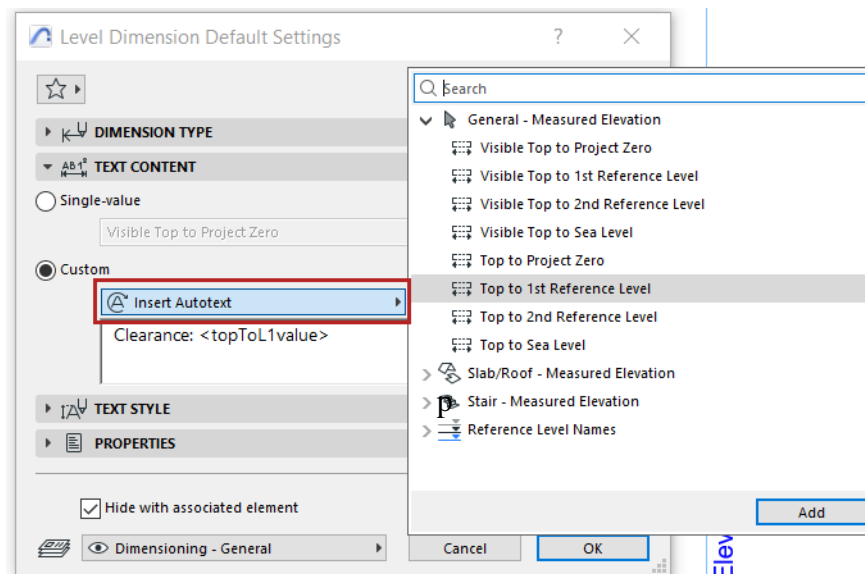
Insert Measured Value (Dimension Text)

Click to insert the dimension’s measured value into the multi-line text field.



Insert Autotext (Level Dimension)

Click to choose a different elevation level (other than visible top) and reference level.

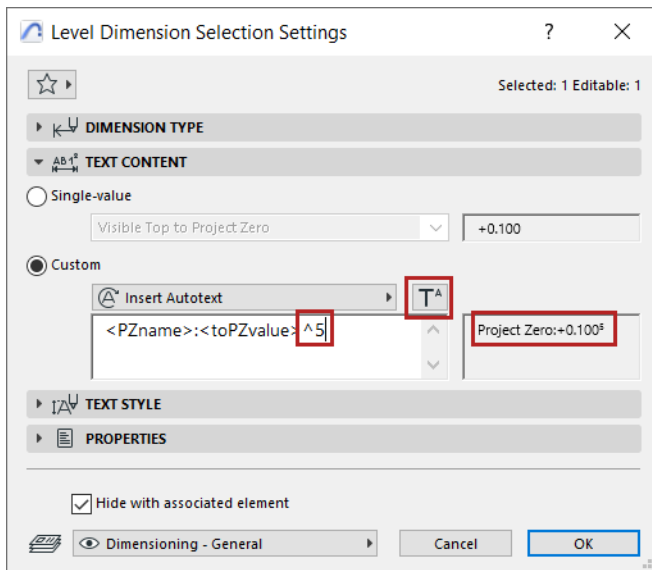


- When editing a placed level dimension: available Autotexts depend on the element type.
- You can display reference level names via Autotext (these are text strings)
- You can display several lines of text (click Enter for line break), and/or add custom text.

Add superscript to dimension text

Click **Insert Superscript Symbol**, then add your superscript text. (The symbol is shown in the dialog as a caret.)

To continue the text in regular font, click Insert Superscript again.



Note: If your dimension already includes a superscript - because you are using Extra Accuracy defined in **Options > Project Preferences > Dimensions** - then add a space before inserting the superscript symbol. Otherwise, the caret itself will appear in the dimension text.

Set dimension unit at Dimensions Preferences (**Options > Project Preferences > Dimensions**).

[See *Dimensions Preferences*](#).

Related Topics:

[Text Style](#)

[Pointer Line](#)

Convert Dimensions Dialog Box (Add-on)

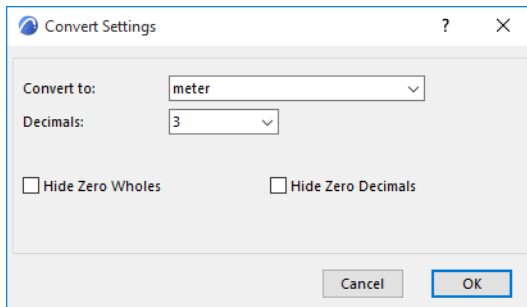
Use **Document > Annotation > Secondary Dimensions > Convert Dimensions** to open this dialog box.

For general information, see [Secondary Dimensions \(Add-on\)](#).

Convert Dimensions

Convert to: Choose the desired dimension unit from the list.

Depending on the chosen unit, additional display options for the dimension unit appear below. Choose the desired display option.



Decimals: Enter the number of decimal places to include in each dimension.

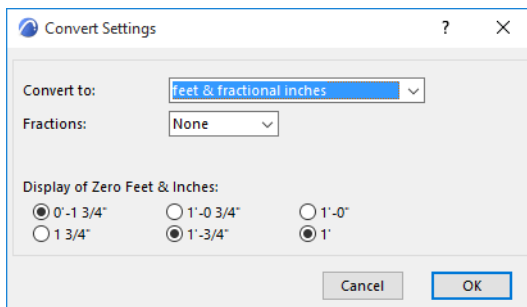
Hide Zero Wholes: Check this box to suppress whole zero values:

Example:

- 0.25 meters is displayed as 25
- 25 meters is displayed as 25.00

Hide Zero Decimals: Check this box to hide any zeros at the end of the decimal dimension, regardless of the number of decimal places you set in the Decimals pop-up above.

You cannot use the Hide Zero Decimals options concurrently with Hide Zero Wholes.



Display of Zero Feet & Inches: This section has four pairs of radio buttons. For each pair, choose the option you prefer for displaying dimensions that measure either zero feet or zero inches.

Text Tool Settings

The Text tool offers multiline texts, full-scale font options, multiple styles and justification in any direction.

For details, see [Text Blocks](#).

Text Block Formatting Panel

Settings in this panel define the overall appearance of the Text Block as a whole.

Anchor Point

Click one of the nine buttons in the Anchor Point icon: this point represents the part of the text block that will be anchored when you click to position the text block. The Anchor Point will remain fixed even if you change the size of the text block (for example, by editing font sizes). A Text Block with a non-fixed angle will rotate along with the oriented view; in this case, the Text Block will rotate around the Anchor Point you set here.

Rotation Angle

Enter a rotation angle for the text. Use the pop-up to define how this angle value should be interpreted when the element is placed:

- **Relative to Orientation** (the angle will be measured relative to the Oriented View) - think of it as relative to your screen.

See also [Set Orientation](#).

- **Absolute to Coordinate System** (the angle will be measured from the (0,0) point of the project coordinates). This means the angle value will not change, regardless of how you change the oriented view.
- **Fixed Angle (Text Block):** Keep the Text Block Angle fixed relative to the screen, even if you rotate the view.

Always Readable

Archicad will automatically “flip” the text to make it legible on screen (as opposed to upside down) regardless of the view’s orientation. This automatic “flip” occurs if the text is at an angle between 90 and 270 degrees.

Wrap Text

This checkbox “locks” the width of a placed text (including non-breaking text blocks) and wraps to the next line.

Opaque

Check this box if you want a background color that differs from the screen’s background color. Open the pop-up palette to choose a color.

Note: You can override the Fill Background Color via Graphic Override Rules (**Document > Graphic Overrides**).

See also [Graphic Override Rules](#).

Frame

Display your text or label with or without a frame.

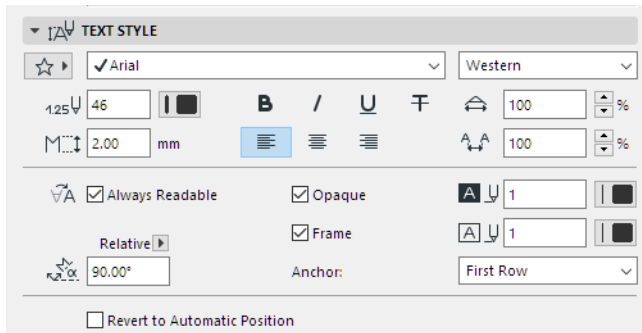
If desired, offset the Frame from the text by entering an offset value in the field at right.

Related Topics

[Text Style](#)

**Model Size vs. Paper Size Elements
Working in Tool Settings Dialog Boxes**

Text Style



This panel is available, with some variations, in the following Tool Settings dialog boxes:

- Section, Elevation, Interior Elevation (Marker Text Style, Story Levels Text Style)
- Detail, Worksheet, Change (Marker Text Style)
- Text, Label, Dimension, Level Dimension, Dimension Text
- Drawing (Title Text Style)
- Door/Window (Marker Text Style)
- Zone (Stamp Text Style)
- Grid

Font and Font Size

Enter the font height, in mm (metric units) or points (imperial units).

- The unit depends on the Length Unit defined in **Options > Project Preferences > Working Units**.
- The number of decimals depends on the setting under “Numbers without Units”, also at **Options > Project Preferences > Working Units**.

Encoding (*Windows only*)

Choose a regional font encoding.

Note: Text elements in Archicad are unicode-based. All encoding systems are supported when you print, plot, edit or display text items, or convert them to DXF/DWG format.

For each language version of Archicad, the default font encoding system is generally the right one.

Text Pencolor

Choose a pencolor with the numeric field or the pop-up control.

Font Style

Select bold, italic, underline, strikethrough.

Alignment

Left, center, right, or justify.

Leading

Modifies the leading value (vertical line spacing) as a percentage. Enter a number or use the arrows to change the value in increments of 25 percent.

Width

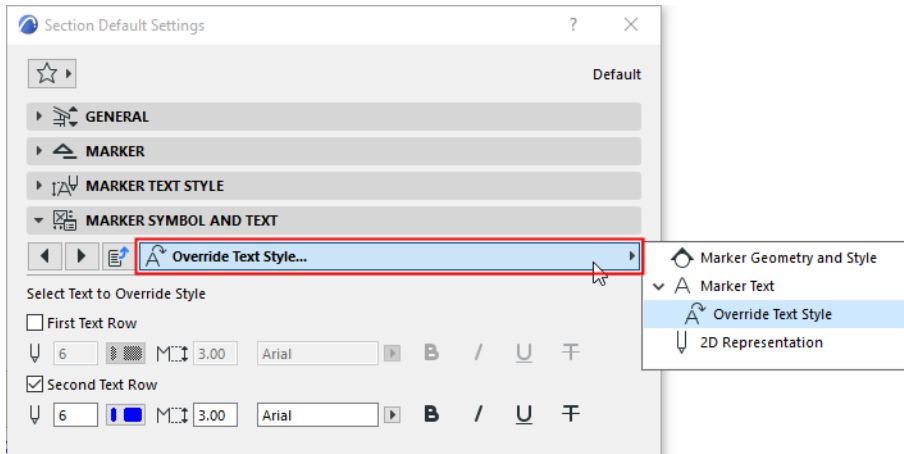
Text character width as a percentage.

Spacing

Modify the spacing between text characters.

Override Text Style

Format settings in the Text Style Panel are uniformly applied to all text in the relevant Marker or textual item. However, in most of these Settings dialog boxes, you can override particular Text Style settings for specific items.



Override Text Style

More Text Style Controls (Dimension, Label)

With/without indicator

Display dimension value with or without the + or - indicator.

Always readable

Archicad will automatically “flip” the text to make it legible on screen (as opposed to upside down) regardless of the view’s orientation. This automatic “flip” occurs if the text is at an angle between 90 and 270 degrees.

Opaque

Check this box if you want a background color that differs from the screen’s background color. Open the pop-up palette to choose a color.

Note: You can override the Fill Background Color via Graphic Override Rules (**Document > Graphic Overrides**).

See also [Graphic Override Rules](#).

Frame

Display your text or label with or without a frame.

If desired, offset the Frame from the text by entering an offset value in the field at right.

Wrap text

This checkbox “locks” the width of a placed text (including non-breaking text blocks) and wraps to the next line.

Rotation angle

Enter a rotation angle for the text. Use the pop-up to define how this angle value should be interpreted when the element is placed:

- **Relative to Orientation** (the angle will be measured relative to the Oriented View);

[See also Set Orientation.](#)

- **Absolute to Coordinate System** (the angle will be measured from the (0,0) point of the project coordinates).

Revert to Automatic Position (Dimension Text)

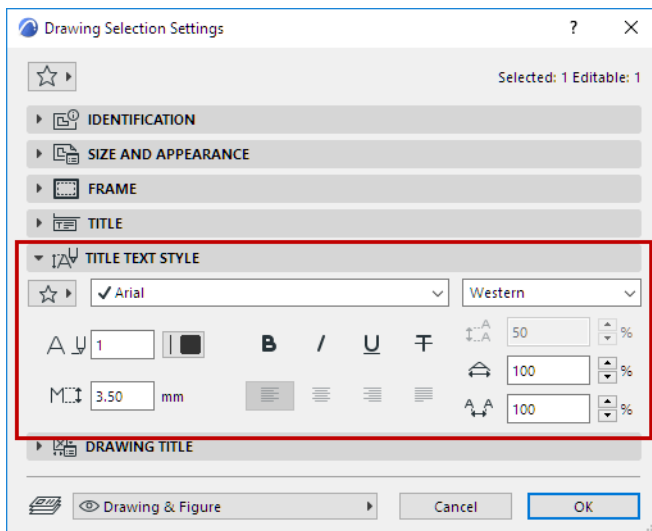
Check this to reset the position of a *selected* Dimension Text.

To undo the position changes of *all* modified dimension text items, go to **Document > Annotation** and click the **Revert All Dimension Text to Automatic Position** command.

[See also Dimension Text Settings.](#)

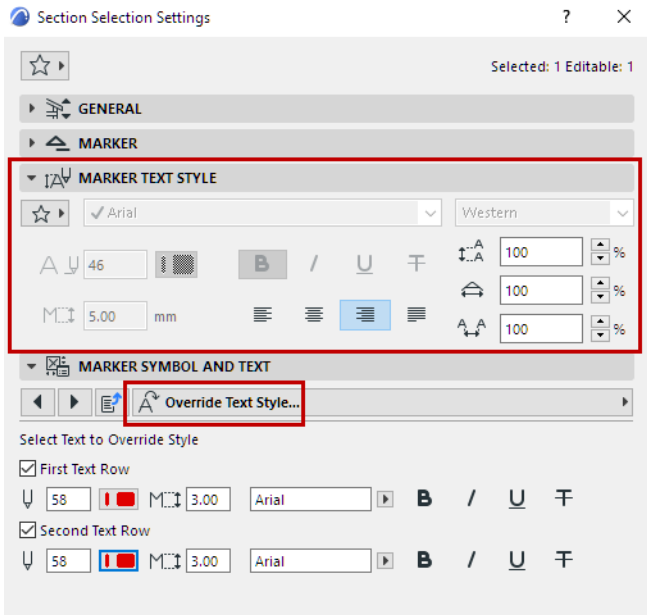
Text Style Controls in Grey

Some controls of the Text Style panel may not be relevant for a particular text item. These will be shown in grey. For example, the Leading and Alignment controls for this Drawing Title:



Irrelevant Text Style Controls in Grey

Controls might also be grey if you have overridden them, as with the font style, pen and size in this Section
Marker text:



Label Tool Settings

For a description of generic settings common to all tools in the Toolbox, see [Working in Tool Settings Dialog Boxes](#).

See also [Labels](#).

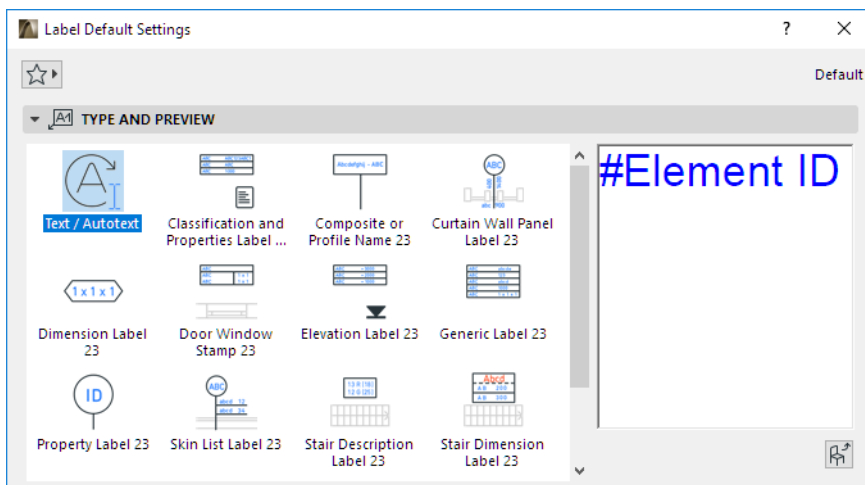
Double-click the Label tool in the Toolbox.



Label Type and Preview Panel

The **Type and Preview** panel defines the labeling options for elements, and displays a preview.

Load another Label: Click the icon in the lower right corner, and browse for a Label-type object.



Text Style

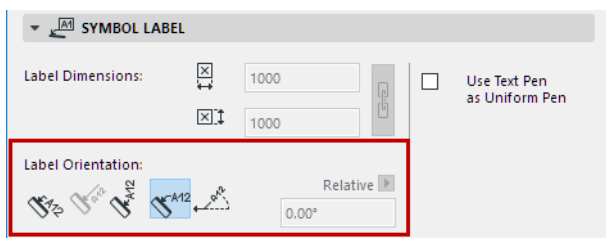
Pointer Line

Symbol Label Panel

(Not active for Text/Autotext Label).

Define Label dimensions.

Enter the Label's orientation. Available options depend on the particular Symbol Label.



See also [Label Orientation](#), below.

Use Text Pen as Uniform Pen: Check this box to use the Text pen (set in the Text Style panel) pencolor for all parts of the marker symbol, regardless of any custom colors set elsewhere for this marker.

Label Custom Settings Panel

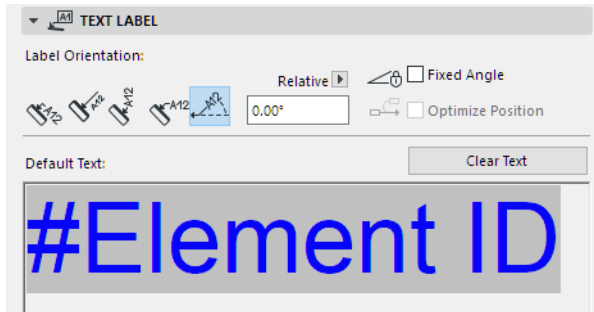
Use Custom Settings to set up the content and geometry of the Label.

For tips on the various controls for Symbol labels, see: [Custom Settings for Symbol Labels](#).

Text Label Panel

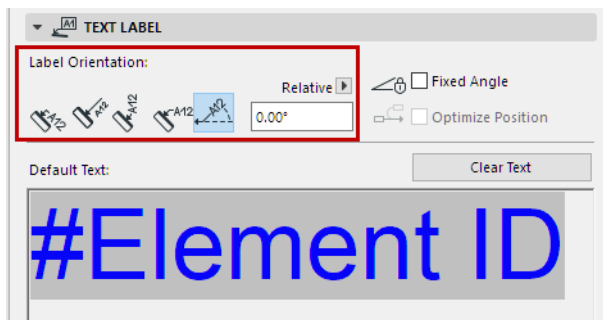
Available only if you have chosen Text Label on the Type and Preview panel.

See also [Text/Autotext Label Content](#).



Note: Favorites of the Text tool can be applied to the text-type items in Dimensions and Labels, and to Fill Text.

Label Orientation



- The **Parallel** method always places the Labels parallel to the labeled elements.
- The **Perpendicular** method places the Labels perpendicular to the labeled elements.
- The **Vertical** method always places the Labels vertically. The text is read from bottom to top.
- The **Horizontal** method always places the Labels horizontally.
- The **Custom** method places the Labels at the angle you define in the edit field next to it.
Use the pop-up to define how this angle value should be interpreted when the element is placed:
 - **Relative to Orientation** (the angle will be measured relative to the Oriented View)
 - **Absolute to Coordinate System** (the angle will be measured from the (0,0) point of the project coordinates).

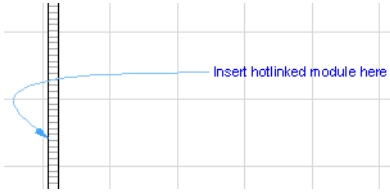
For more information, see [Set Orientation](#).

Fixed Angle (For Text Label with Custom Angle)

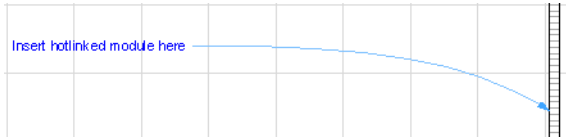
Check this to fix the angle regardless of subsequent rotation of the element.

Optimize Position: Reposition the Label's arrow if it ends up in an awkward position after the Label is moved or if the view is rotated.

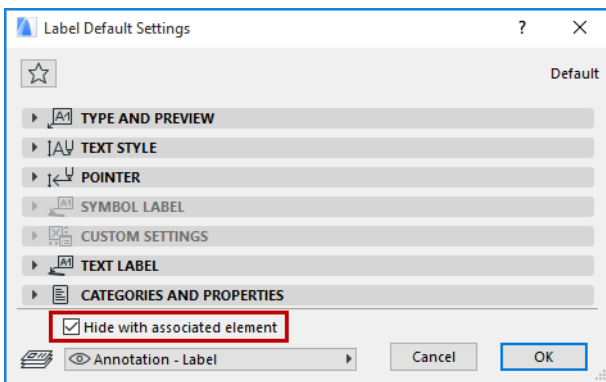
Not optimized Label, after rotation:



Optimized Label, after rotation:



Hide with Associated Element



By default, a Label is hidden if its associated element is hidden (e.g. on a hidden layer). To display the Label anyway, uncheck this box.

Drawing Settings

For a description of generic settings common to all tools in the Toolbox, see [Working in Tool Settings Dialog Boxes](#).

Drawing Identification Panel

Drawing ID

Assign an ID to the placed drawing.

Note: For a Drawing placed in a model view, this field is empty.

- **By Layout:** Drawing ID depends on the Layout Settings of the Layout on which the Drawing is placed. In Layout Settings, consult the “IDs of Drawings on this Layout” tab page.
- **By View ID:** The Drawing ID will be equivalent to the ID of its source View, if any.
- **Custom:** Enter a custom Drawing ID in the text field to the right.

Include Drawing in ID sequence: Check this box to include your drawing in the ID sequence, even if it is a custom ID.

Drawing Name

If the Drawing source is an Archicad view:

- **By View:** Name only
- **By View:** ID + Name
- Custom

If the Drawing source is an external file:

- Custom
- By source file. Read-only field. Click the folder icon to display the file path as absolute or relative.

- **Archicad view:** Displays the view’s path if the Drawing source is an Archicad view.
- **Page No.:** Appears if the Drawing source is a multi-page document (e.g. a PDF file).

Update Type

- **Auto:** Drawing will be updated automatically when the layout is activated. If a drawing comes from a different source than the currently opened project file, then Archicad will check whether it requires updating or not.

- **Manual:** The Drawing is frozen in the state its source view or file was in when placed on the layout. This can be useful if you wish to keep the drawings untouched until they are approved or you are ready to move on to the next stage in your work.
- **Store Drawing in the Project File:** Checked automatically for Manual-update Drawings.
If it is not checked, then an external Drawing's source files must be available if you open the project in a different location.

Drawing Size and Appearance Panel

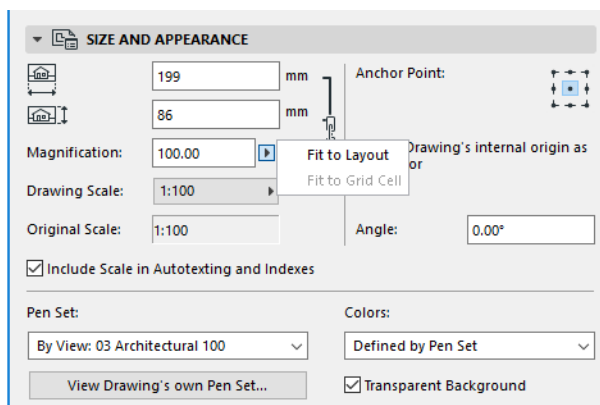
The first four fields - **Width**, **Height**, **Magnification**, and **Drawing Scale** - are interrelated (as indicated by the lock icon). A change in any of the fields will result in corresponding changes in the other three fields, so that the drawing's proportions remain fixed. Use these fields to set the size of the placed Drawing in either measurement units or as a percent of its original size (the Magnification).

Fit to Layout

Use this option from the Magnification pop-up.

Fit to Grid Cell

Available if the Layout contains a Grid.



Drawing Scale

To rescale the Drawing on the Layout (instead of using the original scale of the source view), choose a predefined scale factor from the Drawing Scale pop-up, or enter any other custom scale. (The Width/Height/Magnification values will change to reflect the new Drawing scale.)

Note: If your source drawing size, as defined in Master Layout Settings, is larger than the paper size you choose here, Archicad will automatically resize the drawing's output size to make it fit on the page.

Note: For Drawings placed on Model Views, and for placed Images and PDF documents, no scale is shown. Instead, the drawing's Resolution (in dpi) is indicated.

Customizing the Drawing scale has no effect on the scale of objects within the drawing; it is equivalent to a graphical resizing of the Drawing, like the effect of magnifying a document with a copy machine.

Your customized Drawing Scale is distinct from its Original Scale. Separate Autotext entries and Title parameters are available for Drawing Scale and Original Scale: if you place an Autotext for "Drawing Scale" on a Layout or Master Layout, the Drawing Scales of every Drawing on that Layout are displayed.

See [Autotext](#).

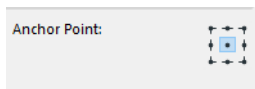
Original Scale: Scale of the drawing's source view.

Include Scale in Autotexting and Indexes: Drawing scale can be included on your output via Autotext (e.g. in an Index or on a Layout). However, for some Drawings, the resulting scale value may be misleading - for example, if the Drawing is placed as part of a PDF file. Since Archicad cannot interpret the content of the PDF, there is no point in displaying a Drawing scale in this case. Uncheck this box for such Drawings.

Anchor Point: The point by which the drawing is placed onto the layout. Serves as the drawing's fixed point in case its elements are modified in its source view.

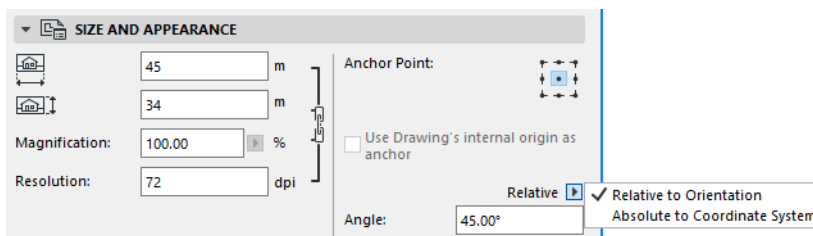
Define the Drawing's anchor point:

- **Use Drawing's internal origin as anchor:** Check this box to use the Drawing's origin as an anchor point after placing it.
- **Anchor Point:** Click one of the nine points to use as an anchor point.



Angle: Enter a rotation angle in degrees for the Drawing position, if desired.

For Drawings placed on the Floor Plan, Detail, Worksheet, or in a 3D Document, the Angle field includes an additional pop-up:



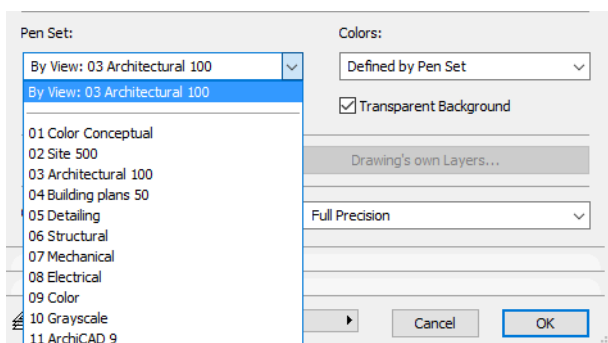
- **Relative to Orientation** (the angle will be measured relative to the Oriented View)
- **Absolute to Coordinate System** (the angle will be measured from the (0,0) point of the project coordinates).

For more information, see [Set Orientation](#).

Pen Set

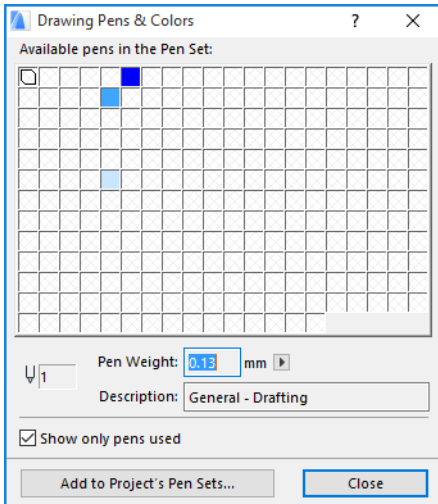
- **By View:** Use the pen set assigned to the Drawing's source view.
- **By Source File:** This is the default if the drawing source is an external vectorial file.

Use the Pen Set drop-down to choose a different pen set.

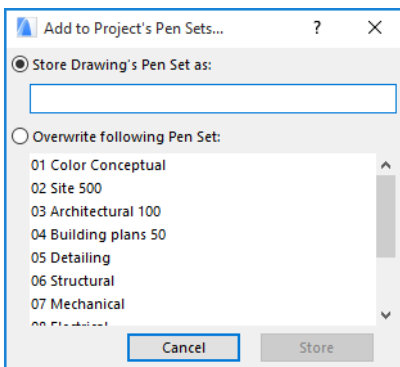


View Drawing's own Pen Set

Click to view and edit the Drawing's Pen Set.



Then **Add to Project's Pen Sets**. In the appearing dialog:



- **Store Drawing's Pen Set:** Enter a name and store it among the project's pen sets.
- **Overwrite following Pen Set:** Choose a Pen Set to overwrite and replace it.

Colors

By default, the Drawing will display the colors as defined in the pen set you have chosen.

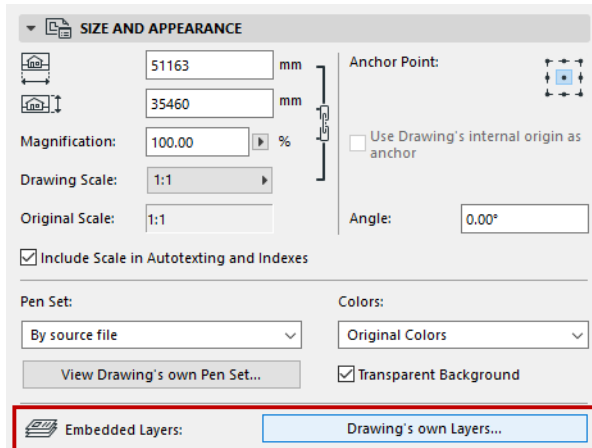
Optionally: use Grayscale or Black & White instead, for this drawing only.

Note: When printing a layout, you have the option to choose black and white output regardless of the color setting in Drawing Settings: activate Black and White checkbox in Print Layout Settings.

See [Print Layout](#).

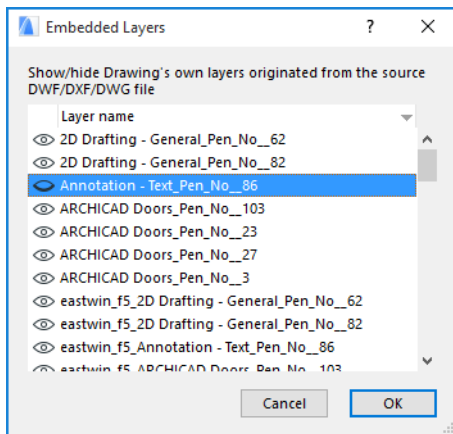
Transparent Background: Check this box to ensure that the Drawing is displayed with a transparent background.

Embedded Layers



This control is active if the source of the selected Drawing is a DWG, DXF, DWF or PDF file.

Click **Drawing's own Layers** to bring up a list of the Drawing's own (embedded) layers. For any layer you wish to hide, click the “eye” icon to close it.



You can sort this list by either Layer name or by show/hide status.

Once you click **OK** to close Drawing Settings, any change you made in the Embedded Layers' show/hide status will be carried out in Archicad; no drawing update is necessary.

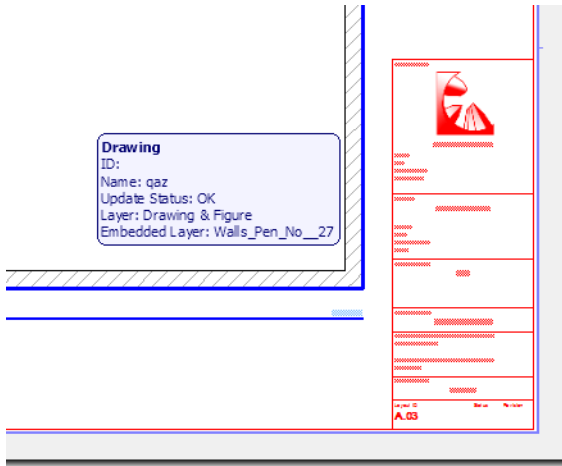
If you do modify the Drawing source and then update the Drawing in Archicad:

- any new layers created in the source file will be displayed in the Archicad Drawing
- any layers deleted or hidden from the source file will be *deleted* from the Archicad Drawing

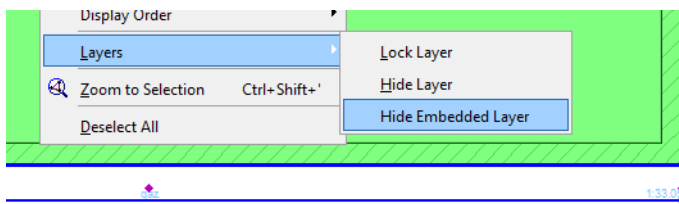
Should you explode this Drawing, then only the elements on currently *visible* layers of the Drawing will be placed into the Archicad project.

Hide Drawing's Embedded Layer from Layout

A quick way to hide the embedded layer of a Drawing component is from its context menu on the Layout itself. First, ascertain the name of the embedded layer: zoom in to the Drawing, use the cursor to identify a point or line in the Drawing, and view the Info Tag to see the name of the embedded layer for that part of the Drawing:



Now, with the cursor at the selected drawing item (e.g. the contour of the wall element), bring up the context menu: in the Layers sub-menu, the **Hide Embedded Layer** command appears. Click this command to hide the embedded layer.



Note: If there are multiple elements at the cursor location, the embedded layer of the topmost item will be hidden.

On-Screen Preview

Choose a format for the on-screen display of this Drawing on the Layout.

With large Layout Books, your choice here can affect navigation speed significantly:

- **Full Precision** (drawing shown with full details, always rebuilt)
- **Quick Preview** (faster navigation; Drawing rebuilt only upon explicit Rebuild of the Layout, or when a Drawing update occurs)
- **No Preview** (fastest navigation)

If you choose **Quick Preview**, the text “Quick Preview” will be displayed. The drawing content is an accurate vectorial preview, but you may lose certain details: certain dashed lines and fills will be shown as solid.

If you choose **No Preview**, the Drawing frame will be shown with the text “No Preview” in place of the Drawing content. (You may prefer this option to reduce redraw time if you are working on a Layout with multiple drawings, and you do not need to see this Drawing’s content as you work.).

This preview display option is an on-screen effect only and does not affect output.

Drawing Frame Panel

Select any edge of the drawing and use the usual pet palette editing commands to modify the frame into any shape. In this case, the **Manually resized frame** button is activated.

To reset the frame to fit the drawing, click **Fit Frame to Drawing**.

By default, **Crop Drawing to Frame** is also checked: as you resize the Frame, the Drawing content is cropped accordingly. (Cropping takes place after you deselect the edited Drawing.) Cropping the Drawing content reduces file size and can speed up Drawing update.

For further information on Frame and cropping, see [Drawing Frame](#).

Split Drawing among Multiple Layouts: This checkbox is active and applicable only if the selected Drawing is created from a Schedule or Index view. If this box is checked, such Drawings will be split into multiple Layouts, if the contents cannot be fit onto a single Layout page.

See [Split Schedule into Multiple Layouts](#).

Add Printable Border: Check this box to add a printable border for your drawing. Use the controls below to choose a line type and pen color for this border.

Border Offset: Enter an offset value for the printable border.

Title Panel

See also [Drawing Titles](#).

Note: Drawing Titles can be applied only to Drawings which are placed on Layouts. Drawings in Model Views do not have Drawing Titles.

Choose a predefined drawing title type from the pop-up list.

Note: You can also create a custom Title Type.

See [Create New Title Type](#).

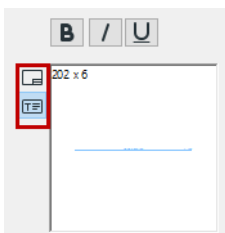
Define Title size and pencolor.

Uniform Title Pen: Check this box to use this pencolor for all parts of the Drawing Title, regardless of any custom colors set elsewhere for this Title.

Flip Title: Check this box to flip (mirror) the title graphically.

Title's Preview Area: The window at right shows a symbolic preview of the Title in one of two formats. Click one of the two buttons to display a preview:

- Show Preview of the Title's position to the Drawing: Click the upper button (Layout icon) to display a preview of the Title's position relative to the Drawing.



- Show Preview of the Title: Click the lower icon (with a T) to display a preview of the Title only.

Back Reference: Click this button to access the Back Reference dialog box. If the current drawing is based on a source marker, then this dialog box will list any Layout whose Drawing displays the source marker of the current drawing.

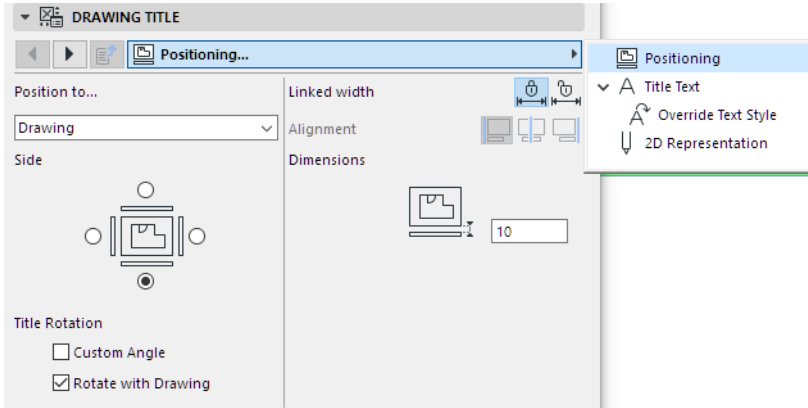
See [Back Reference to Layout Containing Source Marker](#).

Title Text Style Panel

See [Text Style](#).

Drawing Title Panel (Custom Settings)

Use these controls to fine-tune the GDL parameters of the selected Drawing Title, as applicable.



Fill Tool Settings

For a description of generic settings common to all tools in the Toolbox, see [Working in Tool Settings Dialog Boxes](#).

See also [Fills](#).

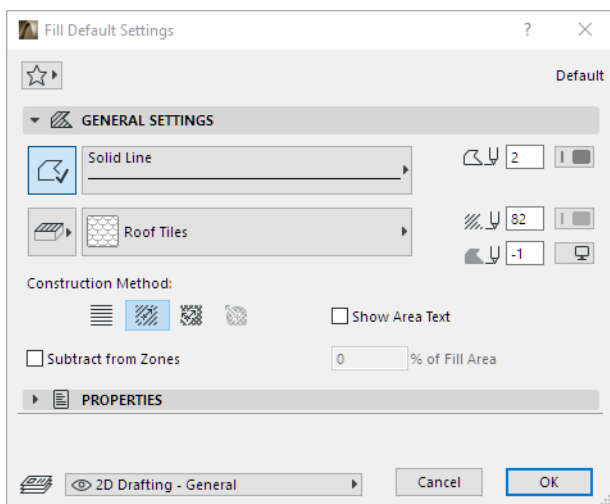
If You Are Editing the Fill Component of a Profile

If you are editing a selected fill component of a Profile attribute, the only available controls are Fill outline on/off, cut line type and color, and Fill type selection (the available Fills here represent Building Materials for the profile you are editing).

