

Profile Builder User Guide

Table of Contents

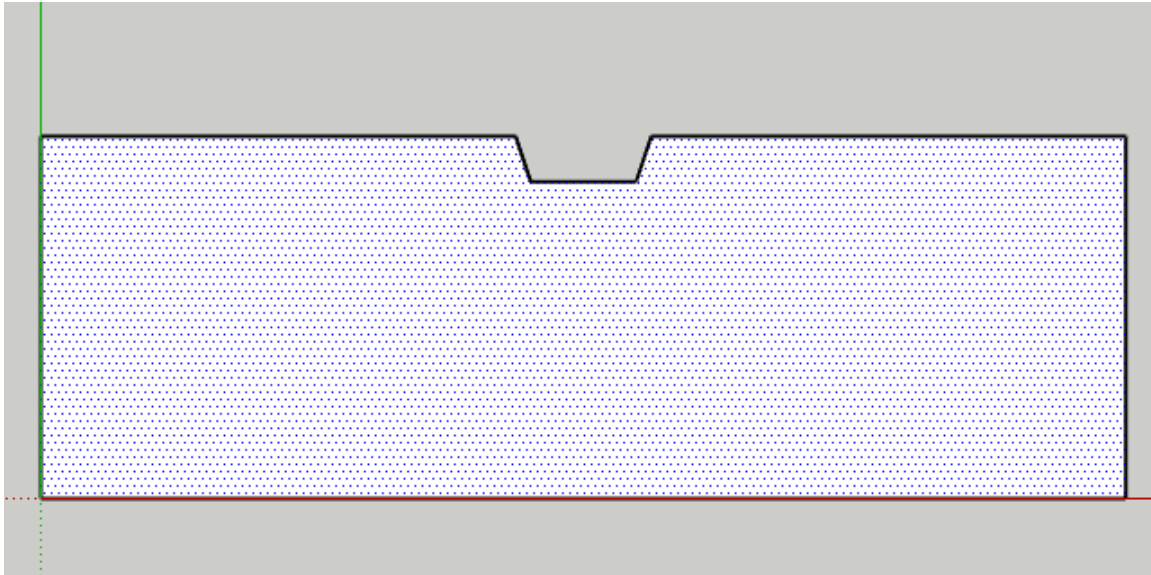
Word did not find any entries for your table of contents.

In your document, select the words to include in the table of contents, and then in the Formatting Palette under Styles, click a heading style. Repeat for each heading that you want to include, and then insert the table of contents in your document. You can also create a table of contents by clicking the Create with Manual Formatting option and then type the entries manually.

Profiles

Create a Profile from a Face

Draw a Face with SketchUp



1. Draw any face in SketchUp at any location.
2. Select the Face.

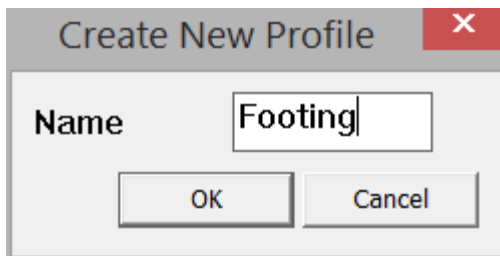
Open the Profile Builder Dialog



Click the 'New Profile' Button

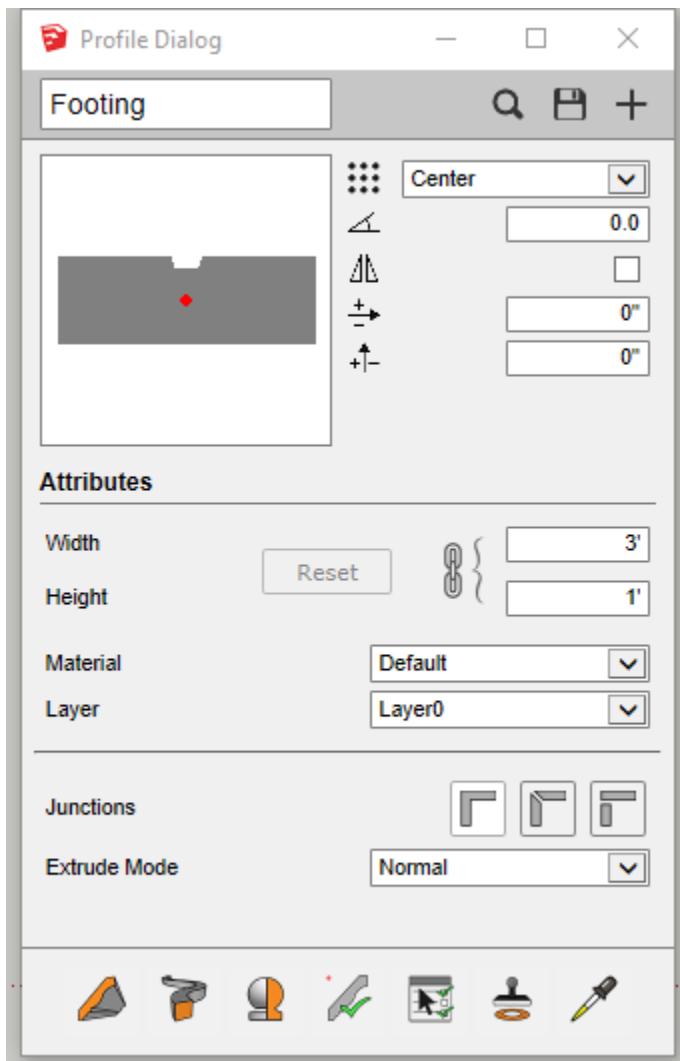


Give the Profile a Name



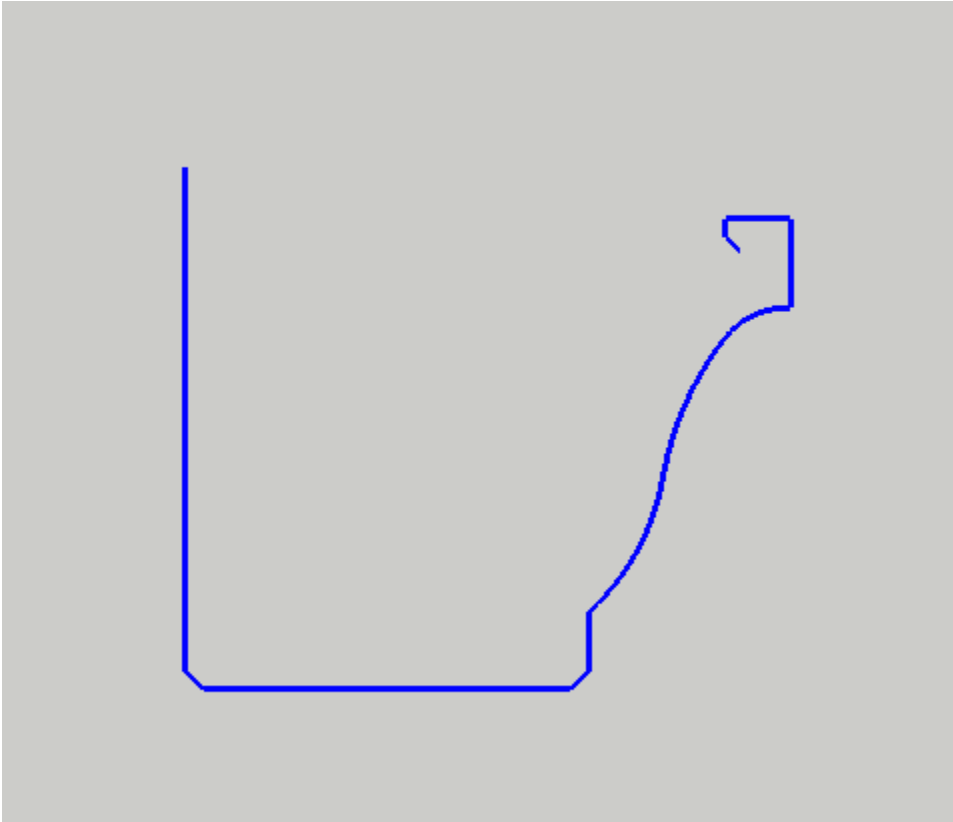
1. Type the name of the Profile and Click OK
2. All profiles must be given a name.

Congratulations! You just made a Profile!



Create a Profile from a Polyline

Draw a Polyline with SketchUp



1. Draw a series of connected lines. The lines must all be located on the XY plane.
2. Select the polyline.

Tip: To control the orientation of the extruded faces, use the Smart-Path Select Tool to select the polyline.

Open the Profile Builder Dialog



Click the 'New Profile' Button

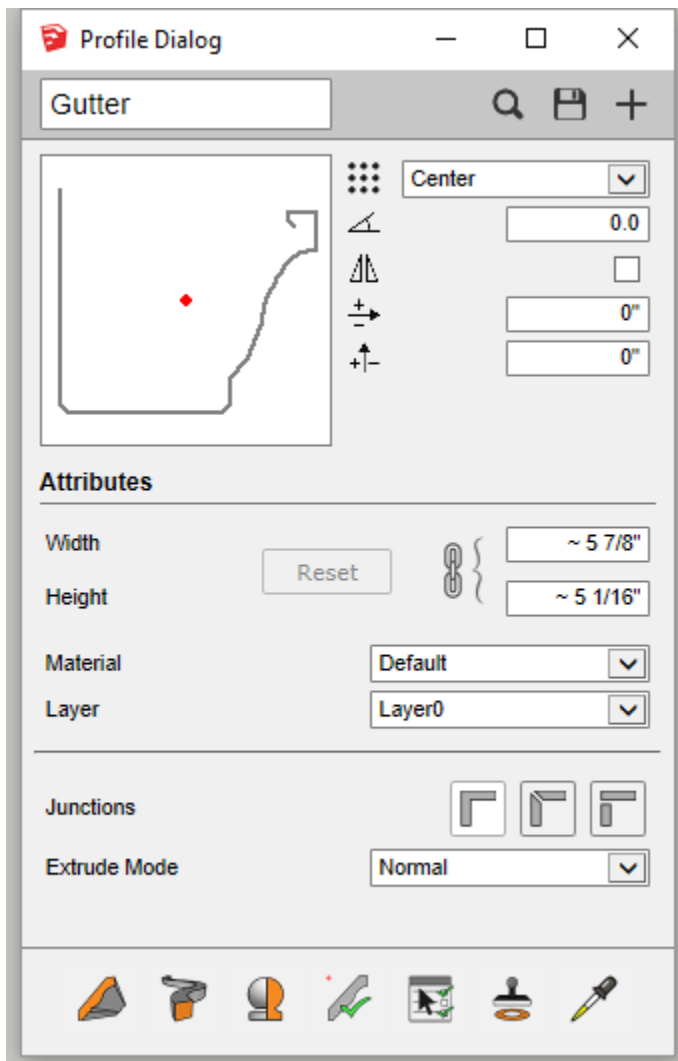


Give the Profile a Name



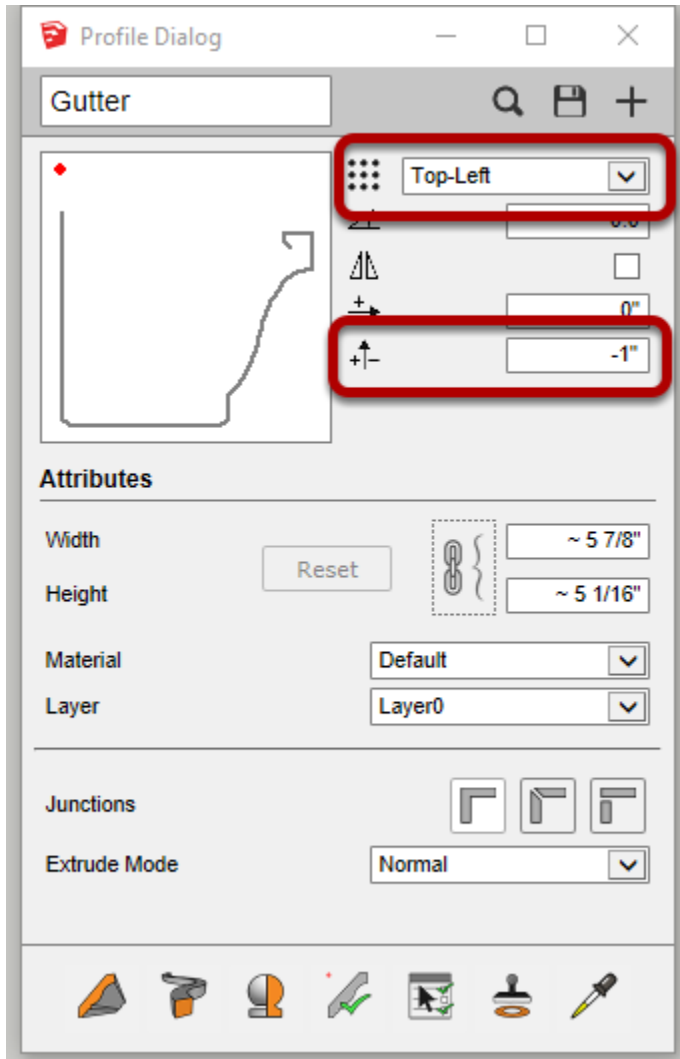
1. Type the name of the Profile and Click OK
2. All profiles must be given a name.

Congratulations! You just made a Polyline Profile!



Saving Profiles

Choose the Default Profile Attributes

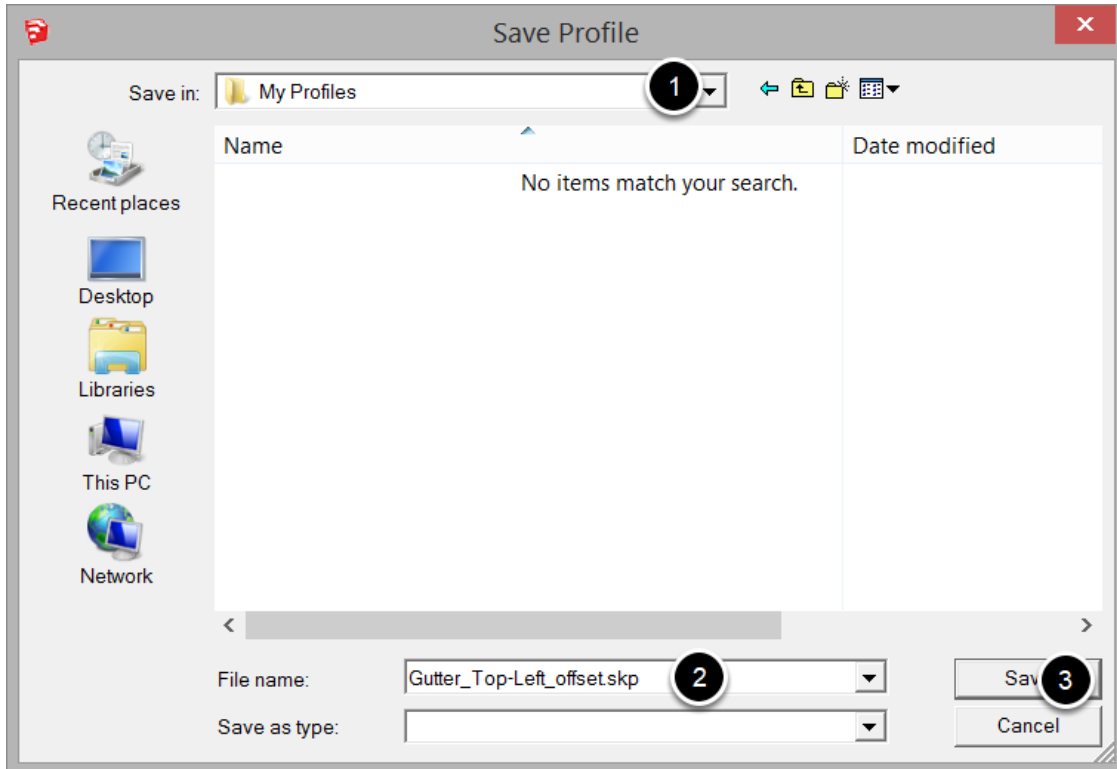


For this profile, the placement point was changed to 'Top-Left' and the Y Offset was changed to -1"

Click the 'Save Profile' Button

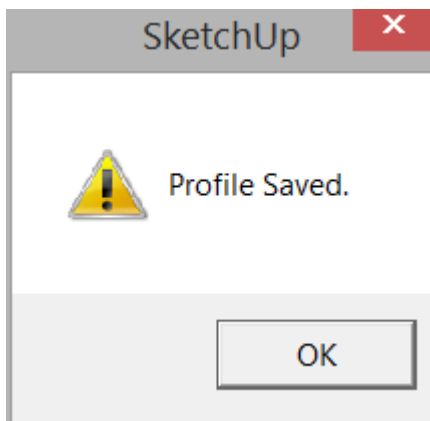


Choose where to save the Profile



1. Select a Folder to save the Profile.
2. Choose a Name for the Profile. You may wish to use a name that also includes important attributes of the Profile.
3. Click 'Save' to save the profile as a SKP file.

Congratulations! You just saved a Profile!

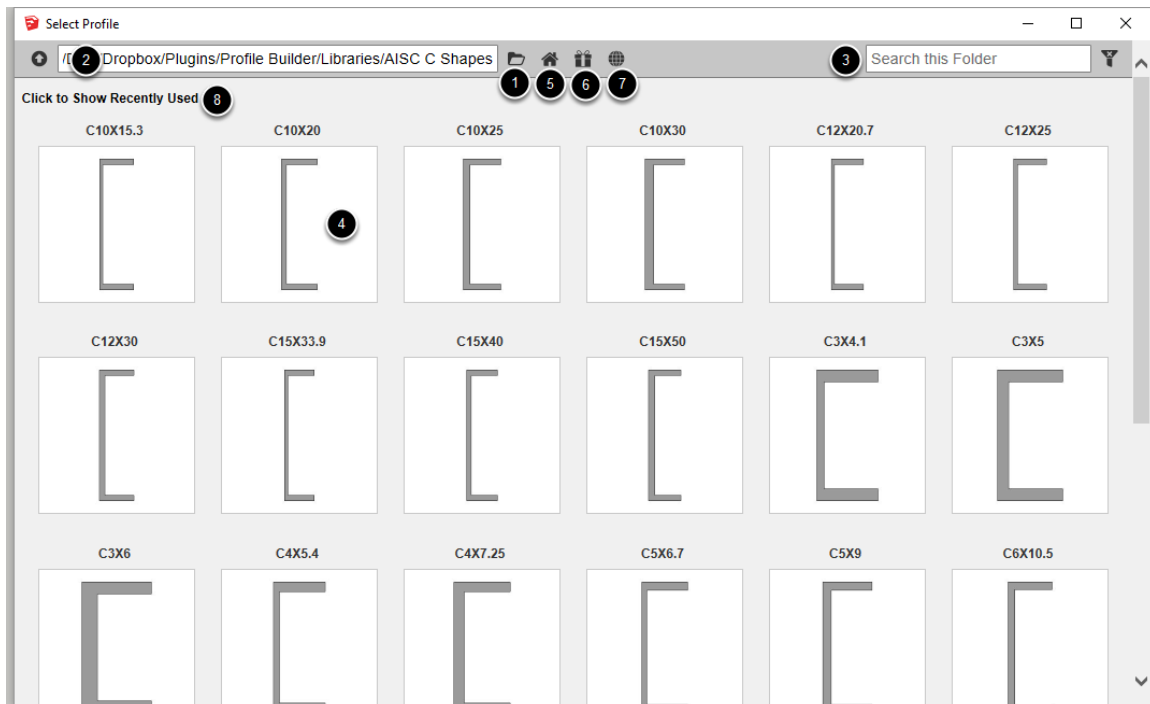


Loading Profiles

Click the Profile Browser Button



Finding a Profile



1. Click the Open Library button to choose a new library folder.
2. The current folder is shown at the top left.
3. Enter text to filter results.
4. Click a profile to load it.
5. Click the Home button to jump to your 'Home' Profile folder. This folder can be set using the Profile Builder Preferences Dialog.
6. Click this button to jump to the sample profiles that are included with the Profile Builder plugin.
7. Click this button to open the Profile Builder website where you can access more free profile libraries.
8. Click this area to expand and show your recently used Profiles.

Loading Other Components as Profiles

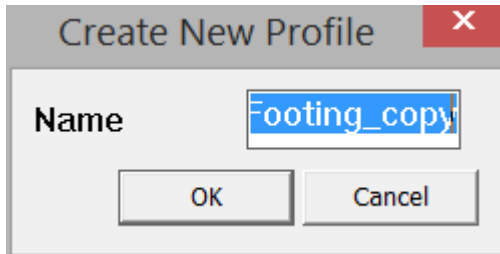
Any SKP file that meets the following criteria can be loaded as a Profile into Profile Builder.

1. The file must contain a face that is not inside a group or component.
- OR
2. The file must contain a polyline that is not inside a group or component.

If more than one face or polyline is found, unexpected results or errors may occur.

Copying a Profile

Create a Copy of a Profile



1. Open the Profile Builder Dialog
2. Ensure that no SketchUp entities are selected.
3. Click the 'New Profile' Button
4. A copy of the current profile will be created. Give the profile a name.

Managing Profiles

Profiles are nothing more than SKP files that contain a single face or polyline. If the SKP file was created from Profile Builder it will also contain some additional meta-data about the other attributes of the profile (orientation, layer, material, etc.)

Editing Saved Profile Shapes

Since profiles are stored as SKP files, to edit the shape, just open the SKP file, make the changes using the SketchUp drawing tools and then re-save.

Tip: Be sure that the SKP only has one face or polyline and it is not inside any group or component.

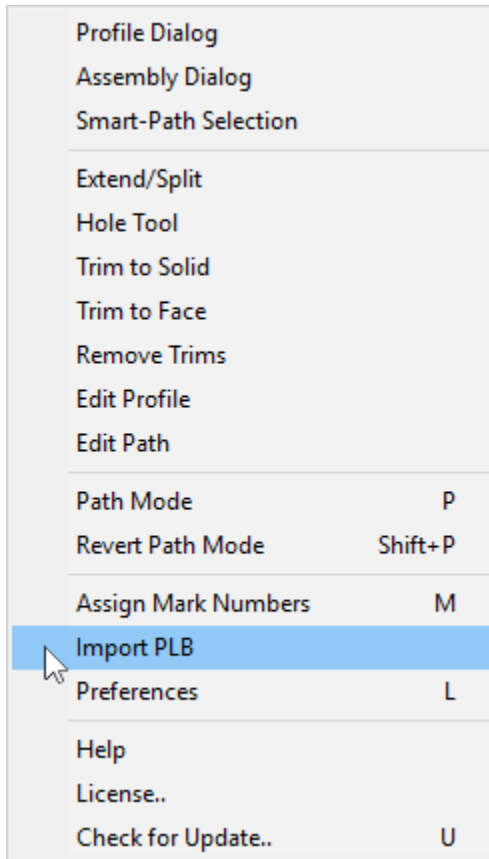
Organizing Profiles

We recommend that your profiles be stored in an organized system of folders on your local computer or network, similar to how you might store your SketchUp Component Library.

Profile folders can be stored at any location as long as you have security permissions to write files to that particular folder.

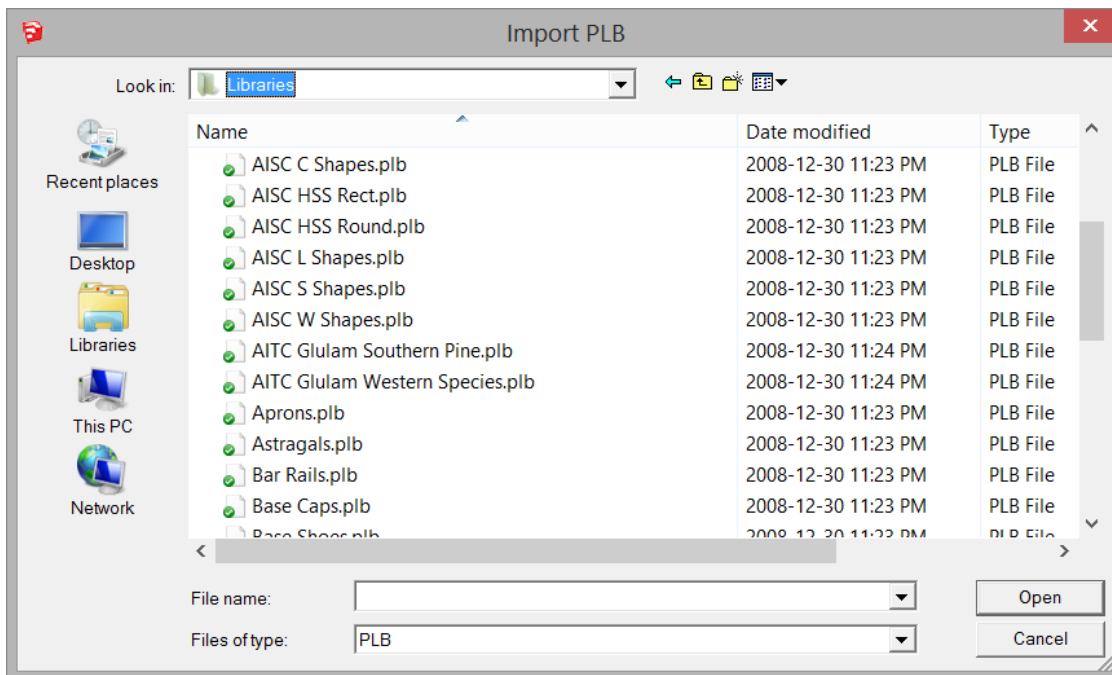
Importing PLB files

Select Import PLB File from the Profile Builder Menu



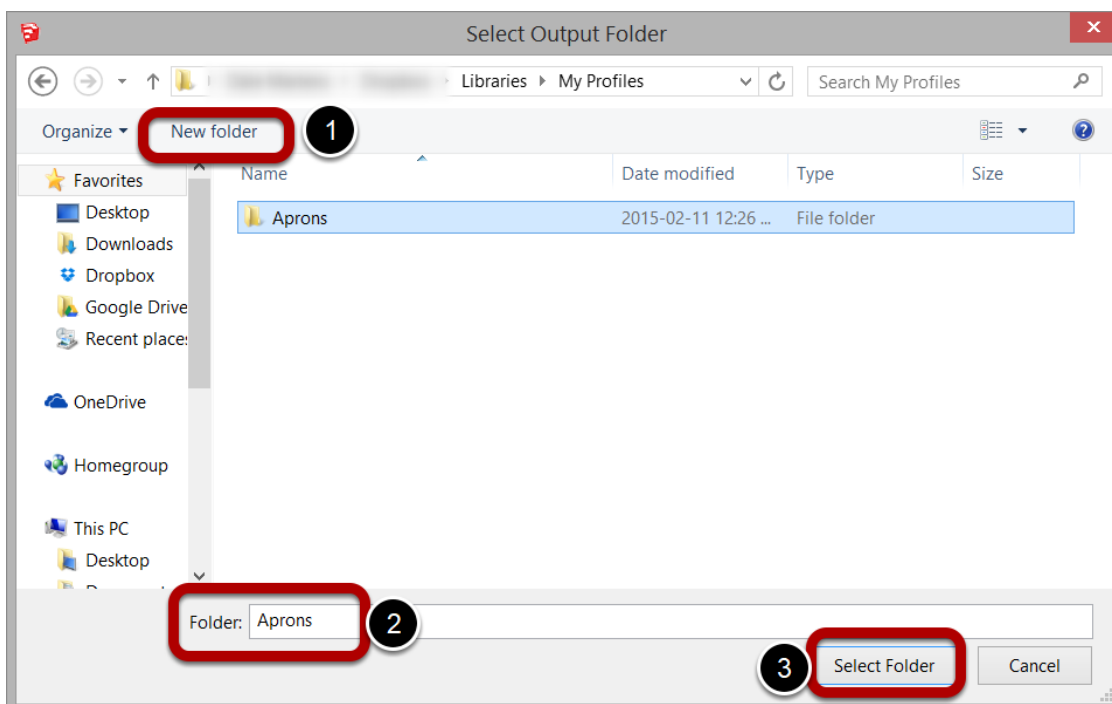
PLB files were used in the original version of Profile Builder to store a library of Profiles. These files are now obsolete but they can be imported and have the profiles converted to SKP files.

Select the PLB file to import



The PLB file must have been created using Profile Builder version 1.

Select the Folder to Output the Profiles



Create a new folder if necessary and give it a name.

A new SKP file will be created for each profile in the PLB file.

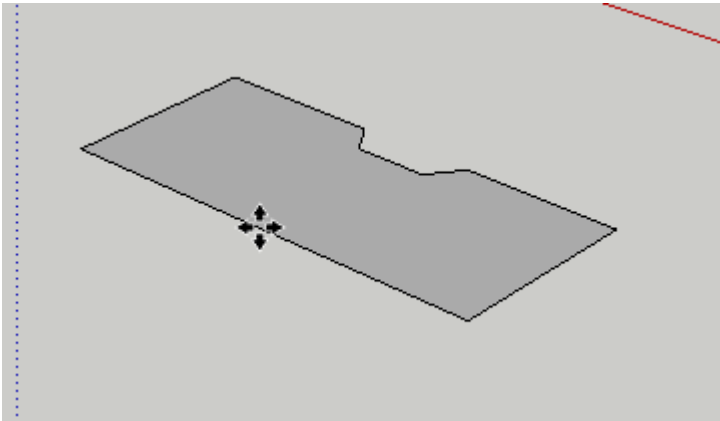
Stamp Profile Tool

Launch the Tool



1. Open the Profile Builder dialog.
2. Click the Stamp Profile Tool Button

Stamp the Selected Profile



After launching the tool, a component will be created of the current profile.

Click anywhere in your model to place the component.

The component will automatically align to any face in your model.

Profile Members

Build Tool

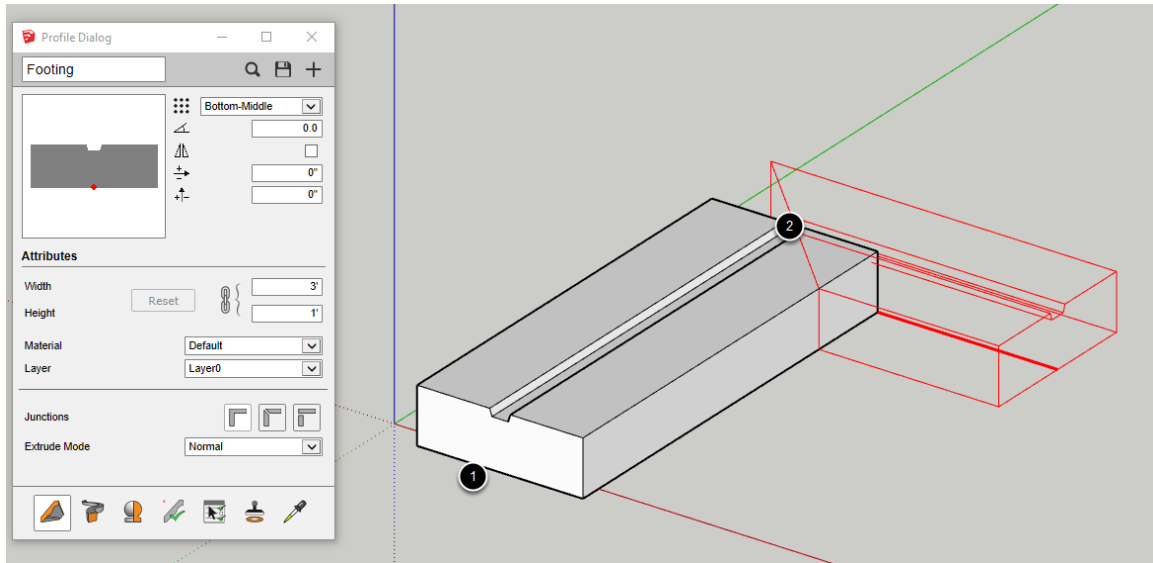
'Profile Members' are parametric smart extrusions. They are the term we use for objects that are created by extruding or sweeping a Profile along a Path.