See [Edit Complex Profile](#).

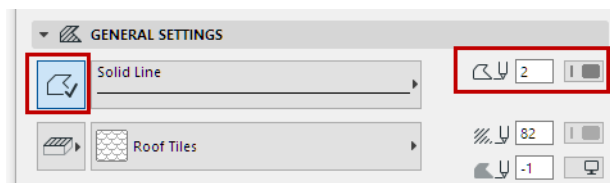
Fill General Settings Panel

This panel controls the appearance of the Fill in the Floor Plan. These Fill settings will also be used if you draw a Fill in a Section/Elevation/IE or Detail/Worksheet window.



Fill Outline

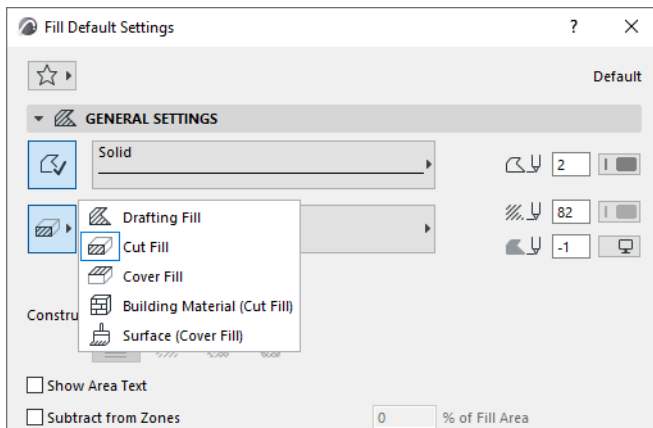
- Click to define a Fill outline.
- Choose its Line Type and color.



Fill Category

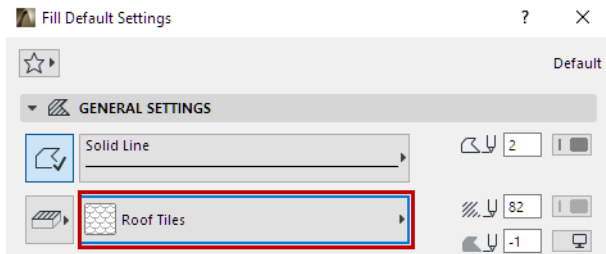
Click the button to the left of the Fill pattern pop-up to choose a Fill Category for this Fill.

For more information, see [Fill Categories](#).



Fill Type

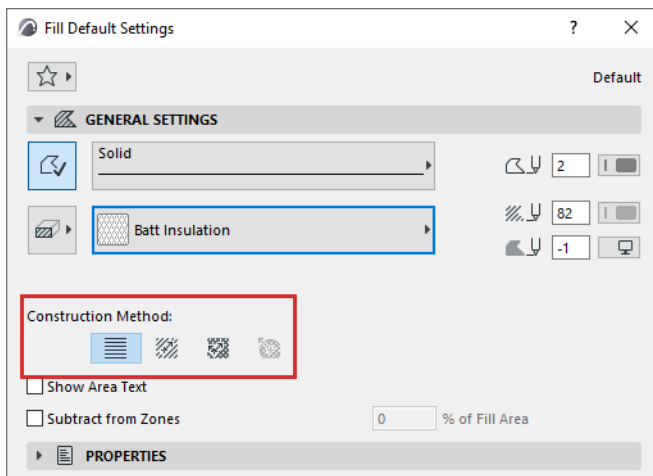
- Choose the fill type that will be used to display the Fill.



- Choose foreground and background pen colors. Depending on the fill type (e.g. those based on Building Materials or Surfaces), certain overrides may be available.
For the fill background, you can also choose the transparent option or the Window Background color.

Construction Method

Set the orientation of the fill's vectorial hatching (if any) or symbol fill pattern.



Choose from the following Construction Method options:

- **Link to Project Origin:** The pattern will always link to the Project Origin, regardless of fill position and rotation.
- **Link to Fill Origin:** The pattern will always follow the fill position and rotation. To modify the fill origin and rotation, select the fill and drag the fill handle.

- **Custom Distortion:** Similar to Link to Fill Origin. Use the two fill handles to distort the fill's X/Y orientation and scaling.

For examples of each option, see [Fill Orientation Methods](#).

- **Radial Distortion:** Graphically adjust the vector's radial distortion. Available only for a **selected symbol Fill** and only if:
 - the symbol fill was used as a skin of a composite structure; and
 - the skin's Fill Orientation was defined as **Fit to Skin** in the Composite Structures dialog box; and
 - you have selected this symbol fill after exploding a curved composite wall.

See [Radial Distortion \(Symbol Fills only\)](#).

Notes:

- Vectorial hatching is displayed only if you have switched to in **View > On-Screen View Options > True Fill Appearance**.
- The use of many radial distorted fills in your project may decrease project performance.

Show Area Text

Check this to show an associative area text with the Fill. If this box is checked, you will place (as part of the fill drafting process) a dimension-text element displaying the Fill's area on the plan.

Note: The format of Area Text and any associated Pointer line is defined in the Dimension Text Settings dialog box.

See [Add Area Text to a Fill](#).

Subtract from Zones

Check to subtract default or selected Fill elements from the Zones.

Type a number (1-100) in the percentage field to define the **percentage** of the Fill element's area that you want to subtract from the Zones in percentage.

Line-type Tool Settings

Applies to Line, Arc, Polyline and Spline.

Uniform Settings for Line Tools: Check this to apply the settings made in this dialog box to all line-type tools: Line, Arc/Circle, Spline, Polyline.

Favorites of any of the four Line-type tools (Line, Arc, Polyline, Spline) can be applied to any of the other Line-type elements.

Line General Settings Panel

- Select a **Line type** from the pop-up selection palette.
- Type a **Pen number** (1-255) for the Line/Arc/Polyline/Spline.
- Click the color for a **Pencolor** selection palette for the Line/Arc/Polyline/Spline.
- For polylines only: Choose a preference for handling the pattern of a dashed line type.
- Check the **Zone Boundary** box to allow lines to delimit automatically placed Zones.

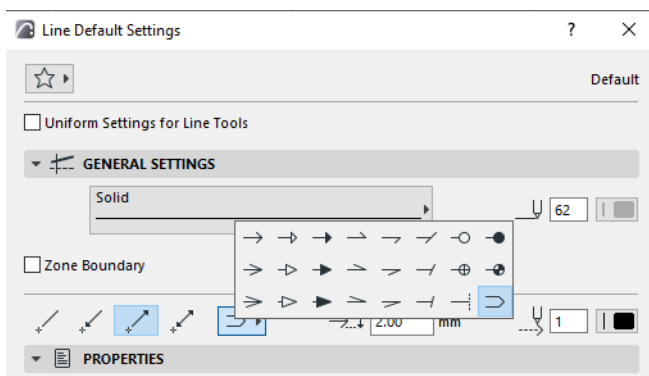
Arrowhead

Use the **Arrowhead** icons to choose a Line with no arrowheads, or a Line with an arrowhead at start, at end or at both ends.

(Not available for Circles and Ellipses).

- Enter a height value for the Arrowhead marker.
- Type a Pen number (1-255) for the Arrowhead marker.
- Click for a Pencolor selection palette for the Arrowhead marker.
- Click the **Arrowhead pop-up** icon for selection of marker types.

The last arrowhead type doubles the line. (Line tool only.)The cursor snaps to the invisible axis line between the two visible lines.



Line Properties Panel

The **Line Properties** panel is identical for the **Line, Arc/Circle, Polyline and Spline** tools.

Related Topics:

[Line Categories](#)

[Lines, Circles, Polylines, Splines](#)

[Create a Manual Zone Boundary](#)

[Working in Tool Settings Dialog Boxes](#)

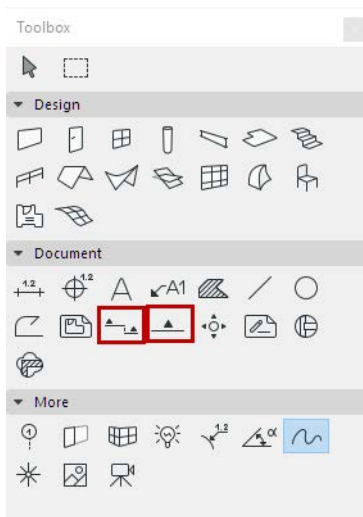
Drawing Arcs and Full Circles

Drawing Polylines and Chained Lines

Dashed Polylines: by Segment or Continuous

Section and Elevation Settings

The following settings work essentially the same way for the Section and Elevation tools.



For information on creating Sections and Elevations, see:

[Sections](#)

[Elevations](#)

For a description of generic settings common to all tools in the Toolbox, see [Working in Tool Settings Dialog Boxes](#).

General Panel

Reference ID: An ID is assigned by default; you can change it by typing any other value into the field.

Name: Appears in the Navigator palette and the title bar of the Section/Elevation window.

Note: If you place a Linked or Independent marker that does not generate a viewpoint, the ID/Name fields are gray.

To Place a New Section/Elevation Marker (Default Settings)

Use the next pop-ups to define what you want to create with the Section/Elevation tool:

If you choose **Create New Viewpoint:**

its Marker Reference can contain either

- the **viewpoint**, identifying it by its location in the Navigator Project Map;

or

- the **first placed drawing of the viewpoint**, identifying it by its location in the Layout Book hierarchy

If you choose **Place Linked Marker:**

its Marker Reference can contain information from of any of the following:

- a selected **viewpoint**
- a selected **drawing**
- the first placed drawing of the selected viewpoint
- the first placed drawing of the selected view

If you choose **Place Unlinked Marker:**

the Marker Reference will not contain any linked information; you can define a custom text (First Text Row/Second Text Row parameters) in the Marker Panel below.

Reference to: shows the path (location in the Navigator hierarchy) of the chosen reference item.

To Redefine a Placed Section/Elevation Marker (with Marker selected):

The pop-up fields give you feedback on the current status of the selected Marker (either **Source Marker**, **Linked Marker**, or **Unlinked Marker**).

Use the pop-ups to change the status as needed.

To redefine the Marker Reference: Choose from among the pop-up choices, or click the Browse button to select the viewpoint/view/drawing whose Navigator path you wish to display in the Marker. The available choices vary depending on the Marker status (Source Marker, Linked Marker or Unlinked Marker).

For an Unlinked Marker, you can define a custom text (First Text Row/Second Text Row parameters) in the Marker Panel below.

Status (for Source markers only)

Choose an option to define the status of the link between the Section/Elevation and the Floor Plan.

- **Auto-rebuild Model:** This view will be automatically rebuilt every time it is activated if the Floor Plan has changed.
- **Manual-rebuild Model:** This view is not rebuilt automatically. It can be rebuilt from the model only by using the **View > Refresh > Rebuild from Model** command.
- **Drawing:** Elements are exploded into 2D drawing elements, which are not linked to the Floor Plan and will not be automatically rebuilt from the model. You can, however, update the drawing to reflect recent changes made to the model.

Show on Stories (for Source markers only)

Choose the stories on which to display the marker and lines.

- If the Vertical Range is **Infinite**: choose **All** stories or a particular custom story (choose the story shown in the pop-up list, or choose **Browse Story**)
- If the Vertical Range is **Limited**, two additional options are active.
 - Entirely in Range:** The marker and line will appear on all stories that are entirely in the vertical range defined in the height value fields.
 - At Least Partly in Range:** The marker and line will appear on all stories that are at least partly included in the vertical range defined in the height value fields.

Horizontal Range (for Source markers only)

The horizontal range defines the depth of model that will be included in the Section/Elevation window:

- **Infinite:** All elements behind the Line will be shown in the Section/Elevation window, provided that they are not hidden by other elements.
- **Limited:** Only the elements between the Line and the limit line will be shown in the Section/Elevation window. (The limit line is defined when you click with the Eyeball cursor after you finish drawing the Section/Elevation line.)

Note: The limit line is a display-only Marker item, and is not shown on the Layout.

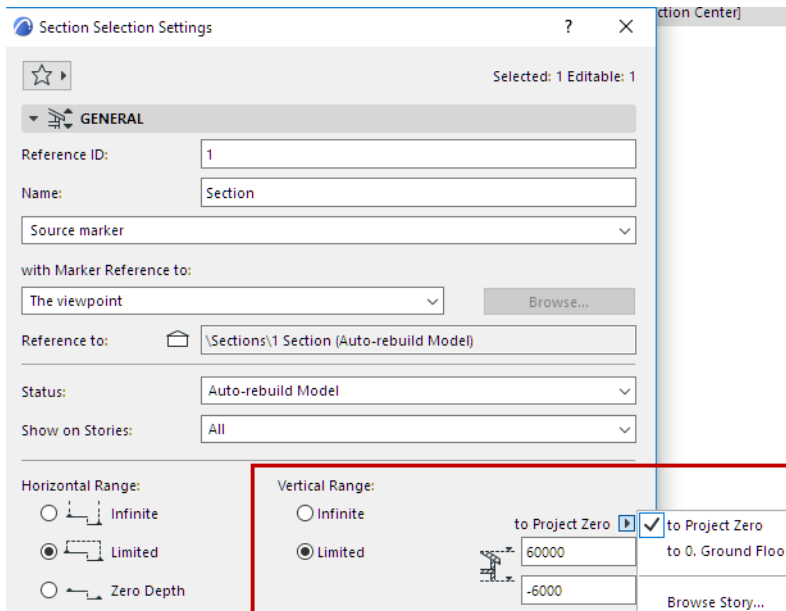
See [Display of Marker Range Lines](#).

- **Zero Depth:** Only elements actually cut by the Section Line will be shown. (Not available for Elevations.)

Vertical Range (for Source markers only)

Define the vertical range of the Floor Plan to include in the Section/Elevation window.

- **Infinite:** Include the entire height of the project.
- **Limited:** Include a limited vertical range. Enter elevation values for the upper and lower limits of the range. (Click the black arrow to choose the reference for the elevation values: to Project Zero, or to a particular custom story.)



Model Appearance Panel

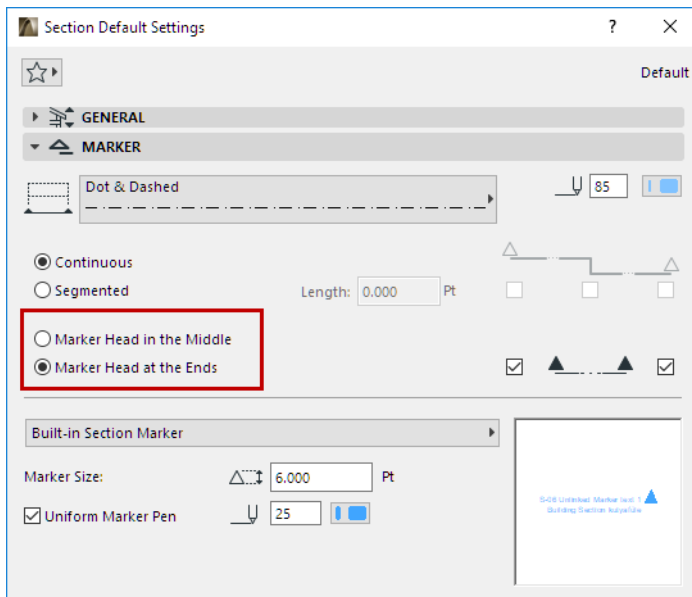
[See Model Appearance \(Section-Type Views\).](#)

Marker Panel

Section Line Settings (Section only)

- Select a Line Type and Penweight/Pencolors for the Section Line and ID.

Note: The Section Depth (limit) line (for a Section of Limited Horizontal range) is an on-screen-only Marker Range item, whose line type and color are set in **Options > Work Environment > On-Screen Options**.



- Choose **Continuous** or **Segmented**.
 - **Segmented:** Use the checkboxes at right to define which segments to display:
 - first segment
 - internal segments at breaks
 - last segment
 - **Length:** of each segment

Marker Head and Type

Place the **Marker Head in the Middle** or **at the Ends** of the marker.

Enable or disable the Markers at either or both ends of the Section/Elevation Line.

Select Marker Type: The chosen marker’s 2D Symbol appears in the preview window.

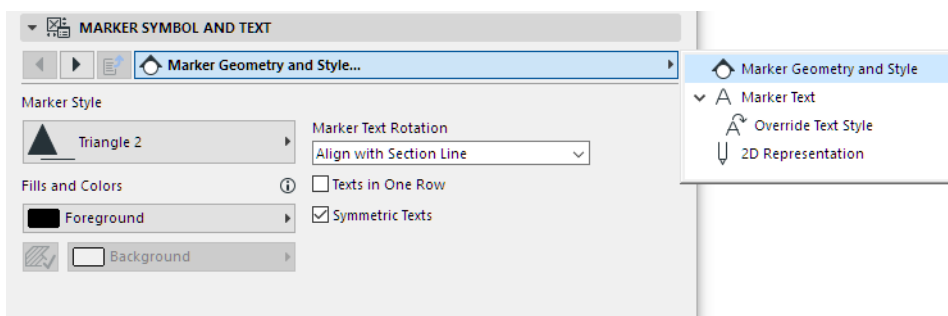
Define Marker size and pencolor.

Uniform Marker Pen: Check this box to use this pencolor for all parts of the marker symbol, regardless of any custom colors set elsewhere for this marker.

Marker Text Style Panel

See [Text Style](#).

Marker Symbol and Text Panel

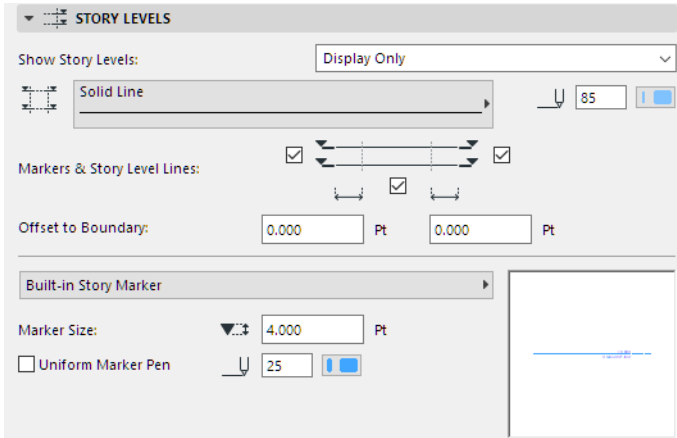


Use the settings pages to customize the appearance and content of the marker chosen in the Marker Panel above.

For Marker Geometry and Style, options vary depending on the Markers loaded in your library. Refer to the preview window of the Marker Panel to see how your choices affect the Marker.

Story Levels Panel

Define the display of Story Level lines and Story Handle Markers in this view.



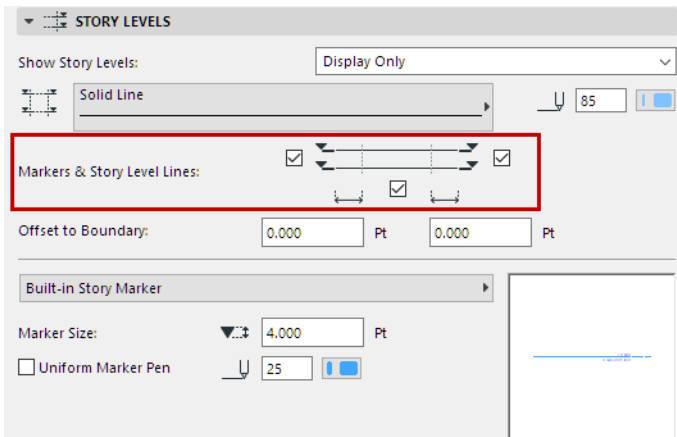
Show Story Levels

- **Display only:** Story lines will appear only on-screen, but will not be displayed in the output.
- **Display and Output.**
- **None**

For more information, see [Story Level Lines](#).

Choose a **line type** and **pencolor** for the story level lines.

Display Markers & Story Level Lines



- Display the left and/or right Story Markers
- Display or hide the Story Level Line

Note: You cannot uncheck this box if the Offset values of the Story Handle Markers are both set to zero.

Use the **Design > Edit Story Levels** command to adjust story levels in this Section/Elevation window.

Offset to Boundary: Enter a value for the offset of the Story Level line beyond the limits of the Section/Elevation.

Select Marker Type: The chosen marker's 2D Symbol appears in the preview window.

Define Marker size and pencolor.

Uniform Marker Pen: Check this box to use this pencolor for all parts of the marker symbol, regardless of any custom colors set elsewhere for this marker.

Story Levels Text Style Panel

See [Text Style](#).

Story Levels Symbol and Text Settings Panel

Use these controls to set the display of the Story Level Marker (selected in the Story Levels Panel).

Grid Tool Panel

Define the display of Grid elements on the Section/Elevation.

Show Grid Elements: Check this to see the Grid Elements.

To narrow the displayed Grid elements:

- **Show Grid Elements by Stories:** Click **Selected** to narrow the set of displayed Grid elements by Story: click **Select stories** to choose the stories whose Grid elements you want to display. (If a Grid element is visible on the selected stories, then it will be visible on this Section/Elevation as well.)
- **Show Grid Elements by Name:** Click **Selected** to narrow the set of displayed Grid elements by Name: click **Select elements** to select individual elements.

Note: These two filtering criteria are related as an “AND” statement: if you set filters both by Story and by Name, then the Section/Elevation will display only those Grid elements which fit the both the Story criteria *AND* the Name criteria.

Dimension lines: Place a dimension between each grid line. Enter the vertical location of this dimension chain and set the reference level.

Total Dimension: Place a dimension line between the two grid lines on the far left and far right end of this Section/Elevation. Enter the vertical location of this dimension chain and set the reference level.

These dimension lines will use the default dimension settings. When a new Grid line is inserted on the Floor Plan and it appears on the Section/Elevation, then it will be included in the dimension chain.

Auto-stagger Grid Markers if they overlap: Automatically stagger Grid markers sideways to avoid overlapping, if the markers would otherwise overlap.

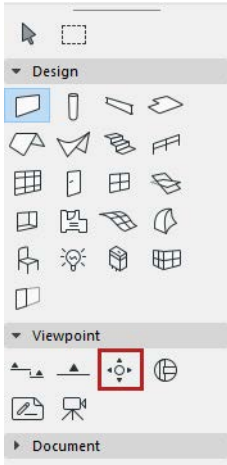
Show non-perpendicular Grid Elements: By default, the Section/Elevation will display only Grid elements that are perpendicular to the Section/Elevation line and are located within the Section/Elevation depth. (By default, no Grid elements that are curved on the Floor Plan will be displayed on a Section/Elevation.)

Check this to show Grid elements which are *not* perpendicular to the Section/Elevation line, but which intersect the Section/Elevation on the Floor Plan. These Grid lines will appear at the intersection point with the Section/Elevation line.

Interior Elevation (IE) Tool Settings

For a description of generic settings common to all tools in the Toolbox, see [Working in Tool Settings Dialog Boxes](#).

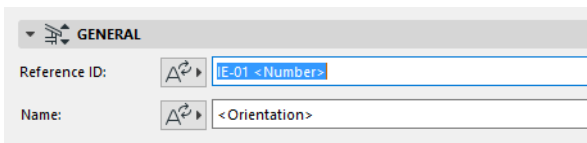
For details, see [Interior Elevations \(IE\)](#).



Interior Elevation General Panel

Settings in this panel depend on whether the selected Interior Elevation (IE) is the whole IE Group, or one of its viewpoints. If nothing is selected, the default settings are those of an IE Group.

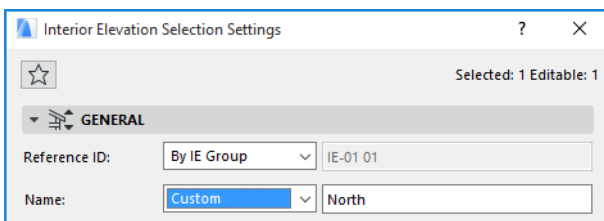
For an IE Group:



Reference ID: Enter any customized ID, if desired, or click the Autotext button to choose an Autotext reference as part of the ID.

Name: Enter any customized Name, if desired, or click the Autotext button to choose an Autotext reference as part of the Name.

For an individual IE viewpoint:



Reference ID/Name: To inherit the ID or Name defined in the parent IE Group's settings, leave the pop-up on "By IE Group."

Any Autotext defined in the parent IE Group's ID/Name settings will be interpreted individually for each viewpoint in the group.

To enter any other ID/Name for this viewpoint, switch the pop-up to Custom and enter the desired text.

Marker Reference to: Choose an item whose information you want to display in the IE marker:

- the Interior Elevation **viewpoint**, identifying it by its location in the Navigator Project Map;
- or
- the **first placed drawing of the Interior Elevation viewpoint**, identifying it by its location in the Layout Book hierarchy

Reference to: shows the path (location in the Navigator hierarchy) of the chosen reference item.

To redefine the Marker Reference: Choose from among the pop-up choices (either viewpoint or first placed drawing of the viewpoint).

Interior Elevation Status

Choose an option to define the status of the link between the Elevation and the Floor Plan.

- **Auto-rebuild Model:** An Interior Elevation with Auto-rebuild status will be automatically rebuilt every time it is opened or brought to the front of the screen if the Floor Plan has changed.
- **Manual-rebuild Model:** An Interior Elevation with Manual-rebuild status is not rebuilt automatically. It can be rebuilt from the model only by using the **Rebuild Elevations/Rebuild from Model** command of the View menu.
- **Drawing:** In an Interior Elevation of Drawing status, elements will be exploded into 2D drawing elements which are not linked to the Floor Plan and will not be automatically rebuilt from the model. You can, however, update the drawing to reflect recent changes made to the model.

Show on Stories

This pop-up list allows you to choose the stories on which to display the Interior Elevation markers and lines.

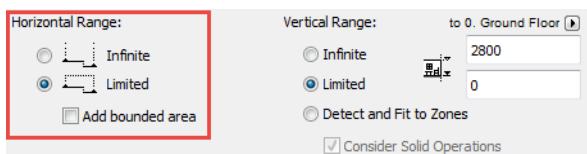
- If you have selected the **Infinite** radio button under **Vertical Range**, choose All stories or a particular custom story (choose the story shown in the pop-up list, or choose **Browse Story** to select another story of the project.)
- If you have selected the **Limited** radio button under **Vertical Range** and entered height values, two additional options are active.

Entirely in Range: The Elevation marker and line will appear on all stories that are entirely in the vertical range defined in the height value fields.

At Least Partly in Range: The Elevation marker and line will appear on all stories that are at least partly included in the vertical range defined in the height value fields.

Horizontal Range

The Horizontal Range can be either Infinite or Limited.



The default choice is **Limited**. In this case, the IE view will extend horizontally until the Room Boundary line.

If the Room Boundary is a Wall, then the view automatically extends to the depth of the Wall.

To adjust the horizontal range, select the Room Boundary line and stretch or shrink it using the commands of the pet palette. (Stretch/shrink modifications to this segment will affect the neighboring line segments accordingly, as when editing any polyline.)

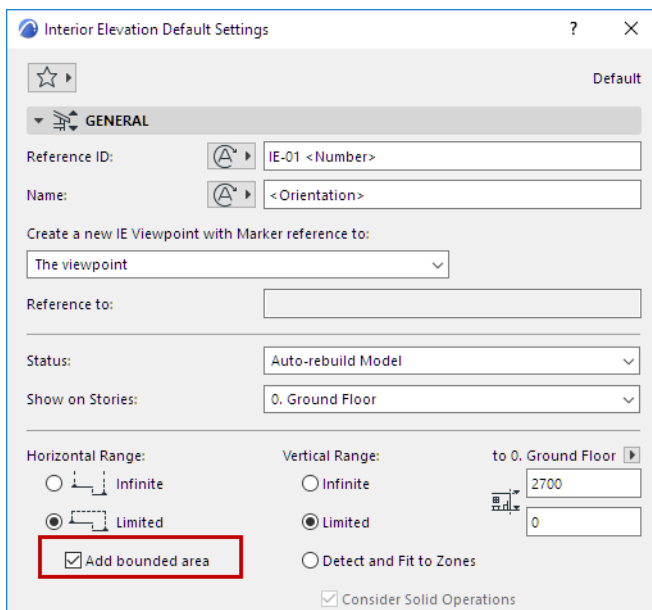
An **Infinite** Horizontal Range means that the Interior Elevation viewpoint will extend horizontally to include all visible parts of the model.

Note: The visibility of elements in an Interior Elevation is defined differently as of Archicad 18. In earlier versions, the bounding box of elements determined whether they would be shown in the viewpoint. To use the older-version logic, turn on the Interior Elevation Legacy checkbox in Project Preferences. ([See Legacy Preferences.](#))

Add Bounded Areas (Interior Elevation Default Settings)

To include connected enclosed areas in the Interior Elevation viewpoint, choose **Add bounded areas**. This checkbox is available

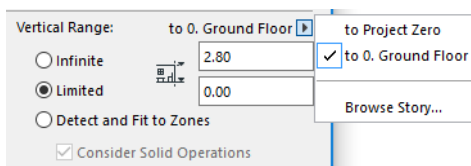
- in Interior Elevation Default Settings (you must set it before you place the IE; you cannot set this control for an already placed IE)
- if the Limited option is checked for Horizontal Range



Vertical Range

The vertical range defines the height of the Floor Plan Elevation that will be included in the Interior Elevation window.

- **Infinite:** The Elevation will include the entire height of the project.
- **Limited:** The Elevation is confined to a limited vertical range. If you click this button, enter elevation values for the upper and lower limits of the Interior Elevation. Elements above and below these limits will not appear in the Interior Elevation.



Use the pop-up to define whether this vertical range should be measured from **Project Zero**, or a particular custom story: choose the story shown in the pop-up list, or choose **Browse Story** to select another story of the project.

- **Detect and Fit to Zones:** The Interior Elevation will use the Zone height (if it detects any zones in the Interior Elevation limit) as the IE viewpoint's vertical range.

If you want this zone calculation to take any Solid Operations into account, you should also check the **Consider Solid Operations** box.

Interior Elevation Model Appearance Panel

Use the Model Appearance controls in Interior Elevation Settings to define the appearance of cut and uncut elements in the view.

These options are largely identical to those in the Model Appearance panel of Section Settings.

Interior Elevations do not have a "Fade Distant Elements" display option.

[See *Model Appearance \(Section-Type Views\)*.](#)

Exclude View Blocking Walls

Check this if you do not want intervening walls to be displayed in your Interior Elevation viewpoint.

Interior Elevation Marker Panel

Marker Placement: Use this control if you are creating a Group of IE viewpoints. Choose one of these options:

- **Individually for Each Viewpoint:** You will place as many markers as viewpoints you are creating in the Group.
- **One common Marker for the IE Group:** You will place a single marker for the entire group of viewpoints.

Select Marker Type pop-up: Select a parametric marker from this list.

The chosen marker's 2D Symbol appears in the **preview window**.

Define Marker size and pencolor.

Uniform Marker Pen: Check this box to use this pencolor for all parts of the marker symbol, regardless of any custom colors set elsewhere for this marker.

Worksheet and Detail Tool Settings

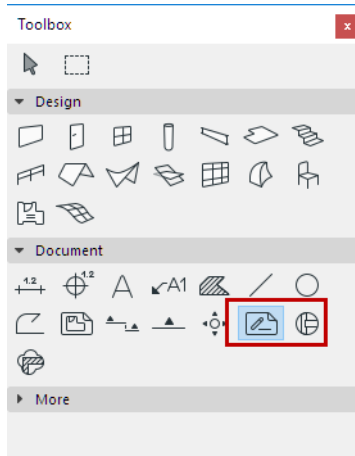
The following settings work essentially the same way for the Worksheet and Detail tools.

For a description of generic settings common to all tools in the Toolbox, see [Working in Tool Settings Dialog Boxes](#).

For more information on these views, see:

Worksheets

Details



General Panel

Reference ID: This editable field contains the ID of the generated Detail viewpoint. The ID serves to identify the Detail in the project window and in the Navigator.

Name: Enter a Name for the Detail Drawing if desired. This name appears in the Navigator palette and the title bar of the Detail window.

Note: If you place a Linked or Independent marker that does not generate a viewpoint, the ID/Name fields are gray.

Use the next pop-ups to either place a new Detail Marker (Detail Default Settings) or redefine the selected Detail Marker:

To Place a New Worksheet or Detail Marker (Default Settings)

If you choose **Create New Viewpoint:**

select one of the items about which the Marker should display information:

- the **viewpoint**

or

- the **first placed drawing of the viewpoint**

If you choose **Place Linked Marker:**

select one of the items about which the Marker should display information:

- a selected **viewpoint**
- a selected **drawing**
- the first placed drawing of the selected viewpoint
- the first placed drawing of the selected view

If you choose **Place Unlinked Marker**:

the Marker will not contain any linked information; you can define a custom text in the Marker Panel below.

Reference to: shows the path (location in the Navigator hierarchy) of the chosen reference item.

Copy construction elements only: If this box is checked, only 2D lines/fills and construction elements from the model view will be exploded and copied into the Detail viewpoint (without dimensions/annotations).

For information on annotation elements cut by the Detail boundary, see [Annotations, Dimensions and Markers Cut by the Boundary](#).

To Redefine a Placed Marker:

As needed, use the pop-ups to change the status of the selected Marker (either **Source Marker**, **Linked Marker**, or **Unlinked Marker**).

To redefine the Marker information: Choose from among the pop-up choices, or click the Browse button to select the viewpoint/view/drawing whose information you wish to display in the Marker. The available choices vary depending on the Marker status (Source Marker, Linked Marker or Unlinked Marker).

For an Unlinked Marker, enter a custom text in the Marker Panel below.

Marker Panel

Select Marker Type pop-up: Select a parametric marker from this list.

The chosen marker's 2D Symbol appears in the **preview window**.

Define Marker size and pencolor.

Uniform Marker Pen: Check this box to use this pencolor for all parts of the marker symbol, regardless of any custom colors set elsewhere for this marker.

Marker Angle: Use these controls to set the desired Marker angle. Define the angle as relative to the screen, or to the model.

- Enter a rotation angle for the Marker.
- Use the two icons to define how this angle value should be interpreted when the element is placed:
 - **Fixed Angle to Screen**
 - **Fixed Angle to Model:** Marker will be rotated along with the view.

Change Tool Settings

For a description of generic settings common to all tools in the Toolbox, see [Working in Tool Settings Dialog Boxes](#).

For more information on this tool, see [Revision Management](#).



Change General Panel

Change ID: This ID serves to identify the Change in the Change Manager, on the Layout Revision and in lists.

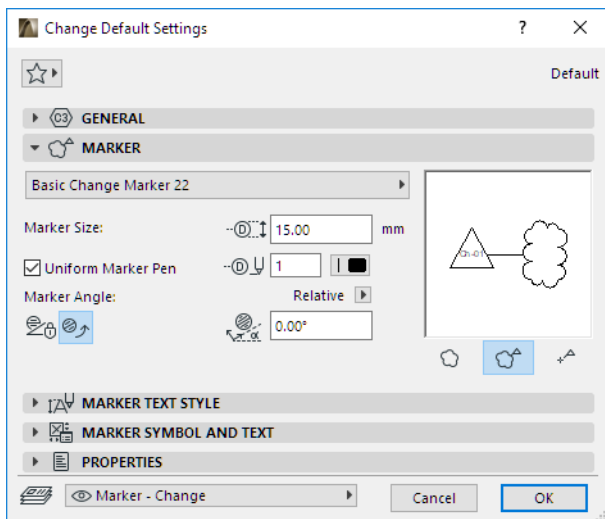
Change Name: Enter any descriptive text for the Change Name.

Note: If you place an unlinked Change marker, the ID/Name fields are gray. For an Unlinked Marker, enter a custom text in the Marker Panel below.

Marker Type: Select either Create new Change or Place unlinked Marker.

- If you choose **Create new Change**, a Change Marker is linked to the Change you just placed. The name of this marker name is shown in the **Reference to** field below.
- If you choose **Place unlinked Marker**, you will create an unlinked Change Marker. This is a graphical element, not linked to anything, but you can link it at any time to an existing Change.

Change Marker Panel



Select Marker Type pop-up: Select a parametric marker from this list.

The chosen marker's 2D Symbol appears in the **preview window**.

Preview: On the right, a preview area shows you how the Marker will look when placed.

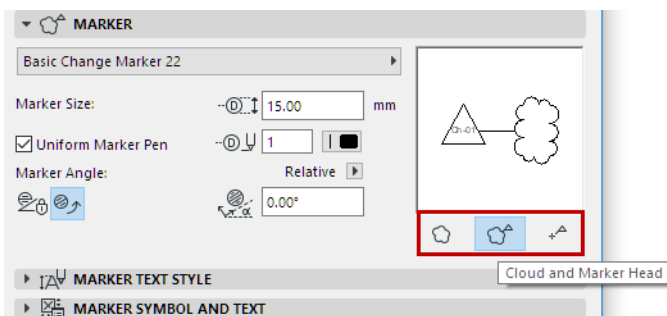
Define Marker size and pencolor.

Uniform Marker Pen: Check this box to use this pencolor for all parts of the marker symbol, regardless of any custom colors set elsewhere for this marker.

Marker Angle: Use these controls to set the desired Marker angle. Define the angle as relative to the screen, or to the model.

- Enter a rotation angle for the Marker.
- Use the two icons to define how this angle value should be interpreted when the element is placed:
 - **Fixed Angle to Screen**
 - **Fixed Angle to Model:** Marker will be rotated along with the view.

Display Cloud/Marker Head: Choose the Change element component(s) you want to place: a Cloud plus a Marker Head, or just one or the other.



Grid Tool Settings

For a description of generic settings common to all tools in the Toolbox, see [Working in Tool Settings Dialog Boxes](#).

For general information on the Grid tool function, see [The Grid Tool](#).



Grid Tool Floor Plan Panel

Use the controls in this panel to define the Floor Plan attributes of the grid element.

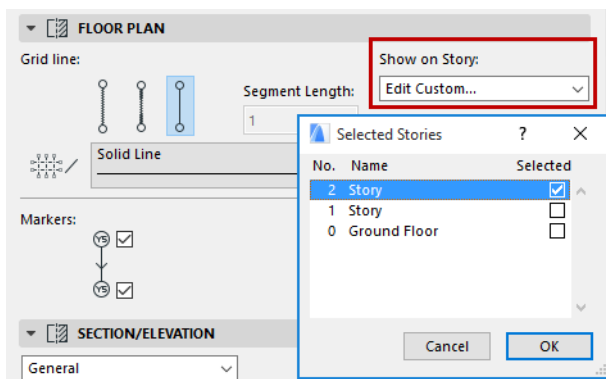
Grid Line: Select the desired representation of the grid line. It can be Hidden, Segmented or Full.

If you select Segmented, enter Segment Length in the field to the right.

Select a **Line Type** and **pen color** for this Grid Element.

Show on Story: From the dropdown list, select where you would like to see the Grid Element.

- **Current Story Only:** The Grid element will be shown only on the current story.
- **One Story Up and/or Down:** The Grid element will be shown on the current story, plus one story up and/or down.
- **All Stories:** The Grid element will be shown on every story of the project.
- **Custom:** Choose this option if you wish to set any other combination of stories. The Selected Stories dialog box appears.



(Once you set these Custom Settings, use the **Edit Custom** option to edit them.)

Markers: Select either, both or none of the two checkboxes to indicate the display of markers on either end of the Grid line.

Staggering: When you stagger grid markers sideways, a given grid line segment will be staggered together with the markers.

Set the default value of this staggering length here.

Grid Tool Section/Elevation Panel

This description applies to the display of Grid elements in both **Section and Elevation** windows.

Use the controls in this panel to define the Section attributes of the Grid element.

General/Custom: Use this pop-up to apply either the General settings to the selected Grid element, or to set Custom settings for the selected Grid element in this Section only.

Every Grid element has one “General” section settings and any number of “Custom” Section settings.

A Grid element using General section settings will be displayed identically in every Section that it appears in.

Custom settings for selected Grid elements are available only if you open Grid Element Settings from a Section window. Custom settings for a Grid element are applied *only* in the current Section window.

Thus, the same Grid element can be displayed in different ways, depending on which Section it is in.

If you apply Custom settings to a Grid element in one Section, then when opening the same Grid element in another Section, it will be displayed using its original settings, because Custom settings apply to a given Grid element only in a single Section window. However, you can use the **Copy from** command to copy those Custom attributes from the other Section, and use them for the Grid element in the current Section, too.

Grid Line: Select the desired representation of the grid line. It can be Hidden, Segmented or Full.

If you select Segmented, enter Segment Length in the field to the right.

Select a **Line Type** and **pen color** for this Grid Element.

Markers: Select either, both or none of the two checkboxes to indicate the display of markers on either end of the Grid line.

In the two fields to the right, enter the height level of each Grid line marker as displayed in Sections, and choose a reference level (default: Project Zero) for measuring this height level.

Note: The actual position of the Grid line (unlike other Archicad elements) will change if the Reference Level is changed.

Staggering: When you stagger grid markers sideways, a given grid line segment will be staggered together with the markers.

Set the default value of this staggering length here.

Grid Tool Naming Rules Panel

Use this panel to define the logic by which each placed Grid element is named.

Generate names automatically: Select this option to auto increase the name every time you place a Grid element.

- **Start at:** This field shows the next name value to be assigned to the next placed Grid element.
- **Prefix:** a static text displayed in front of each auto generated name.
- **Suffix:** a static text displayed following each auto generated text
- **Style:** Choose a style for the auto generated Grid element names: either numbers (1,2,3,...), letters (A,B,C,... or a,b,c,...), or Roman numbers (I,II,III,IV,...)

A **preview** of the next name value to be assigned to the next placed Grid element is visible in the field next to Custom.

Custom: Click this button to override the auto generated name and to assign any other name to the Grid element.

Grid Tool Marker Panel

Use this panel to define the appearance of the Grid element's markers.

Marker Shape: Click this item, then click the black arrow to pop-up a list of available marker shapes. Choose one.

Connect Grid Line to: Click this item, then click the black arrow to pop-up a list of available choices of where to connect the grid line to the marker. (Not available for circular marker shape).

Grid Tool 3D View Panel

Use this panel to control whether and how the Grid Element appears in the 3D window. 3D Documents created from a 3D window showing Grid elements will also display Grid elements.

Display in 3D view: Check this box to display the Grid element in the 3D window. If this box is not checked, all other controls in this panel are grayed.

Vertical position: Enter the vertical position of the Grid Element in 3D. Click the black pop-up arrow to choose a reference level from which to measure its vertical position.

Show as line: Check this box to display Grid elements as line only in the 3D window. If you check this option, Grid elements will not be visible in renderings, and the rest of the Grid line controls in this panel are grayed. Unchecking this option means that Grid lines will be displayed in 3D with surfaces and a physical cross-section, and will be displayed in renderings.

Grid Line Surface: Set the surface of the Grid line's body.

Grid Line cross-section: Select between circular and rectangular cross sections and set their size

Marker Text Surface: Set the marker text's surface

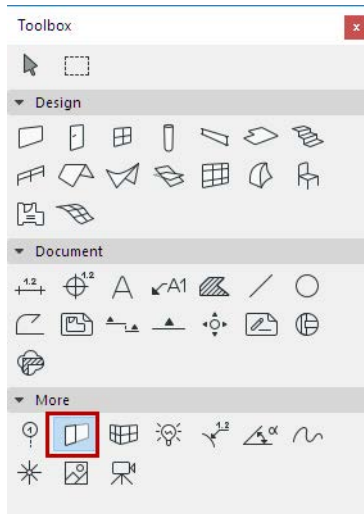
Text Follows View: Check this box to rotate the marker texts to always face the viewpoint. Note: This option is effective only in perspective views.

Wall End Tool Settings

For a description of generic settings common to all tools in the Toolbox, see [Working in Tool Settings Dialog Boxes](#).

For more information, see [Wall Ends](#).

The Wall End tool is used for placing wall end items onto walls by clicking one end of a wall.



The left-hand browser area of the **Wall End Settings** dialog box is similar to those of other Library Part elements.

See [Settings of Library Part Elements](#).

Wall End Preview Panel

On the right-hand side of the panel, the **Preview Area** displays the 2D symbol, the hidden line front view, the hidden line axonometry, the 3D shaded axonometry, the predefined preview picture or the optional information notes of the selected Wall End, depending on the button you switch ON.

Wall End Parameters Panel

In the **Parameters** panel, you can adjust the size, thickness, and (where applicable) fills and colors of the wall end.

- The **Fit to wall height** checkbox is checked by default. If you uncheck it, you can use the settings below to enter any value for the height of the wall end and for the distance of the wall end from the base of the wall.
- The **Cut Depth** parameter tells you how much of the wall, in the Floor Plan view, is cut by the wall end.

Wall End Thickness Panel

Where applicable, the **Thickness** panel contains a preview to assist in setting the wall end's thickness.

- Depending on its type, a Wall End can be set to automatically inherit the wall's skin fills, or you can custom-set the Wall End fills instead.

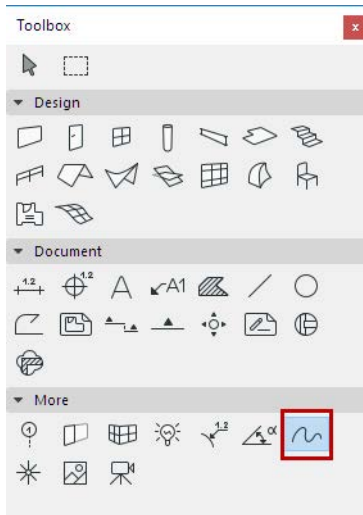
For wall end objects with turning skins, you can set which wall fills should turn together with the wall end.

- As with other objects, you can also set many of these parameters in the Info Box or by using the wall end's editing hotspots.

Spline Tool Settings

For a description of generic settings common to all tools in the Toolbox, see [Working in Tool Settings Dialog Boxes](#).

Geometry methods are: Natural Splines, Bézier Splines and Free-hand.



For more information, see [Drawing Splines](#).

Spline General Settings Panel

The **General Settings** panel is similar to those of the **Line**, **Arc/Circle**, **Polyline** and **Spline** tools.

See description at [Line General Settings Panel](#).

For Splines only, use the two pairs of control buttons to define the type of spline:

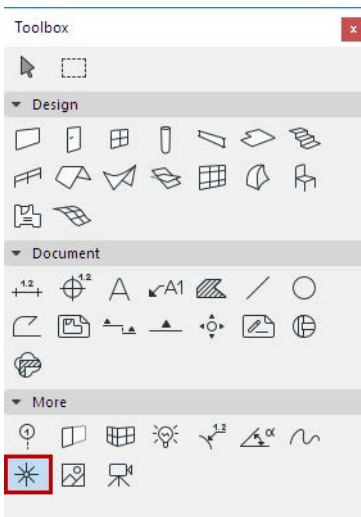
- either a natural Spline or a Bézier curve.
- either an open-ended or a closed spline.

Spline Properties Panel

See description at [Line Properties Panel](#).

Hotspot Tool Settings

Hotspots are drafting aids to mark important nodes anywhere on screen.



To add Hotspots to your plan, activate the Hotspot Tool and click.

Note: Hotspots do not appear on the printed and plotted outputs.

For more information, see [Hotspots](#).

Hotspot Color Panel

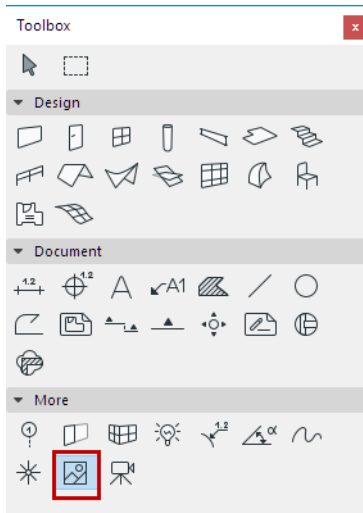
Click the Color drop-down menu to either type a Pencil color number (1-255) for the Hotspot in the text field or choose the appropriate color from the pop-up palette.

For more information on other options of this dialog box, see [Working in Tool Settings Dialog Boxes](#).

Figure Tool Settings

For a description of generic settings common to all tools in the Toolbox, see [Working in Tool Settings Dialog Boxes](#).

The Figure tool is a dedicated tool for placing and manipulating images. Figures are embedded images in the project file.



Images files can be directly imported from within the Figure Settings dialog box.

For more information, see [Figures](#).

Figure Preview and Positioning Panel

You have two options for obtaining image files:

- Click the **Open** button to browse the directory system for image files.
- Click the **Paste** button to use the contents of the Clipboard. (If the Clipboard contains image information, it will be used directly. If the contents is text, it will be first converted to bitmap.)

Note: You can also insert a cropped view of any image file into your Archicad plan. With your Archicad project file open, open any image format file. Click and drag (the Marquee tool is the only available tool) to select the desired image area, then copy it to the clipboard. Close the image file, then paste the clipboard contents into the open project. You can modify it like any other figure.

The Preview area displays the image.

The Name field below displays the imported file's name, or "Pasted Image #" if the figure was inserted from the Clipboard. Click in the field to rename the image if needed. This does not affect the original file's name.

Figure Size: Use these fields to give accurate dimensions to scanned site plans or measured layouts.

- Enter a value for the horizontal size of the inserted image in default length unit.
- Enter a value for the vertical size of the inserted image in default length unit.
- The **Keep Proportions** checkbox below allows you to fix image proportions if appropriate.

Hint: Dimensions are often distorted when drawings or images are scanned. Uncheck the **Keep Proportions** box to unlink the dimensions and adjust figure width and height independently if you want to make sure that dimensions are correct. This is especially important when inserting existing site plans, survey plans or layouts into Archicad Projects.

Resolution: Adjust resolution of the inserted figure if necessary for specific print output quality:

- Enter a horizontal resolution in dpi.
- Enter a vertical resolution value in dpi.

Note: Figure size and image resolution are interdependent. By increasing image size, you decrease resolution, and vice versa.

Use the **Restore Original Values** button to reset the image default parameters for figure size and resolution.

Insert Figure using 1 to 1 pixel correspondence with screen resolution: Every time you insert the selected image with this option, one pixel of the selected image will correspond to one pixel of the screen. The figure size upon insertion will depend on the current zoom level of the screen.

Check **Transparent** to show elements that are located underneath the Figure's white pixels, according to the current display order.

Check the **Mirror** icons to insert the figure flipped horizontally.

- **Rotation Angle:** Enter a rotation angle for the Figure.

Use the pop-up to define how this angle value should be interpreted when the element is placed:

- either **Relative to Orientation** (the angle will be measured relative to the Oriented View)
- or **Absolute to Coordinate System** (the angle will be measured from the (0,0) point of the project coordinates).

[See also Set Orientation.](#)

Click an **Anchor point** button to define the positioning point of the Figure.

Figure Image Format Panel

Use the **Storage Format** field to change the format in which the file is saved within the project file: click on the preferred format and choose OK.

The control to the right of Storage Format displays the **color resolution** of the image.

The controls below display the horizontal and vertical **image size** in pixels.

Use the **Save as** button to save a copy of the selected image file in a different format as an external file.

The field next to the disc icon displays the file size of the selected image in kilobytes.

Camera Tool Settings

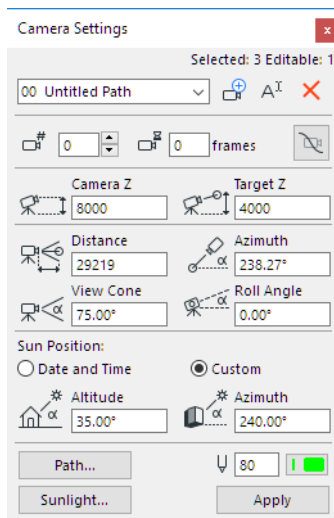
Double-click the Camera tool to open Camera Tool Settings.

Use these settings to define individual perspective views and the Fly-Through path.



For more information, see [Cameras](#).

The Camera Settings palette and the Floor Plan are simultaneously active, so you can alter the settings, the camera locations and orientations without repeatedly opening and closing the window.



Click **New** to add a new path to the project to start a new collection of 3D projections. The **New Camera Path** dialog box opens. Here you can:

- Enter the name of the new Path.
- Add Copy of Current Path: Check this box to duplicate the current set of projections while you define a new one.

Click **Rename** to rename the current path, then type the new name in the dialog box and click **OK** or select a path in the pop-up list.

If you have already defined one or more camera paths, choose the desired camera path from the pop-up menu.

Click **Delete** to delete the current path. The path name will be removed from the list and all its cameras will be deleted.

Camera

To select a particular camera from the current camera path, use the up/down arrows to scroll through the camera numbers, or enter the camera number from the keyboard.

Wait here

This value refers to the number of still frames (time units) during which the fly-through motion will remain frozen at the given camera before moving on to the next one.

Smooth Path at Cameras

Restores the tangent line defining the path to its original position, if you have modified it.

Current Camera Position

Camera Z: Enter a value to specify Camera height from Project Zero.

Target Z: Enter a value to specify Target height from Project Zero.

Distance: Enter a value to specify the horizontal viewing distance between Camera and Target.

Azimuth: Enter a value to specify Camera Azimuth to Target.

View Cone: Enter a value to specify the opening angle of the Camera View Cone.

Roll Angle: Enter a value here to specify Camera Roll Angle.

Sun Position

Use these controls to define a Sun Position for the selected Camera.

[See Sun Position.](#)

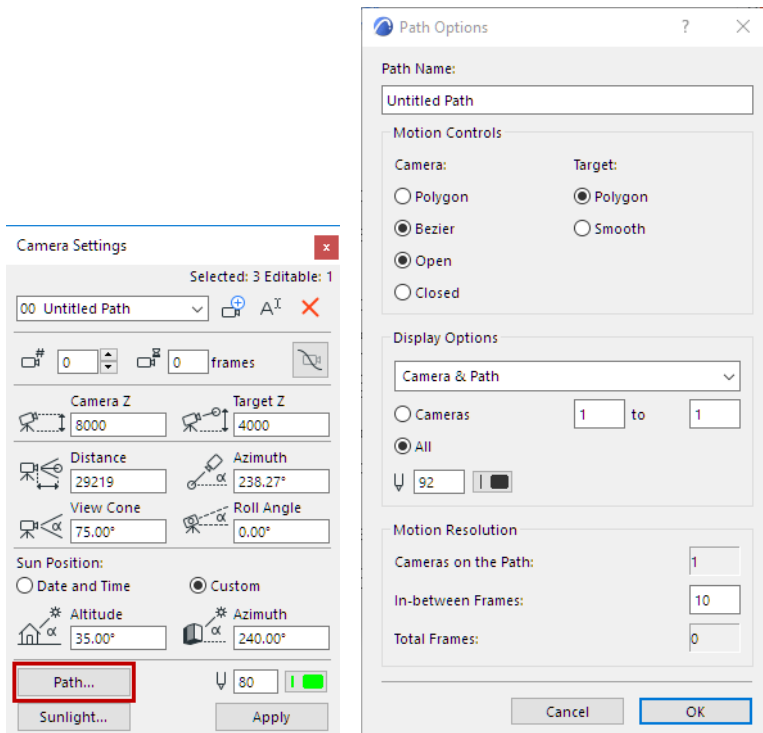
Use the Pencolor/Penweight controls to specify a pen color for the selected camera. (To modify the color of the path line, use the **Path** button for opening the **Path Options** dialog box.)

Sunlight: Click this button to access the Sunlight Settings dialog box.

[For details, see Sunlight Settings dialog box.](#)

Path Options

To open this dialog, click Path from Camera Settings.



Motion Controls

Camera: Choose radio buttons to set the shape of the Camera Path:

- **Polygon:** Click this button to place In-between Frames along a straight line between Key Frames (Polygonal Animation Path).

- **Bézier:** Click this button to place in-between Frames along a Bézier curve between Key Frames (Bézier Animation Path).
Note: Bézier makes for smoother animations, while Polygon paths are somewhat jumpier.
- **Open:** Click this button for an open-ended Path Loop.
Hint: The first and last Key Frames of an Animation Path can also be coincident.
- **Closed:** Click this button for a continuous-loop Path.
Hint: Use this to create smooth endless loop Animations.

The **Target** option defines the shape of the target point path during the Fly-Through animation.

- **Polygon:** This option will place the target points of the in-between frames on the straight line that connects the two consecutive key target points.
- **Smooth:** This option will place Targets along a smooth curve between Key Targets.
Note: Smooth is available only if you have specified Bézier for the camera motion.
Hint: Use Polygon target motion if your camera moves a lot but the target stands stills or moves just a little. This is the case when you fly around a building but keep the same spot in sight. In other cases, try both methods to see which one suits you better.

Motion Resolution

Each 3D Projection you add to the path is considered a keyframe. Archicad can interpolate between keyframes to create In-between views which produce a smoother animation.

Cameras on the path: Shows the total number of Cameras in the current Polygonal or closed loop Bézier Path.

In-between frames: Enter the number of in-between Frames between each Key Frame here.

A larger number of in-between frames makes a smoother animation Path, but uses more memory and hard disk storage space.

Total frames: Shows the total number of Key and in-between Frames.

Display Options

Choose the format in which to display the frames of the current Path: (Cameras are never displayed in printed output.)

- None: Since Cameras cannot be assigned to Layers, choosing None in Display Options here is the only way you can hide them on the Floor Plan.
- Camera only
- Camera & Path
- Everything: the location of the in-between frames will also be shown.

Choose which frames to display:

- Click the **Cameras** radio button to specify a range of keyframes to be displayed.
- Click **All** to display all frames of the path

This setting will be used as a default in the **Create Fly-Through** dialog box: only the visible part of the path will be processed.

Enter a Pencolor for the line of the camera path.

For more information, see [Fly-Through](#).

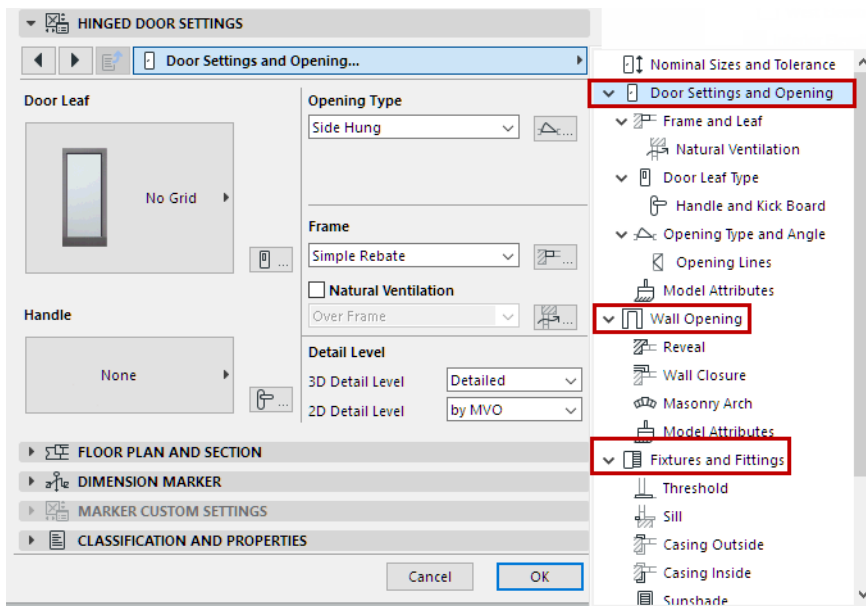
Click **OK** to confirm Path Options and return to the Camera Settings dialog.

Custom Settings Panel for a Door/Window

For every Door/Window in the INT library, you can set parameters using the Custom Settings panel, which has a graphical interface. The name of the panel varies depending on the Door/Window type that is selected.

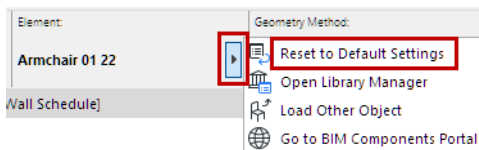
The following sections describe the Custom Settings Panel of Door and Window Settings.

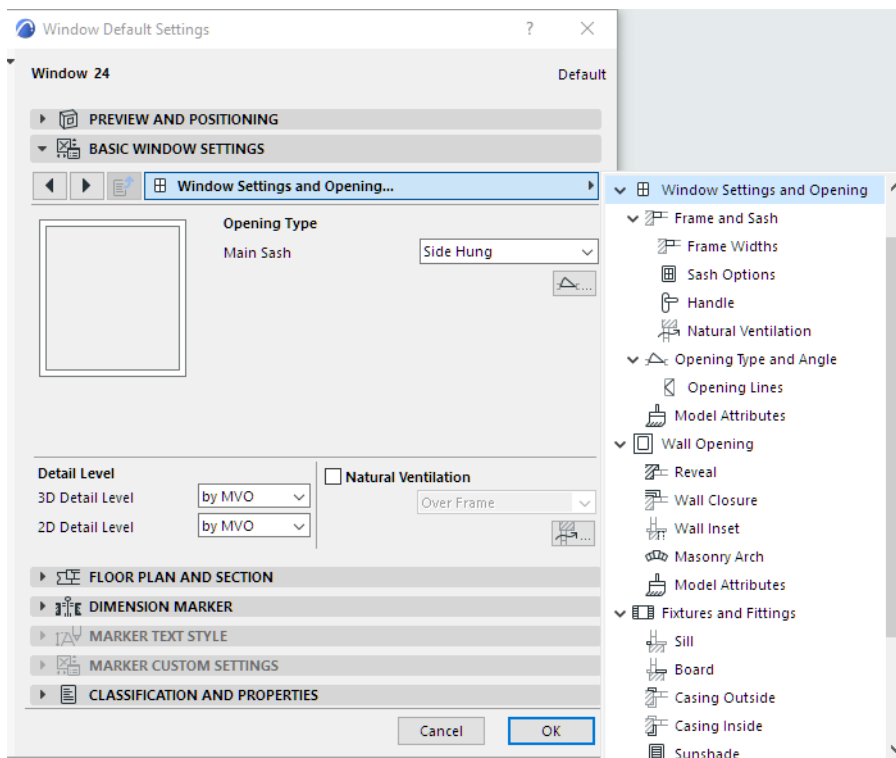
Available Door/Windows and their parameters vary depending on your library version. For information on Door/Window parameters that are not described here, please contact your distributor.



Reset to Default Settings

If you customize any Library Part's GDL settings, you can reset to its default using the Reset command from the Info Box:





For general information on the Custom Settings panel, see [Where to Set Door/Window Parameters](#).

Nominal Sizes and Tolerance

Shape

Door/Window Settings and Opening

Frame Style and Dimensions (for Door)

Natural Ventilation

Door Leaf Type

Handle and Kick Board (for Door)

Leaf and Sash Dimension (for Door)

Sash Type/Sash Options (Door or Window)

Frame and Sash (for Windows)

Frame Widths (for Windows)

Opening Type and Angle (Door/Window)

Opening Lines (Door/Window)

Opening Lines - Sidelight and Transom

Model Attributes of Door/Window Opening

Wall Opening

Reveal Settings

Wall Closure

Wall Inset (for Windows)

Masonry Arch

Tolerances

Model Attributes of Wall Opening

Fixtures and Fittings

Threshold Settings

Sill Settings

Board Settings

Casing Outside/Inside Settings

Sunshade Settings

Model Attributes of Fixtures and Fittings

Floor Plan and Section Tab Page (Door/Window Settings)

Minimal Space

Custom Corner (for Windows)

Nominal Sizes and Tolerance

The following is a description of the Nominal Sizes and Tolerance page of the Custom Settings Panel of Door or Window Settings, from the INT Archicad Library.

Opening size (height and width) can be defined with respect to any of four types of dimensions: Wallhole, Unit, Egress or Leaf dimensions.

The numbered graphics help you identify the dimensions.

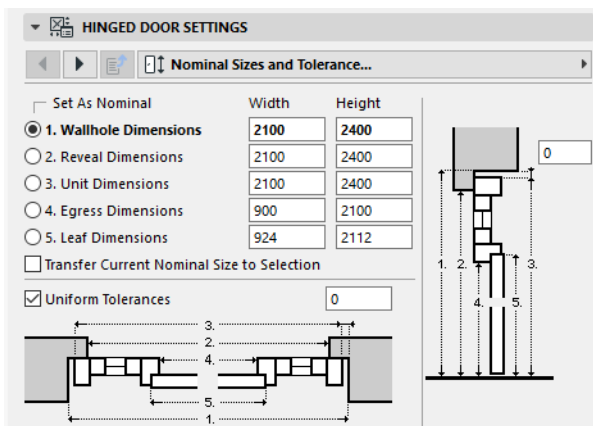
Wallhole Dimensions

The Wallhole is the part cut out of the wall to accommodate the whole opening structure.

Wallhole Width is equal to **Unit Dimension Width** plus the sum of **Side Tolerance** values.

Wallhole Height is equal to **Unit Dimension Height** plus Upper **Tolerance** value.

[See Tolerances.](#)



Unit Dimensions

Unit Dimensions are the width and height of the whole opening structure.

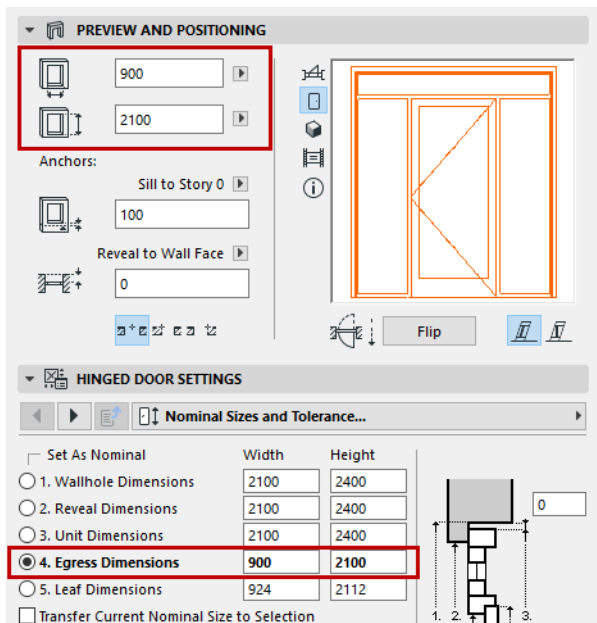
Egress Dimensions

Egress dimensions are the opening's unit width, subtracting the frame width.

Set as Nominal

Use the **Set as Nominal** radio button to choose one of these dimension definitions as the opening's Nominal Size.

- This is the Nominal dimension that will be listed in the Interactive Schedule in the **Nominal W x H Size** field.
- This Nominal dimension will be displayed in the Width and Height fields in the Preview and Positioning Panel or Window Settings.

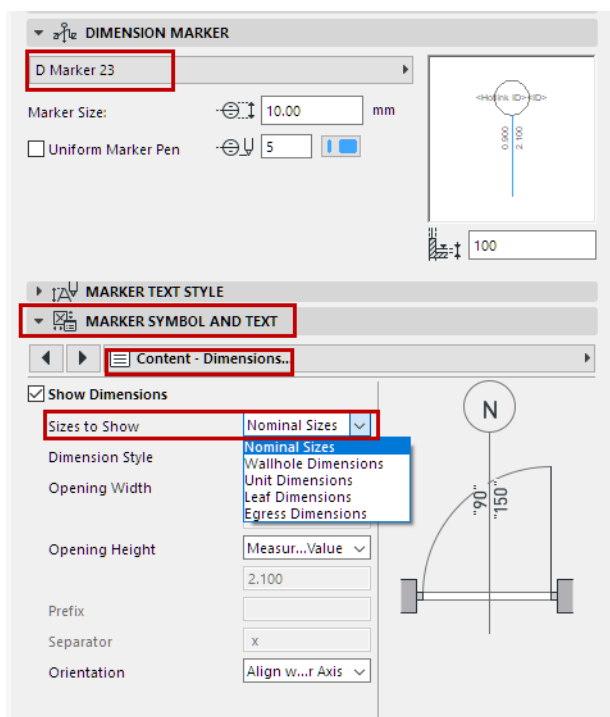


Transfer Current Nominal Size to Selection: This checkbox is applicable if you change your definition of the Nominal Dimension (e.g. from Unit Dimensions to Wallhole Dimensions). If this box is checked, then the values of the original Nominal Dimension (those set for Unit Dimensions) are transferred to the new Nominal Dimension (your Wallhole Dimensions). The other dimension values will be adjusted automatically.

Show Dimensions in Marker

The opening's dimensions (Wallhole, Unit, Leaf or Egress Dimensions) can be displayed in the opening's Marker, if any.

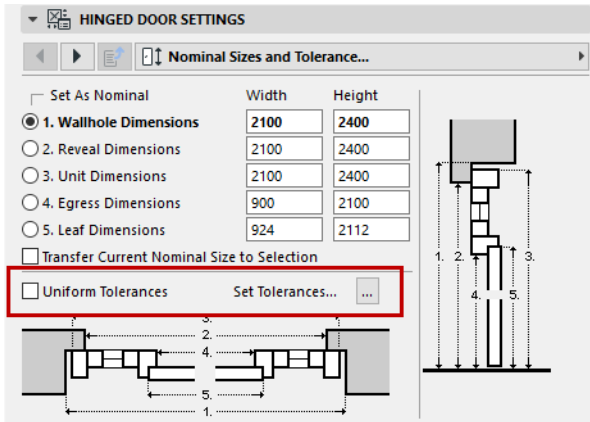
Use the Marker Symbol and Text panel / Content - Dimensions page to choose the preferred dimension to display in the marker.



The dimension definitions chosen here will also be used in Automatic Dimensioning and the Interactive Schedule.

Uniform Tolerances

By default, this is checked, so that you can enter a single value to apply to tolerances on all sides of the opening structure. To vary the tolerance values for the sides of the opening, (as for a complex-shaped opening whose sidelights have varying sill heights), uncheck the box and click Tolerances to go to the Tolerances tab page.

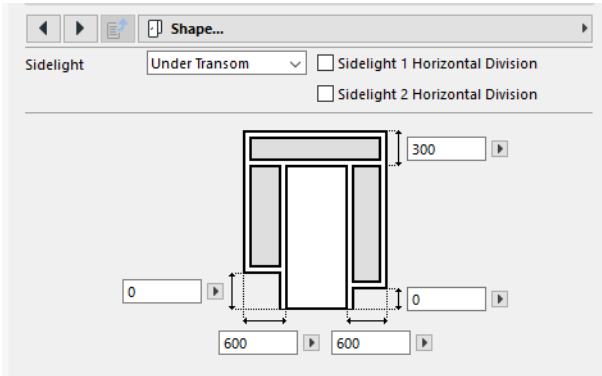


[See Tolerances.](#)

Note: If you wish to enter **Oversize** values, use the Oversize controls on the [Wall Opening](#) page of this Settings dialog. Entering an Oversize does not (unlike the Tolerance value) affect the Nominal Size of the opening.

Shape

Use this page of the Custom Settings Panel of Door or Window Settings, to define the configuration of door leaves/window panes and their divisions.

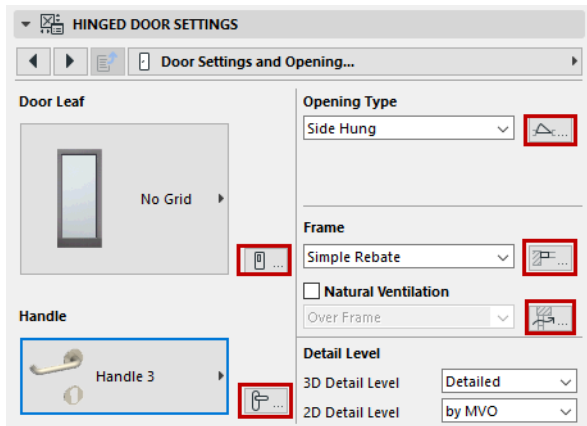


Door/Window Settings and Opening

The following is a description of the Door/Window Settings and Opening tab page of the Custom Settings Panel of Door or Window Settings, from the INT Archicad Library.

You can set all of the main parameters on this tab page.

To set additional, more detailed parameters for any of these components (e.g. Leaf, Frame), click the icon at the right of the setting to access the respective tab page, where you will find additional parameters.



Door Leaf and Handle

Use the pop-ups to choose a **Leaf** style and **Handle** style for the door.

To set additional parameters, click the icon at the right of the respective pop-ups to jump to the Door Leaf Type tab page.

Opening Type

For each component of the opening, choose a type from the pop-up. (Available settings depend on the library part.)

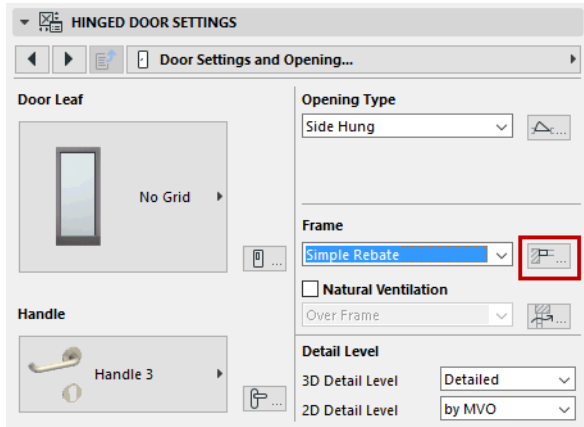
To set additional parameters, click the icon at right to jump to the Opening Type and Angle tab page.

[See *Opening Type and Angle \(Door/Window\)*.](#)

Frame Style: Choose one of three frame styles.

Note: If the chosen door uses a double-egress opening type, the simplest Block style is automatically used. In this case, no other options are available.

To set additional frame parameters, click the icon at right to jump to the Frame Style and Dimensions tab page.



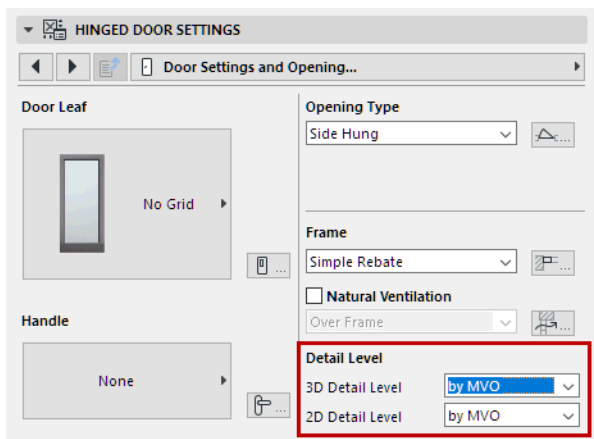
Natural Ventilation: Check this if the opening uses natural ventilation, then choose either “over Frame” or “over Glazing.”

To set additional ventilation parameters, click the icon at right to jump to the Natural Ventilation tab page.

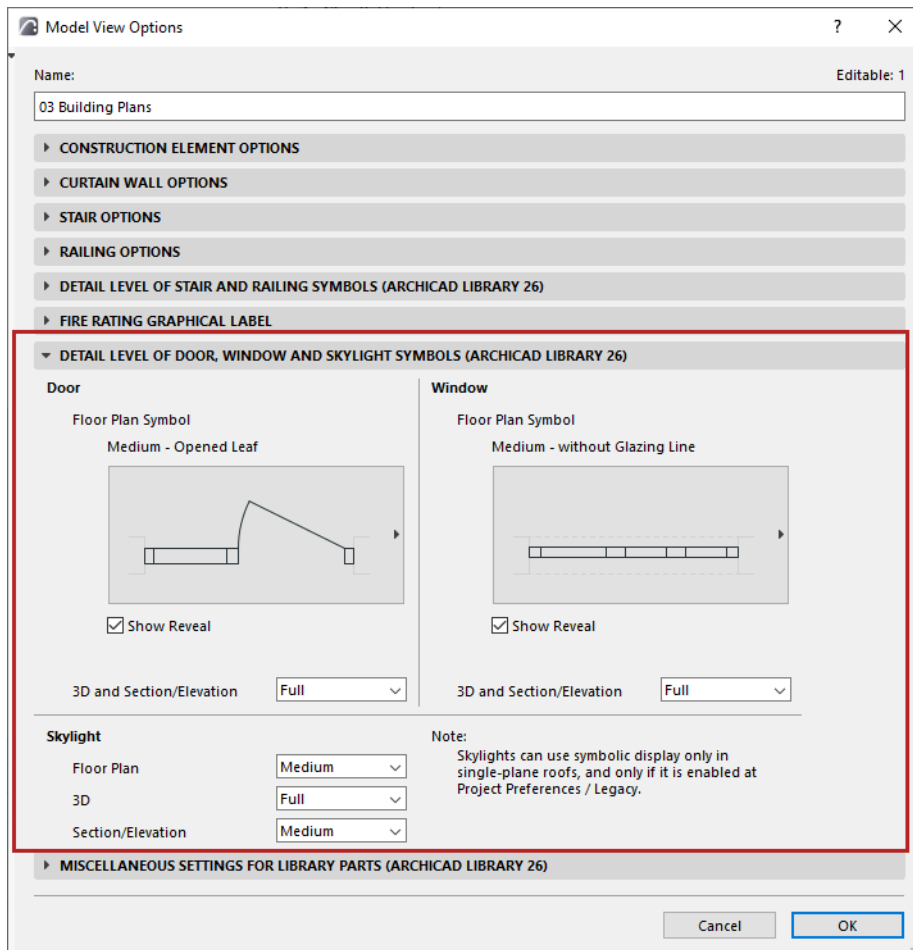
[See Natural Ventilation.](#)

Detail Level

Choose options for the level of detail with which to display the opening in both 2D and 3D.



The default setting for both 3D and 2D Detail level is **By MVO**. This corresponds to the options set in Model View Options (Document > Model View > Model View Options):



3D Detail Level

To overwrite the MVO-based settings for this Door/Window: Choose Schematic, Simplified, or Full.



From left: Schematic, Simplified, Full

If you choose Off, only an empty opening is shown.

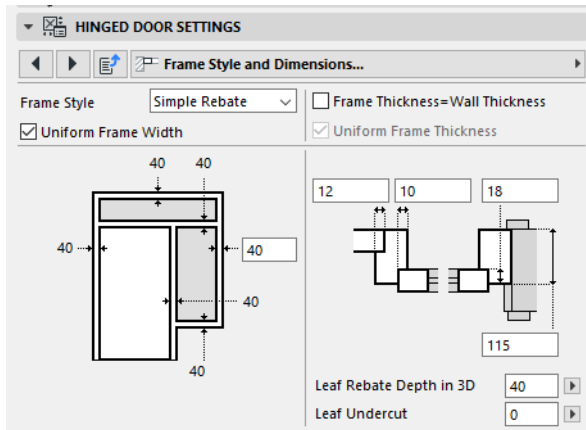
2D Detail Level

Scale Sensitive: The detail level depends on project scale.

Otherwise, you can fix the detail level to a given level (e.g. 1:200), or turn it off (in which case, no 2D opening symbol is shown at all).

Frame Style and Dimensions (for Door)

The following is a description of the Frame Style and Dimensions page of the Graphical Interface Panel of Door Settings for a “Hinged Door”, with sidelight and transom, from the INT Archicad Library.



Frame Style: Choose one of three frame styles.

Note: If the chosen door uses a double-egress opening type (set on the General Settings page), the simplest Block style is automatically used. In this case, no other options are available.

Depending on the chosen Frame Style, set parameters for frame width and thickness, and the thickness of the leaf and glass.

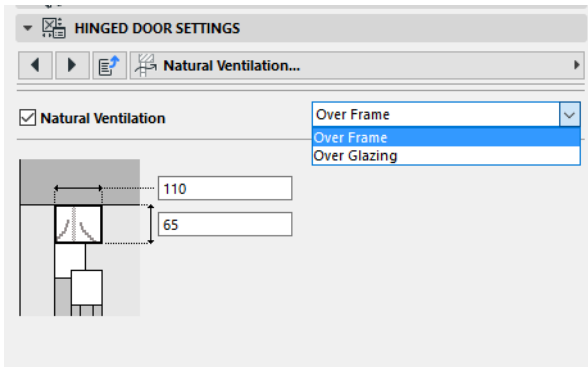
Frame Thickness = Wall Thickness: Check this to make the frame and the wall the same thickness.