Launching the Build Tool



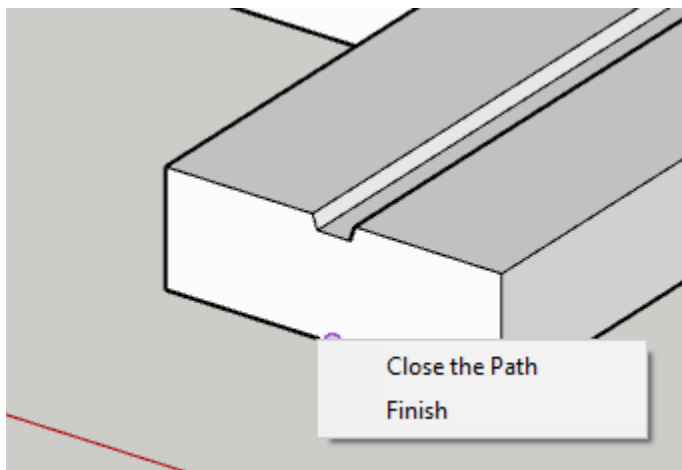
Open the Profile Builder Dialog.
Then, click the Build Tool.

Click Points to Build a Profile Member



1. Click a point in the model to define the start point of the member.
2. Continue clicking points to define the path of the member.

Complete the Profile Member

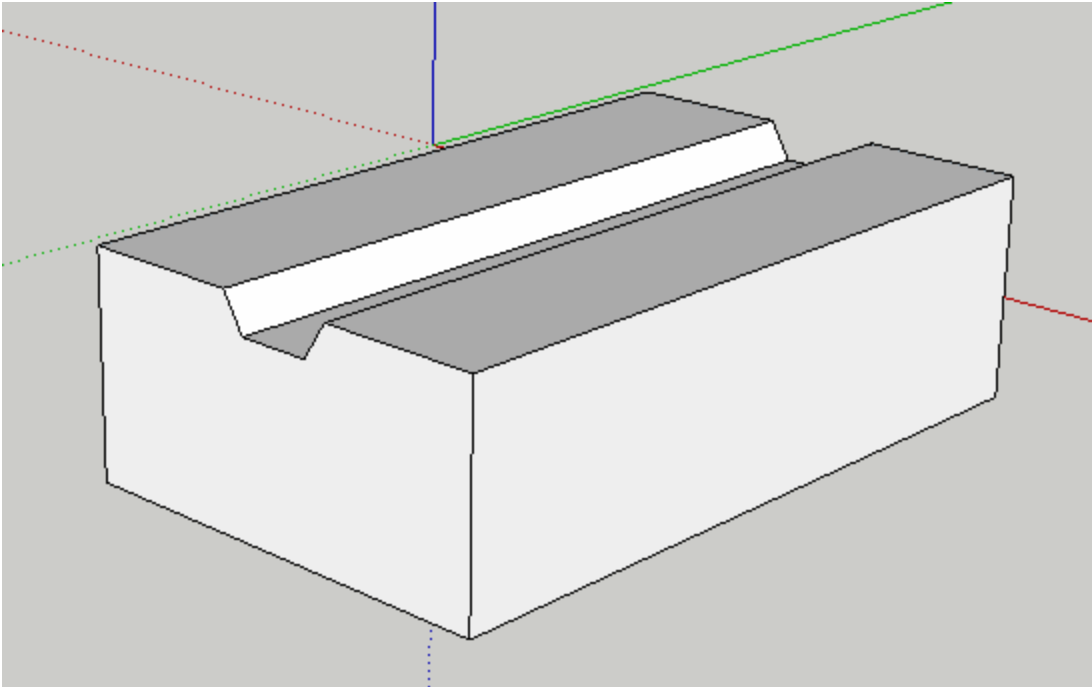


Complete the member by:

1. Pressing ESC, ENTER, or RETURN
- OR
2. Right-click and choose 'Finish' (shown above)
- OR

3. Creating a closed path for the member.

Congratulations! You just built a Profile Member!



Modifier Keys

Arrows Keys = Lock Axis

SHIFT = Lock Inference

HOME = Cycle Placement point

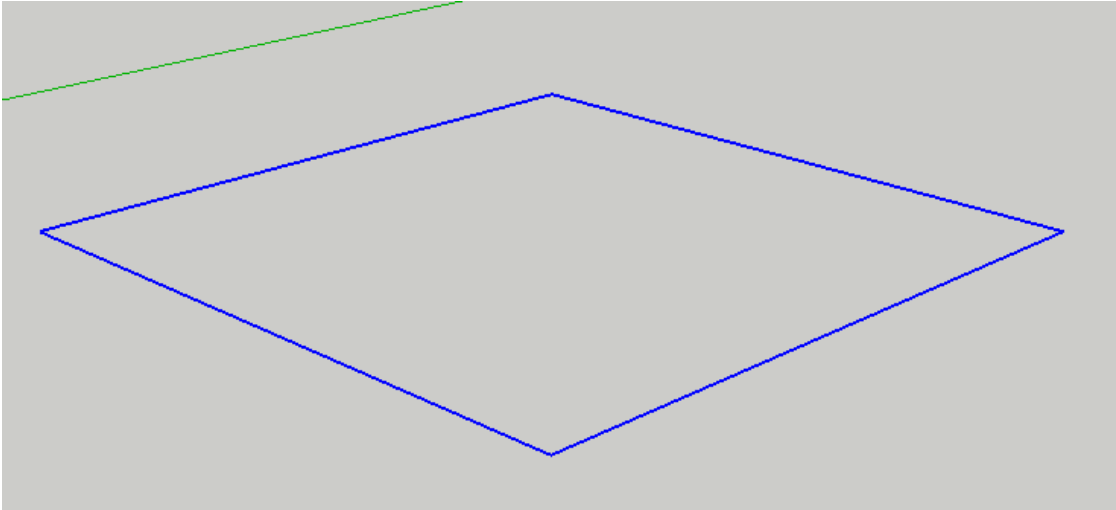
END = Cycle Rotation in 45 degree increments

BACKSPACE / DELETE = Undo the last clicked path point

You can also enter values in the Measurements box to draw precisely.

Build Along Path

Select a Path



First, select a path.

You can select a series of edges using SketchUp's Select Tool.

OR

You can select a face to build along the edges of the face.

OR

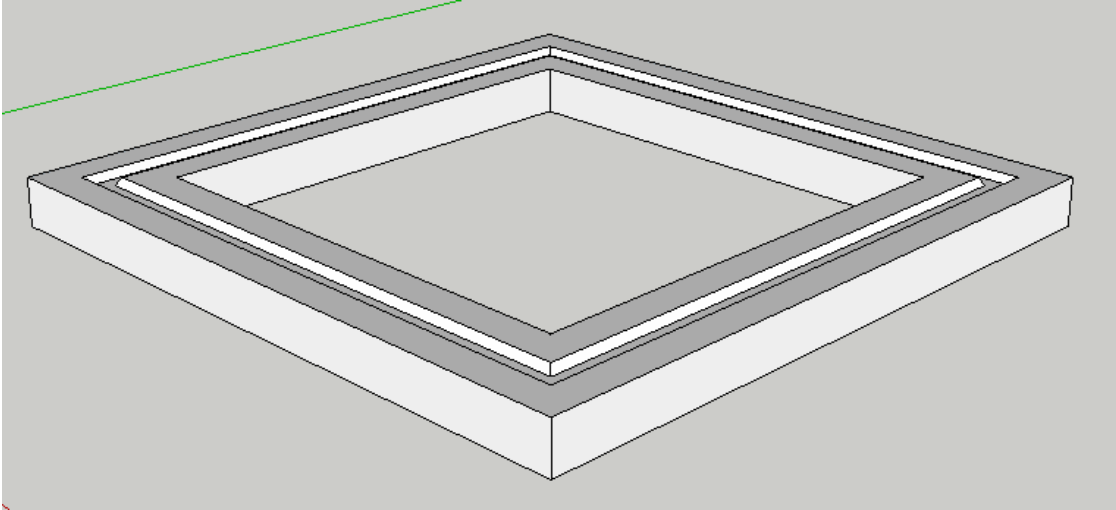
You can use the Profile Builder [Smart-Path select tool](#) to select the path AND control the direction of the path (recommended).

Launch the Tool



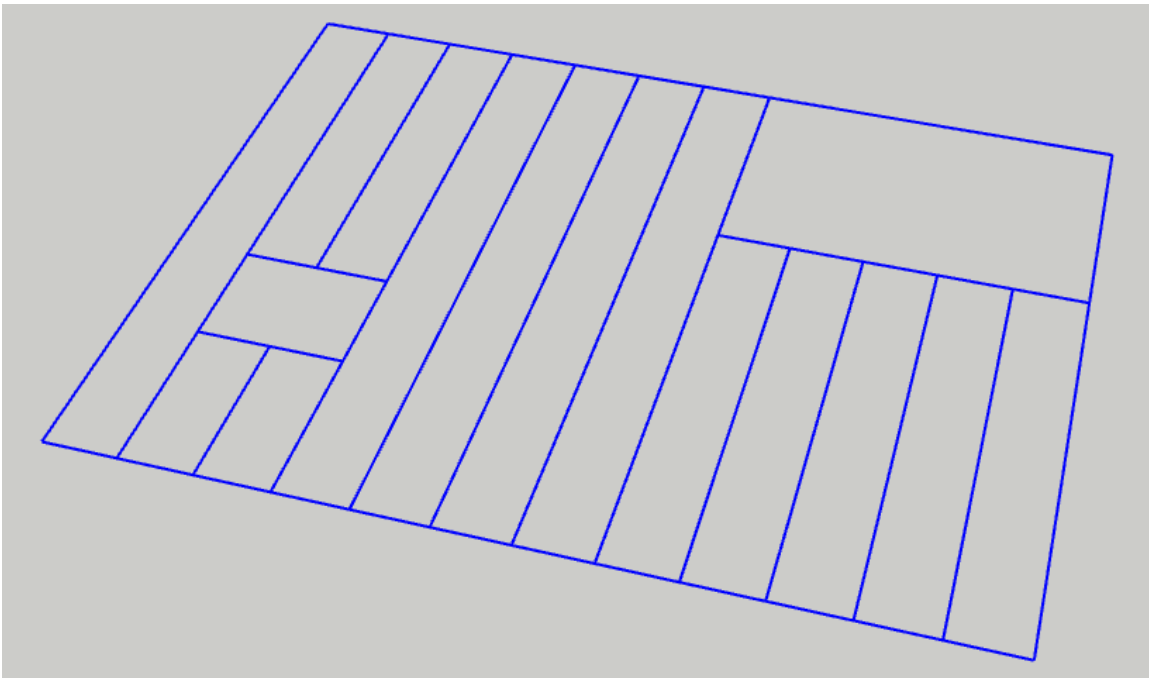
1. Open the Profile Builder Dialog.
2. Click the 'Build Along Path' button.

Congratulations! You just built a Profile Member along a Path!



The currently selected Profile will be extruded along the selected path. Remember to use the Smart-Path select tool if you need to control the direction of the path. This is especially important if the profile is not symmetrical.

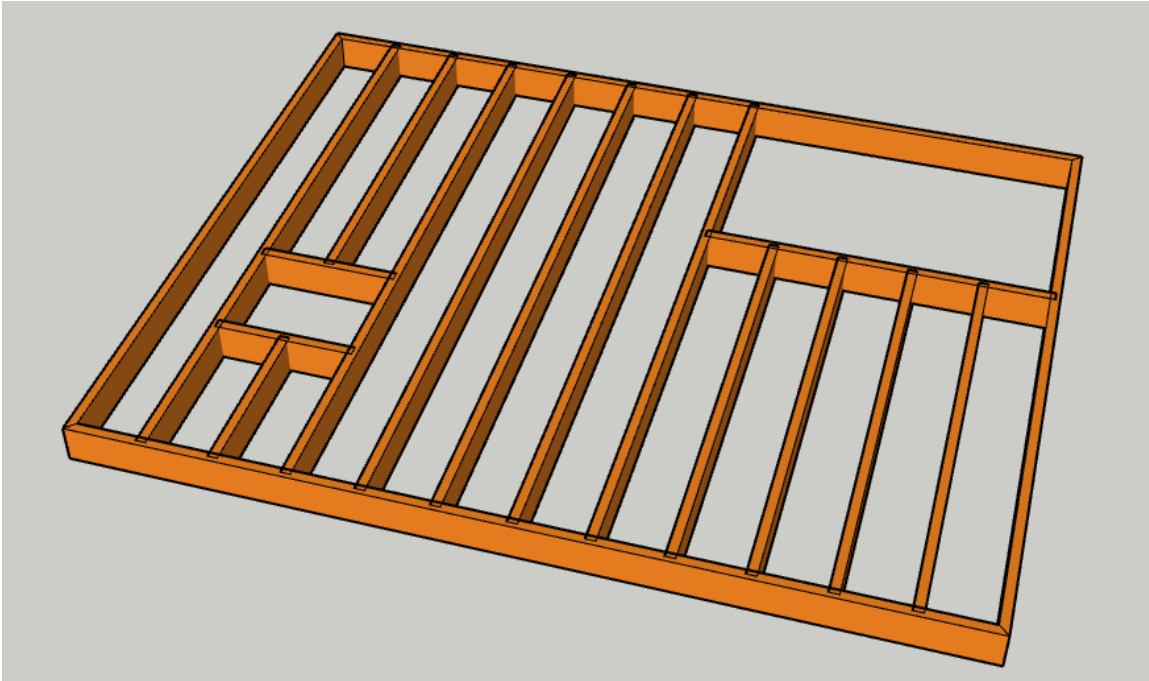
Build Along Multiple Paths



You can select any number of continuous or disconnected edges before launching this tool.

Profile Builder will search the selected edges for various paths and then create a Profile Member along each path that it finds.

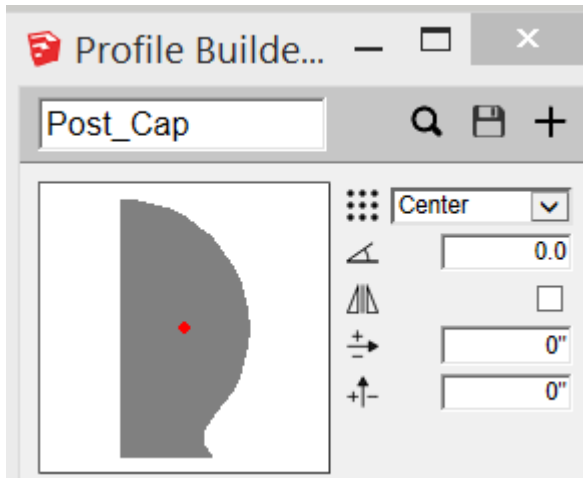
Congratulations! You built Profile Members along Multiple Paths!



This tool can be used to generate 3D framing from line drawings extremely fast!

Revolve a Profile

Select a Profile



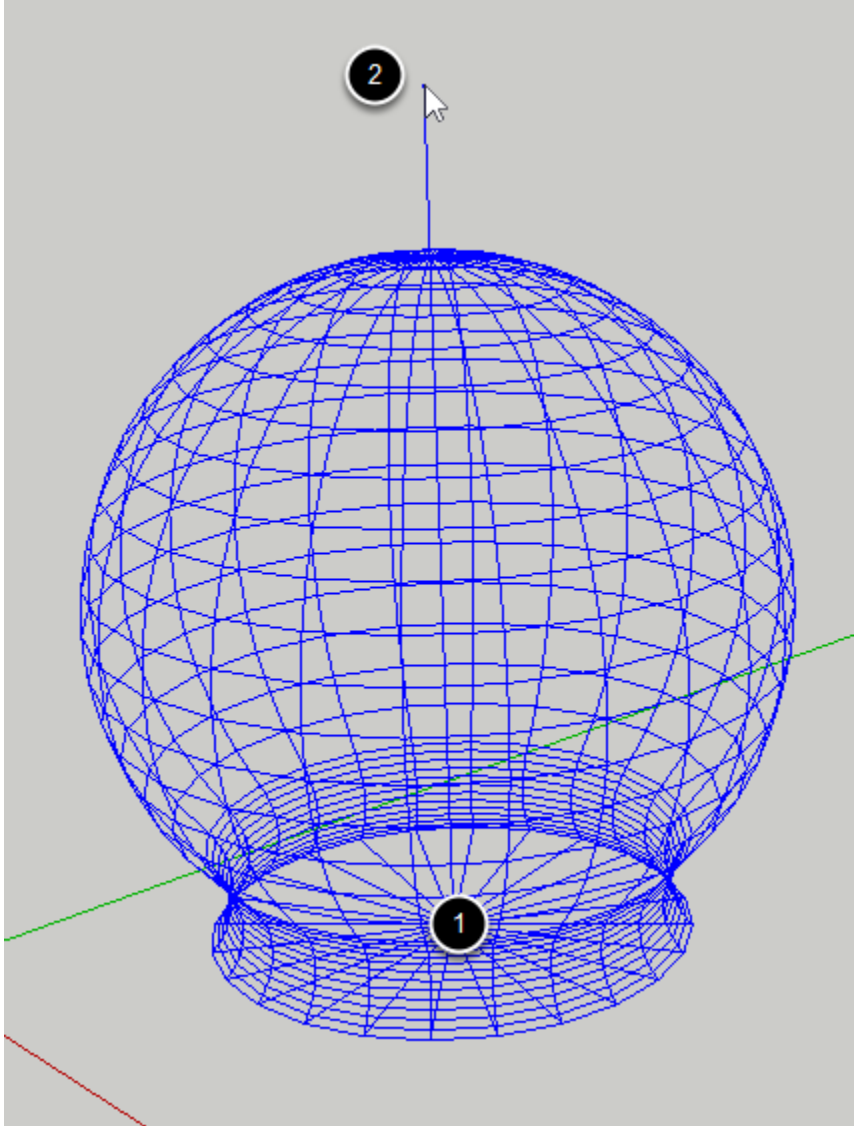
Create or load a Profile that you would like to revolve around an axis.

Launch the Revolve Profile Tool



Click the 'Revolve Profile' button in the Profile Builder dialog.

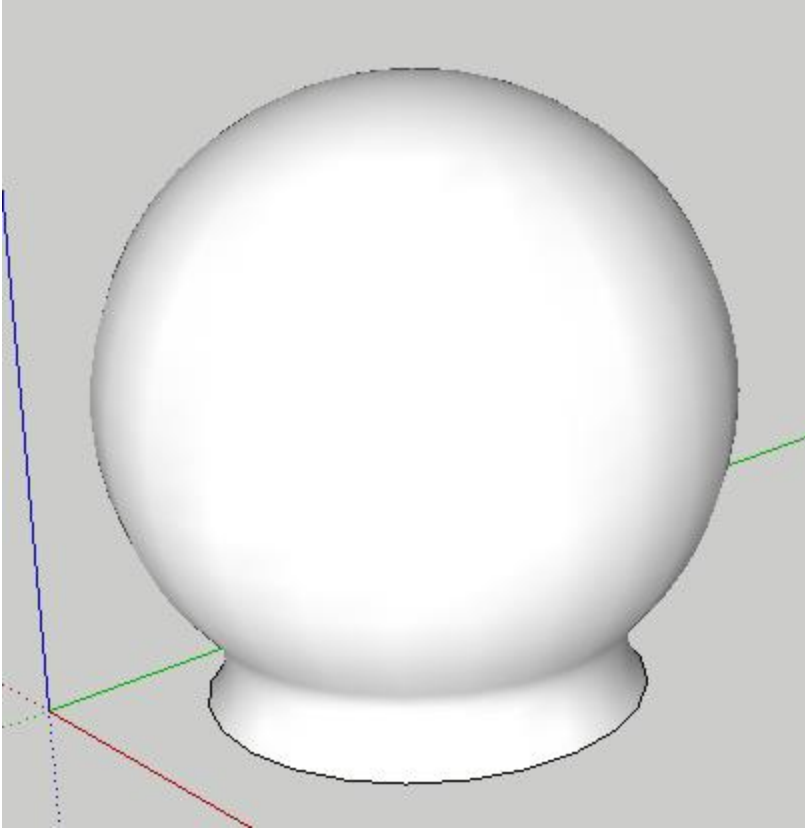
Define the Revolve Axis



1. Click anywhere in the SketchUp model to set the origin of the revolve axis.
2. Click again to define the direction of the axis.

TIP: Use the Measurements box to set the number of sides.

Congratulations! You just revolved a Profile!



Note: No matter what settings you are using for the Profile, the profile will always be revolved about the bottom left corner of the Profile bounding box.

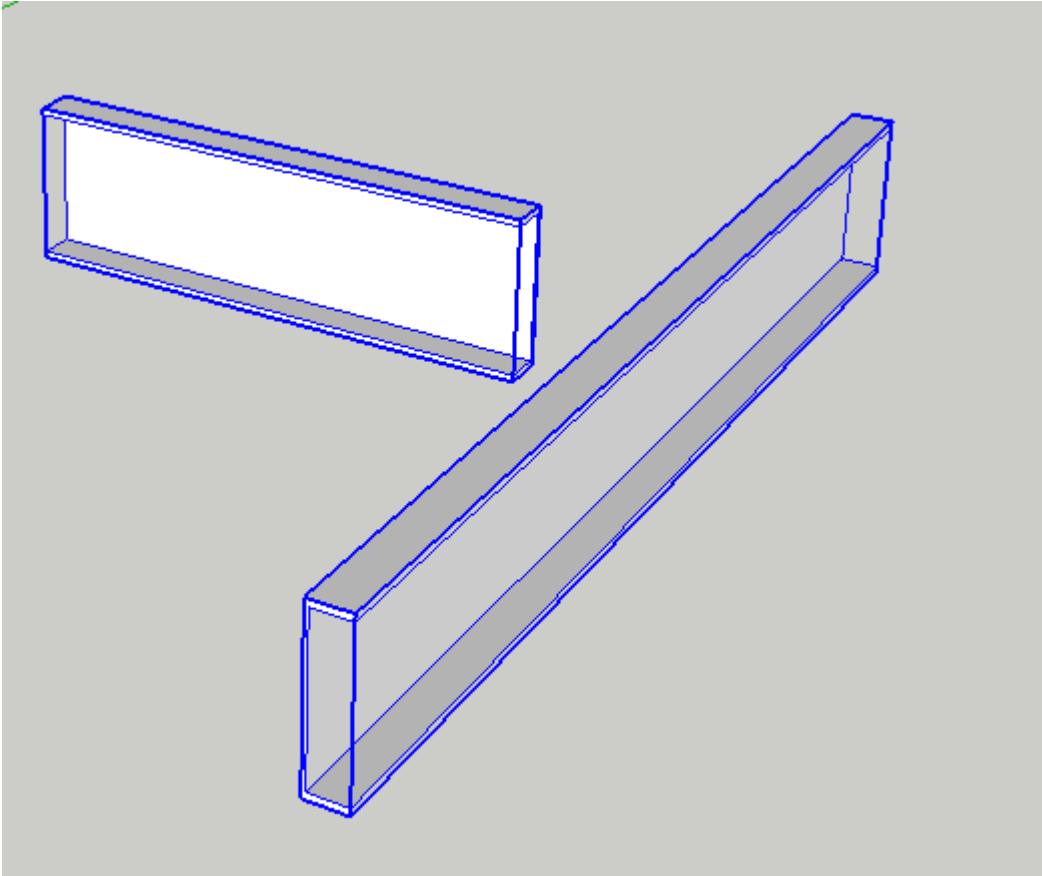
About Revolved Profile Members

Revolved Profile Members behave just like other Profile Members except for one thing:

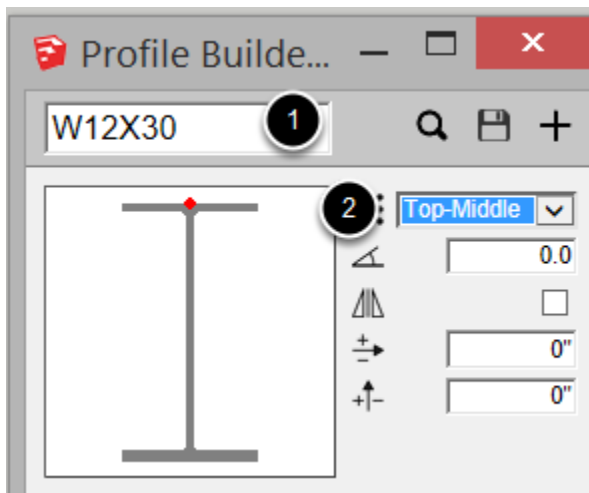
You cannot edit the path of a Revolved Profile Member.

Apply Attributes to a Profile Member

Select Profile Member(s) to Edit



Set the Attributes



Open the Profile Builder Dialog and set the attributes.
In this example:

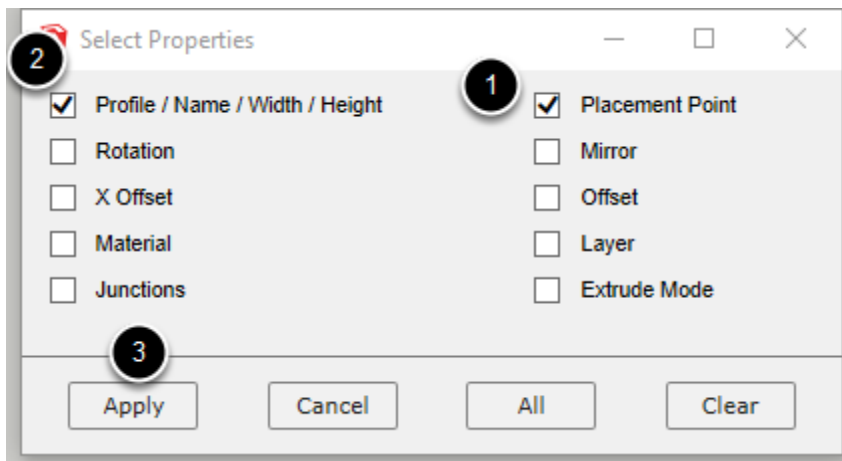
1. The Profile was changed to a W12x30
2. The Placement Point was changed to Top-Middle.

Click the Apply Attributes Button



Click the Apply Attributes Button.

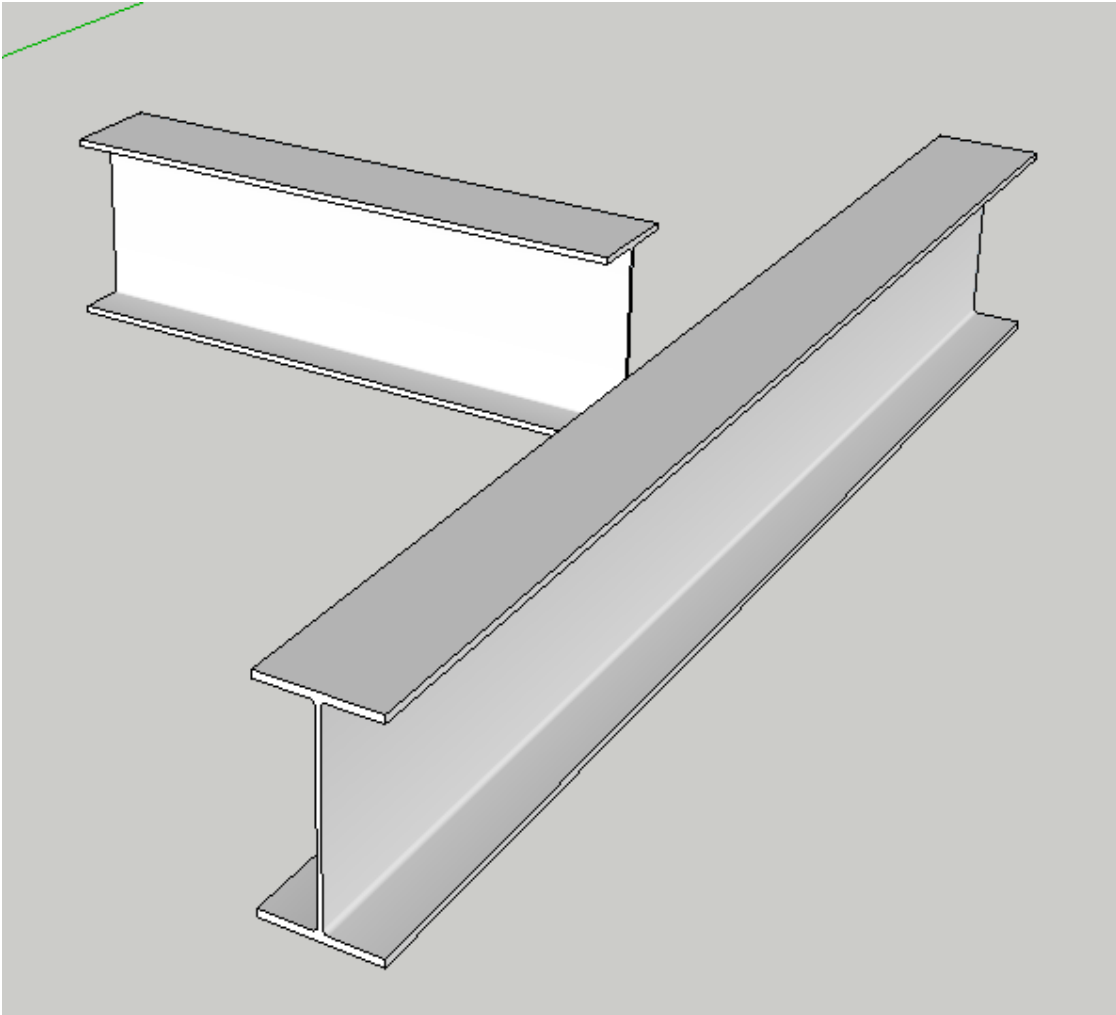
Select the Attributes to Apply



Check the boxes for the attributes that you wish to apply to the selected Profile Members. In this example:

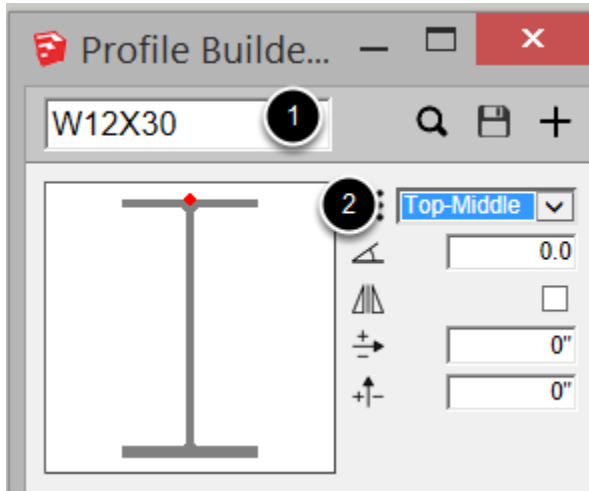
1. The Placement Point is checked.
2. The Profile is checked (this will also modify the member name, width, and height)
3. Click the Apply button to apply the checked attributes to the selected Profile Members.