Uncheck **Uniform Frame Thickness** if you want to vary the frame thickness parameter.

Note: By default, this checkbox is checked and gray (unavailable.) This parameter becomes available only if you have set Frame Thickness to equal Wall Thickness, and if all door sashes open in the same direction. (By default, door leaves and their sidelights/transoms open in opposite directions.)

Natural Ventilation

The following is a description of the Frame Style and Dimensions page of the Custom Settings Panel of Door or Window Settings, from the INT Archicad Library.



Natural Ventilation: Check this if the opening uses natural ventilation, then define its position: choose either “over Frame” or “over Glazing.”

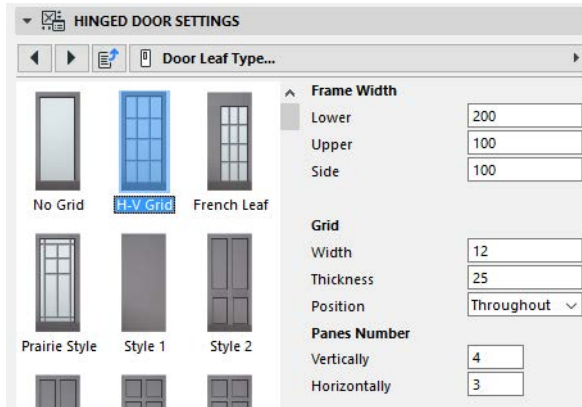
In the fields below, define the height/width of the ventilation component profile (for each sash, if applicable.)

Set fill attributes for the Ventilation profile (in Section views) on the Floor Plan and Section page.

[See Floor Plan and Section Tab Page \(Door/Window Settings\)](#)

Door Leaf Type

The following is a description of the Door Leaf Type tab page of the Custom Settings panel for a “Hinged Door”, with sidelight and transom, from the INT Archicad Library.



Door Leaf Type

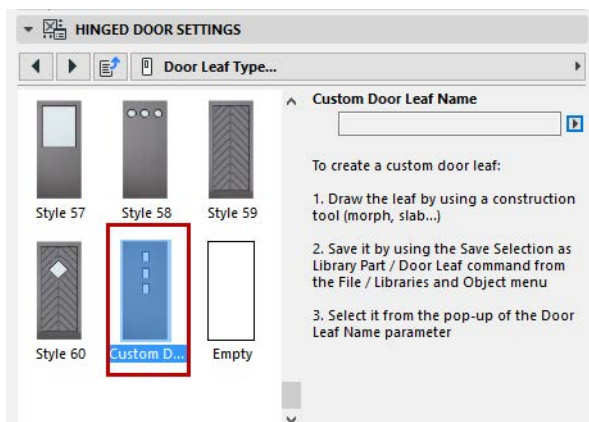
Scroll through the preview panel to find a leaf style that suits your needs. Most of these are predefined; in some cases, you can adjust the width of the frame.

Grid-Style Panels

Choose the **Grid** style (H-V Grid, French Panel or Prairie Style) if you wish to specify the number of horizontal and vertical panes and the mullion configuration. For these panel styles, set the **Position**: place the mullions inside or outside the door, or run them through on both sides (“Throughout”).

Custom Door Leaf

If you have created a **Custom Door Leaf**, choose the Custom style to apply your leaf to the door.



For information on creating a custom door panel, see [Create Custom Components for Doors, Windows, Curtain Wall, Stair, Railing](#).

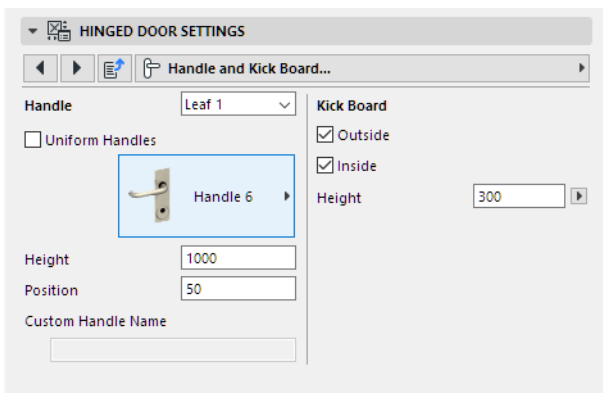
Handle and Kick Board (for Door)

The following is a description of the Handle and Kick Board tab page of the Custom Settings panel for a “Hinged Door”, with sidelight and transom, from the INT Archicad Library.

Door Handle

For a multi-leaf door, set Handle preferences for each leaf separately, or check Uniform Handles to use the same on all door leaves.

From the pop-up list, choose a Handle. Set its height and its position (horizontal distance) from the opening edge of the door.

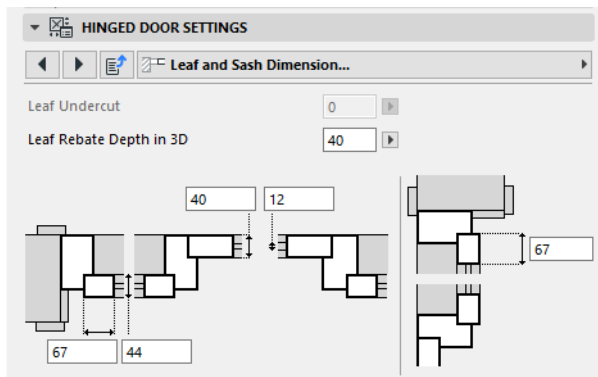


Kick Board

Check to add a kick board on the outside and/or inside of the door, and to define its height.

Leaf and Sash Dimension (for Door)

Use this tab page to define the thickness and width of the door leaf, sash frames, and the glass.

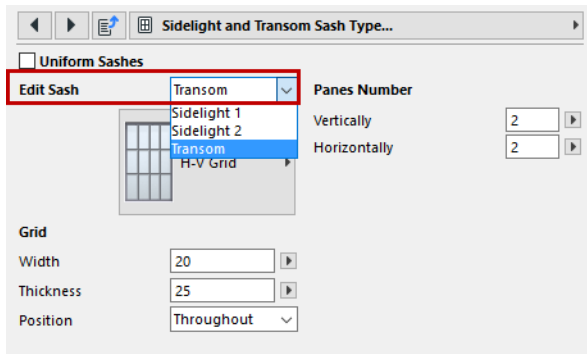


Sash Type/Sash Options (Door or Window)

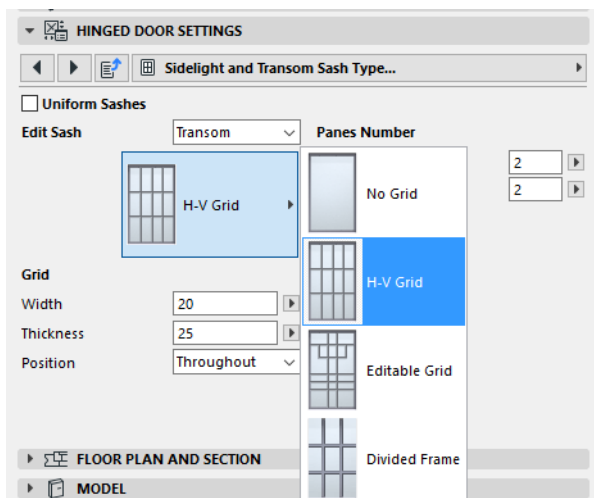
The following is a description of the Sash Type or Sash Options page of the Custom Settings Panel of Door or Window Settings, from the INT Archicad Library.

Start by selecting the sash or transom to edit from the pop-up list.

Note: If you check the **Uniform Sashes** box, all the sashes will use a single type.



Next, choose a pattern of mullions for the selected sash:

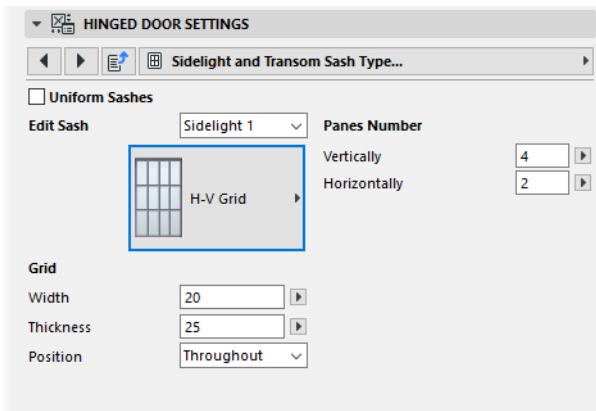


See also [Custom Panel](#).

Note: The “Editable Grid” sash type means that you can edit the sash divisions graphically, in the project, using the editable hotspots.

See [Graphical Editing Using Editable Hotspots](#).

Enter values for the sash pattern: the number of panes horizontally and vertically, and parameters for the grid divisions.

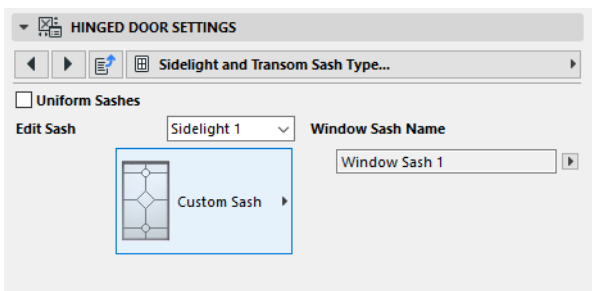


Position: Choose whether to place the grid for this sash inside or outside the opening, or to run it through on both sides (“Throughout”).

Custom Panel

If you have created a custom Window Sash, choose the **Custom Panel** style to apply your sash to the sidelight.

For information on creating a custom panel, see [Create Custom Components for Doors, Windows, Curtain Wall, Stair, Railing](#).

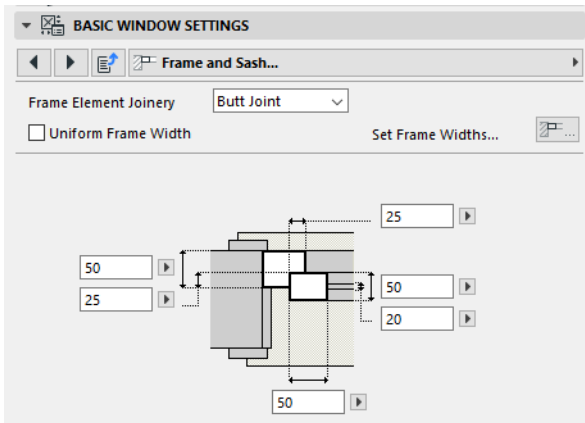


Frame and Sash (for Windows)

The following is a description of the Frame and Sash page of the Custom Settings panel for a Basic Window, from the INT Archicad Library.

Use these controls to adjust the frame and sash width and thickness.

Note: To set options for the sash types and grids, use the Sash Options tab page.



Frame Element Joinery: Choose either Mitered or Butt Joint.

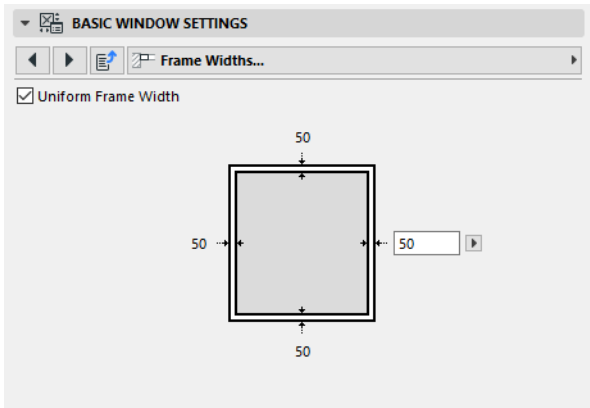
Uniform Frame Width: Check this box if you need a single frame width for the entire window structure. To set a different frame width for any and all sides of the frame, uncheck the box and then click the **Set Frame Widths** button to go to the **Frame Widths** tab page.

[See *Frame Widths \(for Windows\)*.](#)

Frame Widths (for Windows)

The following is a description of the Frame Widths page of the Custom Settings panel for a Basic Window, from the INT Archicad Library.

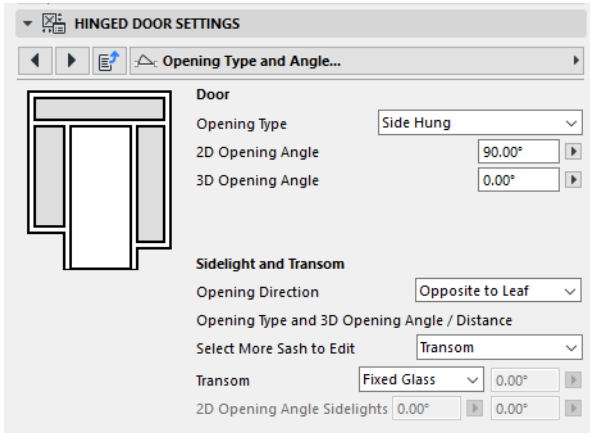
If **Uniform Frame Width** is checked (either on this tab page, or on the Frame and Sash tab page), then you set a single width value for all sides of the frame.



If you uncheck **Uniform Frame Width**, you can set varying width values for the sides, (as for a complex-shaped window).

Opening Type and Angle (Door/Window)

The following is a description of the Opening Type and Angle tab page of the Custom Settings Panel of Door or Window Settings, from the INT Archicad Library.



Use this to set options for the openings of the opening as a whole, and for the sidelight and transom (if used).

Opening Type

For each component of the opening, choose a type from the pop-up. (Available settings depend on the library part.)

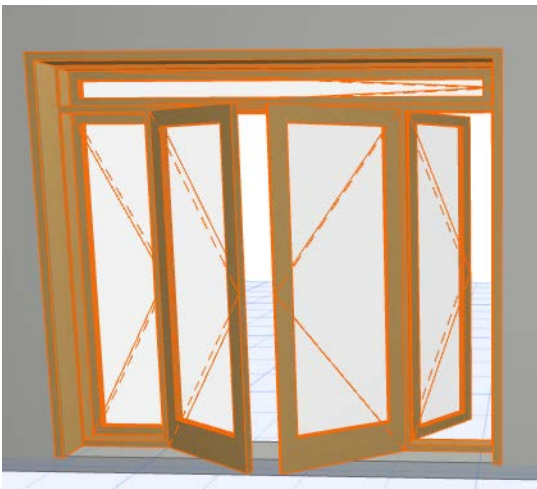
Note: The Opening Type chosen here may limit the possibilities for the door's Frame Style.

2D/3D Opening Angle

Optionally, for each component of the opening, set an opening angle in 2D and 3D display. (You can also edit the opening graphically, using the editable hotspots.)

Sidelight and Transom: Opening Direction

For the Sidelight and Transom, choose "Same as Leaf" or "Opposite to Leaf" from the pop-up to define the opening direction.



Sidelight and Transom: Opening Type and 3D Opening Angle/Distance

Use the pop-up lists to choose a type for the sidelights and transom. Enter an opening angle (or opening distance, for a sliding door) as needed.

Opening Lines (Door/Window)

The following is a description of the Opening Lines tab page of the Custom Settings Panel of Door or Window Settings, from the INT Archicad Library.

Use these controls to set the display of Opening Lines for each opening.

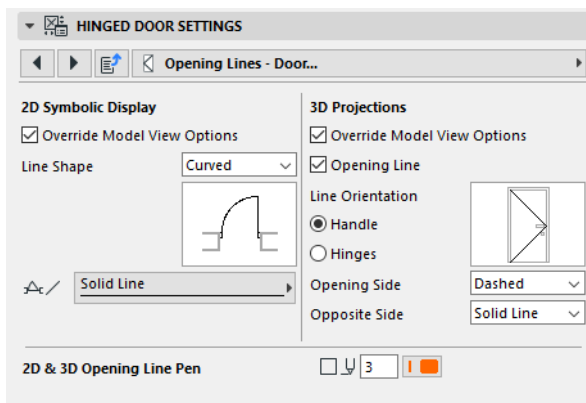
By default, these controls are grayed, because Opening Line display is set globally for all doors and windows in Model View Options.

For details on these settings, see [Miscellaneous Settings for Library Parts](#).

However, to adjust Opening Line settings for this opening only, check the **Override Model View Options** box and set the options as needed.

Separate options are available for 2D Symbolic Display and 3D Projections.

The **2D & 3D Line Pen** control affects the opening line display in both 2D and 3D.



Opening Lines - Sidelight and Transom

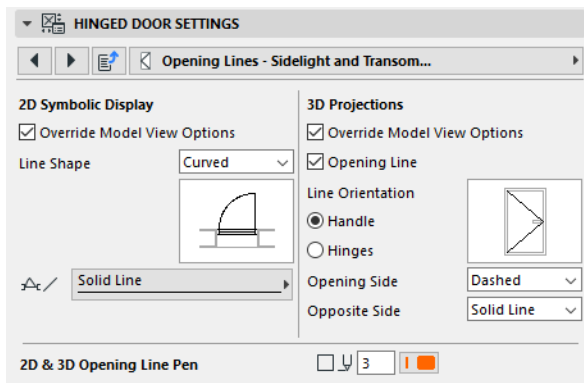
The following is a description of the Opening Lines - Sidelight and Transom tab page of the Custom Settings panel for a “Hinged Door”, with sidelight and transom, from the INT Archicad Library.

Use these controls to set the 3D Projection display options of Opening Lines for the sidelight and transom (if applicable) for each door.

By default, these controls are grayed: Opening Line display is set globally for all doors (including their sidelights and transoms) in Model View Options.

For details on these settings, see [Miscellaneous Settings for Library Parts](#).

However, to set separate Opening Line options for the transom/sidelight of this door only, check the **Override Model View Options** box and set the options as needed.

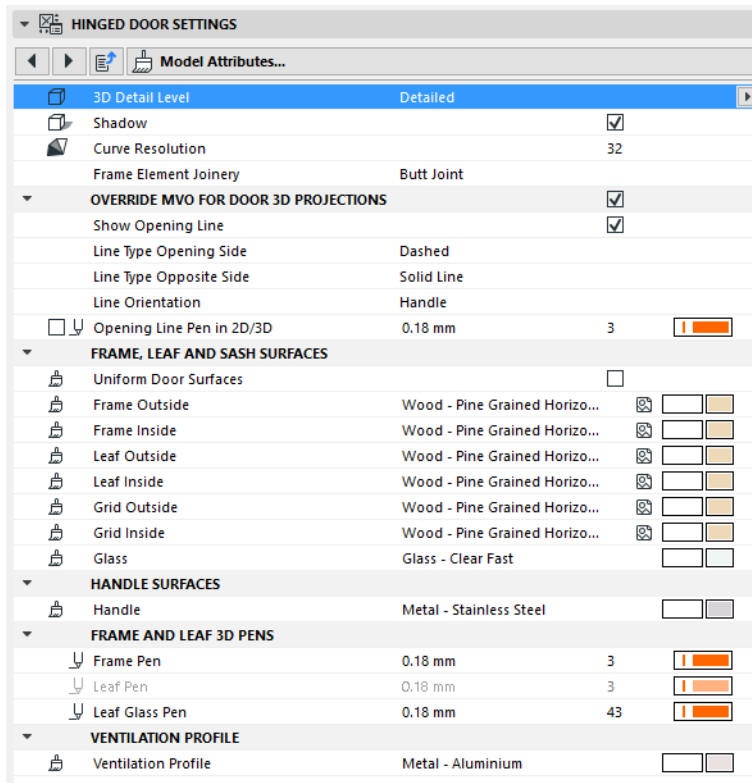


Model Attributes of Door/Window Opening

The following is a description of the Model Attributes tab page (in the “Door Opening” or “Window Opening” section) of the Custom Settings Panel of Door or Window Settings, from the INT Archicad Library.

Use this page to define the opening's 3D appearance. The available controls depend on the components of the particular opening.

In addition to general attributes, you can set specific surface materials and 3D pens for the door leaf, frame, grid and glass components, and for the handle and handle accessory, if any.



3D Detail Level

This is the same setting as on the main Settings and Opening page.

See [Detail Level](#).

Shadow

Turn on to display a shadow effect for this opening in 3D.

Curve Resolution

Applied to the opening's curved components when generating their 3D display. (The circle formed out of the curved component will be divided into the given number of segments.) A higher curve resolution can improve the 3D appearance, but increases file size.

Frame Element Joinery

Choose either Mitered or Butt Joint.

Override MVO for 3D Projections

Check this box to override, for this particular opening, the Model View Options for Opening Lines defined at **Document > Model View > Model View Options > Miscellaneous Settings for Library Parts** (Opening Lines in 3D Projections).

Frame and Leaf (Sash) 3D Pen

These controls affect the parts of the Frame and Leaf/Sash that are displayed

- in the 3D window
- and in 3D-like form on the Floor Plan - that is, the Projected parts, if the opening's Floor Plan Display option is set to Projected or Projected with Overhead.

Wall Opening

The following is a description of the Wall Opening tab page of the Custom Settings Panel of Door or Window Settings, from the INT Archicad Library.

Reveal Type and Depth

From the pop-up list, choose a Reveal Type to use for this opening, or choose one of the No Reveal options. Enter a value for the Reveal Depth.

- You can also choose a Reveal Type and set Reveal Depth on the Reveal tab page.
- Reveal Depth can also be set on the Preview and Positioning Panel.

Wall Closure

This control is applicable if you have placed the opening in a composite wall.

Choose a predefined or custom Closure from the pop-up list. Available closure types vary depending on the type of reveal used for the opening.

- Closure options can be defined in the Wall Closure tab page.

[See *Wall Closure*.](#)

Masonry Arch

Check this to include a Masonry Arch in the opening, and choose the desired type from the drop-down list.

To set additional Masonry Arch parameters, click the icon at right to jump to the Masonry Arch tab page.

Opening Oversizes

Enter values if you want to increase the wall hole cut by the opening, on its left, right, upper and/or lower sides.

Note: Entering an Oversize does not (unlike the Tolerance value) affect the Nominal Size of the window.

Wall Opening Surface

The surfaces of the host Wall can be applied to the opening in two ways.

- If you choose the **Same as Wall Edge** option, Archicad will apply the surface of the wall's edge to the opening's edges.
- If you choose the **Same as Wall Side** option, the external and internal wall surfaces will be applied to the opening's edges, with the reveal depth as the division line between them.

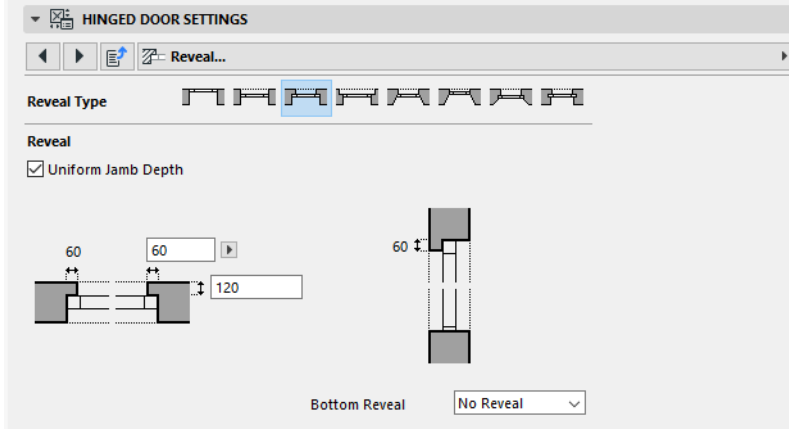
If you are programming a Door/Window in GDL for a curved wall, see also [Hole Surface's Split is Curved](#).

Reveal Settings

The following is a description of the Reveal tab page of the Custom Settings Panel of Door or Window Settings, from the INT Archicad Library.

Use this tab page to choose a **Reveal Type** and to set its parameters, such as Head and Jamb dimensions and Reveal Depth.

- You can also choose a Reveal Type and set Reveal Depth on the Wall Opening tab page.
- Reveal Depth is also set on the Preview and Positioning Panel.



Wall Closure

The following is a description of the Wall Closure tab page of the Custom Settings Panel of Door or Window Settings, from the INT Archicad Library.

Use this tab page to define how the layers of a composite wall will close around the opening that is placed into it. (These settings are relevant only for openings placed in composite walls.)

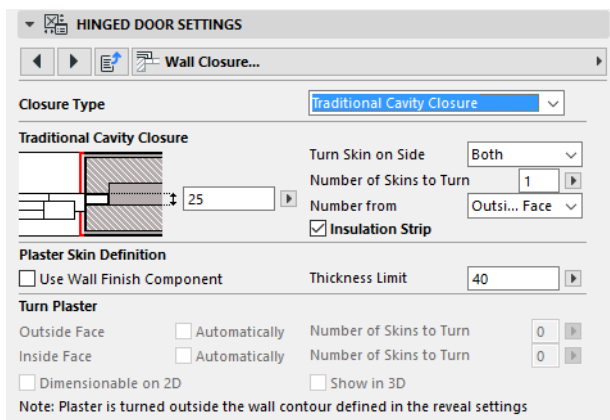
- These settings only affect the display of openings in 2D views.
- The effect of your Turn Plaster settings will be displayed in Section/Elevation views as an extension of the composite wall into the frame. (To see the effect of these settings in the 3D window, you must check the “Show in 3D” box among the Turn Plaster controls.)

[Watch the Video](#)

Wall Closure Tab Page Settings

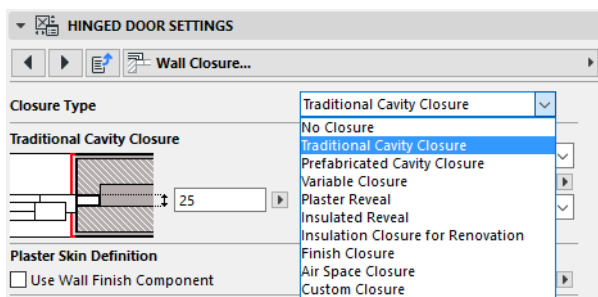
There are two types of settings on this page:

- **Closure Settings** (if you choose to use a Wall Closure); and
- **Turn Plaster** settings, which are independent of whether you are using a Wall Closure type.



Closure Type

Choose a Closure Type from the drop-down menu, or No Closure if you are not using one.



Available closure types can vary depending on the structure of the reveal.

Closure Settings

These options vary depending on the chosen Closure Type.

Turn Skin on Side

Choose whether the wall skins should be turned in on the right, the left or on both sides of the opening.

Number of Skins to Turn

Choose the number, starting from either the Outside face or Inside face of the wall. (Switch this preference using the **Number from** pop-up.)

Profile Width

This parameter is available for the prefabricated cavity closure. Enter a width for the closure element.

If you have chosen Custom Closure: [see *Custom Closure for information on these settings.*](#)

Insulation Strip

Check this box to add a strip of insulation.

Enter a thickness value for this insulation strip.

Turn Plaster

Use these settings if you want the wall's plaster to turn in at the end of wall - irrespective of whether you are using a Wall Closure.

Notes:

- The turned plaster skin will be turned outside of the wall contour, as defined in Reveal Settings.
- You cannot use both Casing and Turn Plaster options at the same time. If Turn Plaster settings are grey, go to the Casing pages of the Settings panel and turn off Inside and Outside Casing. ([See *Casing Outside/Inside Settings.*](#))

Plaster Skin Definition

First, define which wall skin should be considered the Plaster Skin:

- **Use Wall Finish Component:** the component of the composite wall that is defined as “Finish” will be considered plaster.

Alternatively:

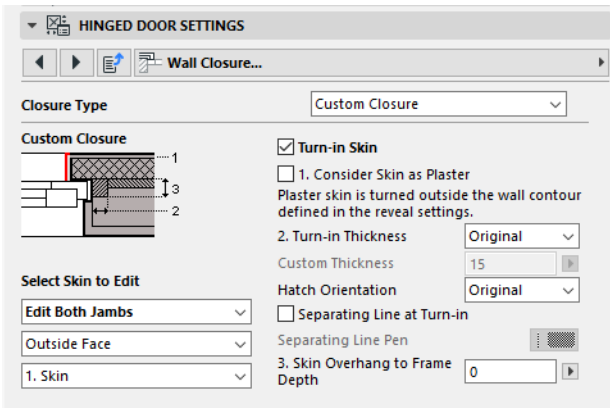
- **Plaster Thickness Limit:** Define the Plaster skin by a given thickness - e.g., all skins up to 40 mm thick will be considered plaster.

Once you have defined which wall skins are considered plaster, use the controls under Turn Plaster to define which of the plaster skins should be turned in:

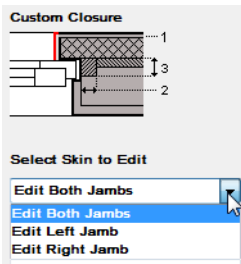
- either **Automatically** (*all* skins defined as plaster under the Plaster Skin Definition above will turn), or
- the specified **Number of Skins to Turn**.

Custom Closure

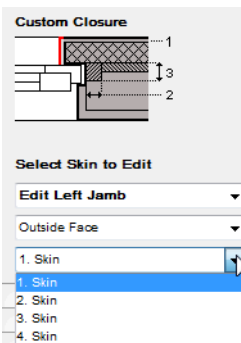
If you want to use a custom closure type, you will set the closure preferences separately for each skin.



First, under **Select Skin to Edit**, choose whether you want the closure on the left jamb, right jamb or both.



Next, select the skin you want to turn in. (Use the pop-up to number the skins starting from either the wall's Outside Face side or the Inside Face.)



On the right side, check **Turn-in Skin** if you want the currently edited skin to be turned in (wrapped); then set the rest of the options (thickness, hatch orientation, separating line pen) for the selected skin.

Check **Consider Skin as Plaster** if you want the skin to behave as plaster: that is, it will turn in along the wall opening as defined by the reveal.

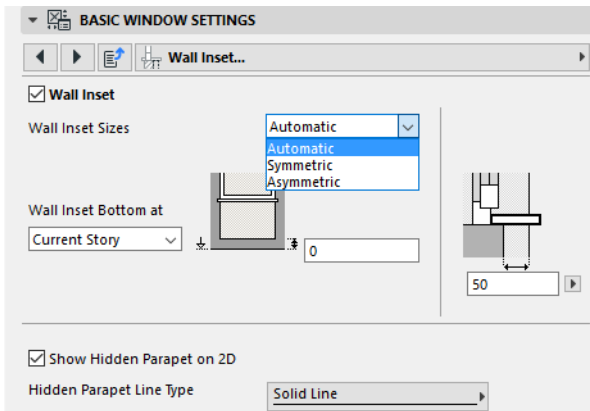
Note: If the wall skin is defined as “Core”, it cannot be considered plaster, and this checkbox will have no effect.

See [How to Define Components as “Core” or “Finish”](#).

Wall Inset (for Windows)

The following is a description of the Wall Inset page of the Custom Settings panel for a Basic Window, from the INT Archicad Library.

Use this page to define a wall inset below the window (e.g. to make room for a radiator).

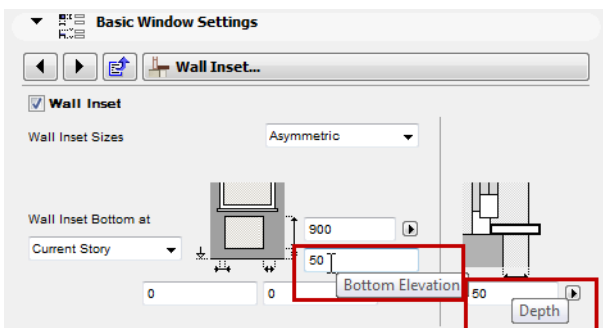


Choose a Wall Inset Size:

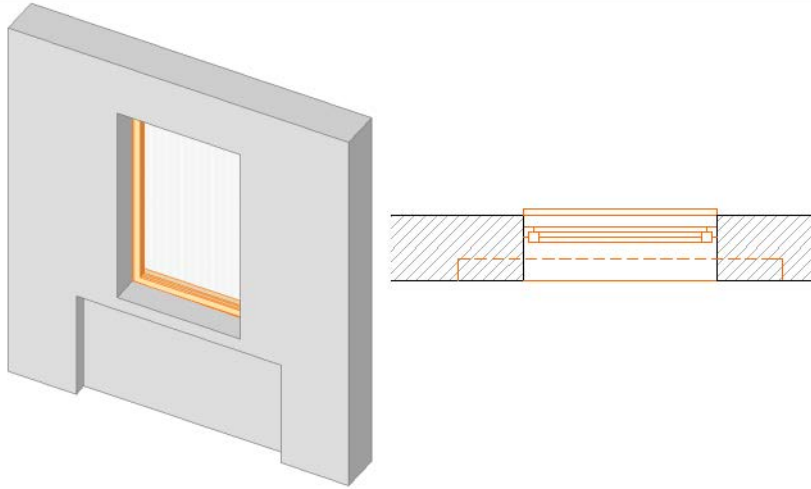
- **Automatic:** The width of the inset is always equal to the window width.
- **Symmetric:** Define a fixed width and a fixed height for the inset, which will be created symmetrically to the window opening.
- **Asymmetric:** Define different overhangs (from right and from left) for the horizontal location of the inset. Define a single fixed height for the inset.

For all inset sizes:

- Define the **bottom elevation** of the inset, or leave it 0 to extend it to the floor.
- Define the **depth** of the inset.

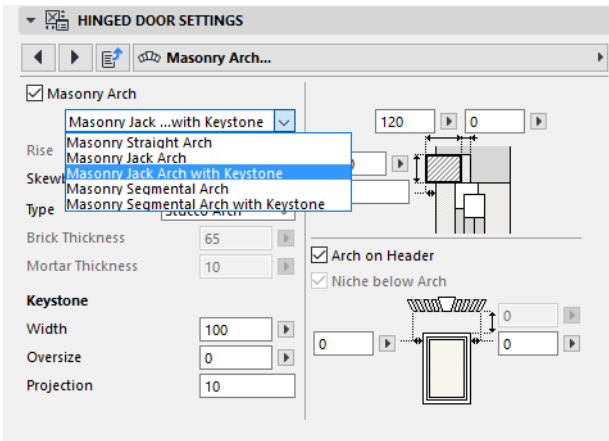


Show Hidden Parapet on 2D: Check this box to display, in 2D views, the parapet line created by the wall inset even if its cut view is blocked by the wall and/or window opening.



Masonry Arch

The following is a description of the Masonry Arch tab page of the Custom Settings Panel of Door or Window Settings, from the INT Archicad Library.



Masonry Arch: Check this box, then choose a style for the arch from the drop-down menu. The rest of the controls vary depending on the arch you choose here.

Rise: The value of the arch's rise.

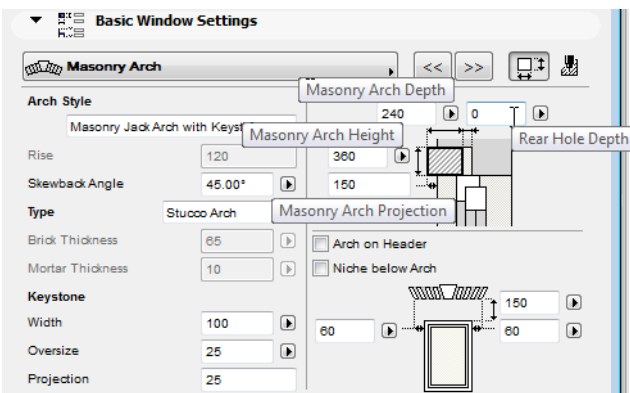
Skewback angle: For "Jack" arch styles.

Type: Choose Stucco or Brick. If you choose Brick, you can set values for the Brick and Mortar thicknesses below.

Note: Using brick arches in a large number of windows may increase your model size significantly.

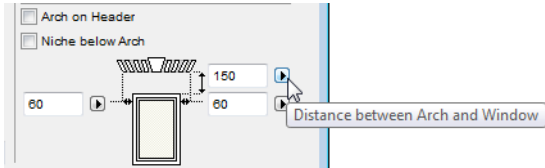
Keystone: For arch styles with a keystone, set values for its width, oversize (if the keystone extends vertically above the arch) and projection (if it protrudes horizontally from arch).

Using the fields on the right side of the dialog box, set the height and depth values for the arch; the depth of any hole behind it ("Rear Hole Depth"); and the arch projection horizontally, if any.

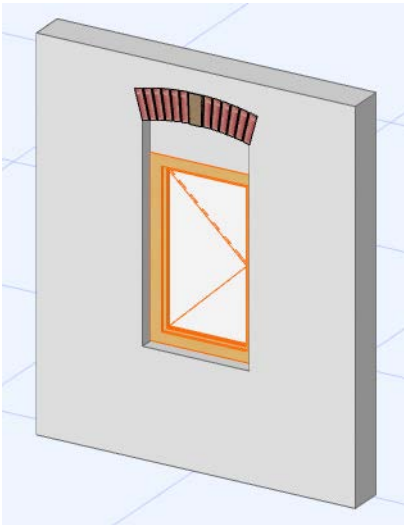


Arch on Header: Check this box to place the arch directly atop the opening header.

If it is not checked, then you can use the field below to define the distance between the arch and the header.



Niche below Arch: Check this box to create a niche (or inset) beneath the arch.



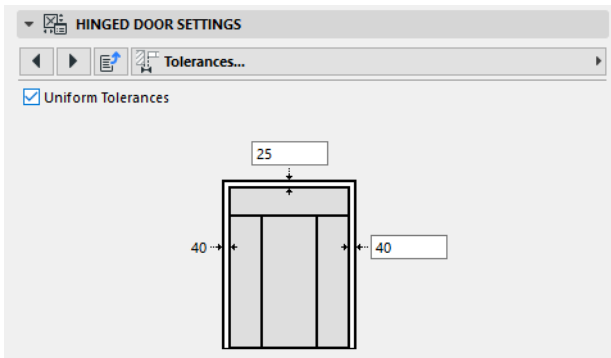
Horizontal Overhang: Use the two fields on either side of the drawing to define the arch overhang on the right and left side of the opening.

Tolerances

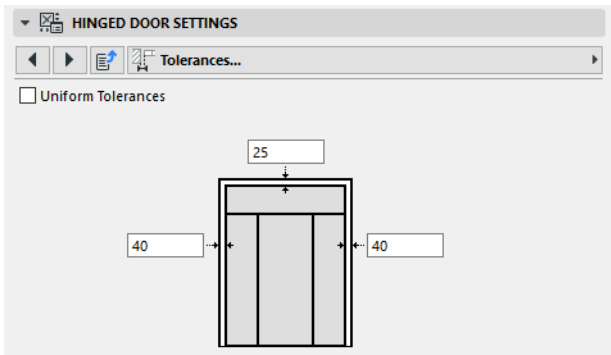
The following is a description of the Tolerances tab page of the Custom Settings Panel of Door or Window Settings, from the INT Archicad Library.

Use this tab page to set the side and upper tolerance values for the opening structure.

If **Uniform Tolerances** is checked (either on this tab page, or on the Nominal Size and Tolerances tab page), then you set a single tolerance value for all sides of the structure, and another value as the Upper Tolerance.



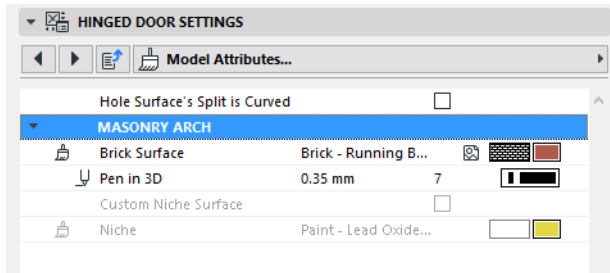
If you uncheck Uniform Tolerances, you can set varying tolerance values for the sides, (as for a complex-shaped opening whose sidelights have varying sill heights).



Model Attributes of Wall Opening

This is the Model Attributes tab (in the Wall Opening Section) of the Custom Settings Panel of Door or Window Settings, from the INT Archicad Library.

Use this page to define the 3D attributes of the opening's masonry arch, if any.