Congratulations! You just applied attributes to a Profile Member!



Select by Attributes

Set the Attributes



Open the Profile Builder Dialog and set the attributes.
In this example:

1. The Profile was changed to a W12x30.
2. The Placement Point was changed to Top-Middle.

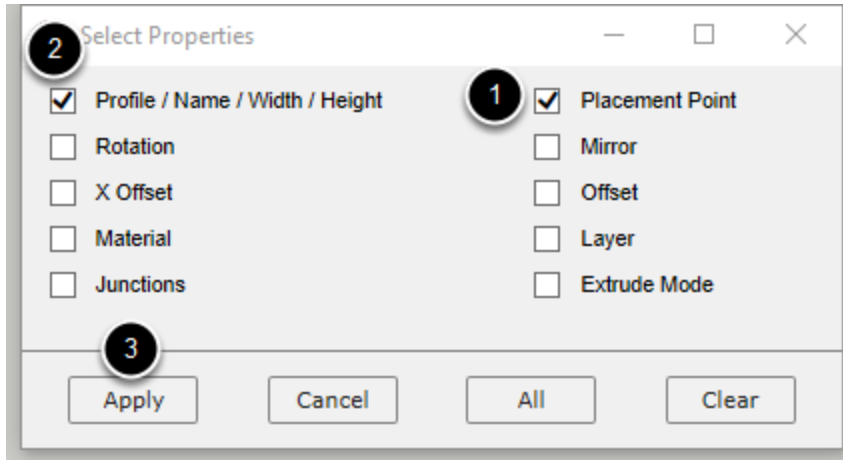
Click the Select by Attributes Button



Click the 'Select by Attributes' Button.

TIP: First ensure that there are Profile Members within the current model context (active group or component).

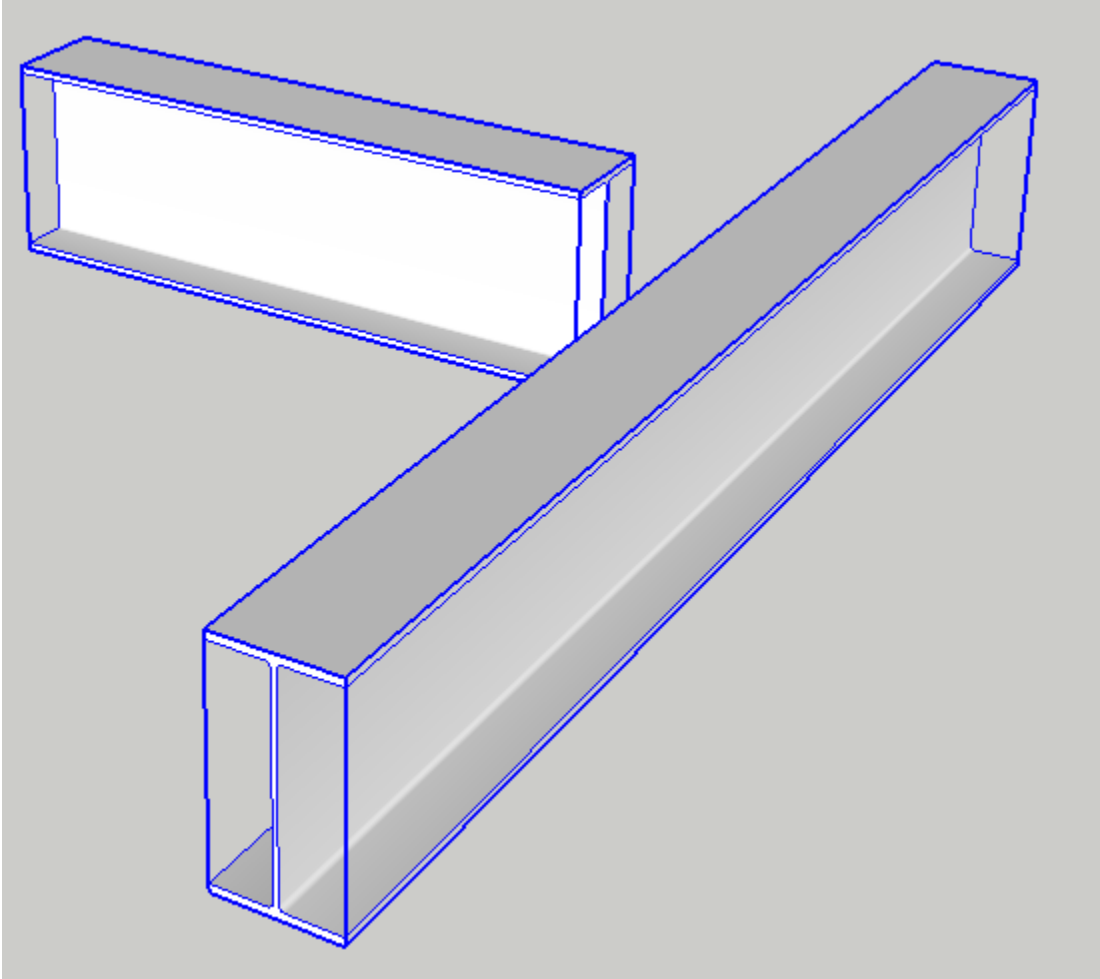
Select the Attributes to Filter the Selection



Check the boxes for the attributes that you wish to use as filters for the selection. In this example:

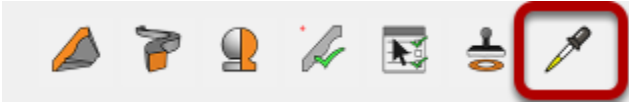
1. The Placement Point is checked.
2. The Profile is checked.
3. Click the Apply button to use the checked attributes as filters for selecting Profile Members.

Congratulations! You just selected Profile Members by Attributes!



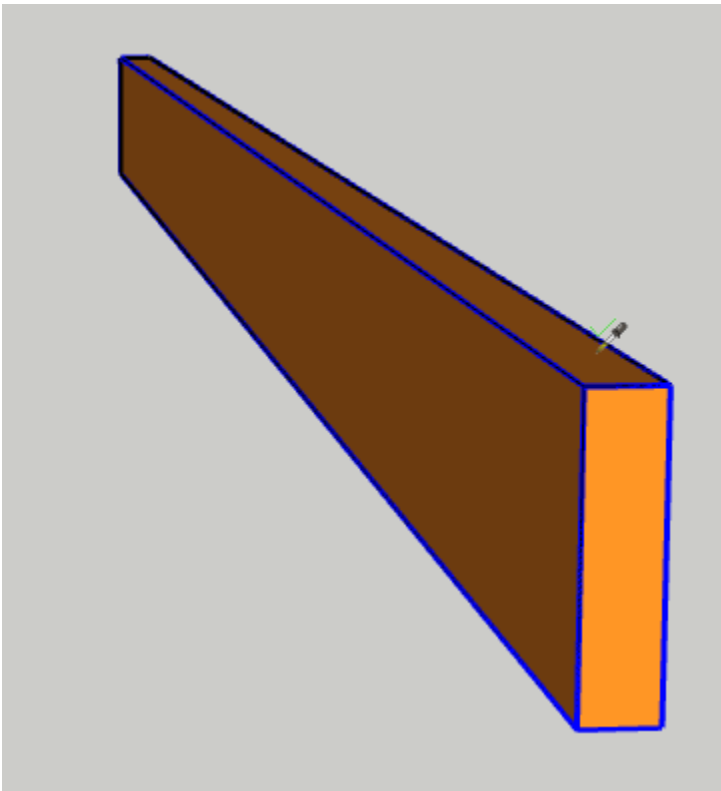
Get Attributes from a Profile Member

Launch the Tool



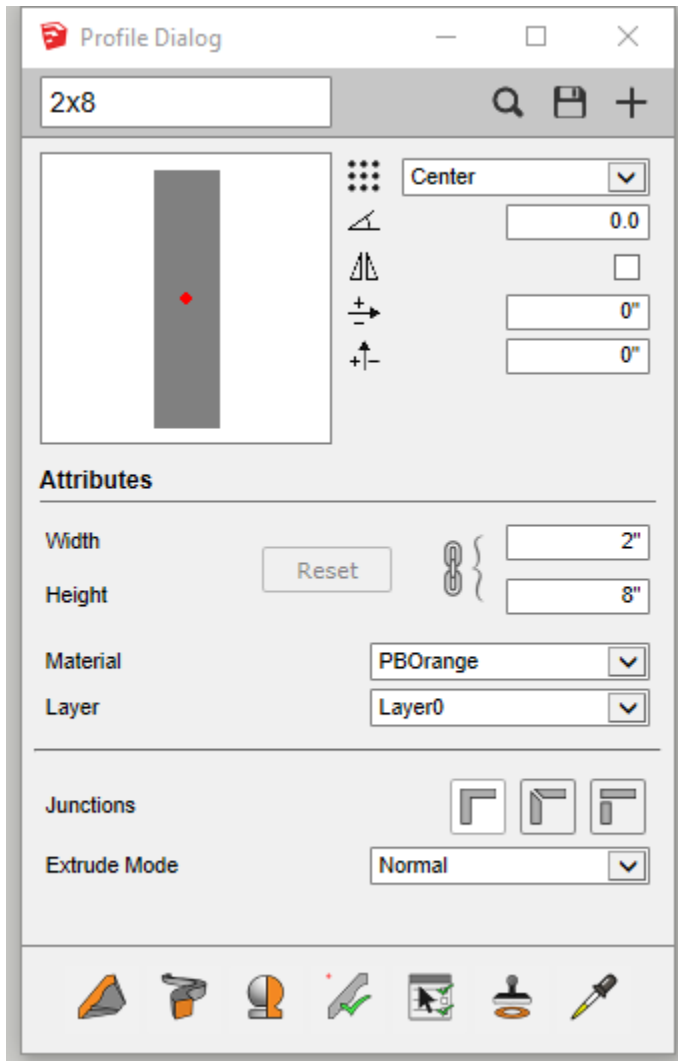
Open the Profile Builder Dialog.
Click the 'Get Attributes' button.

Click a Profile Member in your Model



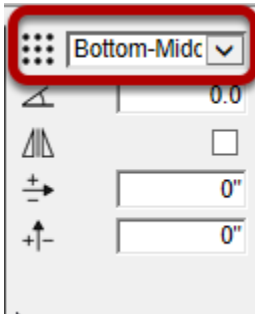
With the tool active, click on any Profile Member in your model.

Congratulations! You just got the Attributes from a Profile Member!



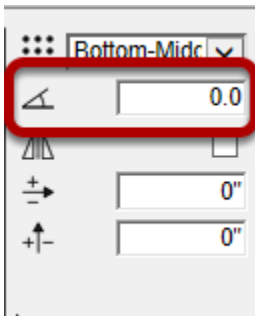
Profile Member Attributes

Placement Point



The placement point controls the location of the path in relation to the profile.

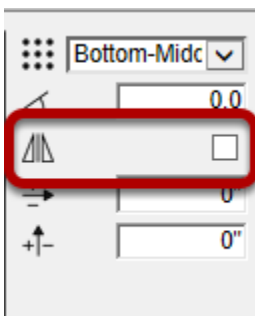
Rotation



Rotation allows you to specify a rotation value to the Profile. The rotation should be set in degrees and will be applied counter-clockwise.

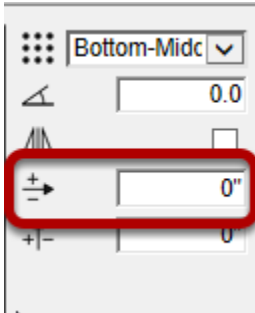
You can also specify a rotation as a V:H ratio (eg. 1:12)

Mirror



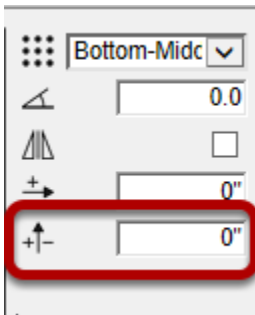
The mirror attribute allows you to mirror the orientation of the profile about the vertical axis.

X Offset



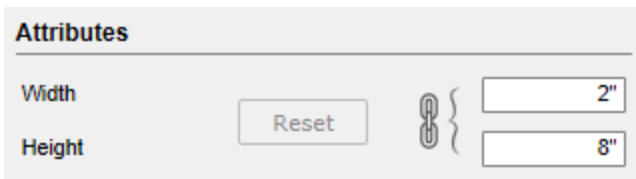
The X Offset value allows you to offset the location of the placement point in the horizontal direction.

Y Offset



The Y Offset value allows you to offset the location of the placement point in the vertical direction.

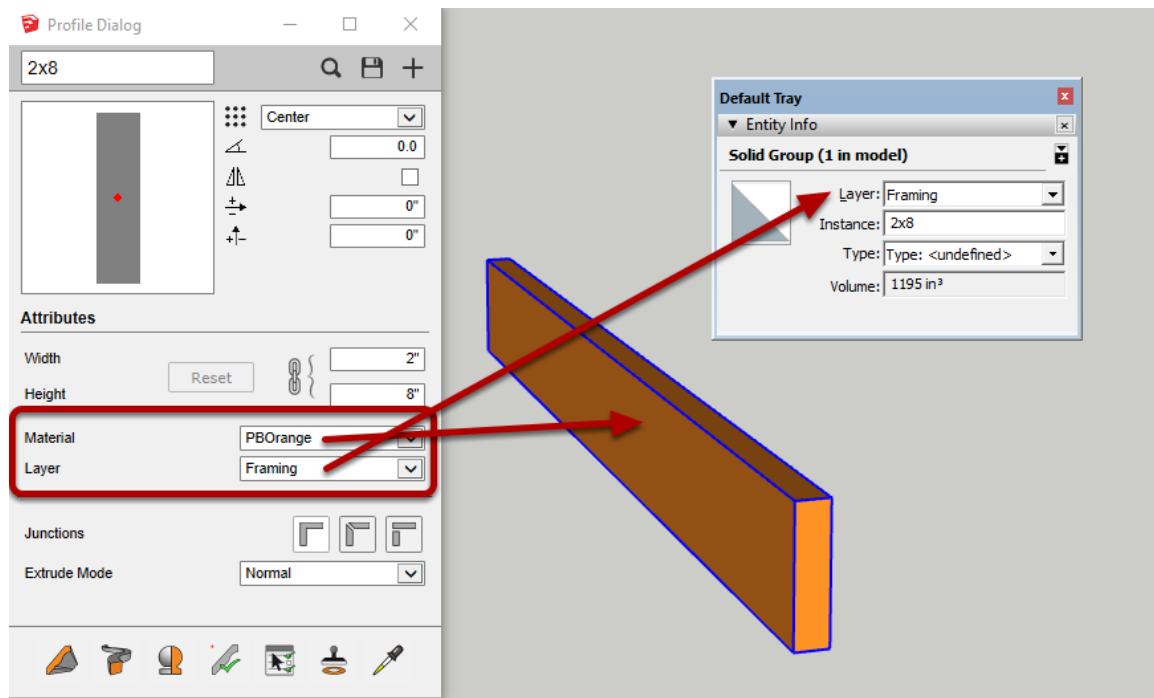
Dimension Attributes



The width and height attributes allow you to scale the profile to your desired dimensions. By default, the width to height ratio is locked unless you click the Lock / Unlock button.

Click the 'Reset' button to return to the default scale of the Profile.

Material and Layer Attributes

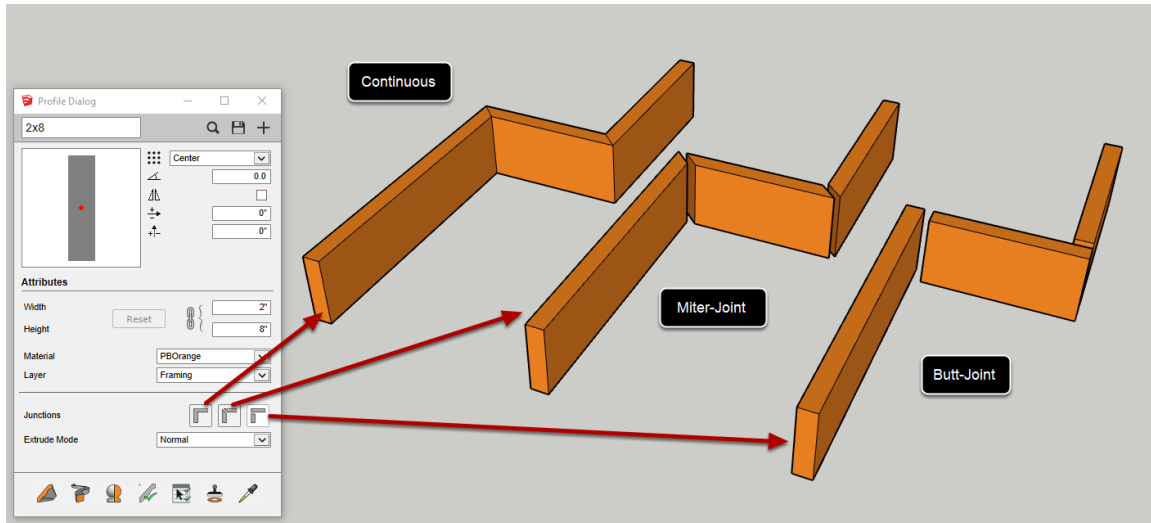


The Material and Layer attributes allow you to associate a default material and layer to be used when creating a Profile Member with this Profile.

The Material will be applied to the faces of the Profile Member only, and not the object (Group or Component) itself.

The Layer is applied to the Profile Member Instance (Group or Component). The faces of the object will be placed on the current active drawing layer.

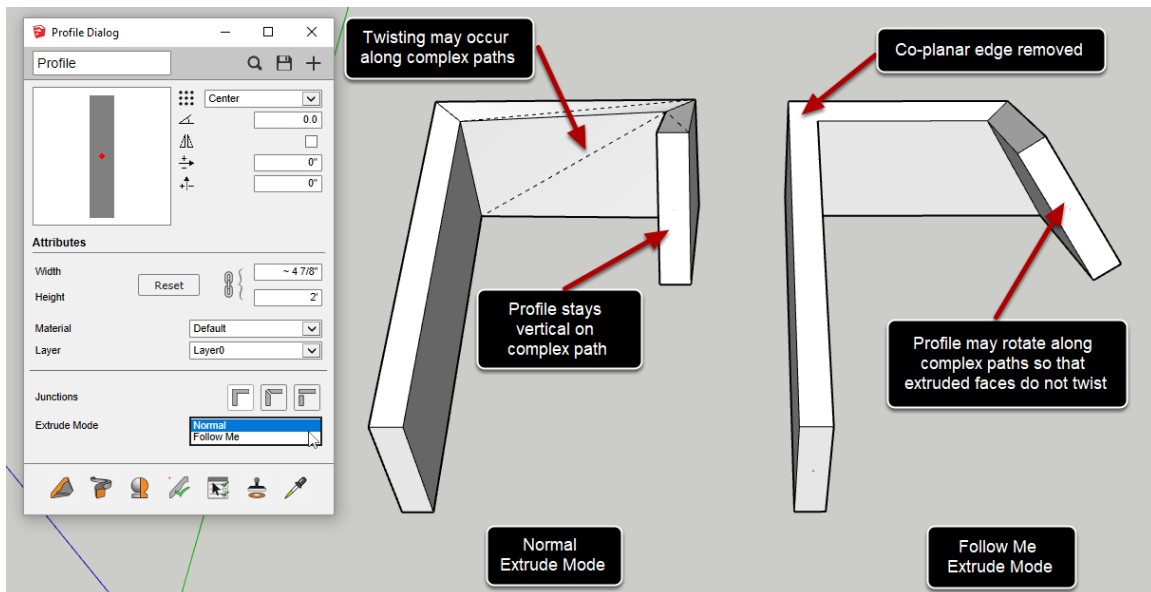
Junction Style



The Junctions attribute allows you to set whether the Profile Member is discontinuous or continuous and also what junction style to use. Choose from Continuous, Miter-Joint, or Butt-Joint.

If set to miter-joint or butt-joint, a separate Profile Member will be created along each segment of the path.

Extrude Mode



The Extrude Mode attribute allows more control over how a profile gets extruded along a path. Depending on the type of objects you are

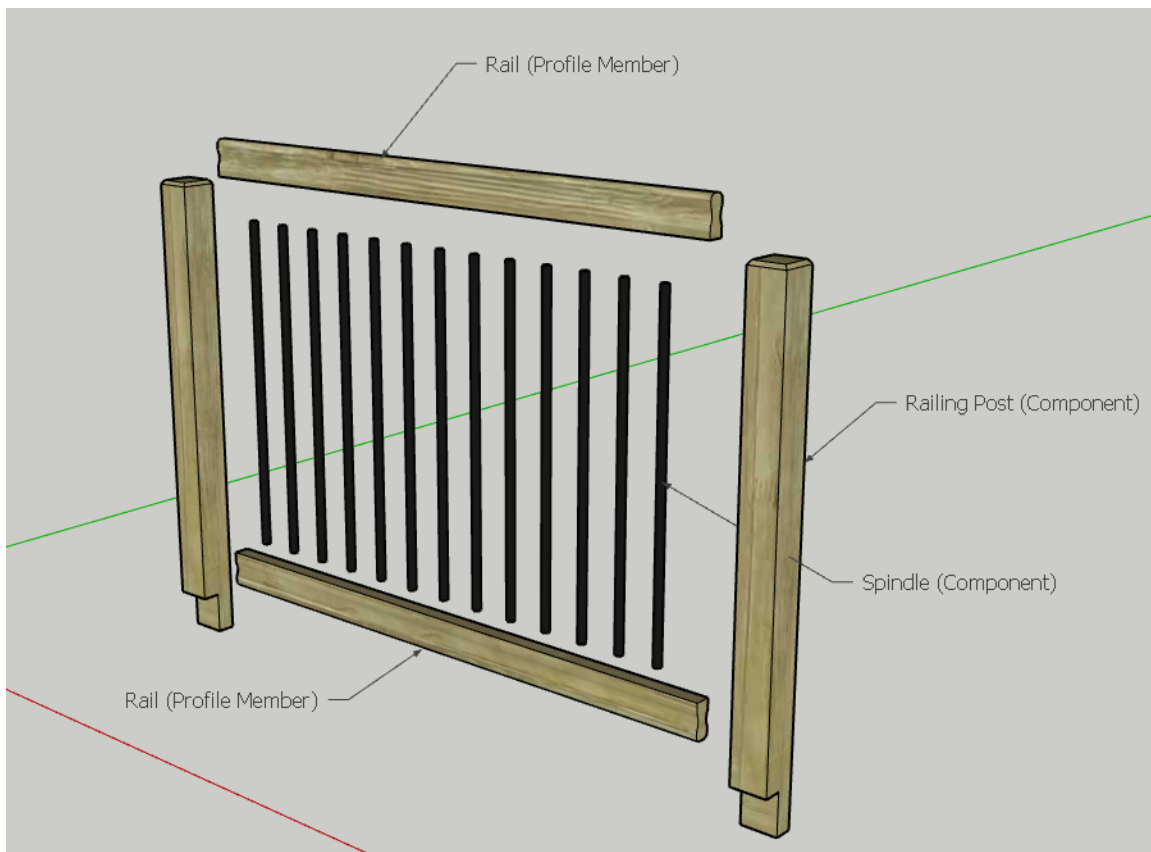
trying to create, you may find that a particular extrude mode works better. For example:

- The Normal mode is a good 'all-purpose' mode that works great in most situations. Since it keeps co-planar edges, material textures can be nicely wrapped along the extrusion.
- Follow Me mode may be preferred for creating simple wall extrusions because it automatically removes co-planar edges and creates simpler geometry.
- Follow Me mode may be preferred for creating circular piping extrusions along complex 3D paths because it can result in cleaner geometry with no twisted faces. Since piping profiles are circular, if the profile rotates along the path, there is no visual difference.

Assemblies

Creating Assemblies

Draw the Parts that will make up the Assembly



Draw a series of Profile Members (using the Profile Builder Build Tool) and other Components that you wish to use for the assembly. In this example, we will make a railing composed of rails, posts and spindles.

The parts for the assembly can be drawn anywhere in your model but it is much easier to create the assembly if you draw all of the parts in the same position that you want for the assembly. That way, you can use the SketchUp tape measure tool to easily calculate the offset and setback parameters for the various parts.

Open the Assembly Dialog

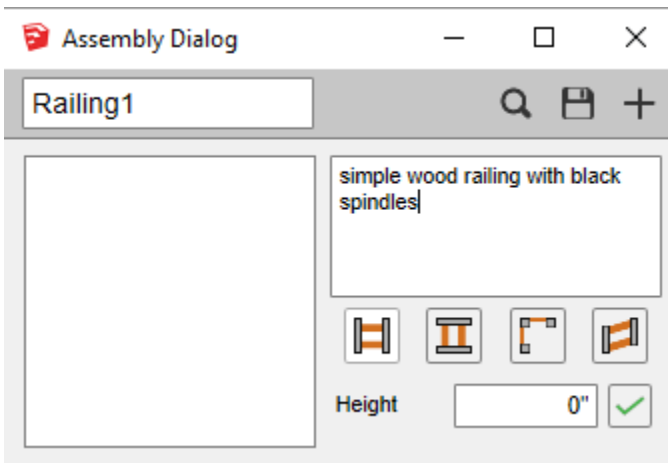


Create a New Assembly

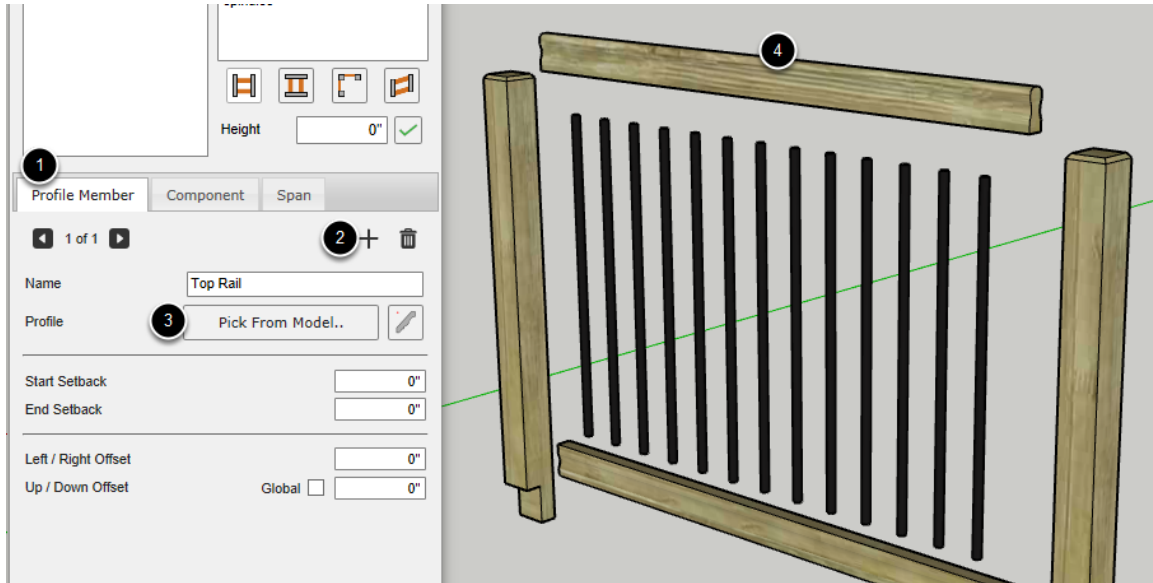


Click the 'New Assembly' button to create a new Assembly

Give the Assembly a Name and Description



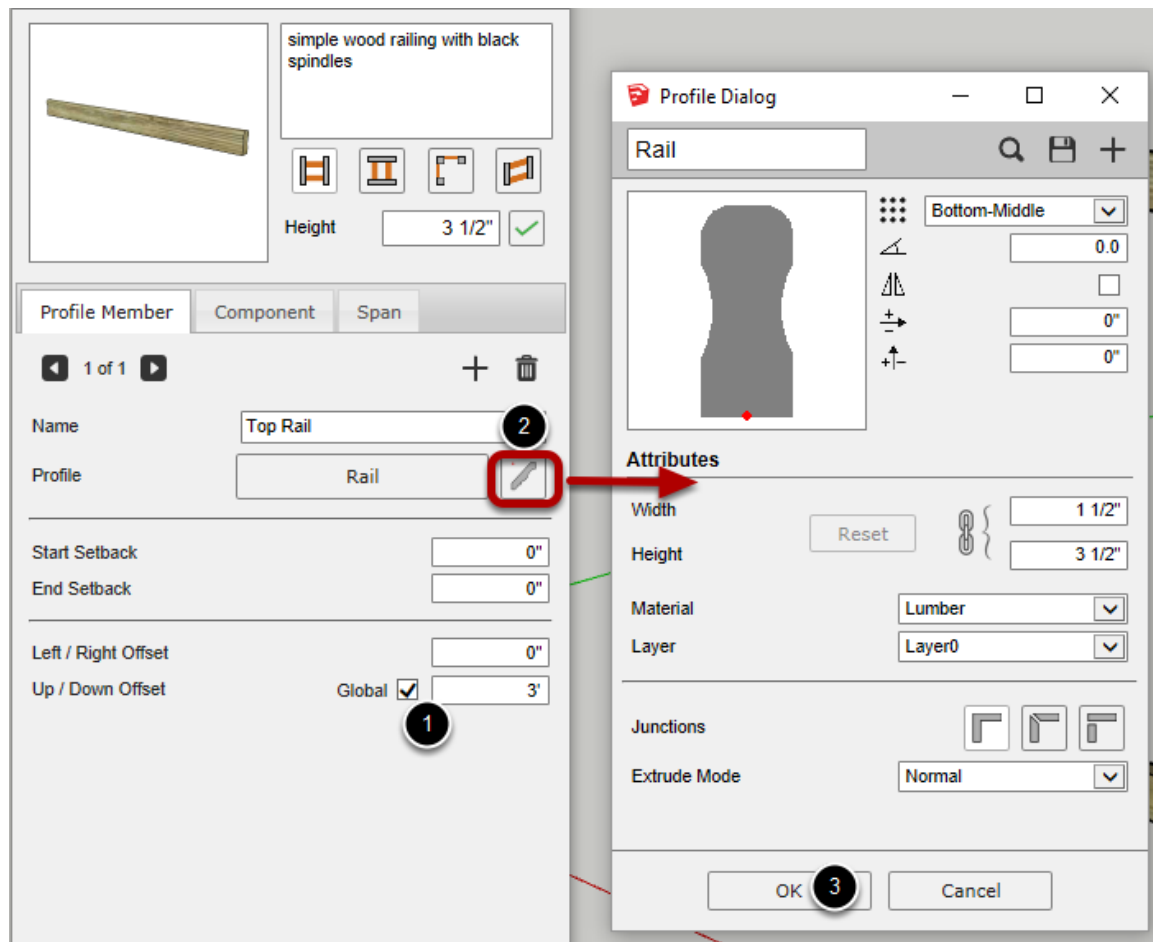
Add a Profile Member to the Assembly



1. Click the Profile Member Tab.
2. Click the 'Add Profile Member' button.
3. Click the 'Pick from Model' button.
4. Click a Profile Member in your model to add it to the assembly.