Hole Surface's Split is Curved

This control may be relevant when creating an opening in GDL and setting its defaults.

In case of a curved wall whose wall opening displays the Wall Side surfaces ([see Wall Opening Surface](#)), the two surfaces - where they meet in the wall hole - can be split either along a straight line or a curve.

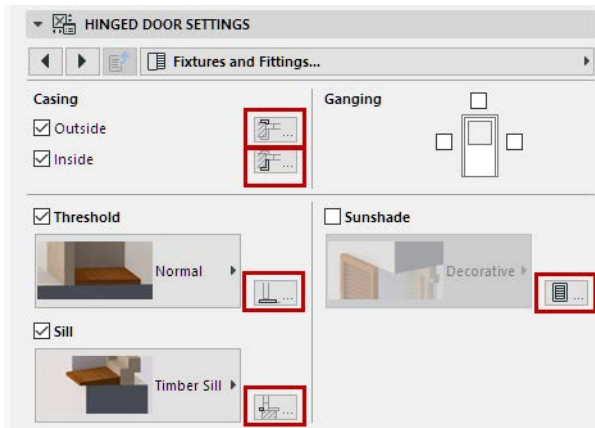
Check **Hole Surface's Split is Curved** to split the surfaces along an arc.

Fixtures and Fittings

The following is a description of the Fixtures and Fittings tab page of the Custom Settings Panel of Door or Window Settings, from the INT Archicad Library.

Use this tab page to define the main fixtures of the opening.

To set additional, more detailed parameters for any of these structures (e.g. Casing, Sunshade), click the icon at the right of the setting to access the respective tab page, where you will find additional parameters.



Casing

Use the **Outside** and **Inside** checkboxes to add Casing to the outer and/or inner side of the opening.

[See Casing Outside/Inside Settings.](#)

Ganging

You can align adjacent openings horizontally and vertically with these parameters. You must check these boxes both for this opening and for the adjacent one so that the opening symbol is displayed correctly.

As a result of ganging, the connecting lines between adjacent opening components (e.g. casing, sill, threshold) disappear - as if they were a single unit.

If you use ganging, some other structure options for this opening may no longer be applicable and will be grayed or eliminated.

Threshold/Sill/Board/Sunshade

Check this box, then use the pop-up to choose the desired component for the opening as needed. To set more detailed parameters, click the icon at the right of the setting to access the respective tab page.

[See Threshold Settings.](#)

[See Sill Settings.](#)

[See Board Settings.](#)

[See Sunshade Settings.](#)

Custom Corner (For Windows)

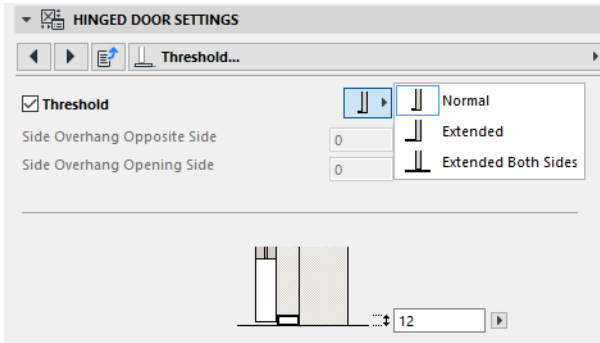
Check "Right" or "Left" to add a custom corner to the window.

To set more detailed parameters, click the icon at the right of the setting to access the Custom Corner tab page.

[See Custom Corner \(for Windows\).](#)

Threshold Settings

The following is a description of the Threshold tab page of the Custom Settings panel for a “Hinged Door”, with sidelight and transom, from the INT Archicad Library.



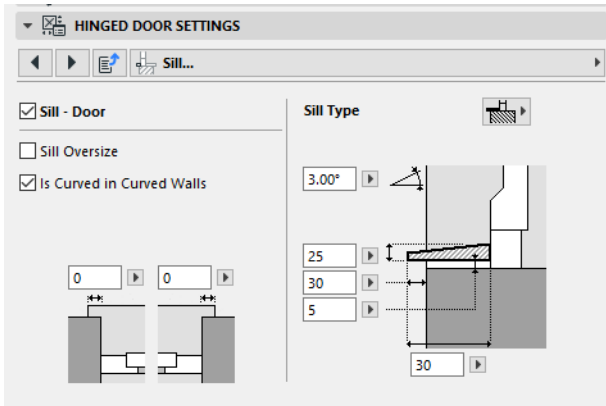
Use the **Threshold Type** pop-up to choose a threshold.

Threshold Settings include the Threshold Thickness and - for Extended threshold types - the nosing and side overhangs on either side of the door.

Sill Settings

The following is a description of the Sill Settings of the Custom Settings Panel of Door or Window Settings, from the INT Archicad Library.

Check the Sill box to assign a sill.



Use the **Sill Type** pop-up to choose a sill type. Depending on the type you choose, the settings will vary.

Choose either Simple or Detailed for the Sill's 3D Detail level.

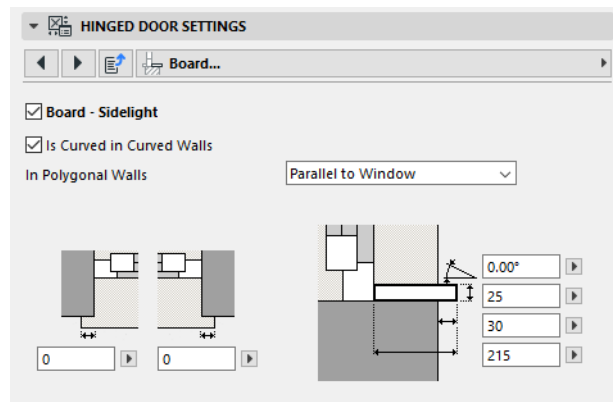
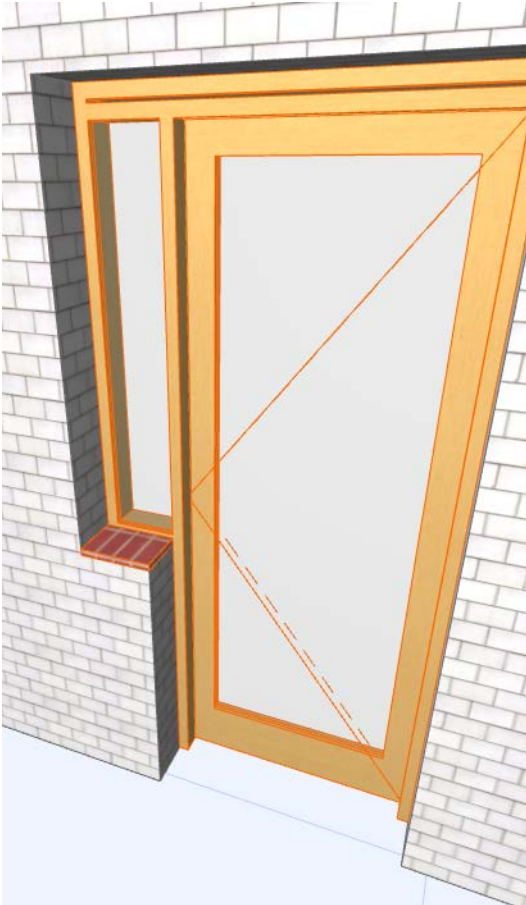
Sill Settings include thickness, width, angle, and nosing, as well as other settings for particular sill types.

Slip in Depth: The distance by which the sill extends beneath the frame (its maximum value is equal to the frame thickness). This value is added to the Sill Width.

Board Settings

The following is a description of the Board Settings tab page of the Custom Settings Panel of Door or Window Settings, from the INT Archicad Library.

Use this page to define a board if you are using an opening with one or more raised sidelights.



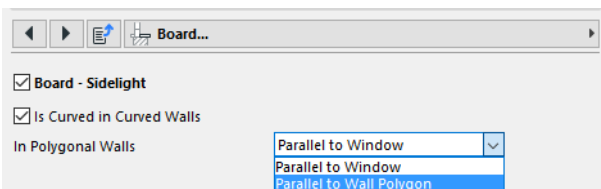
Check **Board** to assign one to the sidelight(s).

Board in Curved Wall

You can opt to curve the board itself if the window is placed in a curved wall: check the box “Is Curved in Curved Walls.”

Board in Polygonal Wall

If the opening is in a polygonal wall, set the board edge to either follow the wall polygon (Parallel to Wall Polygon) or to stay parallel with the sidelight window.



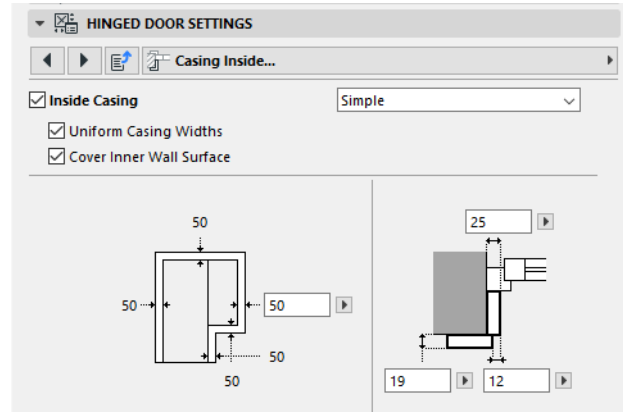
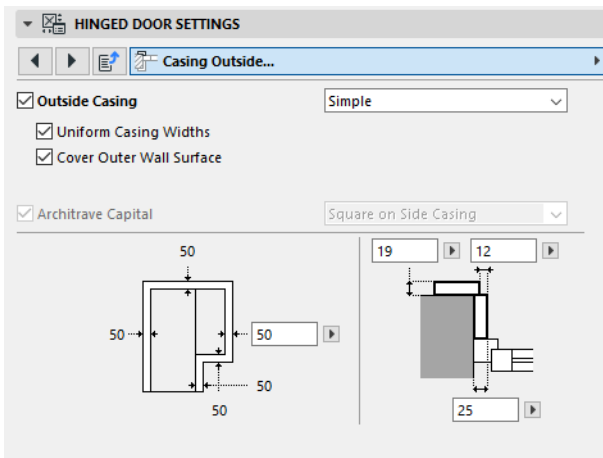
Casing Outside/Inside Settings

The following is a description of the Casing Outside and Casing Inside tab pages of the Custom Settings Panel of Door or Window Settings, from the INT Archicad Library.

Note: You cannot use both Casing and Turn Plaster options at the same time.

See [Turn Plaster](#).

Check **Outside Casing** and/or **Inside Casing** to assign it and to set the casing's parameters.

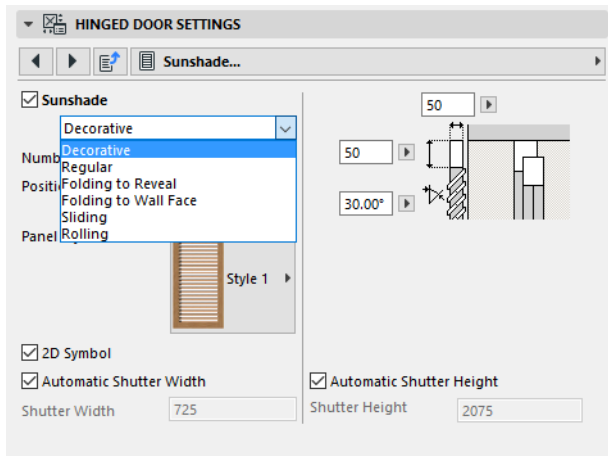


Sunshade Settings

The following is a description of the Sunshade tab page of the Custom Settings Panel of Door or Window Settings, from the INT Archicad Library.

Sunshades are not available if the opening also has a raised sidelight.

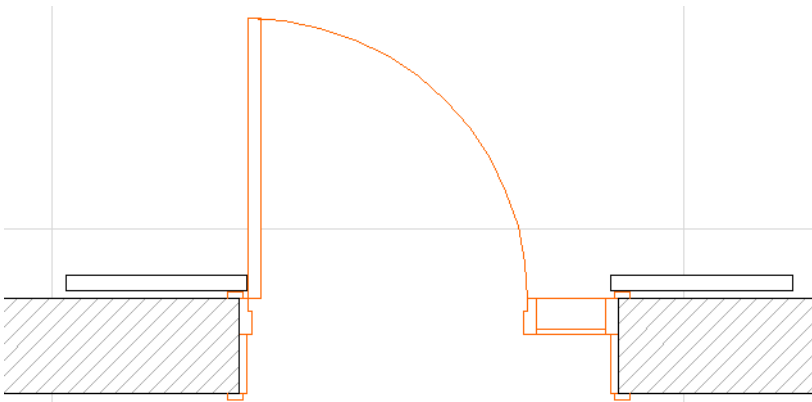
Check the Sunshade box to assign one, then choose a type from the pop-up. The rest of the controls vary depending on the sunshade type.



Open in 3D: (For shutters that open.) Display the shutters in open position in the 3D window.

3D Opening (Angle): For movable shutters, enter how far to open it in 3D views. (You can also edit the shutter opening graphically in the 3D window, using editable hotspots.)

2D Symbol: Check this to display the shutter symbolically in the 2D window.



Extra Frame: Check this box to display (in the 3D window) the sunshade's frame adjacent to the opening.

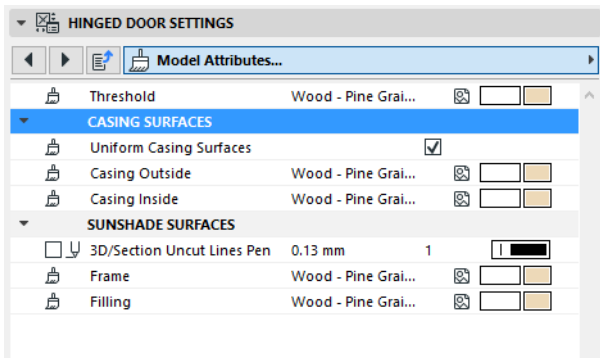
Shutter Height: Enter a value for the shutter height, or click **Automatic Shutter Height** to make it the same height as the opening.

Model Attributes of Fixtures and Fittings

This is the Model Attributes tab page (in the Fixtures and Fittings section) of the Custom Settings panel for a Door or Window from the INT Archicad Library.

The available controls depend on the fixtures assigned to the particular Opening.

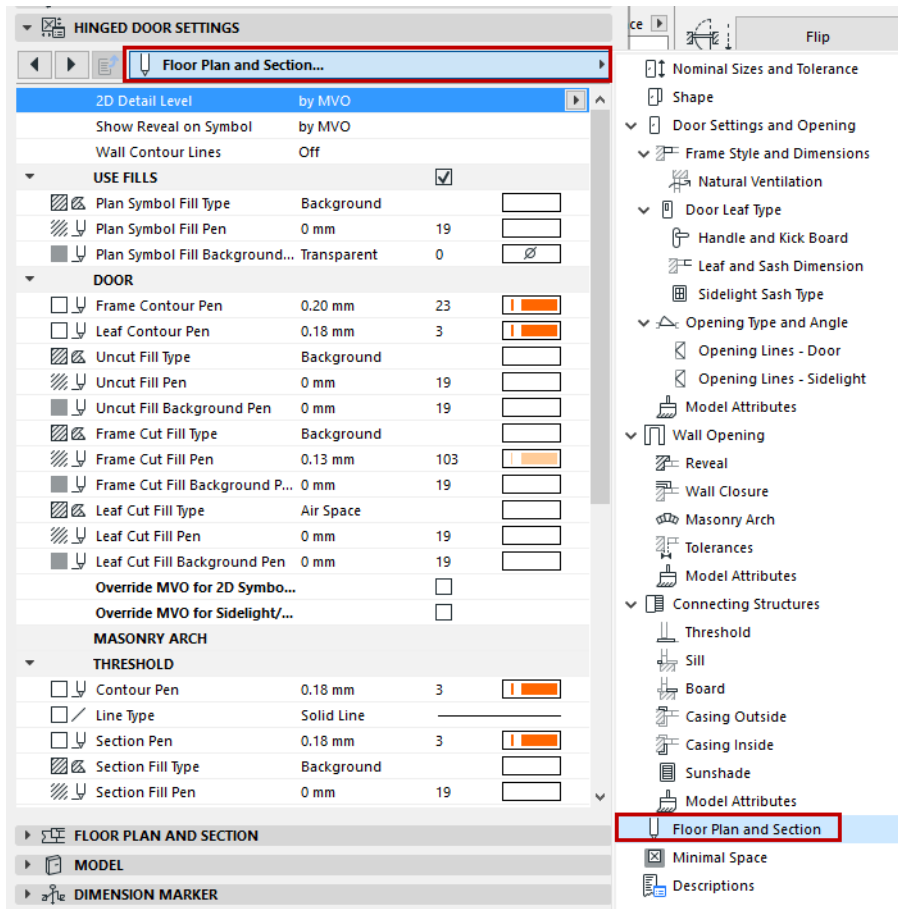
Set specific surface materials and 3D pens, as applicable, to the Threshold, Sill, Casing, Board and/or Sunshade.



Floor Plan and Section Tab Page (Door/Window Settings)

The following is a description of the Floor Plan and Section tab page of the Custom Settings Panel of Door or Window Settings, from the INT Archicad Library.

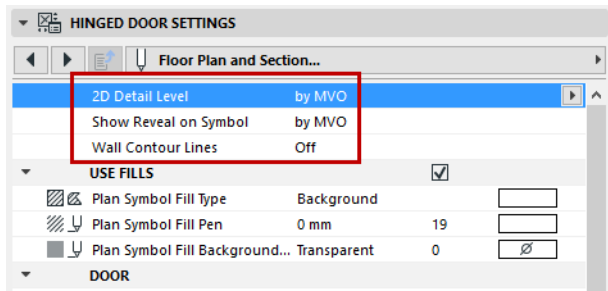
This tab page contains all of the 2D display parameters for the various parts of the opening, as defined for this GDL object in its user interface script. The available parameters vary depending on the opening's components and their geometry.



Note: Alternatively, you can use the 2D attributes on the **Floor Plan and Section** panel of Door/Window Settings. This method involves a reduced number of parameters for simplicity. [See *Override GDL Object Attributes of Doors/Windows*, below.](#)

While setting the parameters, it is worth displaying the Preview from Door/Window Settings to see how your definitions will affect the display.

Note that some display parameters are “by MVO”, that is, corresponding to the options set in Model View Options (**Document > Model View > Model View Options**), but you can override some of these attributes for the particular opening being edited.



The first three parameters apply to the opening as a whole.

2D Detail Level

This is the same setting as on the main Settings and Opening page.

See [Detail Level](#).

Show Reveal on Symbol

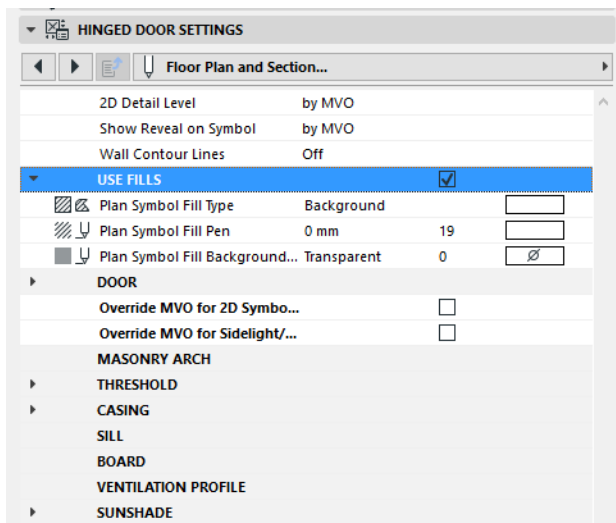
Choose the scale at which the reveal should be displayed on the Floor Plan. (Reveal will not be displayed if the scale is larger than the chosen value.)

Wall Contour Lines

Set the Wall Contour lines for the opening display. If you turn on any of the contour lines, you can set separate lines for hidden/visible lines.

Use Fills on Doors/Windows

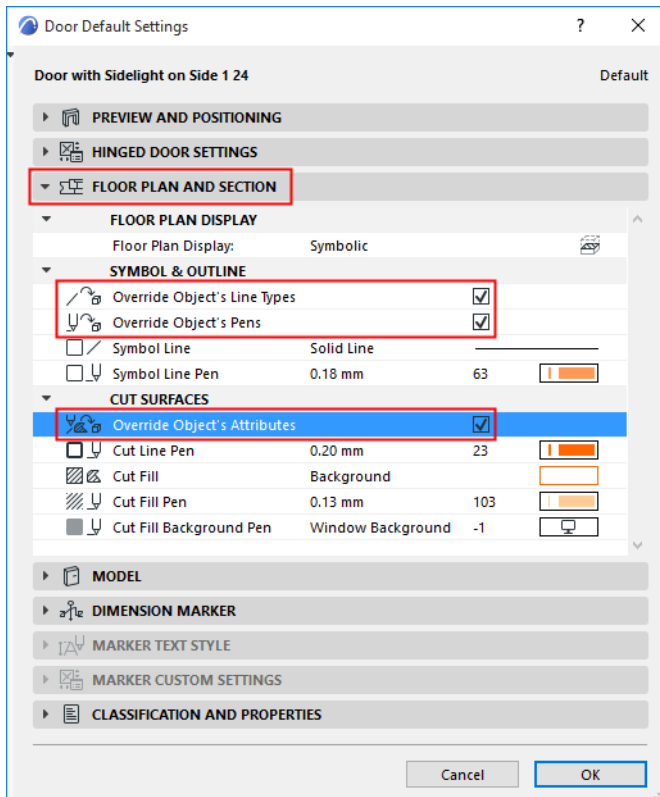
To apply any fills to the door/window, check the **Use Fills** box to enable all fill parameters. If **Use Fills** is off, the opening display will not use any fills on any of its components.



Override GDL Object Attributes of Doors/Windows

Typically, you will define the 2D attributes (line type, pen, fill) for each component of the Window/Door on the Floor Plan using the Floor Plan and Section tab page of the Custom Settings panel, described above.

However, you may wish to override these, using the simplified settings of the Floor Plan and Section Panel of Door/Window Settings.



Set the attributes for cut, uncut and overhead lines and pens as needed on the Floor Plan and Section panel of Door/Window Settings (or the Info Box).

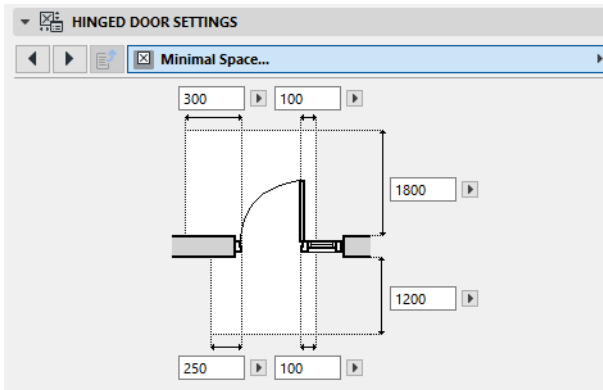
For these settings to take effect, make sure that the applicable **Override Object's** /Line Types/Pens/Attributes checkbox(es) are *on*. Otherwise, the Door/Window will be displayed with its GDL Object attributes.

Note: For some doors/windows, no relevant GDL parameter is defined. In this case, the object will be displayed using the attributes defined in the Floor Plan and Section panel (regardless of the Override checkboxes).

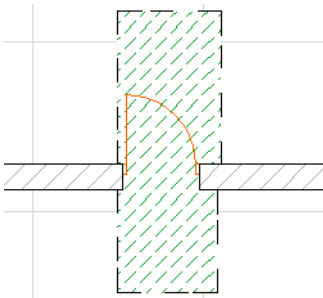
Minimal Space

The following is a description of the Minimal Space tab page of the Custom Settings panel for a “Hinged Door”, with sidelight and transom, from the INT Archicad Library.

Use these controls to define the minimal space requirements for clearance on either side of this door.

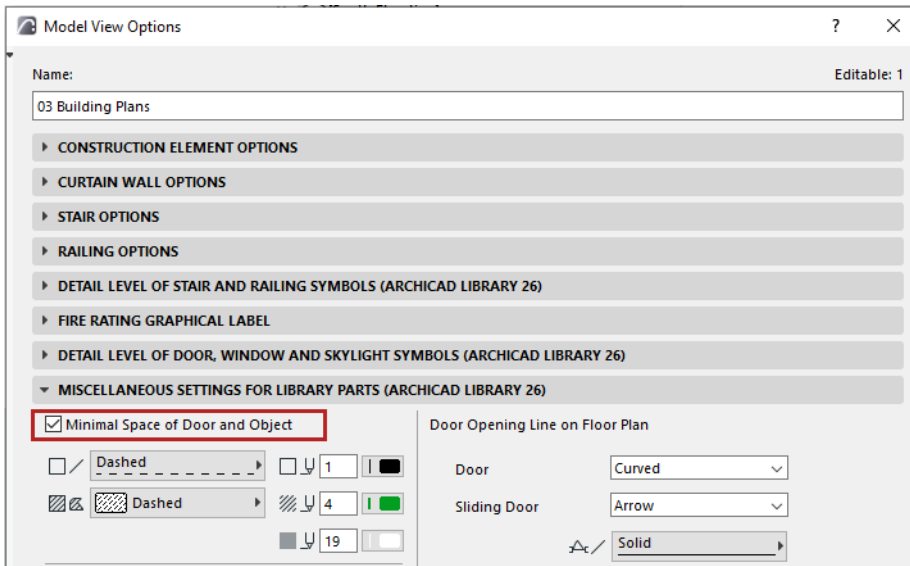


As a drafting aid, you can then turn on a Model View Option to display the minimal space clearances (for both doors and objects) on the Floor Plan.



To view Minimal Space limits on the Floor Plan for *all* doors and objects in the project which have a defined minimal space parameter:

1. Go to **Document > Model View > Model View Options**
2. Open the **Miscellaneous Settings for Library Parts** panel.
3. Check the **Minimal Space of Door and Object** box.



Use the controls below the checkbox to define pens, line type and fills for the display of minimal space.
 For general information on the Custom Settings panel, see [Where to Set Door/Window Parameters](#).

Custom Corner (for Windows)

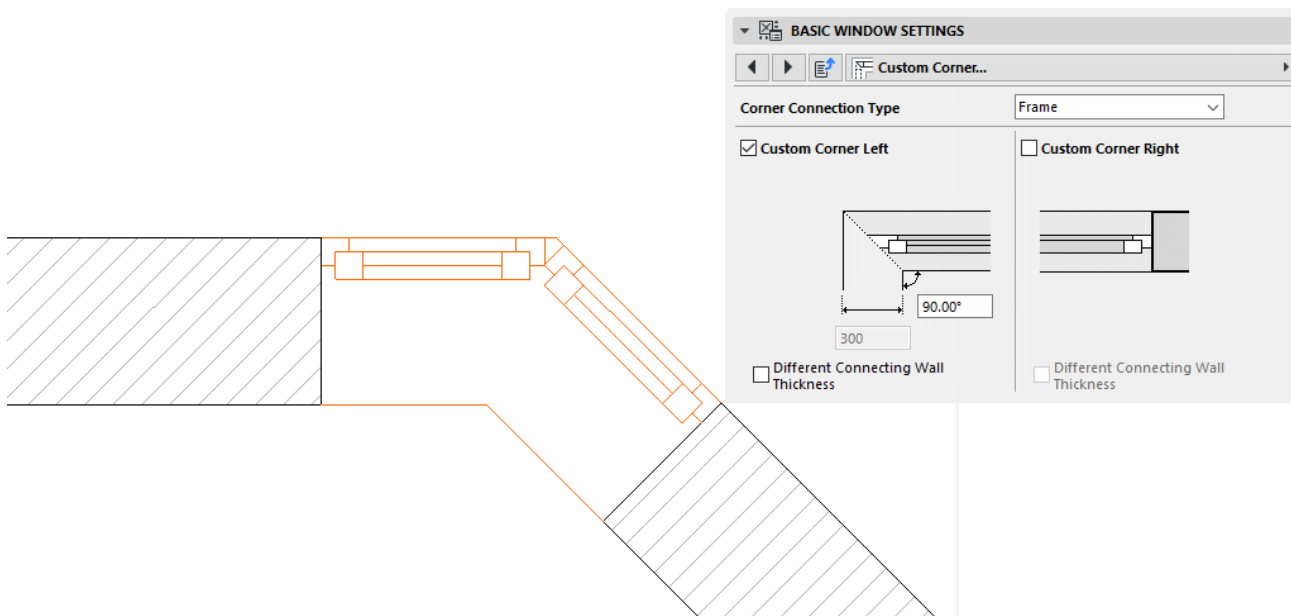
The following is a description of the Custom Corner page of the Custom Settings panel for a Basic Window, from the INT Archicad Library.

Use this tab page to define how two windows which meet at a corner should connect to each other. Set the options for both connecting windows.

The corner connection type can be one of three kinds:

- Frame: join the Frames directly to each other;
- Corner Column: insert a column element which each window frame will abut
- Glass: insert a glass element between the frames.

Set the angle at which the two walls meet.



If the two walls that form the corner are of differing thicknesses, check **Different Connecting Wall Thickness** so you can enter the thickness values for each wall.

Extra Topics

Functions in this section may not be available in all markets.

Solibri Connection

Solibri is a leading model checking software used for BIM validation, compliance control, design process coordination, review, and analysis. With the Archicad-Solibri Connection add-on, you can connect your Archicad model directly to Solibri for fast and automated code- and constructability checking.

Solibri Connection automatically detects and sends only those elements that are changed in the Archicad model, resulting in faster round-trip collaboration.

The workflow supports Issue reporting based on BCF 2.1, enabling designers to assign the detected issues or solve them later on.

This add-on allows fast BIM updates and the use of Solibri while working in a real time design scenario.

No longer do you need to save files and upload back and forth between programs. Simply design, check and amend as necessary.

Key Features and Benefits of Solibri Connection

- IFC-based data exchange:
 - Archicad Classifications
 - Archicad Properties
 - IFC Type Products
 - Colors
 - Space boundaries
 - Junctions, priority-based connections
 - Hierarchical elements
- Fast synchronization between the two applications
- Ability to manage the exported content
- Selection of elements in Archicad based on the selection in Solibri
- Selection of elements in Solibri based on the selection in Archicad
- Connection between the applications can be disconnected and re-established
- BCF-based issue import into Archicad

Related Topics

[Install Solibri Connection Add-on](#)

[Connect to Solibri](#)

[Export Model to Solibri](#)

[Select Archicad Elements in Solibri](#)

[Get Selection Basket](#)

[Import Issues from Solibri](#)

[Break Solibri File Link](#)

Note on Archicad Design Checker

Design Checker is also available for all users as of Archicad 27. Design Checker uses a cloud service called Solibri Inside for integrated design quality assurance.

[See *Archicad Design Checker*.](#)

Install Solibri Connection Add-on

1. Go to the Downloads page of your local Graphisoft website.
2. Go to the Add-ons page and download your language version of Solibri Connection.

See [graphisoft.com/downloads/addons](https://www.graphisoft.com/downloads/addons).

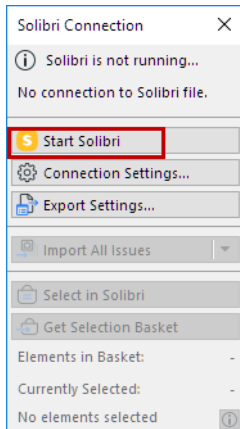
Before installing the Add-on, make sure that the relevant Archicad version (Archicad 24 build 4000 or newer) is installed on your computer.

Please be aware that Solibri 9.10.3.5 or newer is required and Solibri Anywhere version is not supported.

When installing Solibri Connection Add-on, you must be logged on as a member of the Administrator Group.

Connect to Solibri

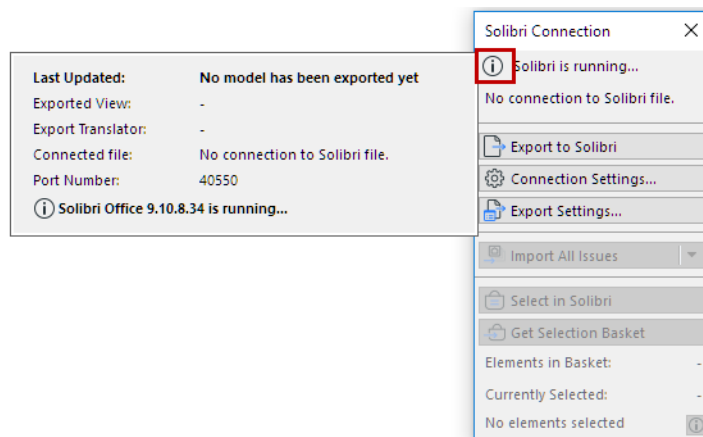
1. Open the **Solibri Connection** palette from
 - File > Interoperability > Solibri Connection or
 - Window > Palettes
2. Click **Start Solibri**.



Please note: Solibri must be started from Archicad to enable the communication between the two applications.

Connection Status

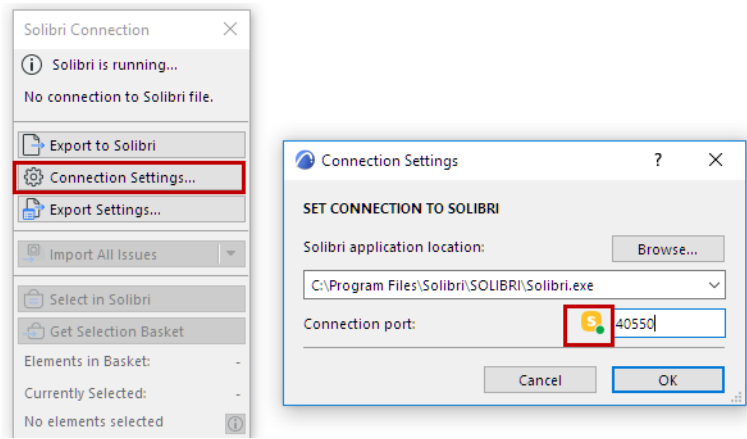
Check the top of the palette for the current status. If needed, click the info button for further details.



Connection Settings

If there is a problem with the connection, click **Connection Settings**:

- Check if Archicad has found the location of Solibri.app. If not, click **Browse** and select your Solibri application.
- The communication between the two application requires a free Connection port. The green dot indicates that the port is free and ready to use.
If it is red, enter a different port number.

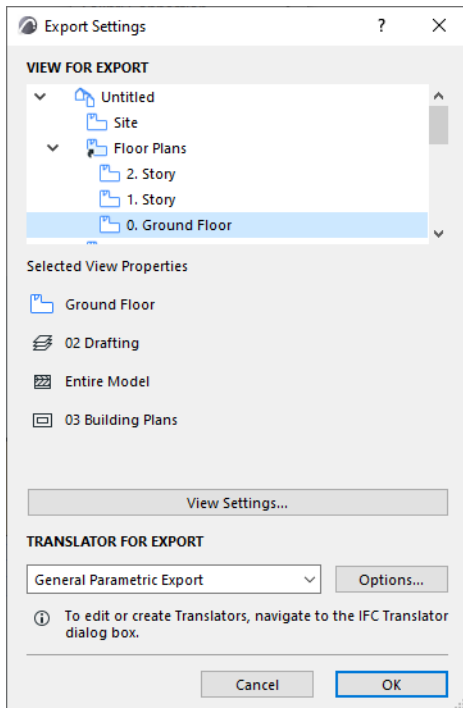


Export Model to Solibri

1. In the Solibri Connection palette: Click the **Export Settings** button.
2. Select the view for export.

The selected view can either be 2D or 3D. In case of floor plan view, the *visible* elements on the selected story will be exported.

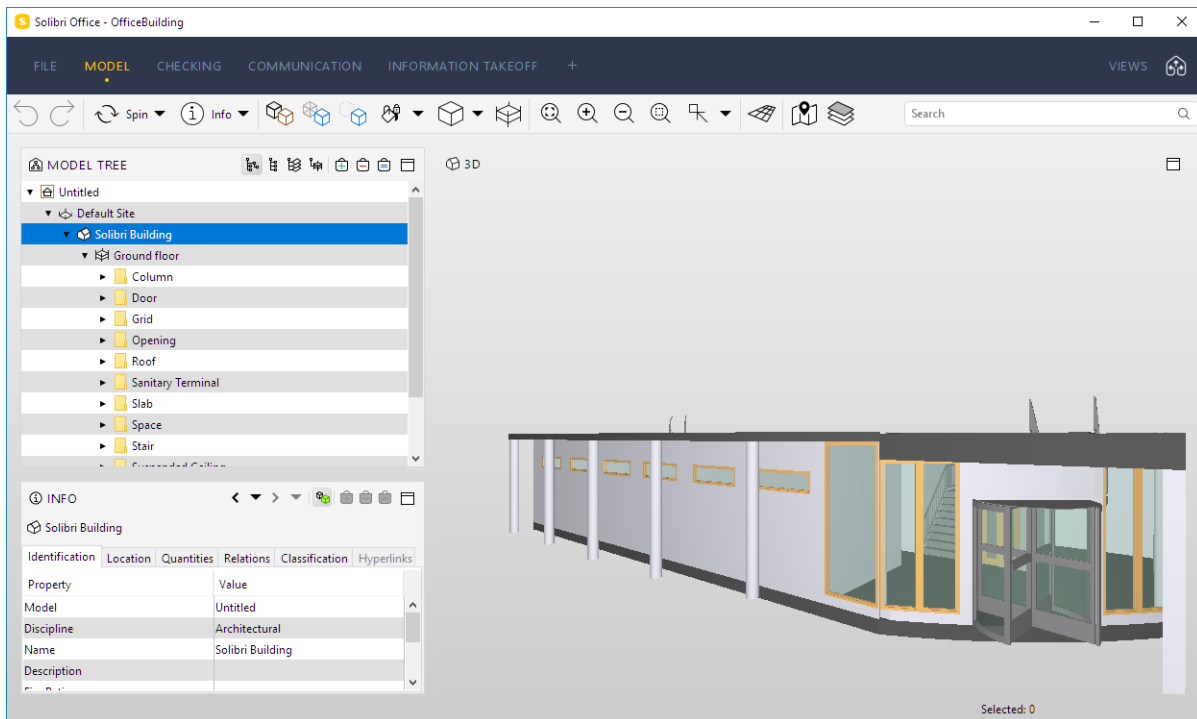
Note: Even if you select a Structural Analytical model view, all of the view's physical elements (not the Structural Members) will be exported to Solibri.



3. Select the IFC translator for the Export. (Click **Options** to see a summary of the IFC Translator's settings.)
Translators can be managed from **File > Interoperability > IFC > IFC Translators**.
4. After you have selected the view and the translator: click **OK**.
5. In the Solibri Connection palette: Click **Export to Solibri**.



6. The model is exported.

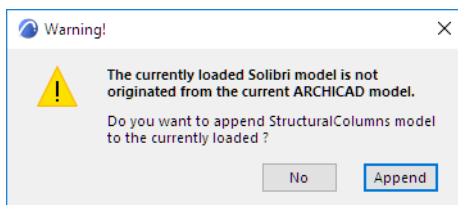


7. Make sure to save the model in Solibri: the next time you open it, the Solibri model can resume its connection to the Archicad model.

Export multiple models into one Solibri

Users can connect multiple Archicad instances with one Solibri. Once the connection between the Archicad and Solibri is established, the second Archicad can be connected as well.

1. Launch the second Archicad.
2. From the Solibri Connection palette, click on **Export to Solibri**.
3. A Warning appears: another Archicad model is already connected to the currently loaded Solibri model.