Be sure to give the part a meaningful name. Here we named the part 'Top Rail'.

Edit the Profile Member part



1. Set the Up / Down Offset to 3 feet and click the 'Global' checkbox. By setting the offset to global, the part will be offset along the model blue axis even if the assembly is built along a sloping path.
2. Edit the Profile Attributes if desired.
3. Click 'OK' when finished editing.

Add the Bottom Rail Part

Assembly Dialog

Railing1

simple wood railing with black spindles

Height 2' 11" ✓

Profile Member Component Span

2 of 2

Name Bottom Rail

Profile Rail

Start Setback 0"

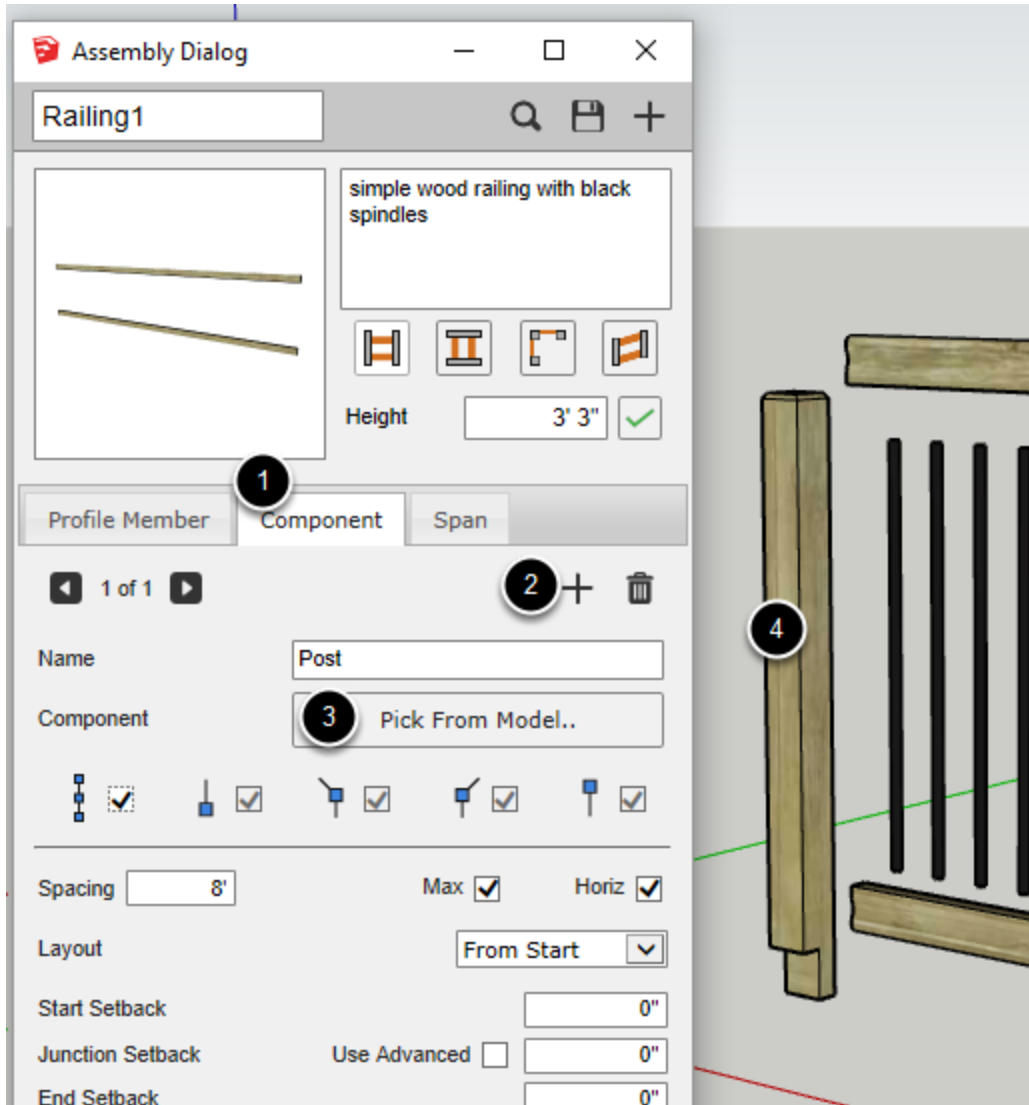
End Setback 0"

Left / Right Offset 0"

Up / Down Offset Global ☒ 8"

Repeat the previous steps to add the Bottom Rail.
Set the Up / Down offset of the bottom rail to 8" and check global.

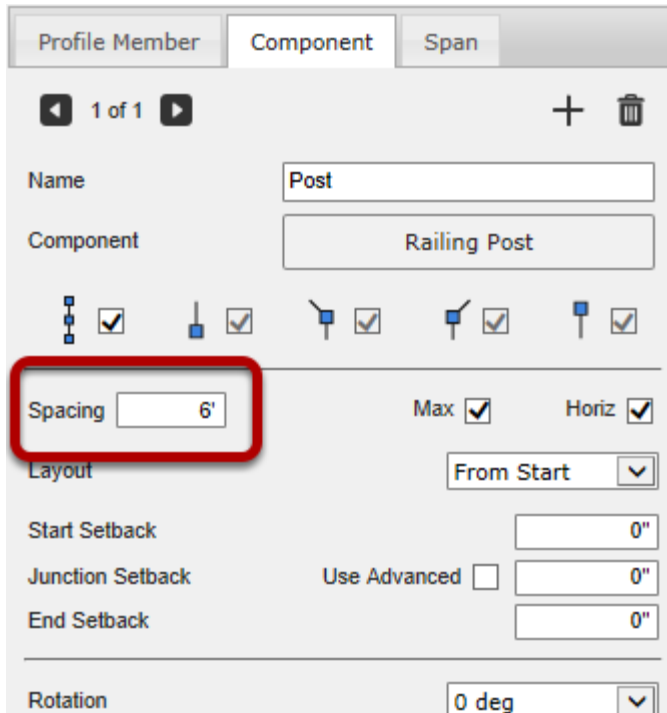
Add a Component to the Assembly



1. Click on the 'Component' tab.
2. Click the 'Add Component' button.
3. Click the 'Pick from Model' button.
4. Click on a component in your model.

Be sure to give the component part a meaningful name.

Edit the Component Part








Profile Member Component Span

◀ 1 of 1 ▶ + 🗑️

Name

Component

 ☒  ☒  ☒  ☒  ☒

Spacing Max ☒ Horiz ☒

Layout ▼

Start Setback

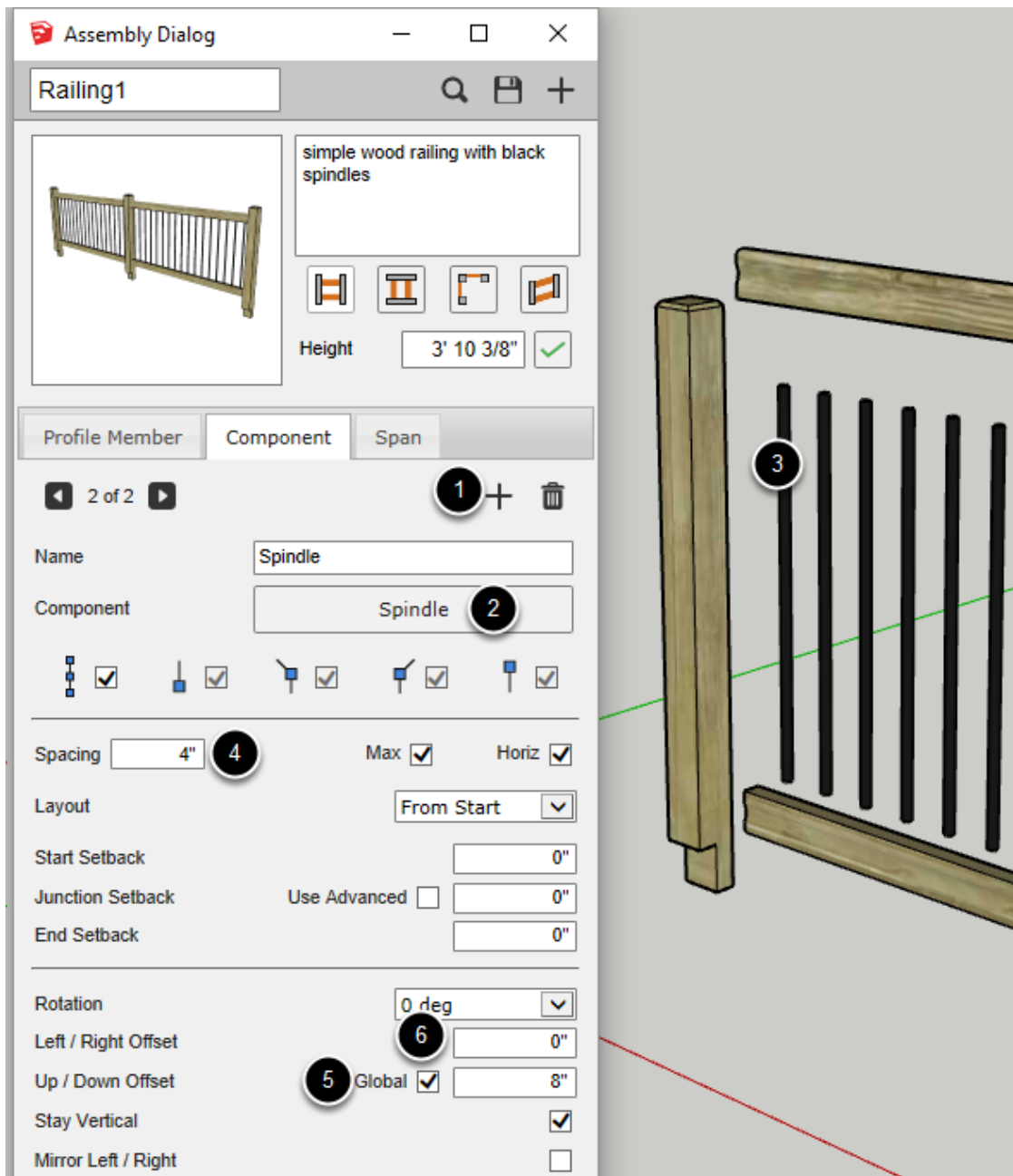
Junction Setback Use Advanced ☐

End Setback

Rotation ▼

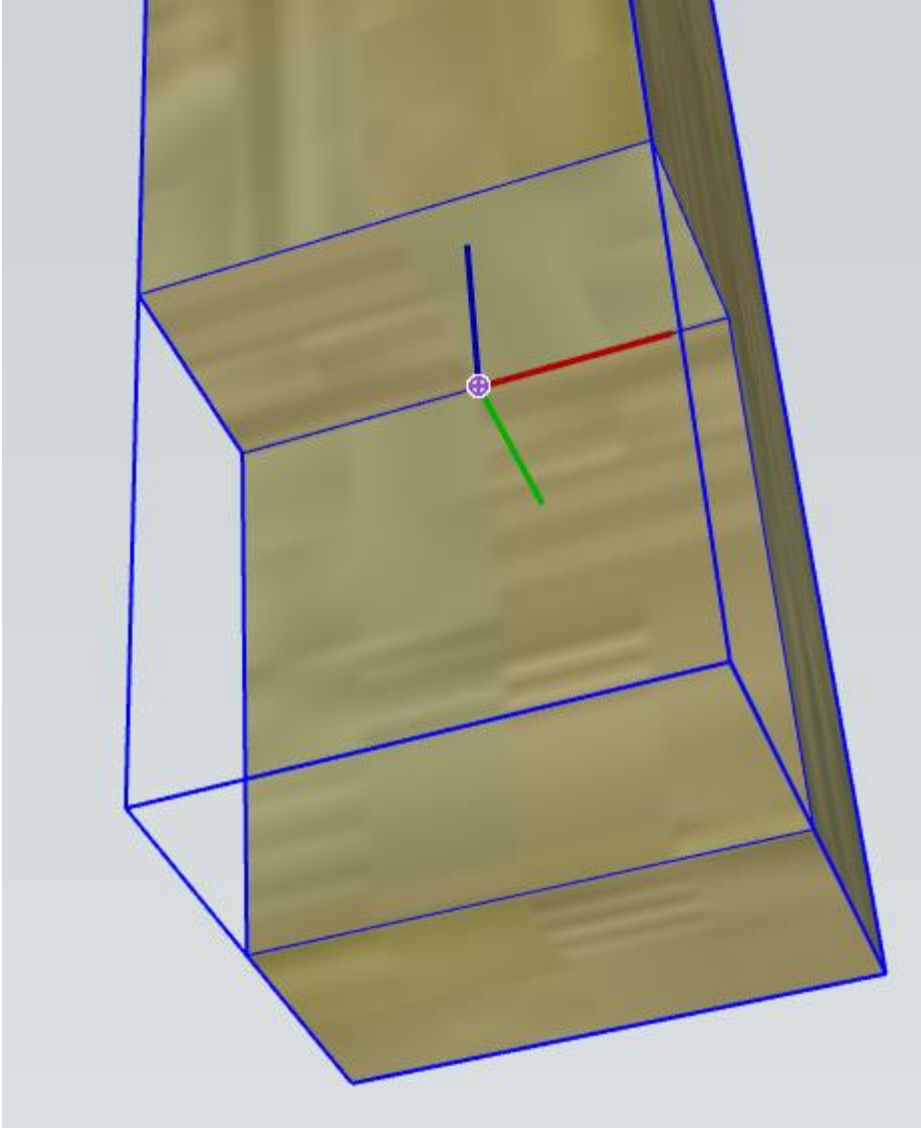
Change the spacing to 6'. Leave the other settings as is.

Add the Metal Spindle Part



1. Click the 'Add Component' button. This will create a copy of the current component part that is stacked beside the current component part.
2. Click the 'Pick from Model...' button.
3. Click the Metal Spindle Component.
4. Change the spacing to 4 inches.
5. Change the Up / Down offset to 8 inches and check global.
6. Change the Left / Right offset to 0 inches.

Edit the Axis of the Component

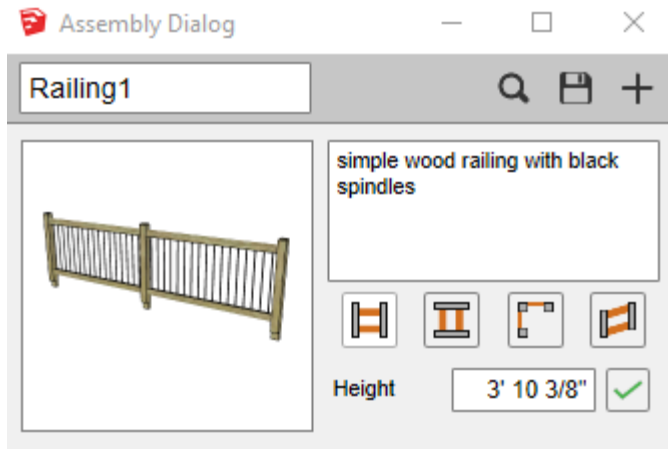


It will usually be necessary to edit the axis of the Component so that it is positioned correctly. To edit the axis of a component, right-click on the component in the SketchUp model and choose 'Change Axes' from the context menu.

The component will be positioned on the Assembly path as follows:

- The red axis will point in the direction of the path of the assembly (forward)
- The blue axis will point 'up'
- The green axis will point 'left' from the perspective of someone looking in the direction of the assembly path.

Congratulations! You just created an Assembly!