4. To add a second Archicad model: Click **Append**.

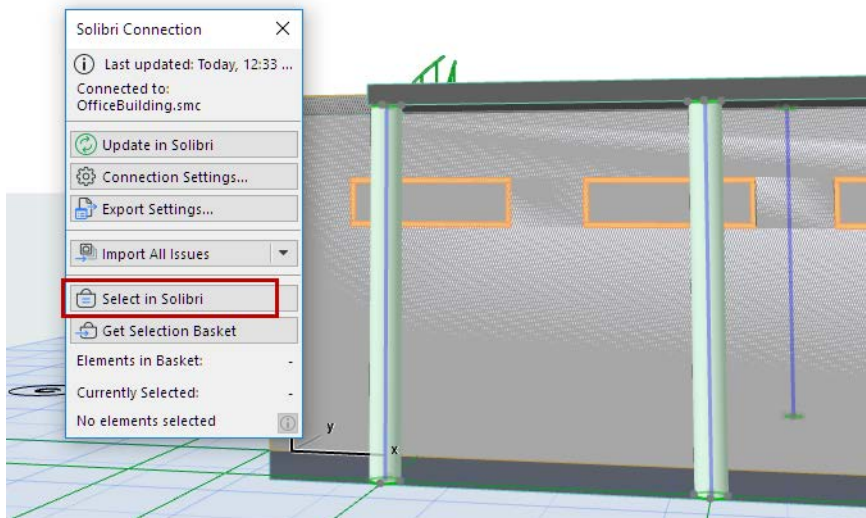
Note: In this situation, Connection Settings indicates that Solibri is using the port to connect to another Archicad. The same connection is used for the second Archicad model, too.

This model will appear in Solibri next to the first one.

Select Archicad Elements in Solibri

Selected elements in Archicad can be selected in Solibri as well.

1. Select elements in Archicad.
2. Click the **Select in Solibri** command on Solibri Connection palette.

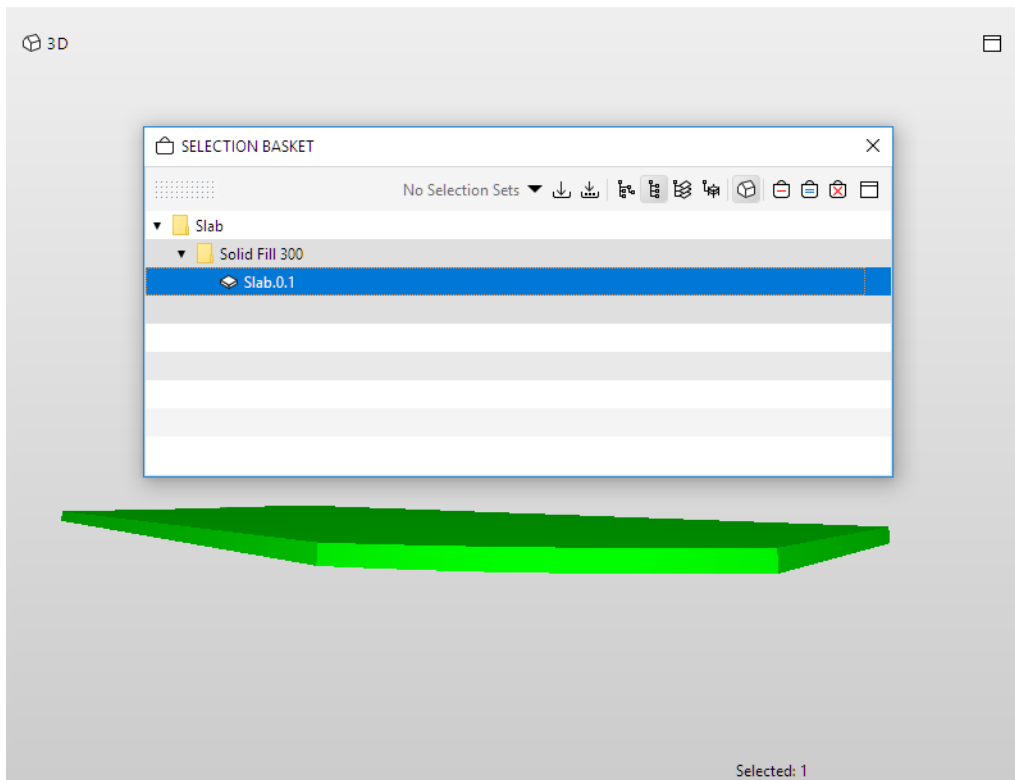


Solibri's Selection Basket is filled with the selected elements from Archicad.

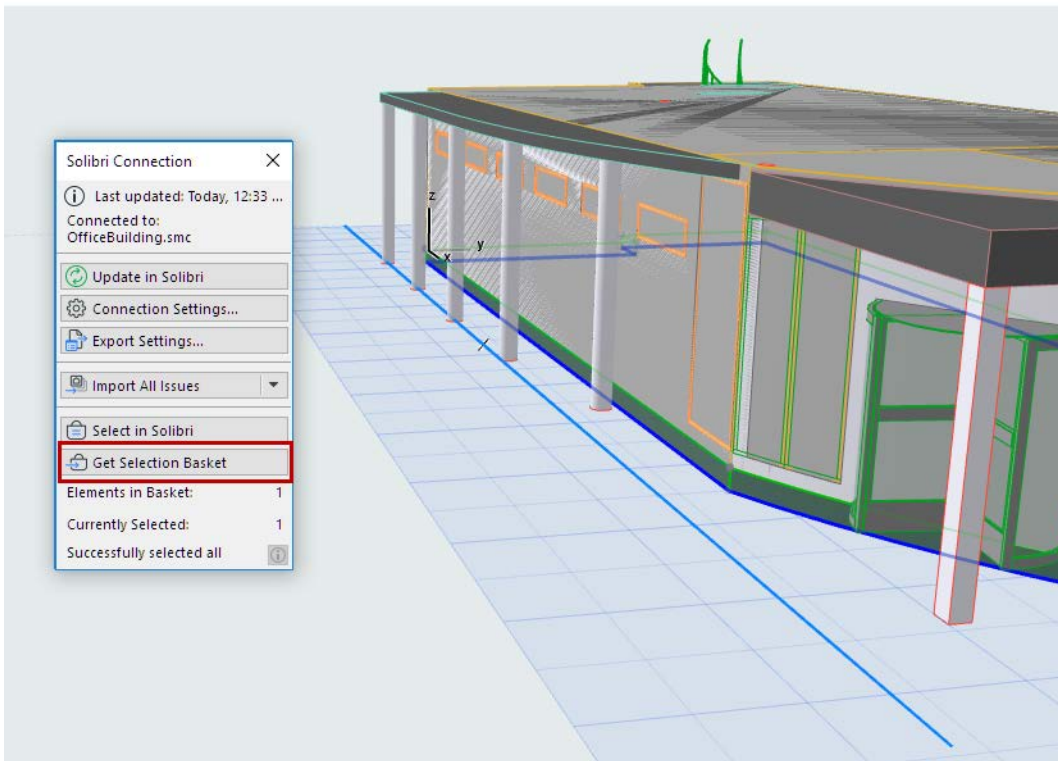
Get Selection Basket

When you find components that need to be fixed in Solibri, use **Get Selection Basket** to easily select them in Archicad.

1. In Solibri, add elements to the Selection Basket.



2. In Archicad, click **Get Selection Basket**.



All components in the Selection Basket will be selected in Archicad, provided that they are visible in the current view.

Note: You can also opt to include the Camera position of the view. (With Solibri versions 9.12.1 and up only.)

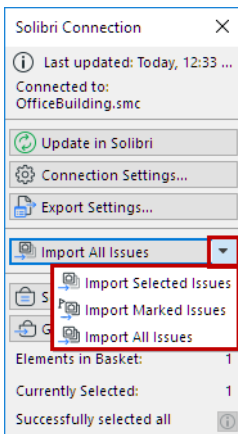
The Solibri palette gives you feedback on

- the number of elements in the Selection Basket
- the number of elements currently selected in Archicad
- whether any elements are not visible in Archicad, and therefore cannot be selected.

Import Issues from Solibri

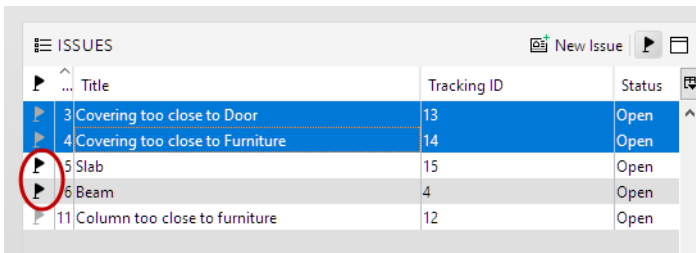
Issues created in Solibri can be imported to Archicad's Issue Organizer, where you can handle them as needed.

You can import all issues, or limit the import to issues that are either selected or marked in Solibri.



Click one of the following commands from the pop-up:

- Import Selected Issues
- Import Marked Issues
- Import All Issues



“Selected” and “Marked” (flagged) Issues in Solibri

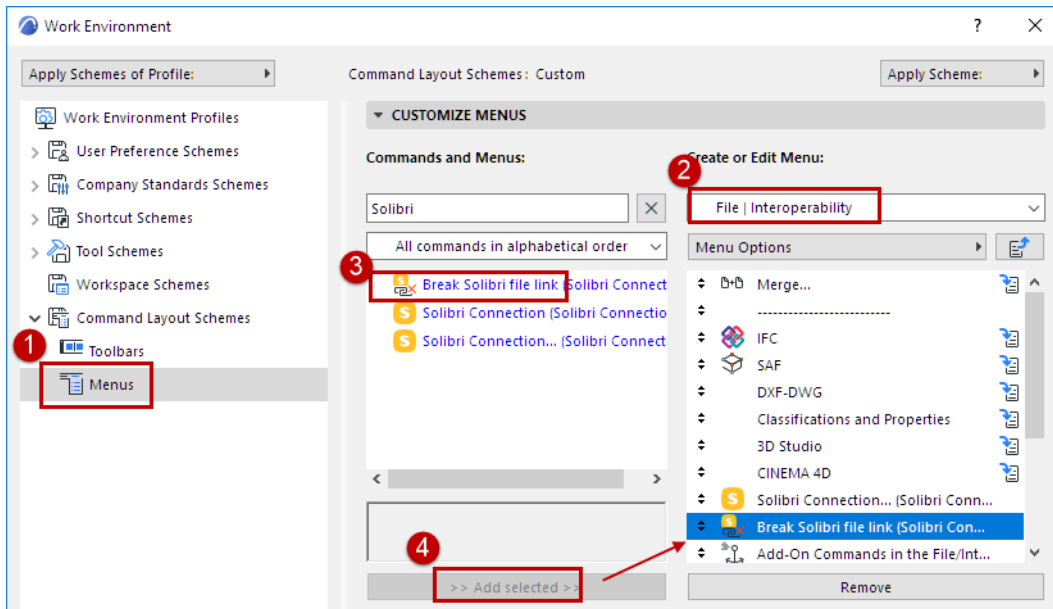
The imported Issues are now added to Archicad's Issue Organizer.

Break Solibri File Link

This is an extra function that breaks the current connection between Archicad and Solibri.

This command is not available by default. To use it, first you must add it to an Archicad menu:

1. Go to **Options > Work Environment > Menus**
2. Choose the menu where you want to place the command (here, the File > Interoperability menu)
3. Search for the command: Break Solibri file link
4. Add it to the menu.



Bluebeam Connection

Bluebeam Connection is a PDF-based collaboration and markup solution. This add-on provides an easy connection between Archicad (version 21 and later) and Bluebeam Revu (any version).

Recommended Workflow

From Archicad, save or publish 2D views or layouts as PDFs, then send the PDFs to consultants for markup in the Bluebeam Revu application.

Re-import the PDF markups to Archicad using Bluebeam Connection. The Bluebeam Issues will appear in the right places, on the original 2D source views or Layouts in Archicad. Manage the Issues, using Archicad's Issue Management functions.

Repeat the process as needed. Each time you reimport Issues from a PDF, you can decide whether to skip or include duplicate or modified Issues from earlier rounds.

Install Bluebeam Connection

Download the Bluebeam Add-on from <https://graphisoft.com/downloads/addons/bluebeam>

Note: For the USA version only, the Add-on is included in Archicad (version 21 Update series 5000 and later). All other language versions must download the add-on separately, using the link shown above.

Bluebeam Connection Palette

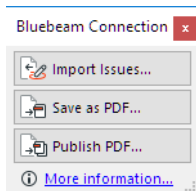
The Bluebeam Connection Palette contains three commands used in the Bluebeam Connection workflow. Use these commands to save Archicad view(s) in PDF format, and to import PDF markups to Archicad.

Note: These commands are also available from the Publisher, the File menu and Issue Organizer.

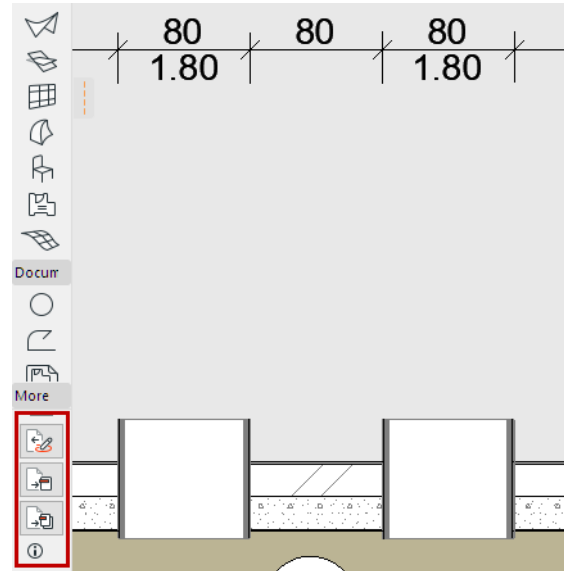
However, if you are using the Bluebeam workflow, this palette conveniently groups these commands for you.

Open the palette using one of these commands:

- File > Interoperability > Bluebeam Connection
- Window > Palettes > Bluebeam Connection



Bluebeam Connection Palette



Bluebeam Connection Palette - Docked

Publish Archicad View(s) for Bluebeam

From the Bluebeam Connection palette, use one of these two commands:

- Save as PDF
- Publish PDF

to publish one or more Archicad views in PDF format, optimized for markup in Bluebeam Revu.

In particular, the published PDF will include Zone polygons from the Archicad Floor Plan view, which can then be annotated in Bluebeam Revu.

Both commands are described below.

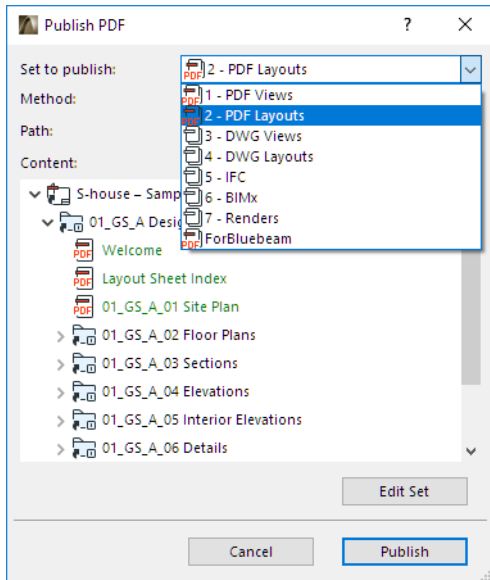
Save Single Archicad View in PDF Format

1. From the Bluebeam Connection palette, click **Save as PDF** to save the current view in PDF format.
2. Forward the PDF for markup in Bluebeam Revu.

Publish PDF Set

To publish an entire Publisher Set in PDF format, for use in Bluebeam Revu:

1. From the Bluebeam Connection palette, click Publish PDF Documentation.
2. The Publish PDF dialog appears.



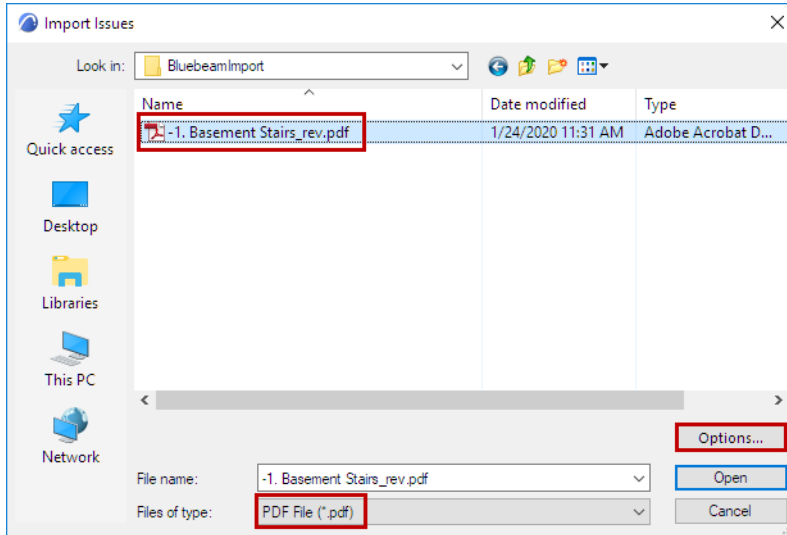
3. From the **Set to publish** drop-down, choose a Publisher Set.
 - You must choose a Publisher Set in which all items are in PDF format. If the set you choose includes items in other formats, the Publish button is grey and you cannot continue.
 - In this case, click **Edit Set** to open the Organizer, where you can set up a new Publisher Set and/or redefine Publisher Set items in PDF format.
4. Click **Publish**.
5. Forward the resulting PDF for markup in Bluebeam Revu.

Import Issues to Archicad

Once the PDF has been marked up in Bluebeam Revu, follow these steps in Archicad to import the Issues:

1. From the Bluebeam Connection Palette: click **Import Issues**.
2. In the appearing **Import Issues** dialog box, set the File Type as PDF.
3. Browse for and select the marked up PDF file.

4. Click **Options**.



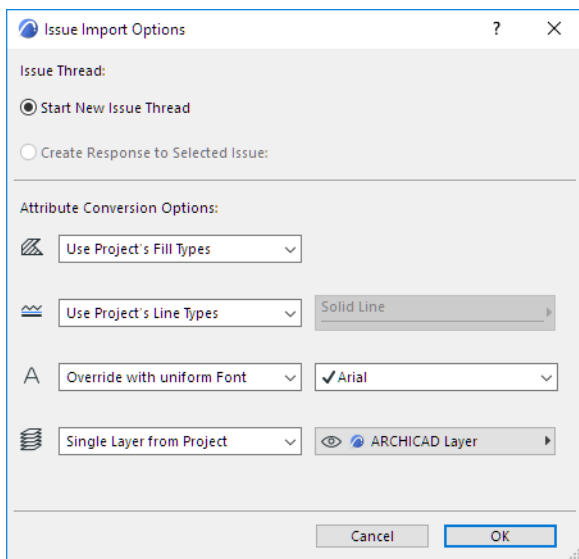
The **Issue Import Options** dialog box opens:

Issue Import Options

In this dialog, define how the issues imported from Bluebeam should appear in Archicad.

Issue Thread

Choose an option for how the imported Issues should appear in Issue Organizer.

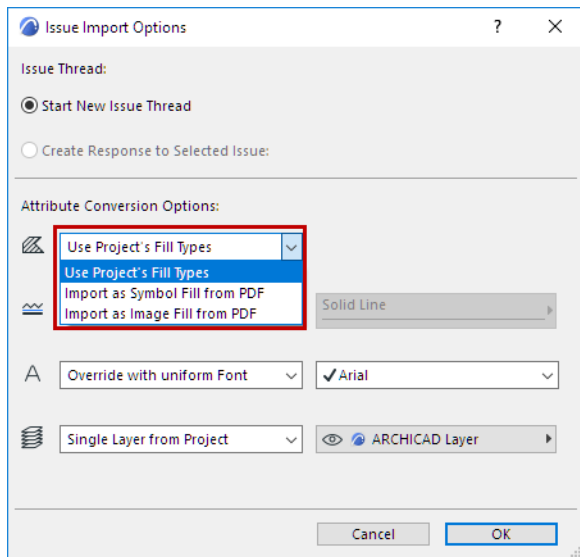


- **Start New Issue Thread:** The imported Issues will constitute their own thread. (In the Issue Organizer, you can sort entries by thread.)
- **Create Response to Selected Issue:** This option is available if an Issue is currently open in Issue Manager. Choose this to list the imported Issues underneath the currently active Issue, as sub-Issues.
Tip: To create a separate “category” for the imported Issues, first create a new Issue in Archicad, and give it a name (e.g. “Imported Issues from PDF”.) With this new entry selected, import the PDF Issues, and use the “Create Response to...” option. The imported Issues will be listed under this Issues (use **Sort by:** Thread).

Attribute Conversion Options

Choose how to convert the fills, line types, fonts and layers of the Issues you are importing.

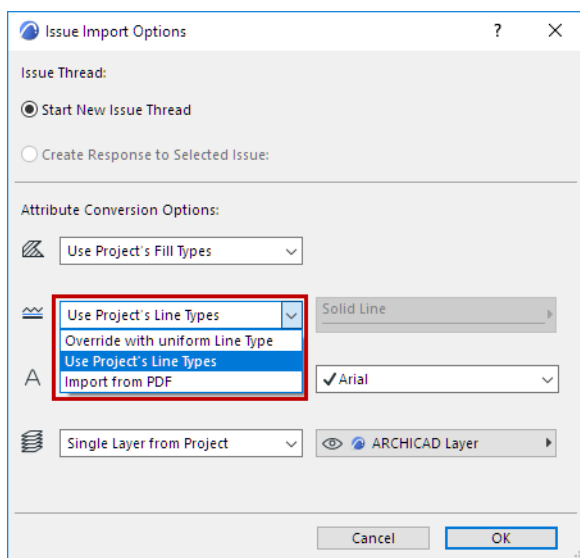
Fills



- **Use Project's Fill Types:** No new Fill Types will be imported from the PDF. Any imported fills that have no counterpart in the project will be transformed into solid fills.
- **Import as Symbol Fill from PDF:** A new vectorial fill attribute will be created in the project and used for imported fills. (For example: "60% PDF Solid Fill")
- **Import as Image Fill from PDF:** A new image fill attribute will be created for any fills, among the drawing primitives, which have no counterpart among the project's attributes.

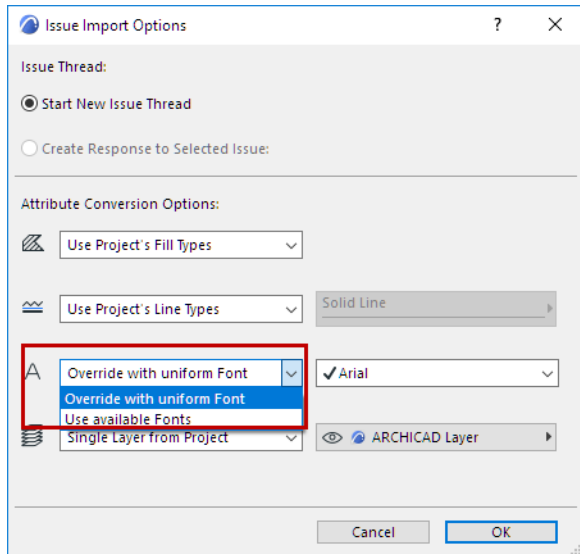
Note: If an image fill is generated, then this file is added to the project's Embedded Library. If you are working in a Teamwork project, you must have the needed access right: Library Parts – Create.

Line Types



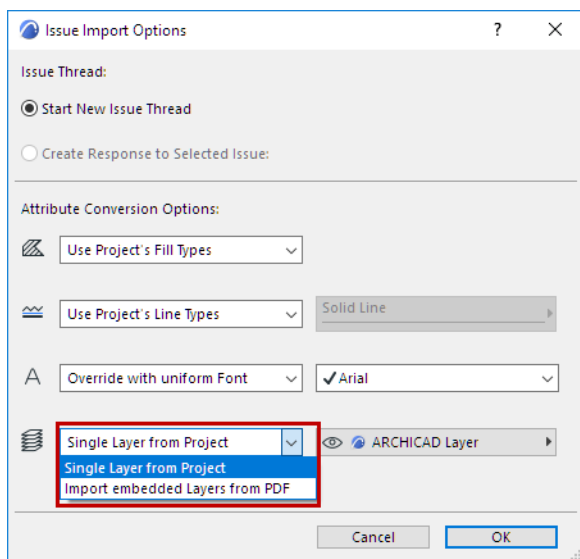
- **Use Project's Line Types:** No new Line Types will be imported. Issues are displayed using similar, existing Archicad line types (e.g. solid, dashed, dotted). If a corresponding Line Type does not exist in the Archicad project, a solid line is used.
- **Override with uniform Line Type:** Choose one of the project's Line Types. Imported Issues will be displayed using this Line Type.
- **Import from PDF:** Line Types of Issues are imported and added to the Archicad project.

Text Font



- **Override with uniform Font:** Choose one of the project's font types to display text items on imported Issues.
- **Use available Fonts:** Imported Issue texts are displayed using similar fonts available in the Archicad project.

Layer of Issues



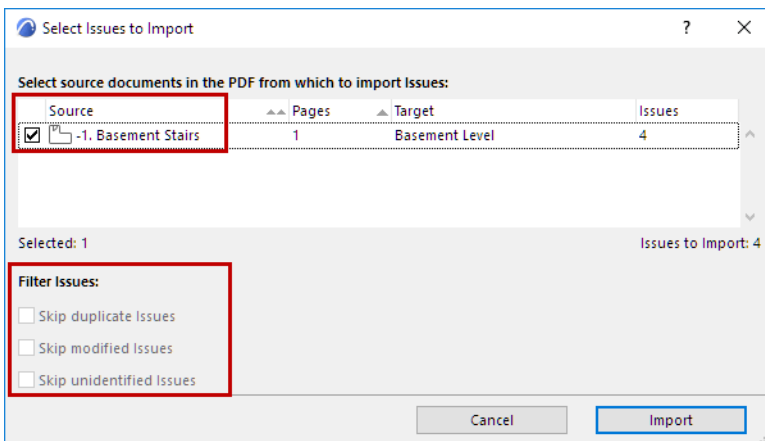
- **Single Layer from Project:** Choose one of the project's Layers. Imported Issues will be placed on this Layer.

- **Import embedded Layers from PDF:** Issues will be placed on layers corresponding to their embedded layer assignment (if any) in the PDF source. Those embedded layers will be added to your Archicad project.

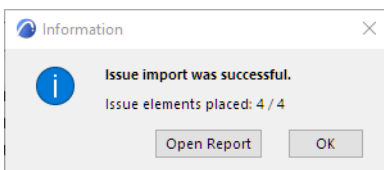
After setting Issue Import Options, click **OK** to return to the Import Issues dialog.

Import Issues Open File

1. From the Import Issues dialog, select the Bluebeam file to import. Click **Open**.
2. The **Select Issues to Import** dialog appears:

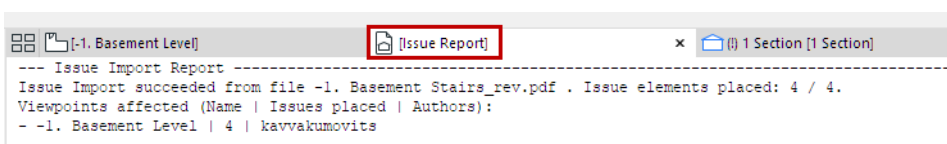


3. Select the source documents in the PDF from which to import the Issues.
4. Filter the Issues as needed:
 - **Skip duplicate Issues:** Duplicate Issues have already been imported to Archicad, and they have not changed in the PDF since then. *Check this to skip importing these duplicate Issues.*
 - **Skip modified Issues.** Modified Issues have already been imported to Archicad, but they were later modified in the PDF. *Check this to skip importing these Issues.*
 - **Skip unidentified Issues:** Some markup applications do not assign unique IDs to the Issues. Thus, these Issues cannot be identified as either new, duplicate or modified. (This is not a problem in the Bluebeam Revu workflow.) *Check this to skip importing these Issues.*
5. Click **Import**. An Info dialog appears:



6. Click **Open Report** to see a summary of Issues that have been added to the original Archicad views, or click OK to skip the report.

The new Issues are also listed in Issue Organizer.



7. Navigate to the source Archicad views or Layouts to handle the imported Issues.

Merkmalserver-Verbindung Add-on

The GER and AUT versions of Archicad automatically include the Merkmalserver-Verbindung Add-on.

Note: Merkmalserver-Verbindung is based on the ÖNORM A 6241-2 standard (freeBIM). (*For more information, see www.freebim.at.*)

Functions of the Merkmalserver-Verbindung

Merkmalserver-Verbindung

Load Merkmalserver Data

Set Current Stage

Show/Hide Properties by Stage in Archicad

Update Merkmalserver Database

Merkmalserver Properties in IFC Exchange

Export/Import Scheduled Merkmalserver Properties

Using Merkmalserver-Verbindung in Teamwork

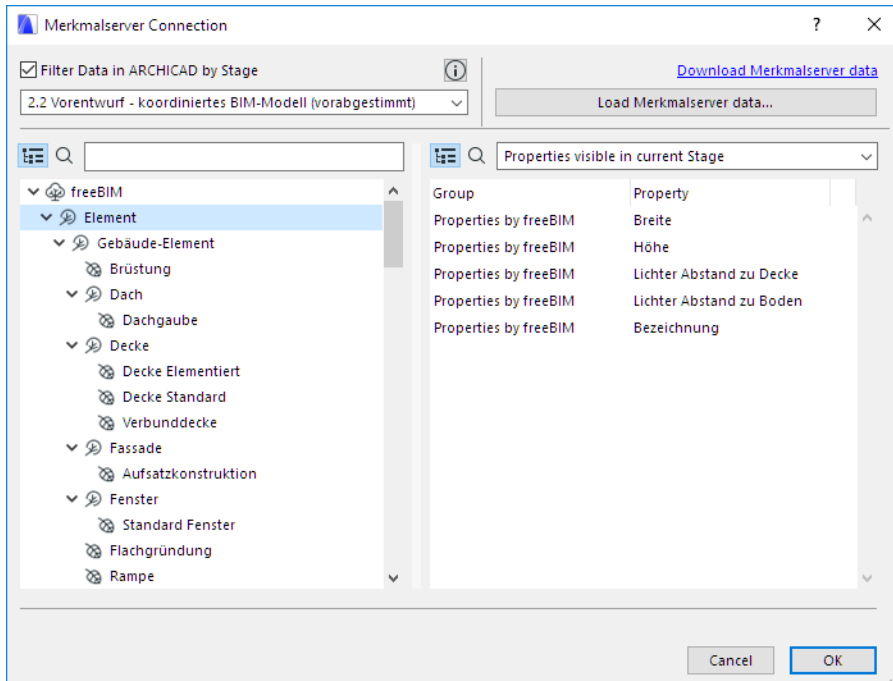
Functions of the Merkmalserver-Verbindung

- Loads the Merkmalserver data file so that you can access the following data in Archicad according to the Merkmalserver (freeBIM) standard:
 - Classifications and available Properties
 - IFC Type mapping and IFC Property mapping (freeBIM Conversion Presets are available for IFC Translators)
- Lets you filter the visibility of Merkmalserver Properties, depending on the current stage, in the Archicad Tool Settings dialogs
- Lets you export only the visible Merkmalserver Properties, when you export a model to IFC

Merkmalserver-Verbindung

This dialog is available here: **File > Interoperability > Classifications and Properties > Merkmalserver-Verbindung.**

freeBIM Classifications are on the left. Properties are on the right.



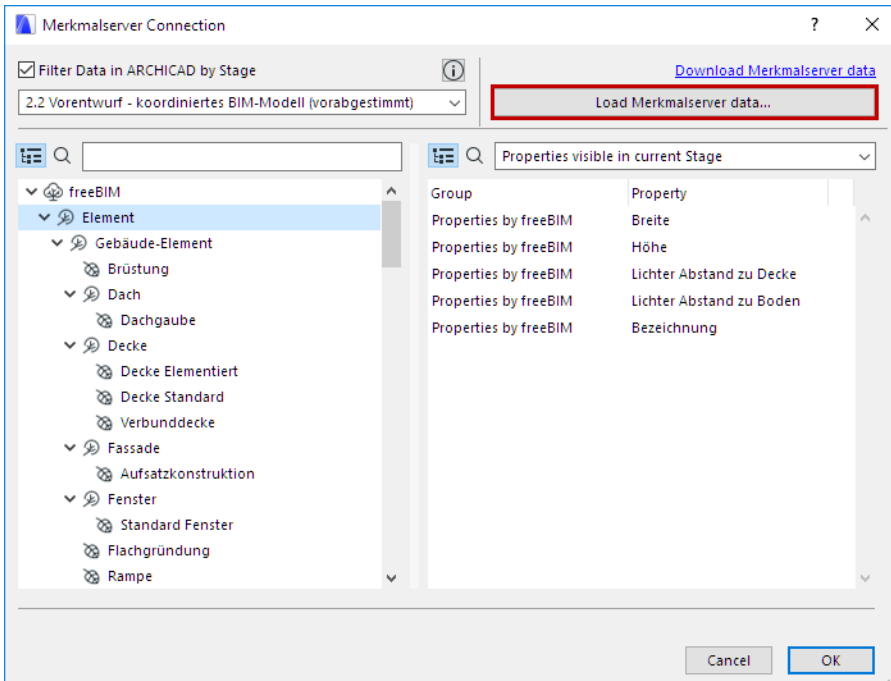
Use the Merkmalserver-Verbindung to do the following:

- Load the latest Merkmalserver database
- Set the current Merkmalserver Stage
- Preview list of Merkmalserver Properties and see information on each
- Enable filtering of visible Properties, by stage, in Archicad Tool Settings dialogs

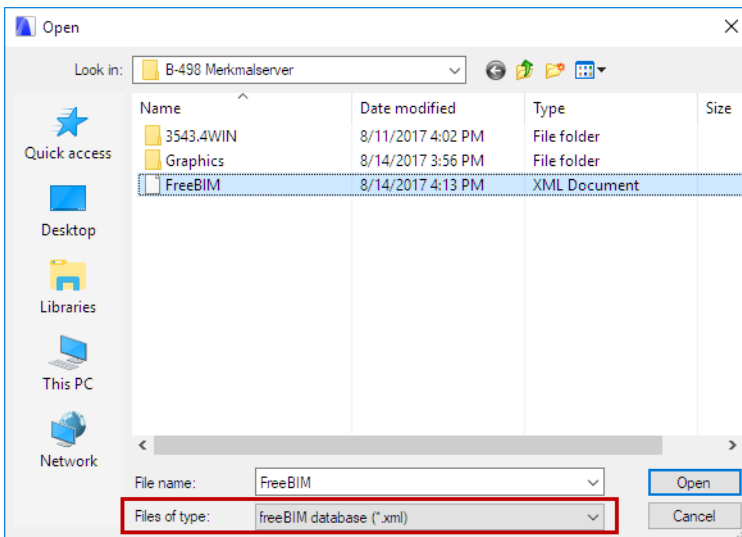
Each function is described below.

Load Merkmalserver Data

1. Open the Merkmalserver-Verbindung here: **File > Interoperability > Classifications and Properties > Merkmalserver-Verbindung**
2. Click **Load Merkmalserver data...**

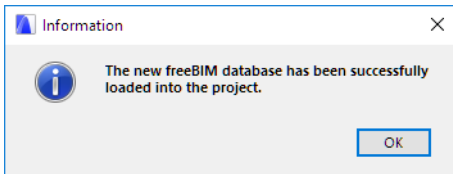


3. Browse for the XML file you need. (To download the latest Merkmalserver XML file, go to the Graphisoft downloads page: [Download BIM Content.](#))



4. Click **Open**.

The data is loaded into the Archicad project.

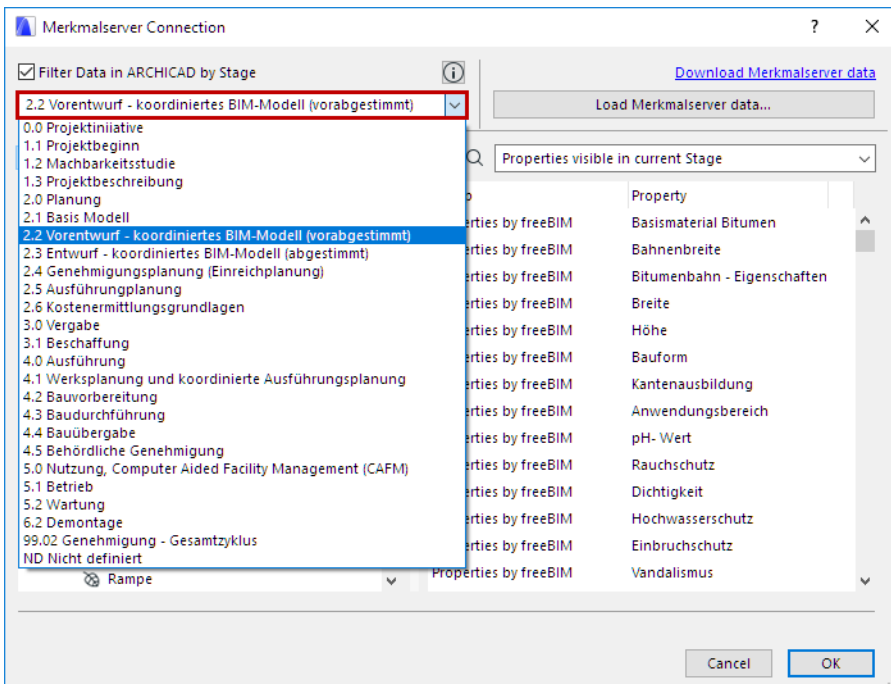


If your Archicad project already contains Merkmalsserver data, choose how to manage existing data: either **Update** or **Replace**.

[See details at *Update Merkmalsserver Database*.](#)

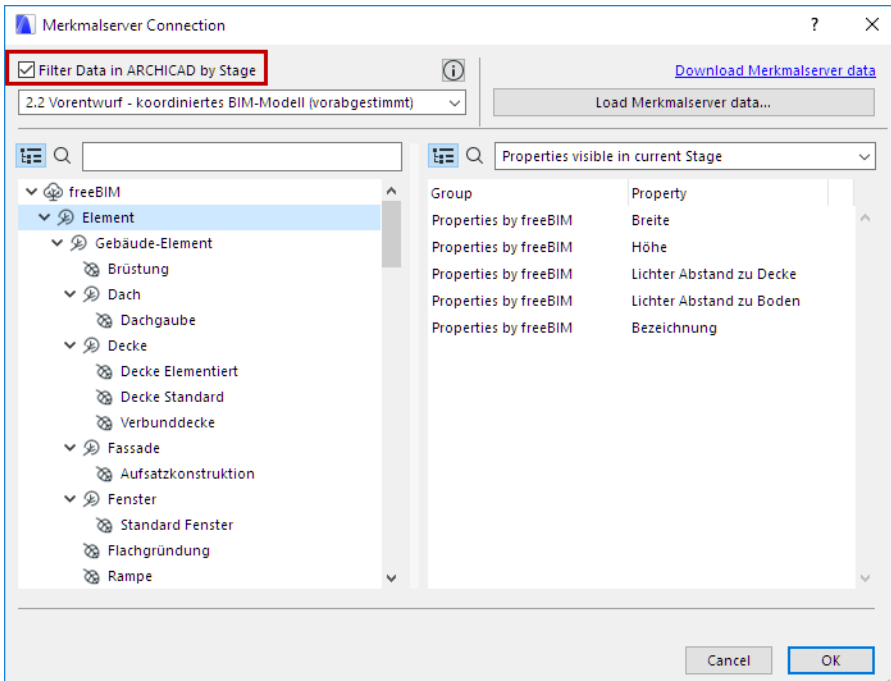
Set Current Stage

1. Open the Merkmalsserver-Verbindung here: **File > Interoperability > Classifications and Properties > Merkmalsserver-Verbindung**.
2. On the left side of the dialog, choose a design stage according to the Merkmalsserver standard.

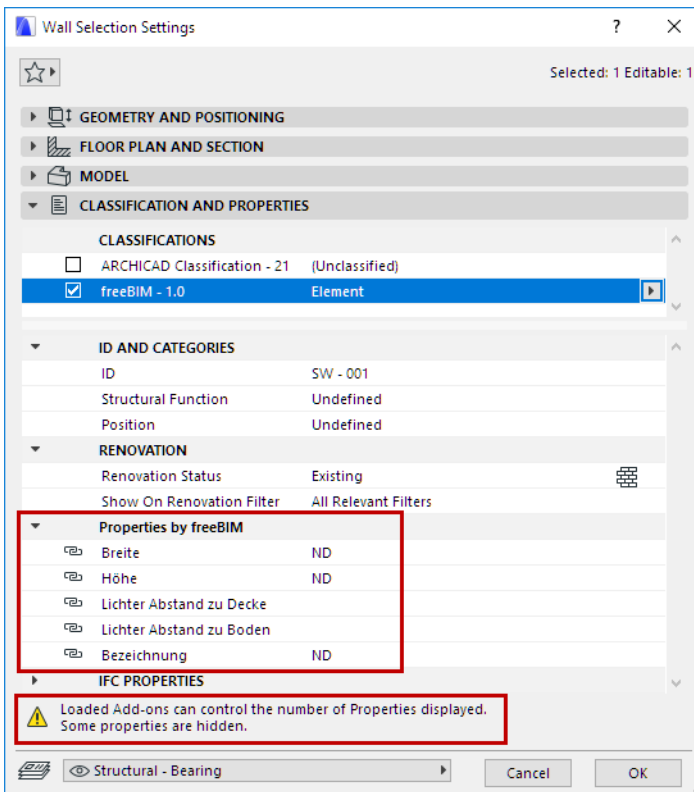


Show/Hide Properties by Stage in Archicad

1. Open the Merkmalserver-Verbindung here: **File > Interoperability > Classifications and Properties > Merkmalserver-Verbindung.**
2. Check the **Filter Data in Archicad by Stage** box.



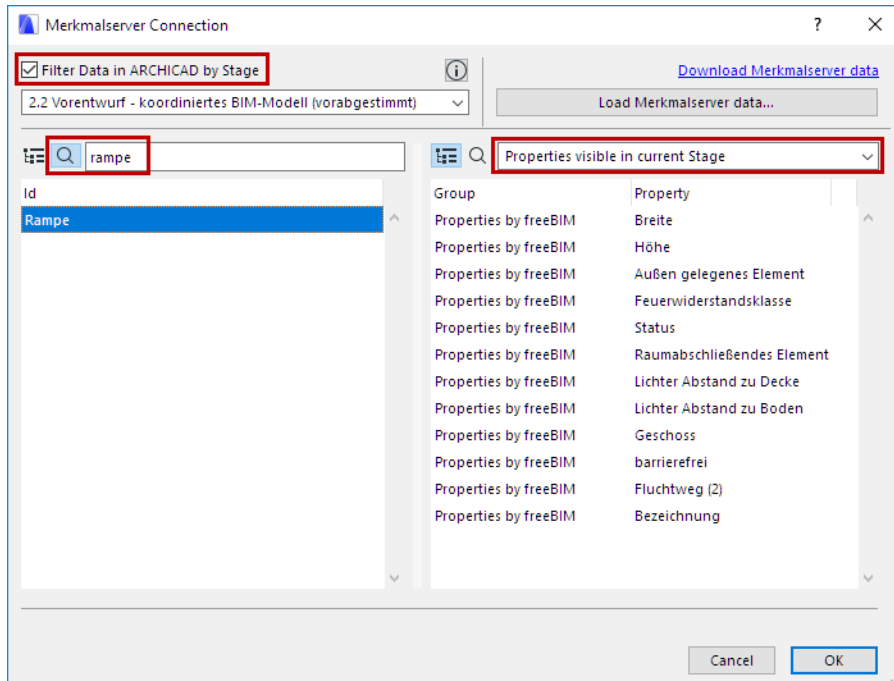
3. As a result, Merkmalserver Properties will be shown or hidden in Archicad Tool Settings according to the current Stage.



Preview Merkmalsserver Properties

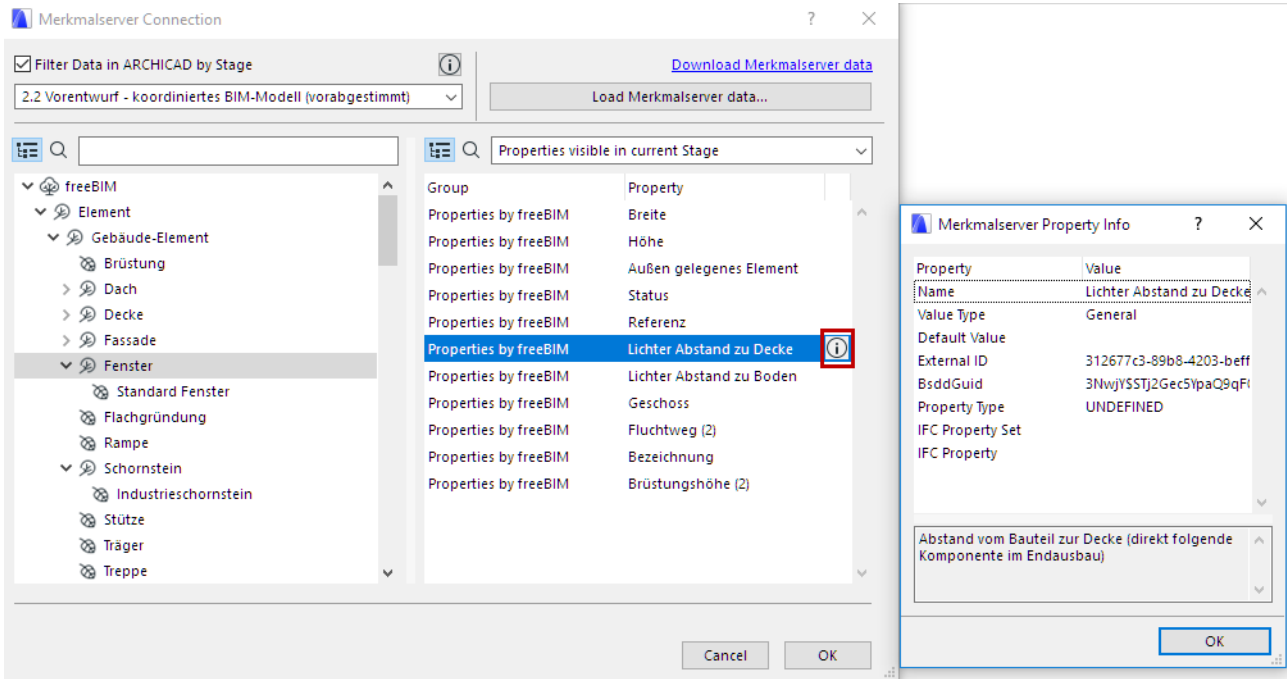
You can easily check which Merkmalsserver Properties are *available* and *visible* for a given Classification item, depending on the current stage.

1. In Merkmalsserver-Verbindung, use the search controls at the top of the Classification list (on the left), to select a Classification item.
2. For any Classification selected (on the left), you can check (on the right) which Properties are available for this Classification, according to the current freeBIM stage.
3. If the **Filter Data by Stage** checkbox is on, the list shows which Properties are visible in Archicad Tool Settings dialogs.



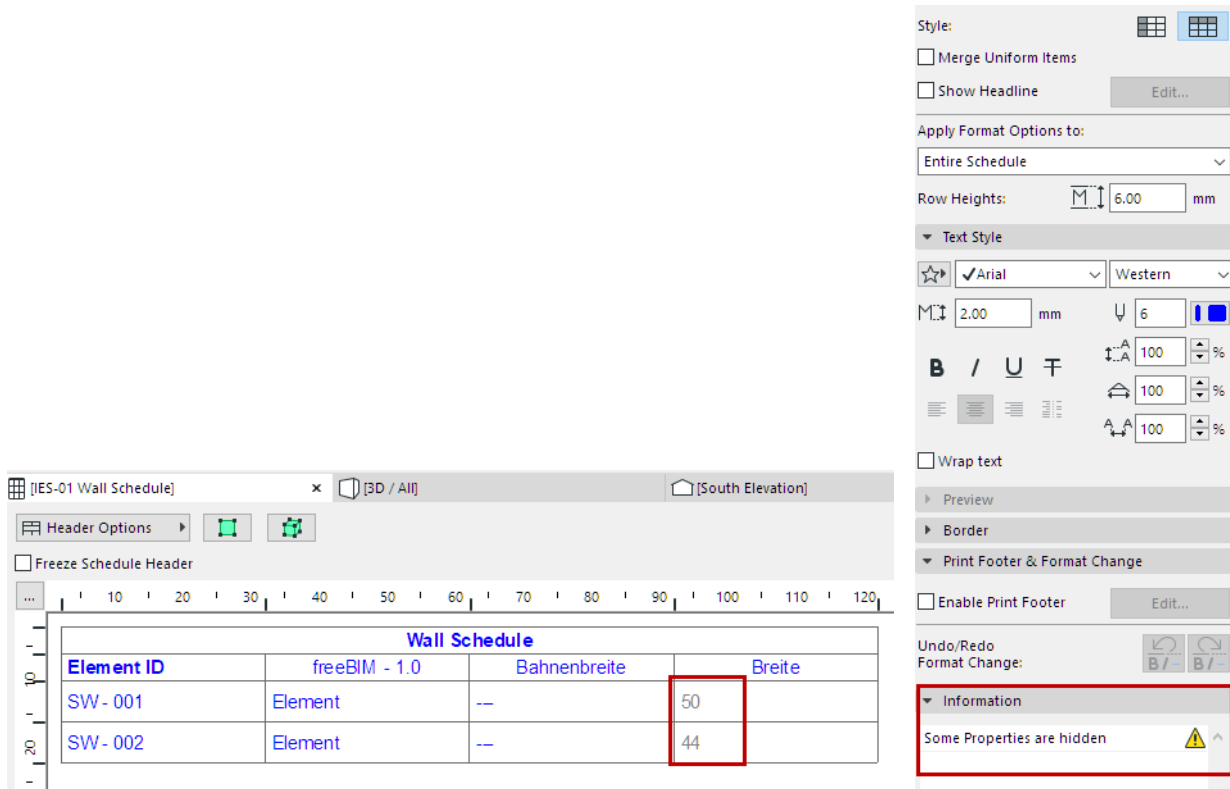
Get Info on a Merkmalsserver Property

- In Merkmalsserver-Verbindung, click the **Info** button next to any selected Property to see its definition according to the Merkmalsserver standard.



Merkmalserver Properties in Interactive Schedule

- All Properties and their values are visible in Interactive Schedule lists. However, if a Merkmalserver Property is hidden in Tool Settings for the current stage, then that Property's value is shown in the Schedule with grayed out fonts. The Schedule's Info field displays a note to this effect.



Update Merkmalserver Database

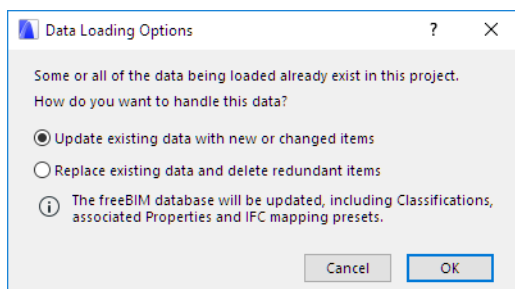
The Merkmalserver definitions are modified from time to time by the ÖNORM standard.

As needed, use Merkmalserver-Verbindung to load the latest XML file and update your database in Archicad. Follow the instructions:

[See Load Merkmalserver Data.](#)

If your Archicad project already contains Merkmalserver data, choose how to manage existing data: either **Update** or **Replace**.

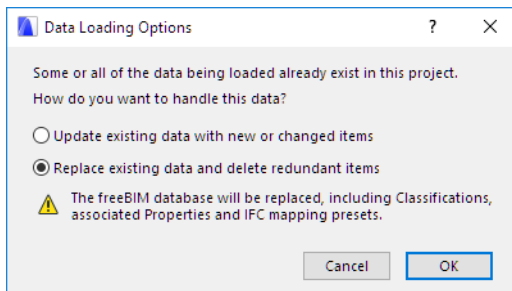
Update existing data



Choose the **Update** option to add any new definitions, and update existing data with changed definitions. The following data are updated: freeBIM Classifications, associated Properties, and IFC mapping presets.

Even if a data item definition is updated, its associations (e.g. property assignment to elements; property use in Schedule Criteria) will be maintained.

Replace existing data



Choose the **Replace** option to delete the Merkmalserver data that exists in the Archicad project, and replace it with the incoming new Merkmalserver database.

The following data are replaced: freeBIM Classifications, associated Properties, and IFC mapping presets.

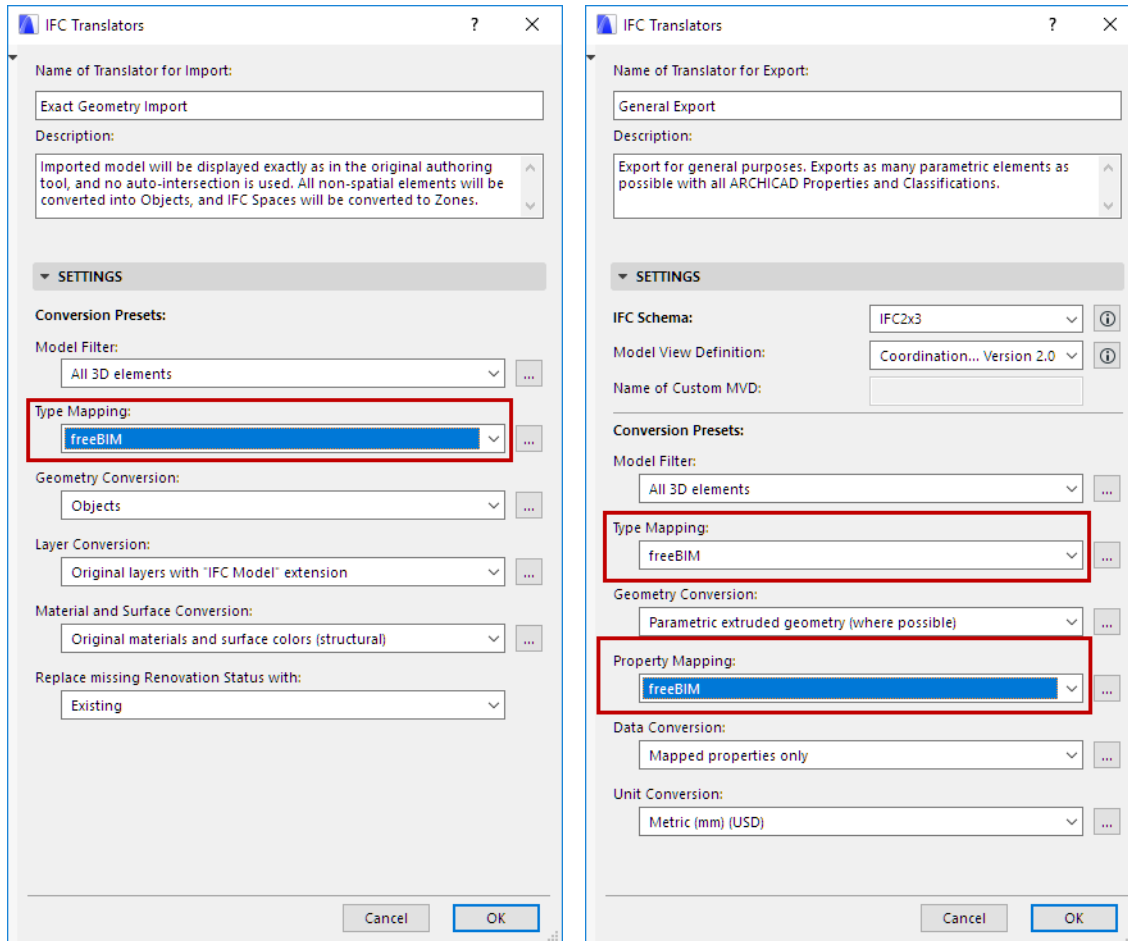
- If you created new data items in Archicad (e.g. a new Property in the freeBIM group), then the Replace function will delete these newly created data types.
- Even if a data item (e.g. property) is replaced with a new one of the same name, its associations (e.g. property assignment to elements; property use in Schedule Criteria) will be maintained.

Merkmalserver Properties in IFC Exchange

When you exchange an Archicad model using IFC, the data export/import is controlled through IFC Translators (**File > Interoperability > IFC > IFC Translators**). With the Merkmalserver-Verbindung Add-on, you can optimize this process by using built-in data mapping presets, and by filtering the exported properties by visibility.

Use freeBIM Conversion Presets

Use the predefined freeBIM Conversion Presets for IFC **Property Mapping** (export translators) and **Type Mapping** (export/import translators).



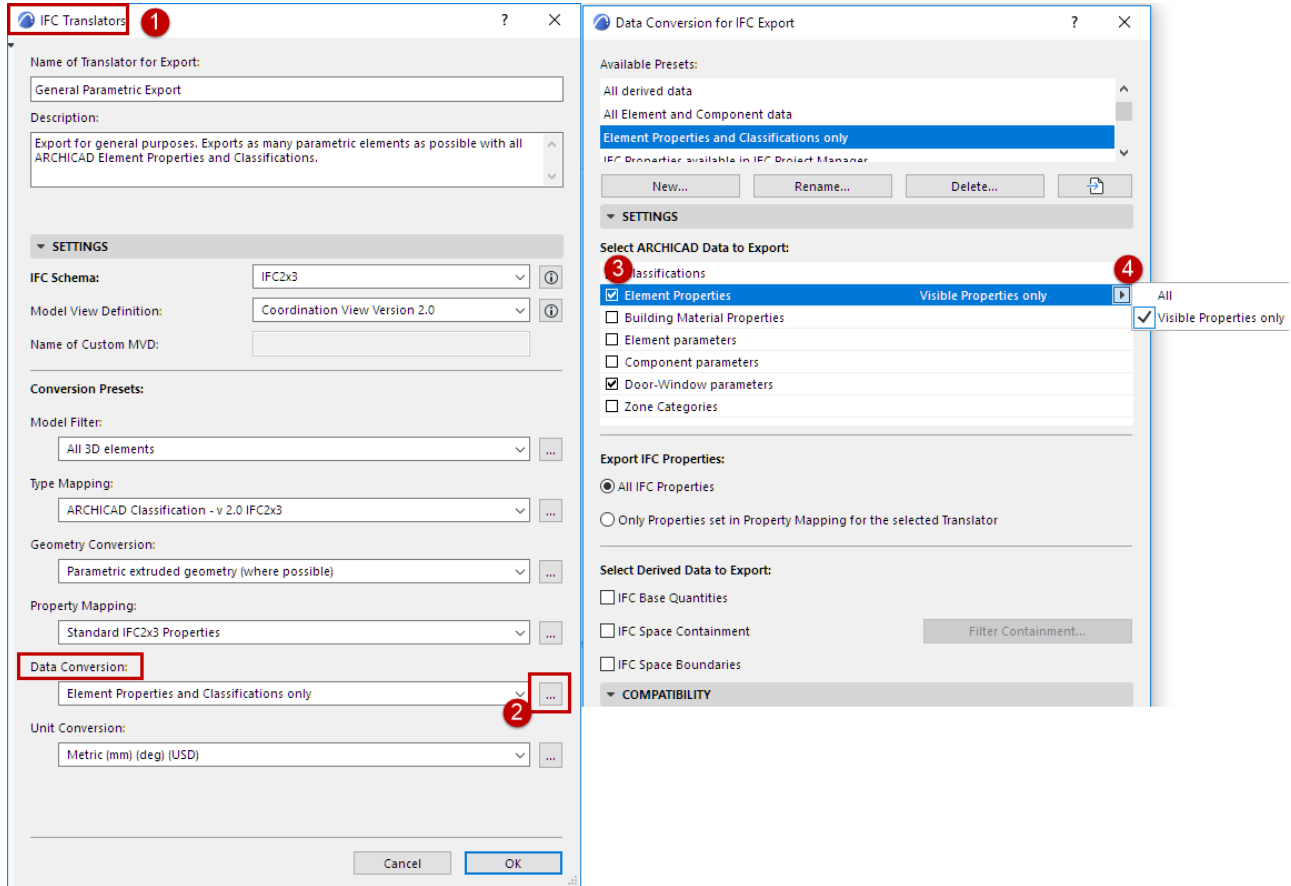
freeBIM Presets for IFC Translators: Type Mapping, Property Mapping

Export Visible Properties Only

Use this option if you are exporting Merkmalsserver properties to IFC, and you wish to export only the visible properties (as set up in Merkmalsserver-Verbindung, using the “Filter by Stage” filtering checkbox).

1. Open **File > Interoperability > IFC > IFC Translators**
2. Open the IFC Translator dialog's **Data Conversion** for IFC Export dialog.
3. Check the box to export **Element Properties**.

4. Choose the **Visible Properties only** option.

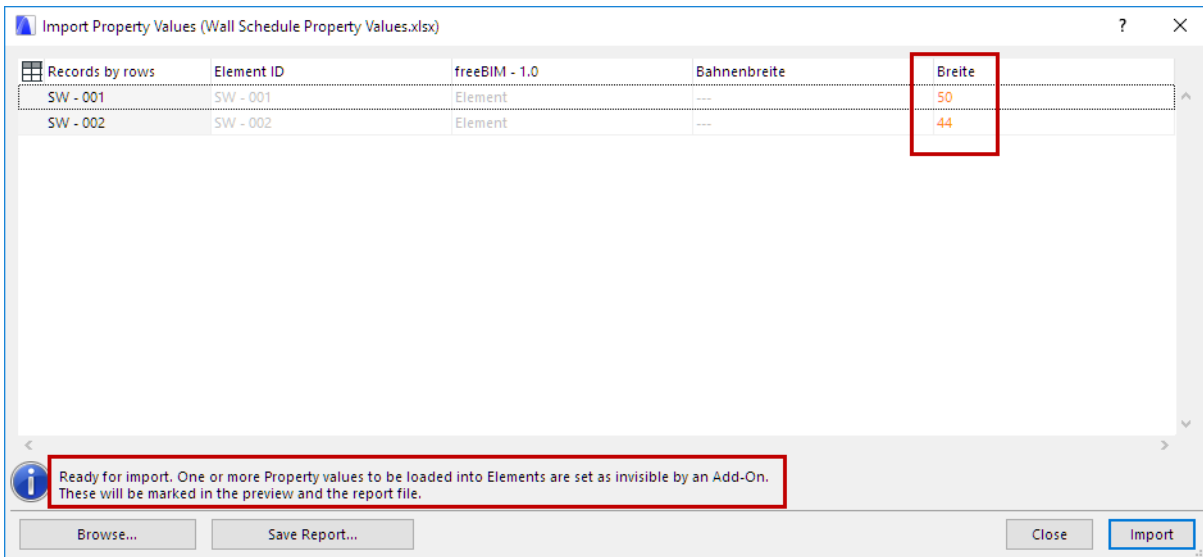


Export/Import Scheduled Merkmalserver Properties

You can export and import scheduled Merkmalserver data via Excel spreadsheet, using the **File > Interoperability > Element Properties > Export Property Values from Schedule/Import Property Values in Elements** command.

In this workflow, all Merkmalserver Properties are exported and imported, regardless of their visibility in Archicad.

However, you will receive colored and textual feedback about any exported/imported values that are hidden in Archicad due to the Merkmalserver-Verbindung Add-on.



Similarly, the Export/Import function's Report (click **Save Report** at the bottom of the list) also displays hidden Merkmalserver properties in a special color.

Legend for Element Property Import Report			
Formatting	Status	Description	
Black headers and fields	Valid data	Valid data, will be imported provided that the property's box is checked at importing.	
"---" in a data field	Not available	The property is not available for that element, and can be ignored by the consultant.	
Grey headers and fields	Identifying data	These are not element properties and will not be imported. Including these at export serves for Element identification.	
Orange fields	Hidden data	Data items belong to Properties set as invisible in the Project by an Add-On.	
Blue headers and fields	Duplicates	Data items belong to duplicate properties or duplicate elements. Only the values of the first instance of this property will be imported.	
Red headers and fields	Invalid data	Invalid or missing data; cannot be imported. Possible reasons:	
		- The data does not correspond to the referred Property's Data type	
		- Data is missing	
		- The model element that the property applies to is not available, because it is locked, placed on a locked layer, part of a Hotlinked Module, not reserved in Teamwork, or has been deleted	

Using Merkmalserver-Verbindung in Teamwork

You must have the needed permissions, and reserve the needed dialogs, before you can use Merkmalserver-Verbindung in Teamwork.

Set Merkmalserver Stage

Reserve Merkmalserver-Verbindung.

Load Merkmalserver Database

Reserve all of the following dialogs:

- Classification Manager
- Property Manager
- IFC Translators

Required permissions to load Merkmalserver database:

- Classifications and Properties - Create
- Classifications and Properties - Delete/Modify
- IFC Translators - Manage

Note: If you do not have permission to manage IFC Translators and/or you have not reserved the IFC Translators, you can still load the Merkmalserver database - however, in this case only the Classifications and Properties will be loaded. No Translator Presets will be created.

Area List Add-on

Introduction

This section describes how to use the Area List Add-on for Archicad - specifically, to create area and volume calculations in conformity with the DIN 277 standard.

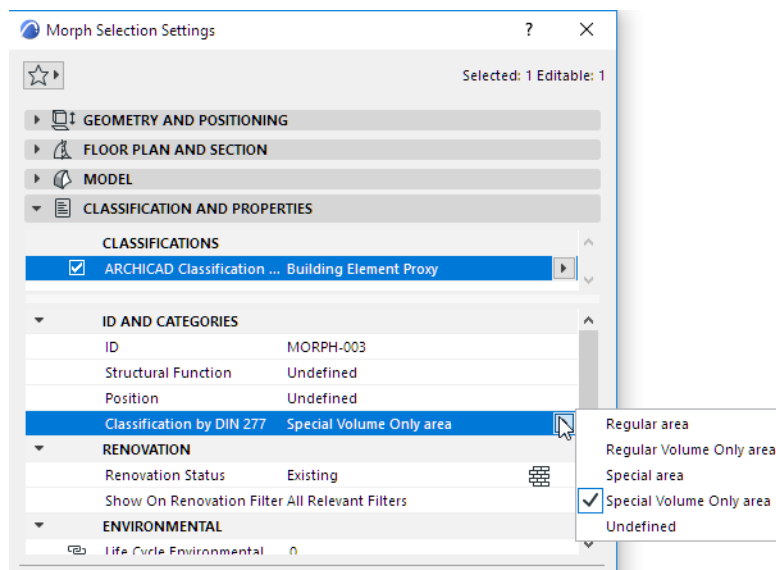
Calculation Basics

The Zone Tool

In order to use the Area List add-on, you must create Zones in the project, using the Zone Tool. Net areas and volumes will be calculated based on these Zones. Make sure that Zones are placed correctly. Usually, the Zone should extend from the top of the floor Slab to the bottom of the floor Slab of the story above. Zones below the Roof are trimmed by Roof planes.

The Morph Tool

Morph bodies are used to calculate gross volumes and gross areas. In order to conform to DIN 277, the Morphs should extend to the outside surfaces of the building. For these Morphs, set their “Classification by DIN” value to one of the Regular or Special values.

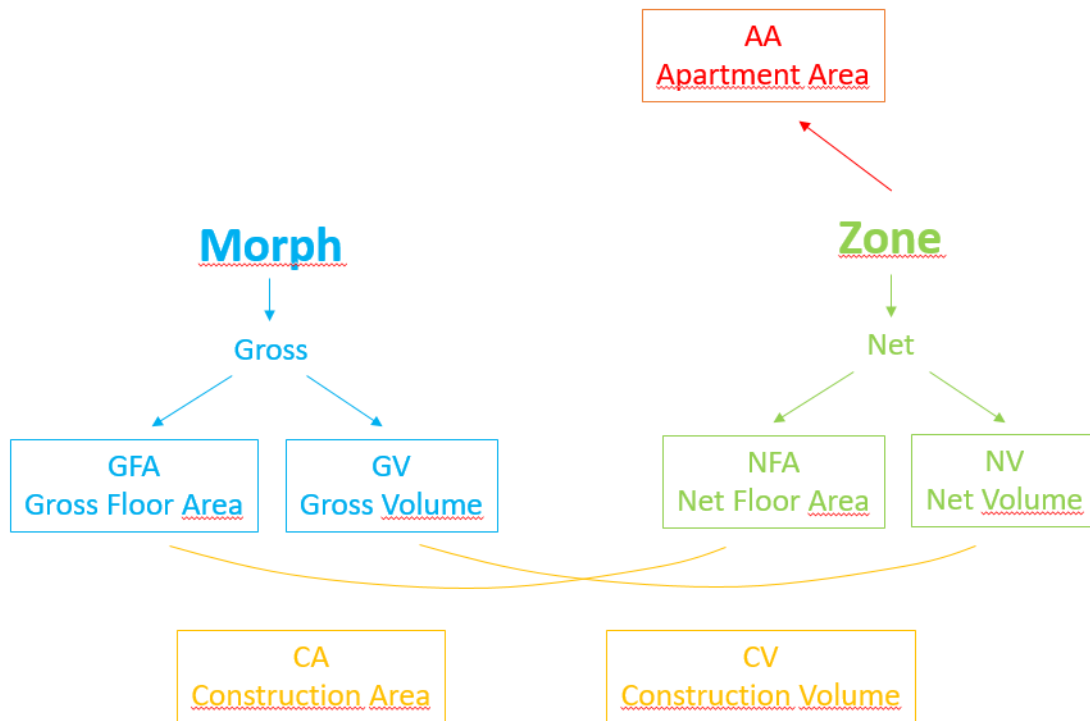


When modeling Morphs, make sure:

- Morphs should be placed on every story
- Use separate Morphs for Regular and Special calculations

Relation between Zones and Morphs

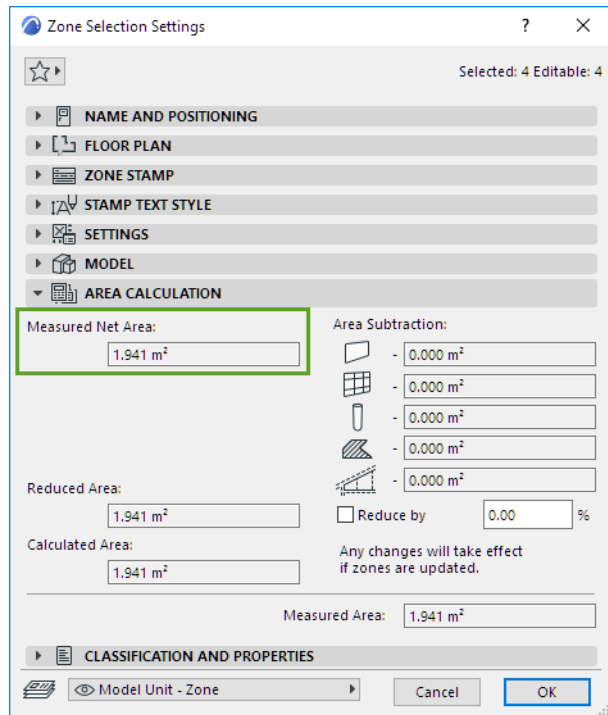
The following chart illustrates the relationship between Zones and Morphs. You will need both in order to produce complete calculations for Areas and Volumes.



Area and Volume Calculation Options

Net Floor Area

The Net Floor Area is the Measured Net Area of the Zone, as shown in Zone Settings. Subtractions due to Fills, Walls, Roofs etc. are not considered. Sums are calculated by stories.



Gross Floor Area

This is equal to top projected surface of Morphs that are classified as Regular or Special. Morphs classified as “Volume only” are not considered. Sums are calculated by stories.

Construction Area

The Gross Floor Area minus the Net Floor Area.

Net Volume

Volume of Zones, calculated by stories.

Gross Volume

Volume of Morphs that are classified for DIN.

Construction Volume

Gross Volume minus Net Volume.

Apartment Area

Archicad allows you to produce area calculation based on living area rather than by story. Here, subtractions due to Roof crops etc. are considered. It is equal to the Calculated Area as shown in Zone Settings.

Where to Find the Lists

Generated lists can be found

- under the **Area List** item in the Navigator
- or at **Document > Listing Extras > Area Lists**, from Area List Settings

Click on a list to open it.

Apartment Area Calculation
according to "Verordnung zur Berechnung der Wohnfläche" from 01.04.2004 (WöFM)

1. Building Project

Training Series 3, #PIn	#Site Full Address
-------------------------	--------------------

2. Client

#Client Company, #Client Full Address	#Client E-mail, #Client Phone Number
---------------------------------------	--------------------------------------

3. Architect

#Contact Full Name, #Contact Full Address	#Contact E-mail, #Contact Phone Number
---	--

4. Apartment Description

Apartment: <Apartment Name>; House Number: <House Number>; Function: <Function>; Story: 0. Ground Floor, 1. Gallery; Position: <Position>

5. Calculation (Based on a CAD model)

Nr.	Room Description	Area [m²]	To be filled by authorities	Overbuilt
			Area [m²]	
			Area to be calculated [m²]	
01	Exhibition space	63.24		
Gross Area Sum:				63.38
Rounding Correction:				-0.14
02	Entrance	7.24		
Gross Area Sum:				7.37
Rounding Correction:				-0.13
03	Washroom	3.50		
Gross Area Sum:				3.54
Rounding Correction:				-0.04
04	WC	1.94		
Gross Area Sum:				1.94
Rounding Correction:				0.00
05	Gallery	24.88		
Gross Area Sum:				25.79
Low Ceiling Reduction Sum (50%)				0.15
Rounding Correction:				-0.78
Net Apartment Area [m²]				100.78

There are _____ apartments in the building.
 The apartment includes _____ garage(s) and _____ utility room(s).
 The apartment is completed not completed

Place and date	Signature
----------------	-----------

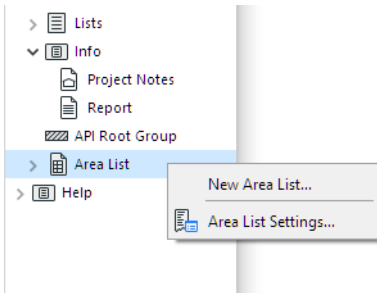
- Training Series 3
 - Stories
 - Sections
 - Elevations
 - Interior Elevations
 - Worksheets
 - Details
 - 3D Documents
 - 3D
 - Schedules
 - Project Indexes
 - Lists
- Info
 - Project Notes
 - Report
 - API Root Group
 - Area List
 - new list**
- Help

The following predefined Lists are available in the Archicad 22 Template, but you can create any custom list at any time.

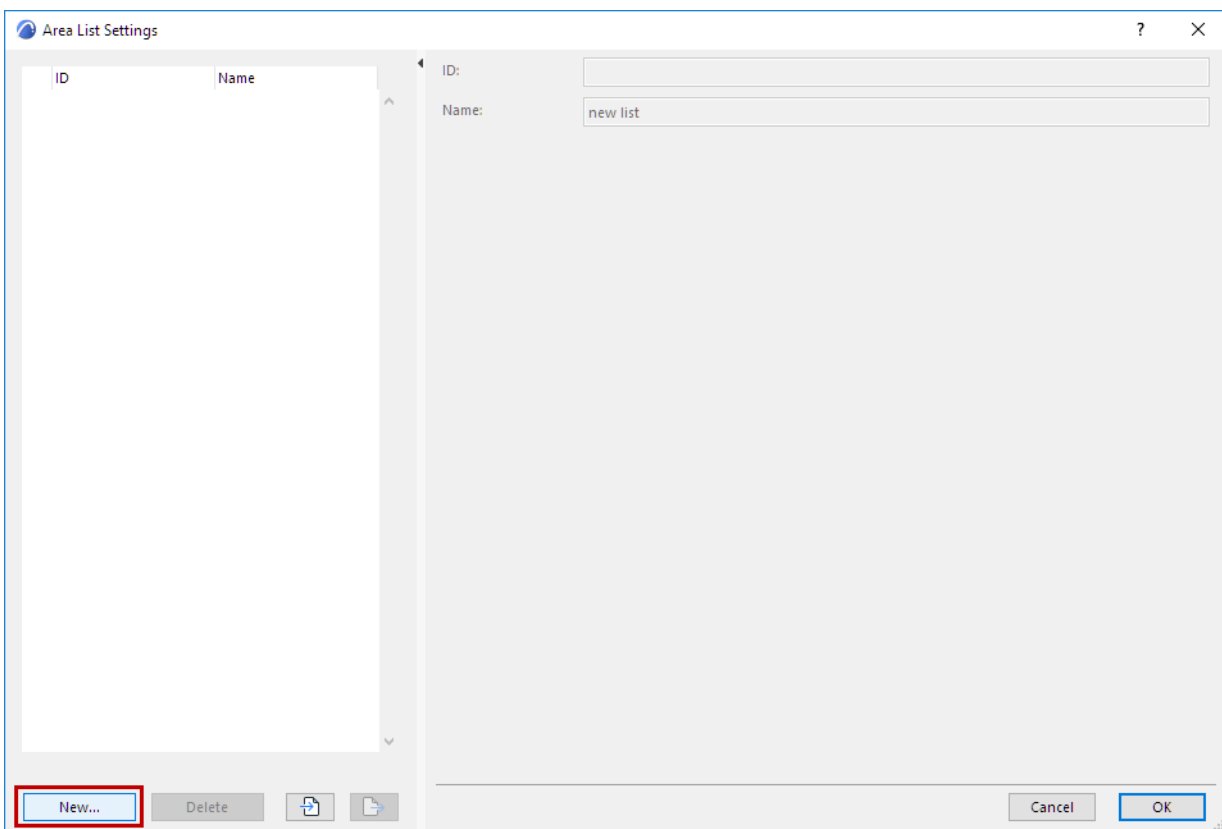
- ▼ Area List
 - 01 BGF - Brutto-Grundfläche
 - 02 NRF - Netto-Raumfläche
 - 03 KGF - Konstruktions-Grundfläche
 - 04 BRI - Brutto-Rauminhalt
 - 05 NRI - Netto-Rauminhalt
 - 06 KRI - Konstruktions-Rauminhalt

Create New Area List

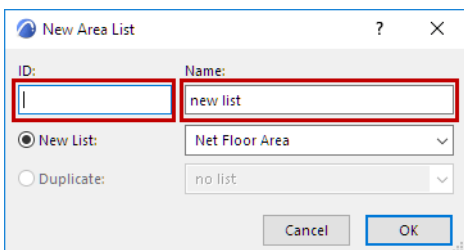
In the Navigator, right-click on the Area List item to access the context menu and the **New Area List** command.



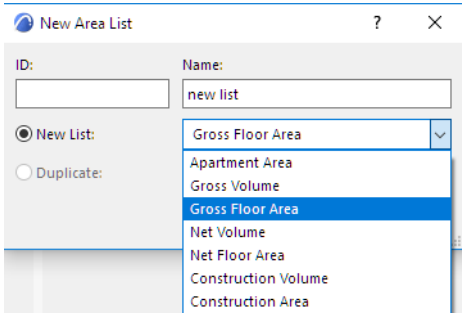
Another way: in Area List Settings, click **New** at bottom left.



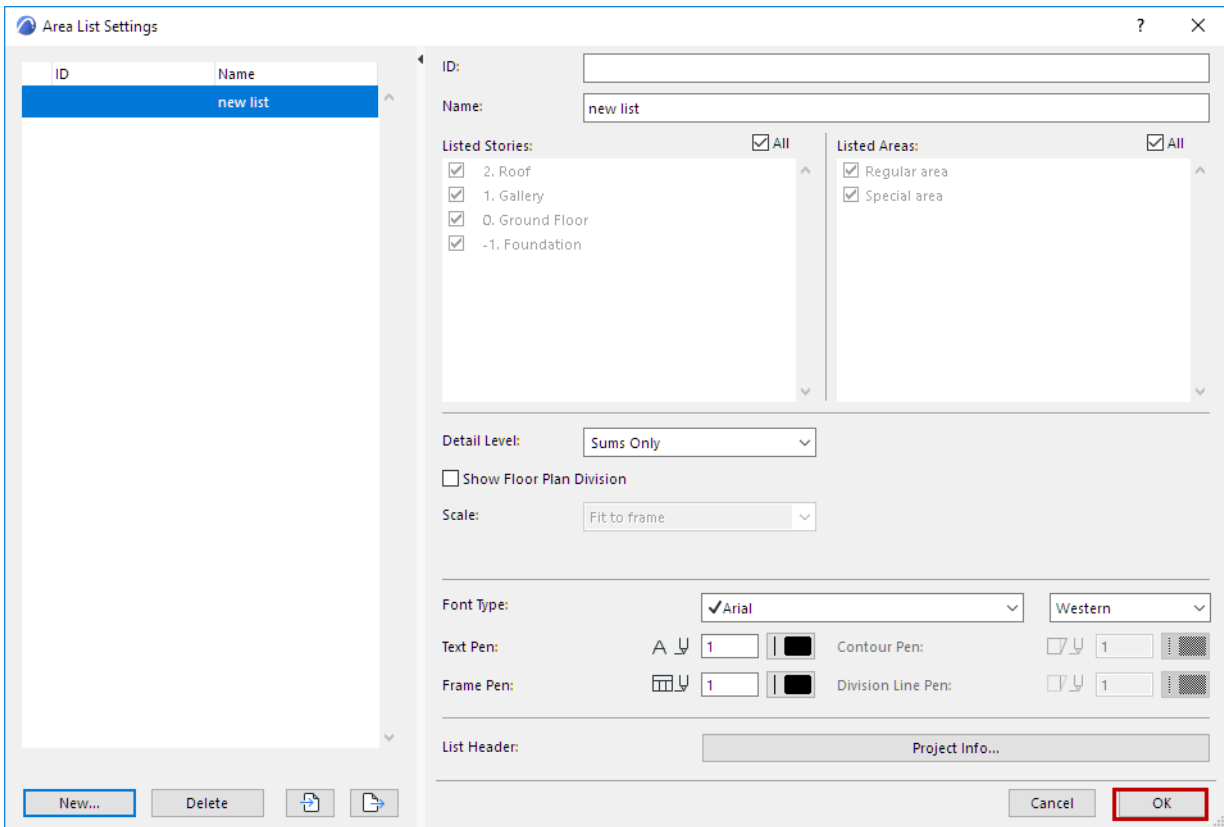
Enter a name and ID for the new List.



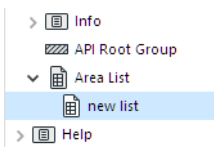
From dropdown menu, choose the List type.



In the Area List Settings dialog, set the options as needed. Click OK.

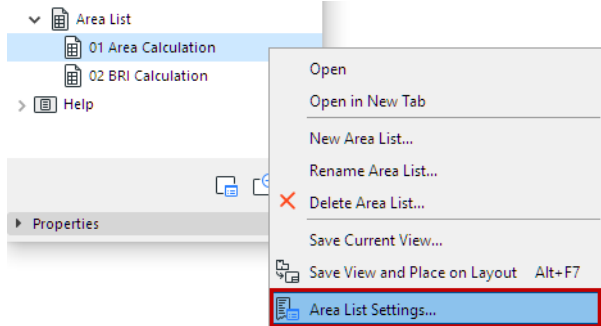


The new List is now available in the Navigator.

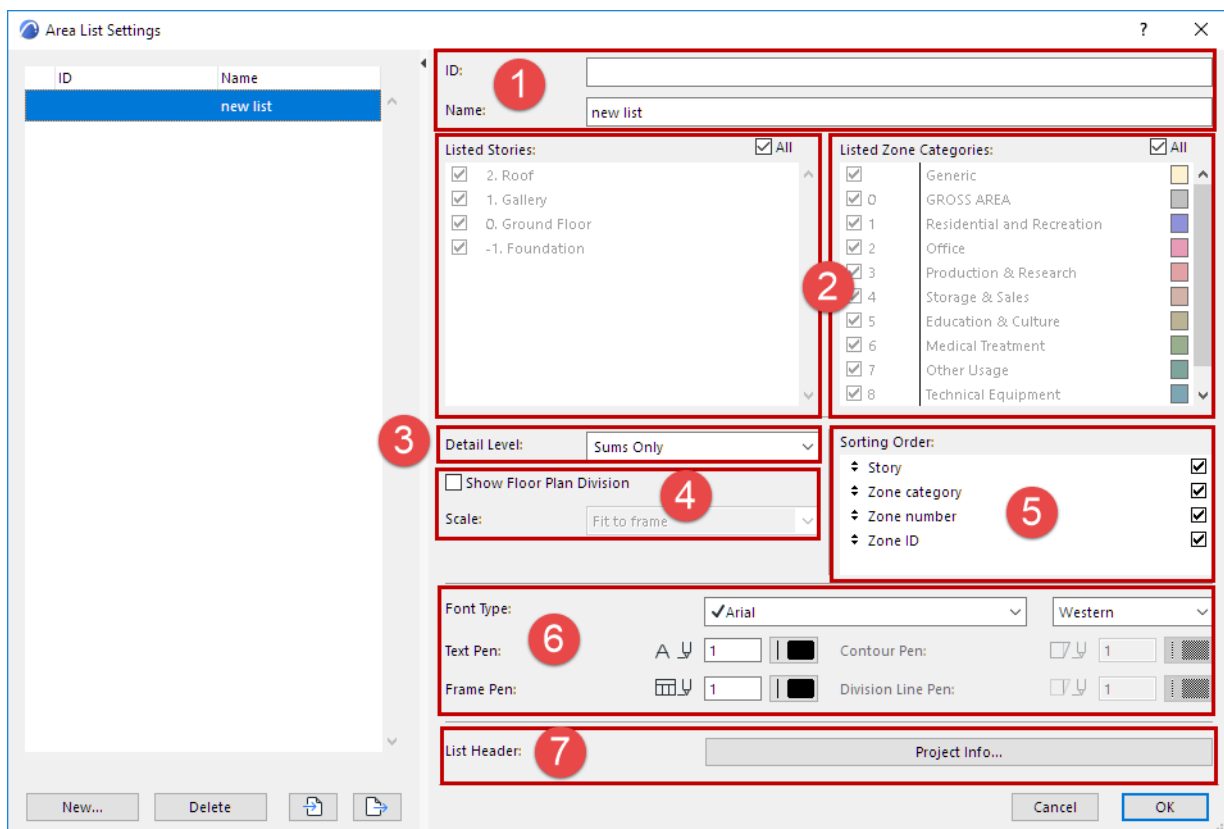


Edit Area List

To edit the settings of a list, select it in the Navigator and right-click to access **Area List Settings**.



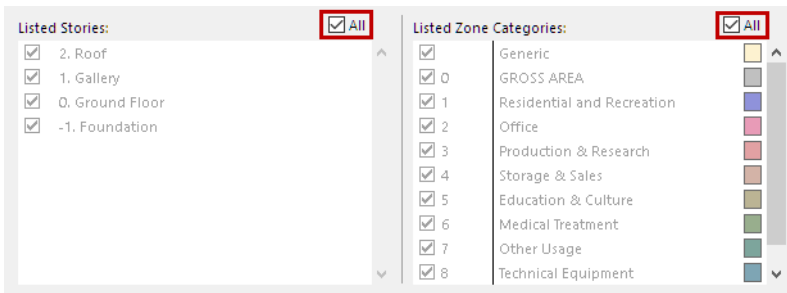
In the upcoming dialog, the Settings are divided into groups:



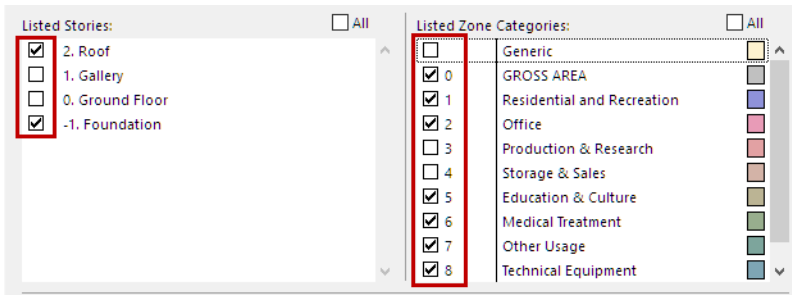
1. Here, enter the ID and Name of the list.

ID:	<input type="text"/>
Name:	<input type="text" value="new list"/>

2. Here, decide which Stories and Zone Categories should be considered in the calculation. To limit the calculation to certain stories/categories, first uncheck the **All** checkbox.

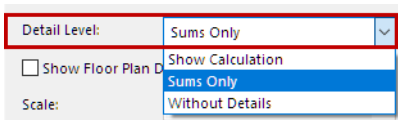


Now you can mark specific Stories and Zone Categories.



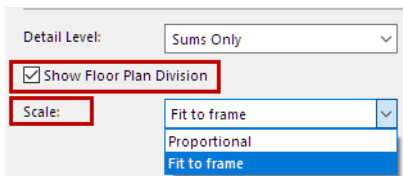
3. In the Detail Level section of the dialog, set how you want to display the calculation result:

- only the sums
- the complete calculation
- or no details (just the final result)



4. **Show Floor Plan Division:** Check this to create 2D illustrations of the Zones/Morphs used to calculate the list.

Choose how to **Scale** the resulting illustrations: either **Fit to Frame** or **Proportional**, to make the area lists easier to understand.



Net Floor Area
by DIN 277 - Jan 2016

1. Building Project

Training Series 3, #Pln	#Site Full Address
-------------------------	--------------------

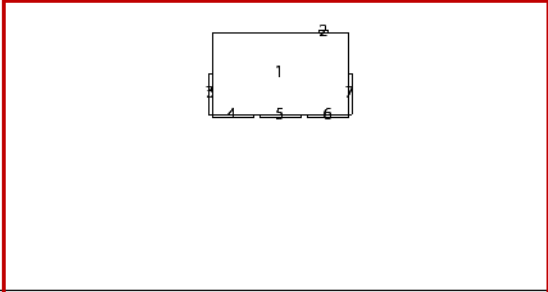
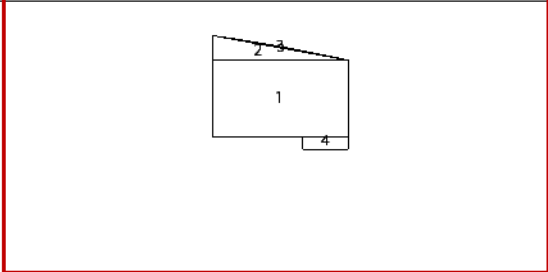
2. Client

#Client Company, #Client Full Address	#Client E-mail, #Client Phone Number
---------------------------------------	--------------------------------------

3. Architect

#Contact Full Name, #Contact Full Address	#Contact E-mail, #Contact Phone Number
---	--

4. Calculation (Based on a CAD model)

	Net Floor Area [m ²]																		
0. Ground Floor																			
Exhibition space 01	63.30																		
	<p align="center">Gross Area</p> <table> <tr><td>1: $9.96 * 5.96$</td><td align="right">59.36</td></tr> <tr><td>2: $0.75 * 0.16$</td><td align="right">0.12</td></tr> <tr><td>3: $0.26 * 3.00$</td><td align="right">0.78</td></tr> <tr><td>4: $3.00 * 0.26$</td><td align="right">0.78</td></tr> <tr><td>5: $3.00 * 0.26$</td><td align="right">0.78</td></tr> <tr><td>6: $3.00 * 0.26$</td><td align="right">0.78</td></tr> <tr><td>7: $0.26 * 3.00$</td><td align="right">0.78</td></tr> <tr><td>Gross Area Sum:</td><td align="right">63.38</td></tr> <tr><td>Rounding Correction:</td><td align="right">-0.08</td></tr> </table>	1: $9.96 * 5.96$	59.36	2: $0.75 * 0.16$	0.12	3: $0.26 * 3.00$	0.78	4: $3.00 * 0.26$	0.78	5: $3.00 * 0.26$	0.78	6: $3.00 * 0.26$	0.78	7: $0.26 * 3.00$	0.78	Gross Area Sum:	63.38	Rounding Correction:	-0.08
1: $9.96 * 5.96$	59.36																		
2: $0.75 * 0.16$	0.12																		
3: $0.26 * 3.00$	0.78																		
4: $3.00 * 0.26$	0.78																		
5: $3.00 * 0.26$	0.78																		
6: $3.00 * 0.26$	0.78																		
7: $0.26 * 3.00$	0.78																		
Gross Area Sum:	63.38																		
Rounding Correction:	-0.08																		
Washroom 03	3.50																		
	<p align="center">Gross Area</p> <table> <tr><td>1: $2.25 * 1.27$</td><td align="right">2.86</td></tr> <tr><td>2: $2.25 * 0.41 / 2$</td><td align="right">0.46</td></tr> <tr><td>3: $30.00 * 30.00 * (PI/360) * 4.36 - 2.28 * 29.98 / 2$</td><td align="right">0.07</td></tr> <tr><td>4: $0.75 * 0.20$</td><td align="right">0.15</td></tr> <tr><td>Gross Area Sum:</td><td align="right">3.54</td></tr> <tr><td>Rounding Correction:</td><td align="right">-0.04</td></tr> </table>	1: $2.25 * 1.27$	2.86	2: $2.25 * 0.41 / 2$	0.46	3: $30.00 * 30.00 * (PI/360) * 4.36 - 2.28 * 29.98 / 2$	0.07	4: $0.75 * 0.20$	0.15	Gross Area Sum:	3.54	Rounding Correction:	-0.04						
1: $2.25 * 1.27$	2.86																		
2: $2.25 * 0.41 / 2$	0.46																		
3: $30.00 * 30.00 * (PI/360) * 4.36 - 2.28 * 29.98 / 2$	0.07																		
4: $0.75 * 0.20$	0.15																		
Gross Area Sum:	3.54																		
Rounding Correction:	-0.04																		
WC 04	1.94																		

Morphs can also be displayed in 3D: check **Show 3D Illustration**.

Brutto-Grundfläche + Brutto-Volumen

nach DIN 277 - Jan 2016

1. Bauvorhaben

#Projektname, #PrjNr	#Grundstück Komplette Adresse
----------------------	-------------------------------


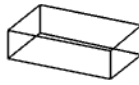

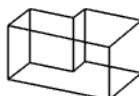
2. Auftraggeber

#Auftraggeber Firma, #Auftraggeber Komplette Adresse	#Auftraggeber E-Mail, #Auftraggeber Telefon
--	---

3. Architekt

#Architekt Kompletter Name, #Architekt Komplette Adresse	#Architekt E-Mail, #Architekt Telefonnummer
--	---

4. Berechnung (basierend auf einem CAD-Modell)

	Höhe [m]	Brutto-Grundfläche R [m ²]	Brutto-Grundfläche S [m ²]	Brutto-Volumen R [m ³]	Brutto-Volumen S [m ³]
0. EG					
Morph-001	1.0000	7.96		7.9555	
		Fläche 1: 3.60 * 2.21 7.96 Summe: 7.96 Rundungskorrektur: 0.00			
Morph-008	2.4996	12.66		31.6385	
		Fläche 1: 4.78 * 2.03 9.70 2: 2.71 * 1.09 2.95 Summe: 12.65 Rundungskorrektur: 0.01			
Summe 0. EG		20.62	0.00	39.5940	0.0000

- For certain list types, you can define the sorting order. Check the items which the sort order should consider (all items will be shown in any case). To change the order, click on an item and drag it up or down to a new position.

Sorting Order:

Story	<input checked="" type="checkbox"/>
Zone category	<input checked="" type="checkbox"/>
Zone number	<input checked="" type="checkbox"/>
Zone ID	<input checked="" type="checkbox"/>

- Use the text and frame settings controls at the bottom of the dialog to customize the list appearance (font and pens).

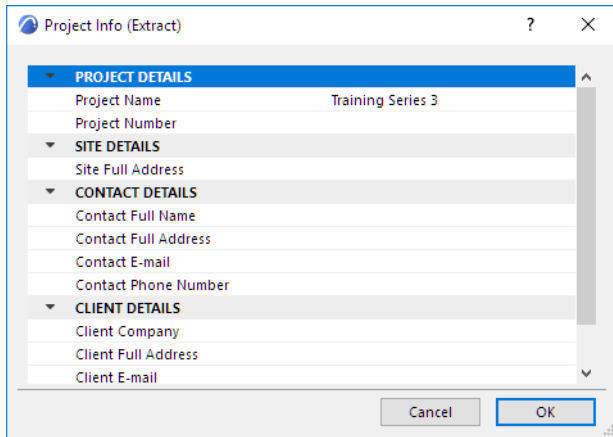
Font Type:

Text Pen: Contour Pen:

Frame Pen: Division Line Pen:

- Certain **Project Info** items can be included in the List Header. Click Project Info to open this dialog, where you can enter or edit these data.

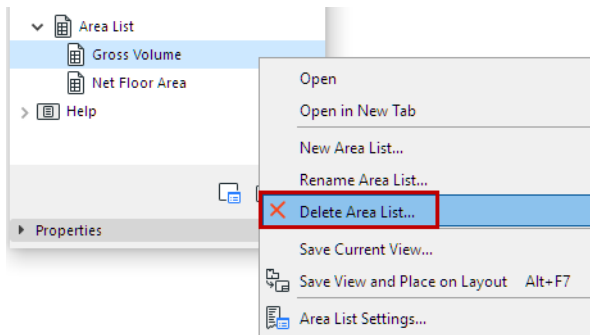
List Header:



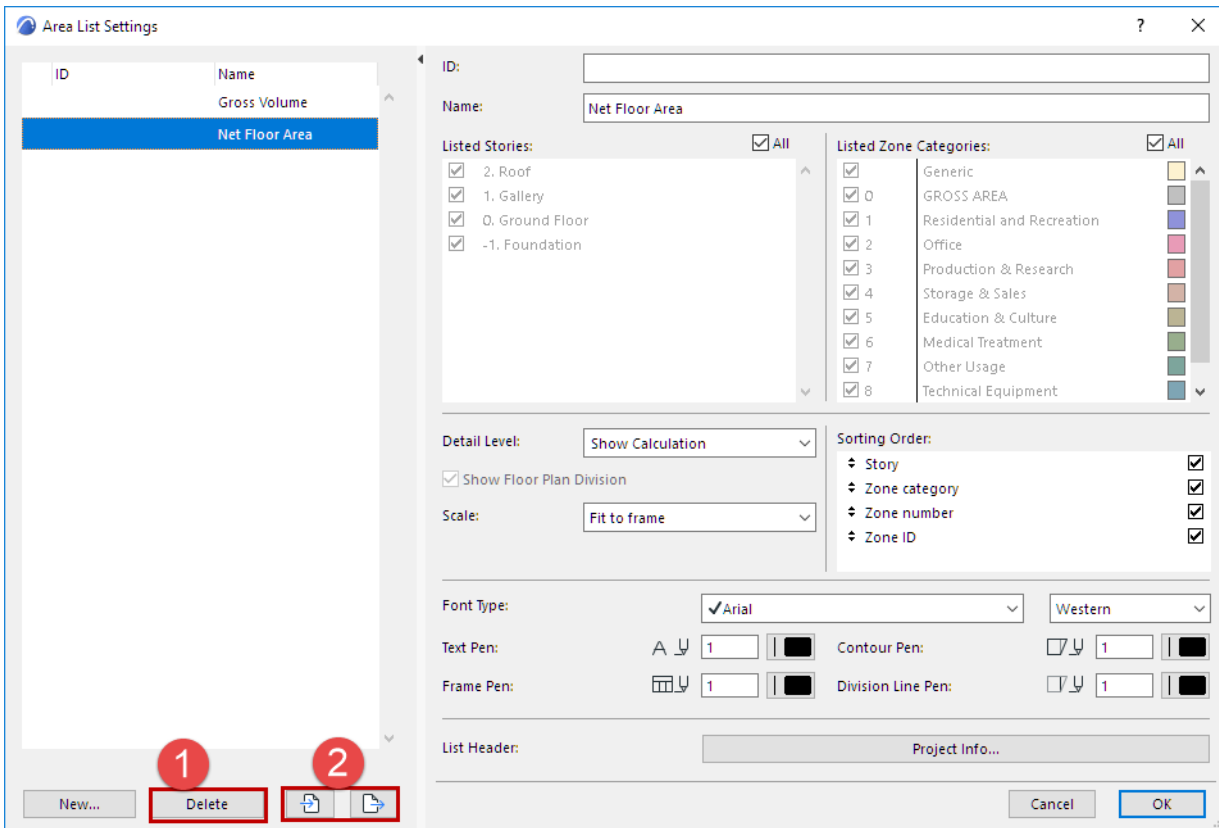
Rename, Delete or Export Area List

Area lists can be renamed, deleted and imported/exported.

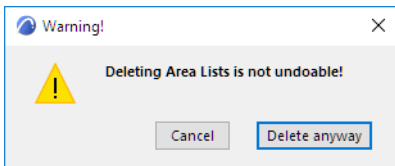
Some of these commands are available from the Navigator item's context menu.



In Area List Settings, commands are available to Delete lists (1) and to Import or Export lists (2) in XML format.



Note: Deleting a list is not undoable.

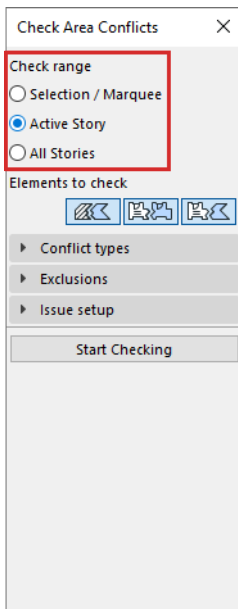


Check for Area Conflicts

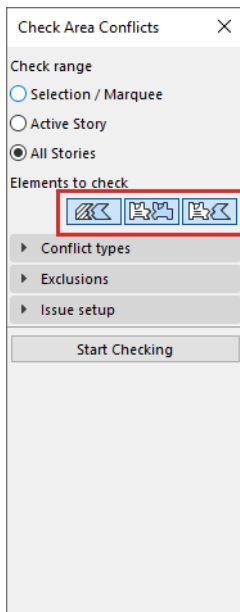
Use this feature to find drafting errors on the Floor Plan - specifically, Zone and Fill polygons that overlap a little, or hang out a little, or have small gaps between them. This way you can easily find/fix modeling problems. Each identified area conflict is added to Issue Organizer.

This function works on Floor Plan only.

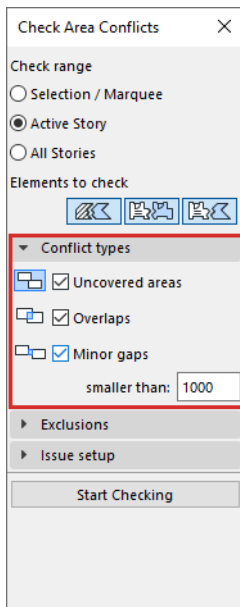
1. Open the Check Area Conflicts palette:
 - **Edit > Check Area Conflicts**, or
 - **Window > Palettes > Check Area Conflicts**
2. **Check range:** Choose which part of the model to check:



- **Selection/Marquee:** Only elements that are currently selected or Marqued.
 - If you activate the Selection/Marquee option, but nothing is selected in the project, the check will consider the Active Story.
 - **Active Story:** All visible elements on the currently active Story.
 - **All Stories:** All visible elements on all Stories.
3. **Elements to check:** Click tool icons of the element pairs to check. Choose at least one pair:
 - Fills vs Fills
 - Zones vs Zones
 - Zones vs Fills



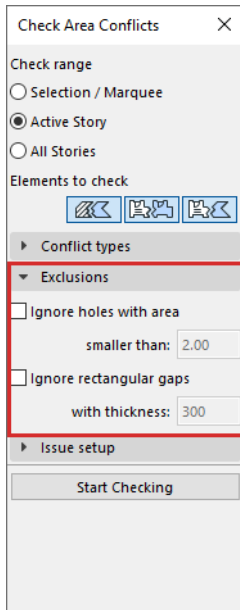
4. Conflict types: Choose one or more conflict types to find:



- **Uncovered areas:** Find areas that should be covered by another polygon, but they are not covered - because one polygon edge extends past the other.
 - For example: a Zone covers a Fill almost entirely, but the Fill edge hangs out, due to a drafting error.
 - How it works: The check signals a conflict only if the larger area covers at least 50% of the smaller area.
- **Overlaps:** Find small overlapping areas between Zones and/or Fills
 - How it works: The check signals a conflict only if one area overlaps the other by less than 50%.
- **Minor gaps:** Find small gaps between Zones and/or Fills.
 - Enter the maximum allowable gap (length). Any gap smaller than this is considered a conflict.

Exclusions

Use these options to ignore certain area conflicts. These are holes or gaps based on intentional modeling - so they should not be handled as errors.



Ignore holes with area

Activate this checkbox to exclude intentional holes (such as chimney hole) from the check.

Enter the area of holes to exclude. Holes smaller than this limit are ignored, and no issue is created.

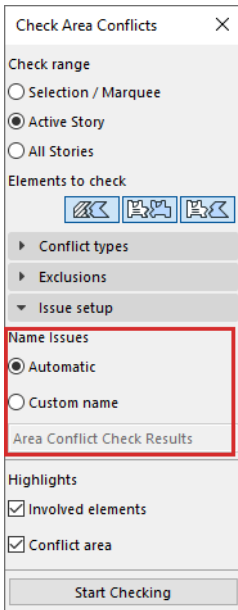
Ignore rectangular gaps

Activate this checkbox to exclude intentional gaps between Zones and/or Fills (e.g. thick walls that are not part of the Zone/Fill area).

Enter the thickness of gaps to exclude (e.g. Wall thickness). Gaps of this size are ignored, and no issue is created.

Issue Setup

Choose how to name the Issues resulting from the check: Automatic naming, or custom naming.



Automatic naming

Names are created based on conflict type and element type.

For example:

- Overlap (Zone-Zone)
- Uncovered area (Fill)
- Minor gap (Zone-Fill)

Issue numbers are added incrementally as a suffix.

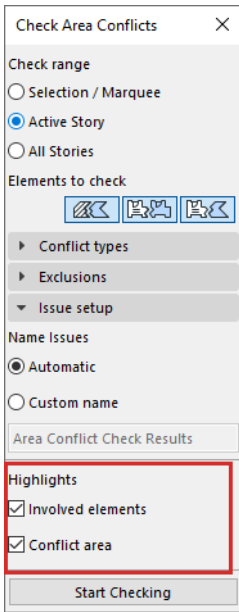
Custom naming

Create single, default custom name for all created Issues. Each Issue will have a unique number.

If needed, you can modify the Issue names manually in Issue Organizer.

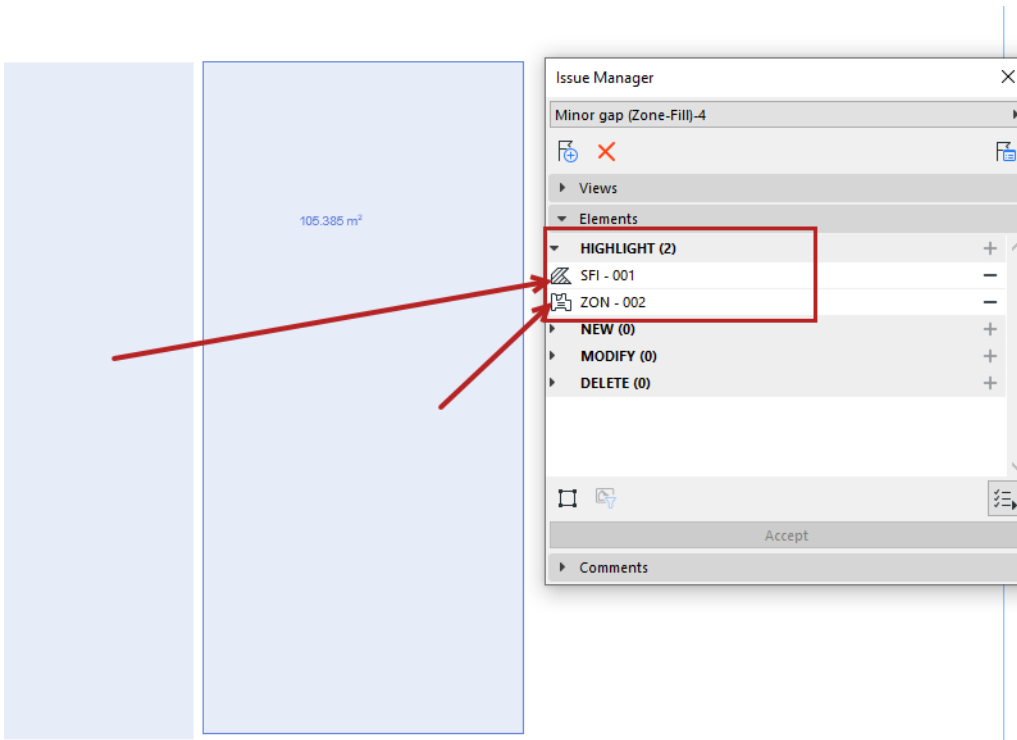
Highlights

Define how to highlight the conflicting elements or areas. Choose one or both of the options:



Involved Elements

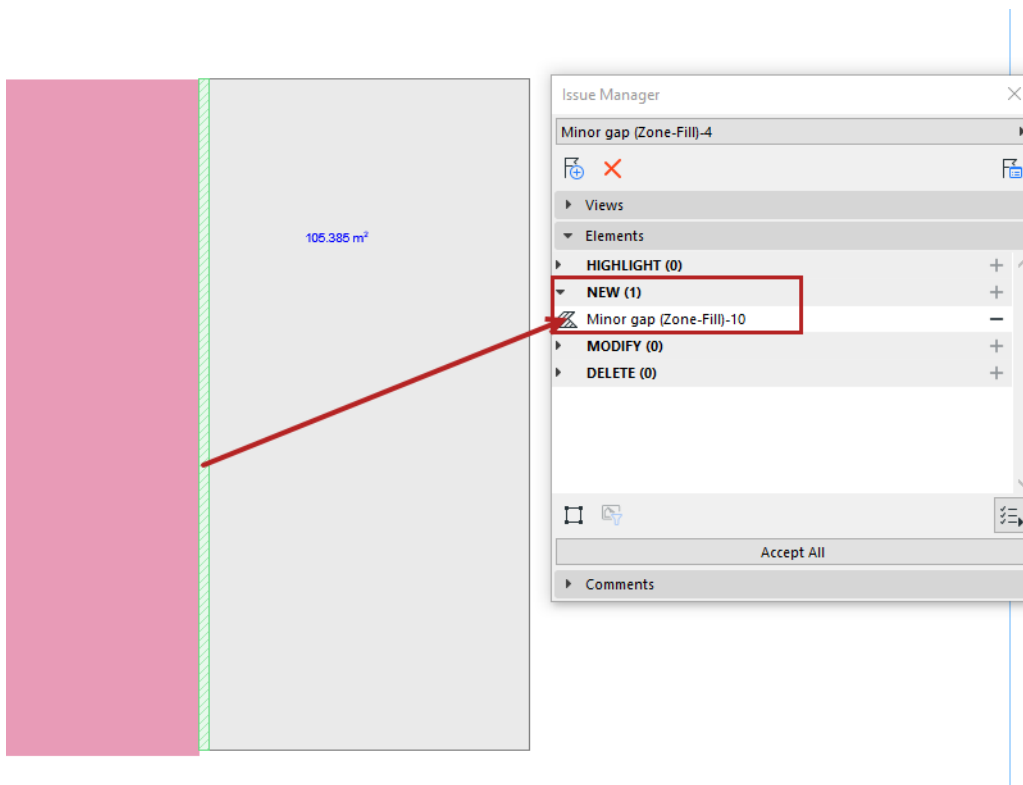
Add the conflicting elements as highlights to the Issue.



Highlight the involved Zone and Fill elements, add elements to Issue

Conflict area

For each conflict area, create a new fill element as part of the Issue. The fill is placed over the conflict area in the floor plan, so you can see it easily.

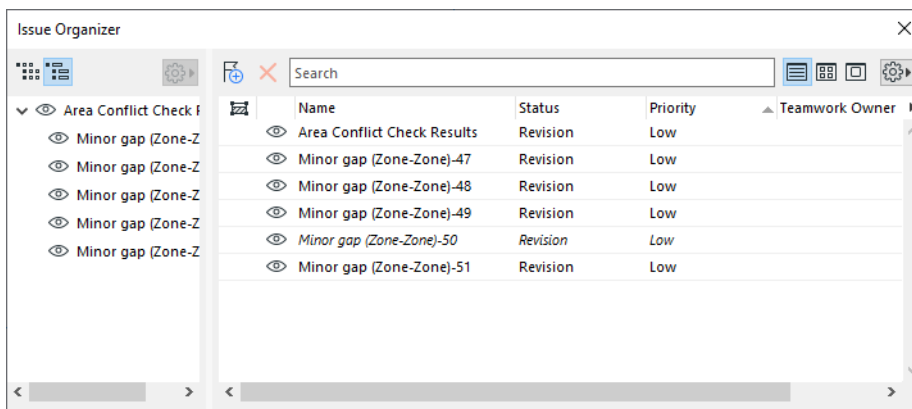


Highlight the conflict area: highlight is added to issue as a Fill element

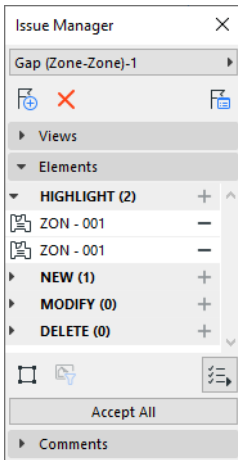
Note: This fill is added to the Issue and is not part of the model. It is visible only as long as the Issue is visible.

5. Click **Start Checking**

6. To see the results, open Issue Organizer (Window > Palettes > Issue Organizer).



- Area Check results belong to a single thread in Issue Organizer. The parent issue is called “Area Conflict Check Results”.
 - After the check, you can click “Undo” to undo all the results in one step.
7. Double-click any Issue to see it in Issue Manager. Zoom to the highlighted elements or areas and adjust the model as needed.



8. Click 'Start Checking' again if needed. After each check, previously resolved Issues are removed from Issue Manager.

Notes for Teamwork users

Permissions required to use the Check Area Conflicts feature:

- Issue - Create/Modify
- Issue- Delete

Permission required to activate the Conflict area highlight:

- Document tool elements

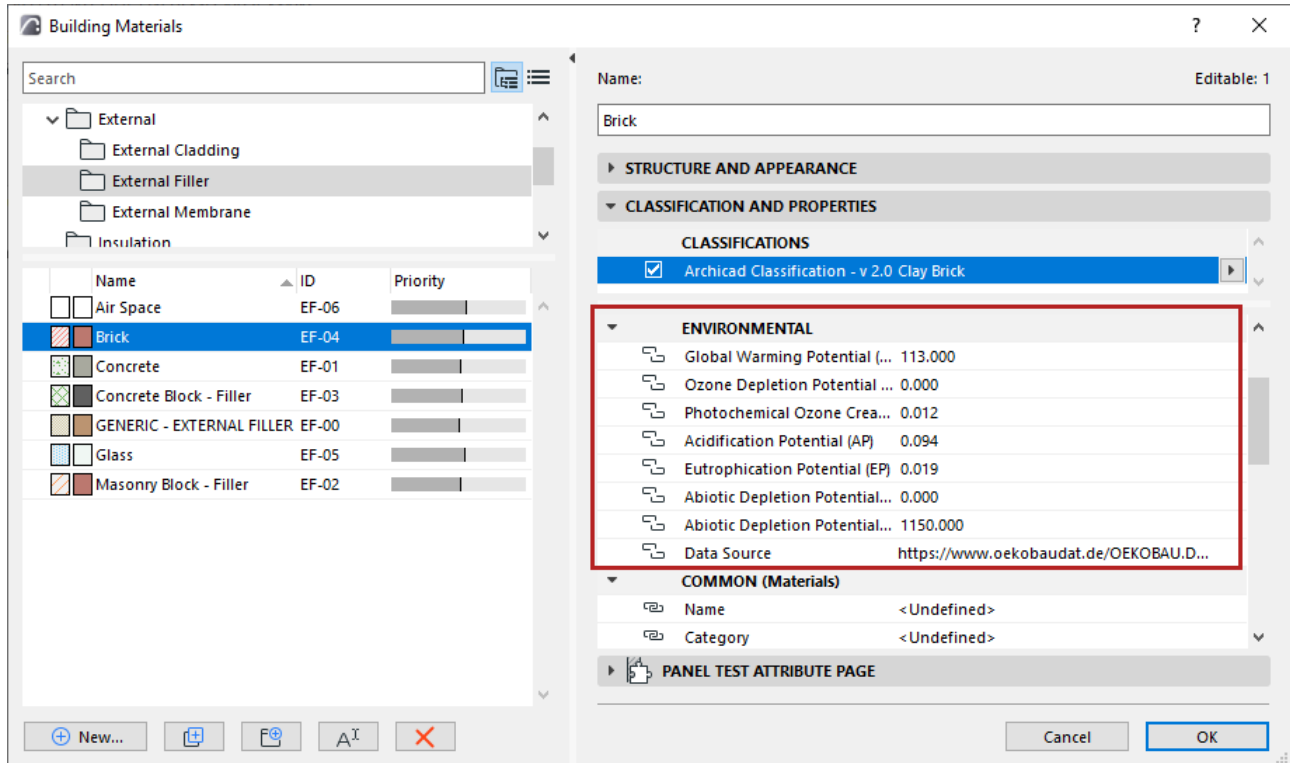
If you need these permissions, please contact your administrator .

Environmental Properties

Building Materials in certain Archicad templates include built-in Environmental properties, corresponding to standard environmental indicators.

Note: Available out-of-the-box for the INT template, plus in language versions based on the INT template. Property values are based on data of the German ÖKOBAUDAT database.

Use these validated properties to document embedded energy and CO2 footprint data for building lifecycle analysis and sustainability reports.



Name	Description
Global Warming Potential (GWP)	<p>Global Warming Potential: the insulating effect of greenhouse gases (GHG) - CO2 and methane - in the atmosphere preventing the earth losing heat gained from the sun. As global temperature rises, it is expected to cause climatic disturbance, desertification, rising sea levels and spread of disease.</p> <p>Unit: kg CO2 eq.</p>

Name	Description
Ozone Depletion Potential (ODP)	<p>Ozone Depletion Potential: depletion of the ozone layer (O₃) in the atmosphere caused by the emission of chemical foaming and cleaning agents allows the passage of greater levels of UV from the sun, causing skin cancer, damage to the immune system and reducing crop yields.</p> <p>Unit: kg R11 eq.</p>
Photochemical Ozone Creation Potential (POCP)	<p>Photochemical Ozone Creation Potential: Creation of ozone in the presence of sunlight, nitrogen oxides and volatile organic compounds. Ozone leads to chemical smog that affect human health, food crops and the ecosystem in general. The effects vary according to geography and climate and are especially problematic in heavily urbanized areas with existing pollution.</p> <p>Unit: kg Ethene eq.</p>
Acidification Potential (AP)	<p>Acidification Potential: emissions, such as sulphur dioxide and nitrogen oxides from manufacturing processes, result in acid rain which harms soil, water supplies, human and animal organisms, and the ecosystem.</p> <p>Unit: kg SO₂ eq.</p>
Eutrophication Potential (EP)	<p>Eutrophication Potential: increased concentrations of nitrates and phosphates can encourage excessive growth of algae and reduce oxygen levels. This increases mortality in aquatic fauna and flora, leads to loss of species dependent on low-nutrient environments, reduces biodiversity and has knock-on effects on non-aquatic animals and humans.</p> <p>Unit: kg Phosphate eq.</p>
Abiotic Depletion Potential (for non-fossil resources) (ADPE)	<p>Abiotic Depletion Potential for non-fossil resources: the over-extraction of minerals and other non-living, non-renewable materials which can lead to exhaustion of natural resources.</p> <p>Unit: kg Sb eq.</p>

Name	Description
Abiotic Depletion Potential (for fossil resources) (ADPF)	Abiotic Depletion Potential for fossil resources: The over-extraction of fossil fuels including all fossil resources. Unit: MJ
Data source	Building life cycle assessment data sheet in the ÖKOBAUDAT database https://www.oekobaudat.de/en.html