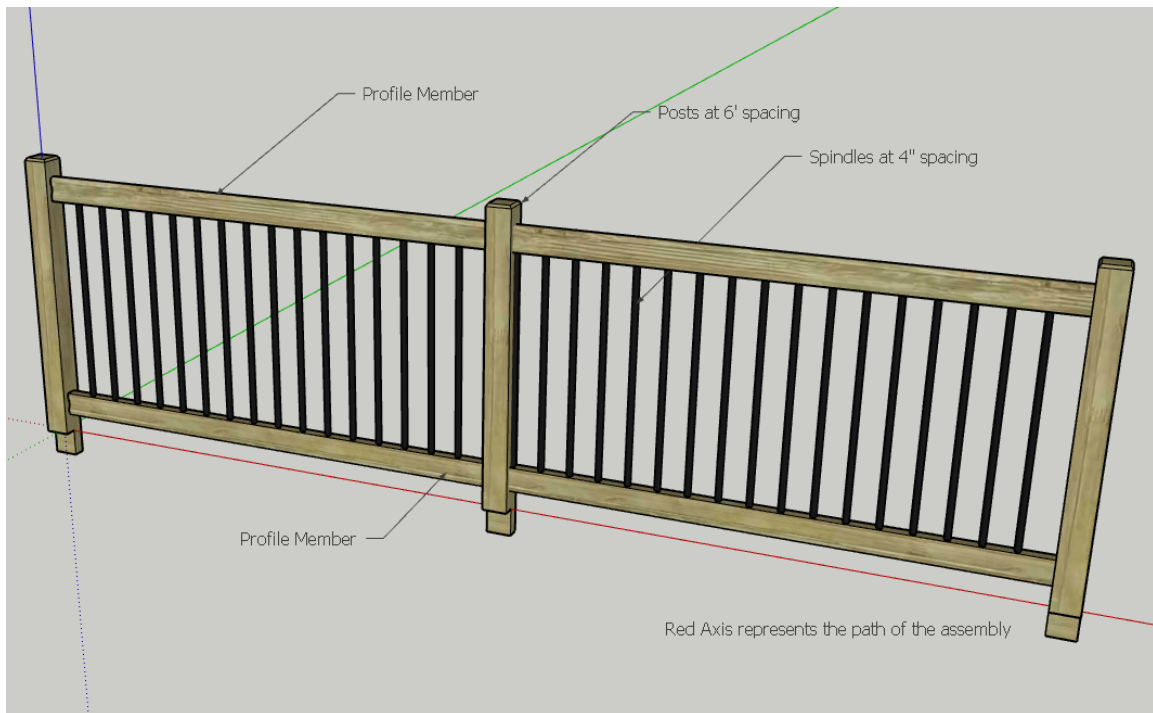


Now that you have created an assembly, you can save it, build it, or share it on the 3D Warehouse!

Creating Assemblies Instantly with Auto-Assemble

Basic assemblies can be created almost instantly using the Auto-Assemble Feature.

Create a Prototype of the Assembly

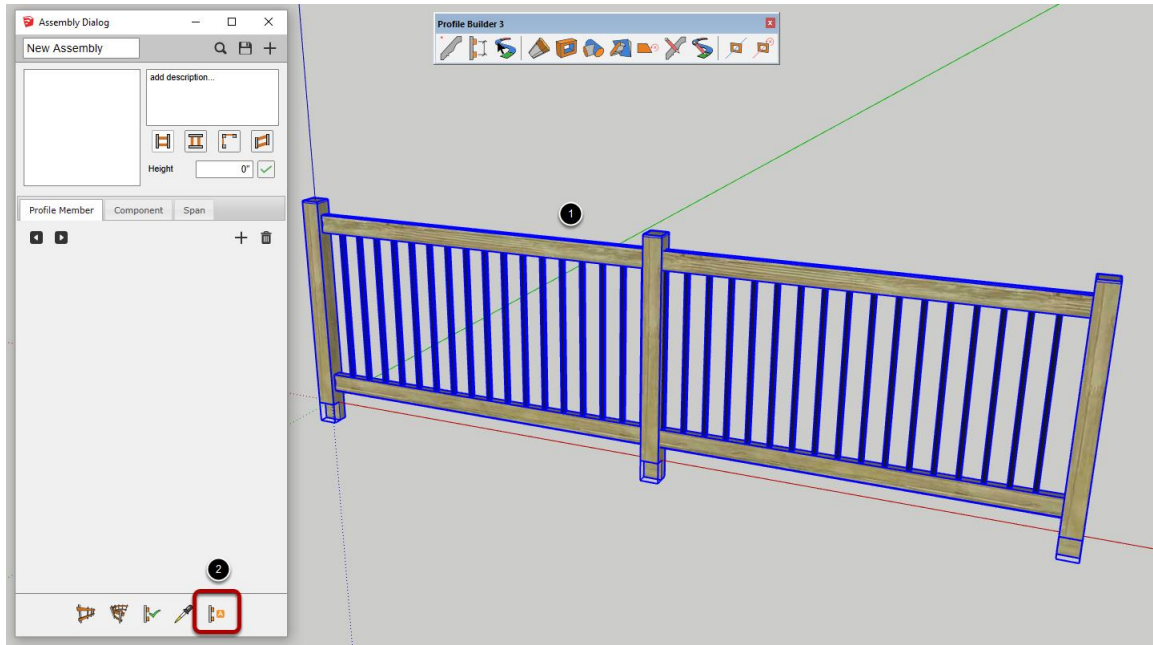


First, create a prototype of the assembly along the red axis of your model. The red axis is used to represent the path of the assembly.

The prototype should be made up of only components and Profile Member groups. The Profile Members must be drawn parallel and in the direction of the red axis.

If you want to create an 'infill' component part that repeats along the path, be sure to create at least 3 copies of the component at an equal spacing within your prototype. If you do not have 3 copies at equal spacing, the component part will be placed only at the start or end of the assembly when it is generated.

Select all of the Parts in your Assembly Prototype and Auto-Assemble

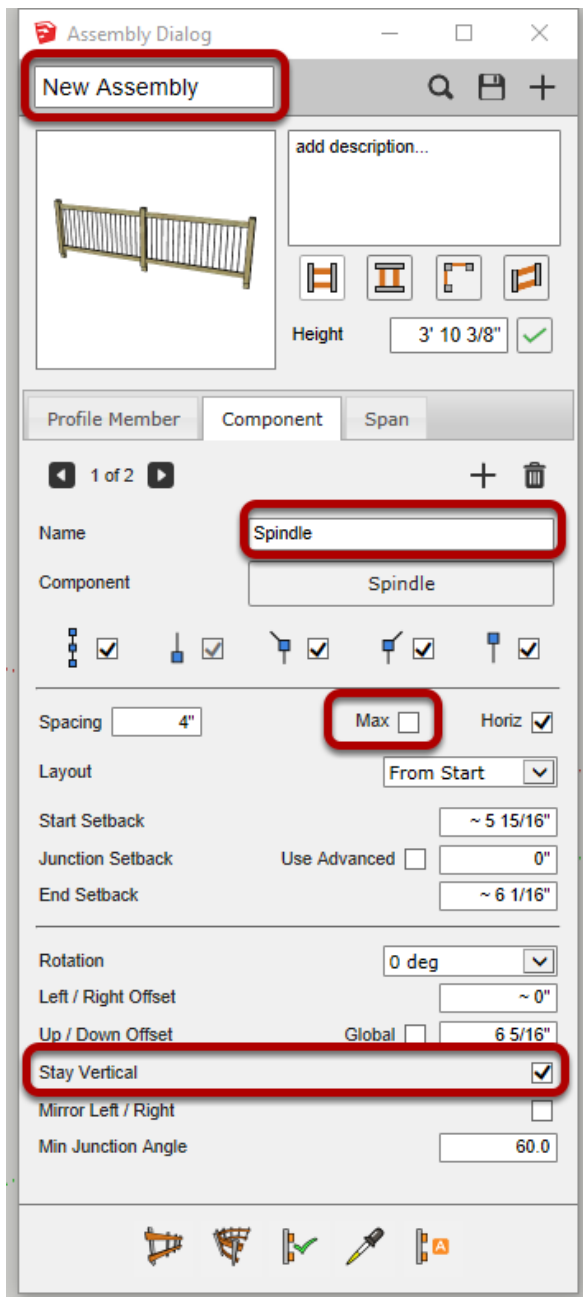


Do not group the parts together before selecting. Make sure each part can be selected individually.

1. Use the SketchUp selection tool to select all parts in the prototype.
2. Click the Auto-Assemble button in the Assembly Dialog.

The selected parts will be automatically combined into an assembly with the offsets and setbacks calculated automatically!

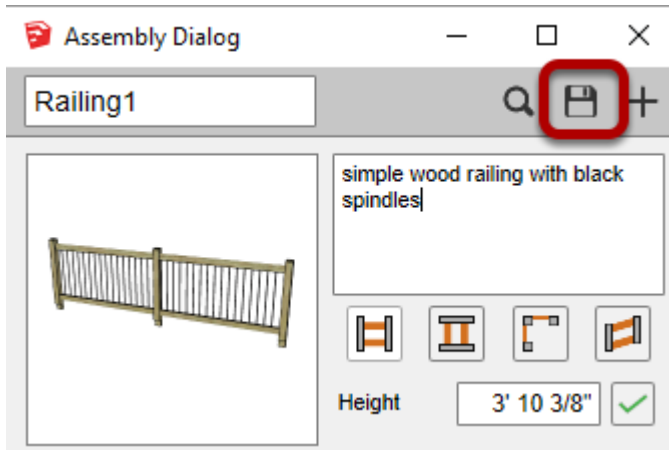
Make Final Adjustments to the Assembly



After creating an auto-assembly, it is recommended to give meaningful names to the assembly and all parts. For component parts, you may want to change the spacing parameter to 'Max' or adjust the 'Stay Vertical' checkbox.

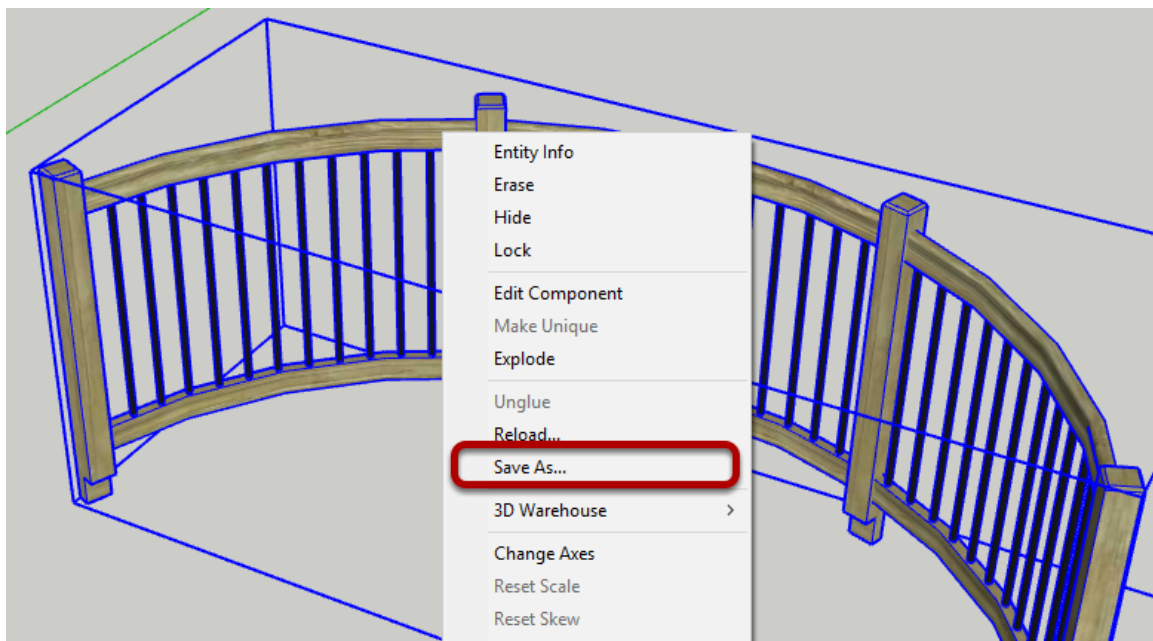
Saving Assemblies

Save the Assembly



Click the 'Save Assembly' button to save the Assembly. A window will open asking where to save the Assembly. The assembly will be saved as a SketchUp component (SKP file).

Save an Assembly Component

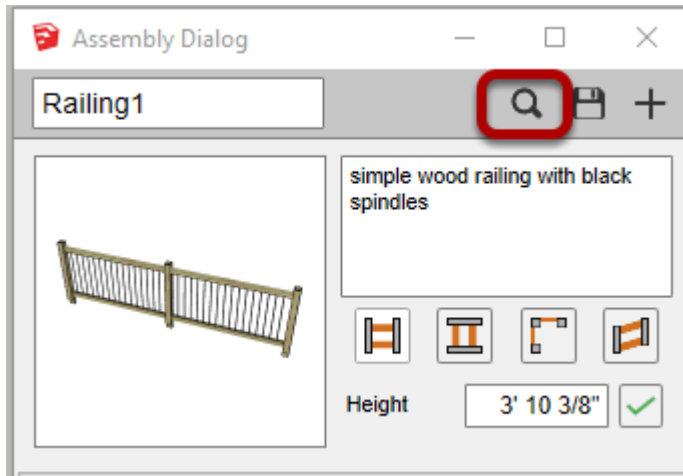


You can also save an Assembly by converting an Assembly to a Component using the right-click context menu and then using the context menu to save the Component.

This is useful for creating specific prototype Assembly thumbnails for your library or before uploading to the 3D Warehouse.

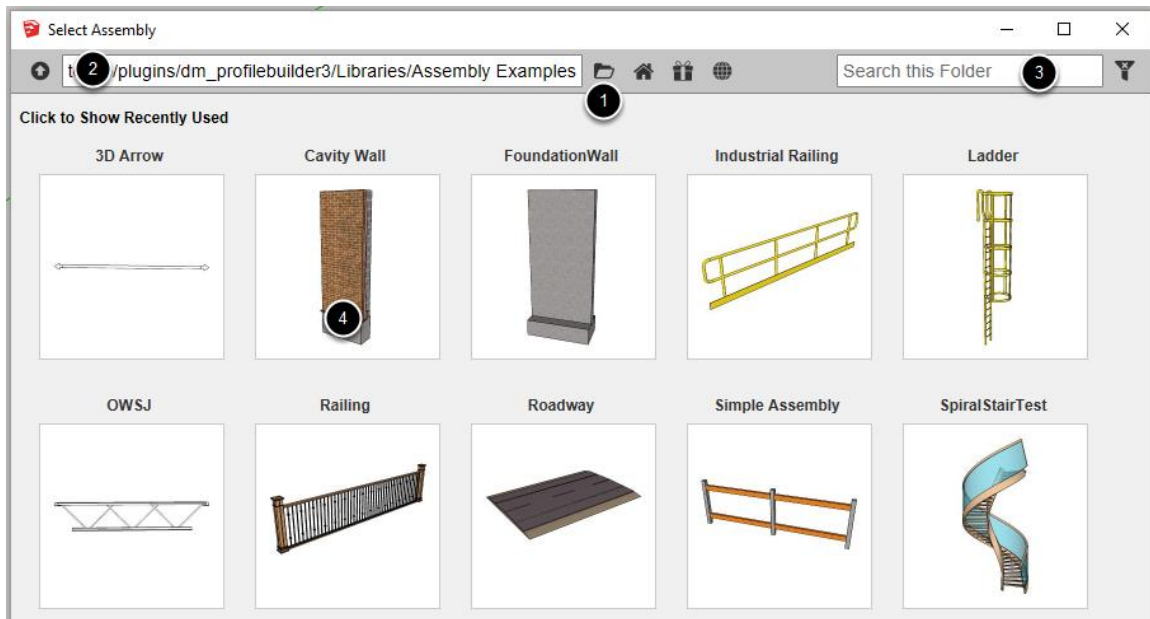
Loading Assemblies

Load an Assembly



Click the 'Assembly Browser' button.
The Assembly Browser window will open.

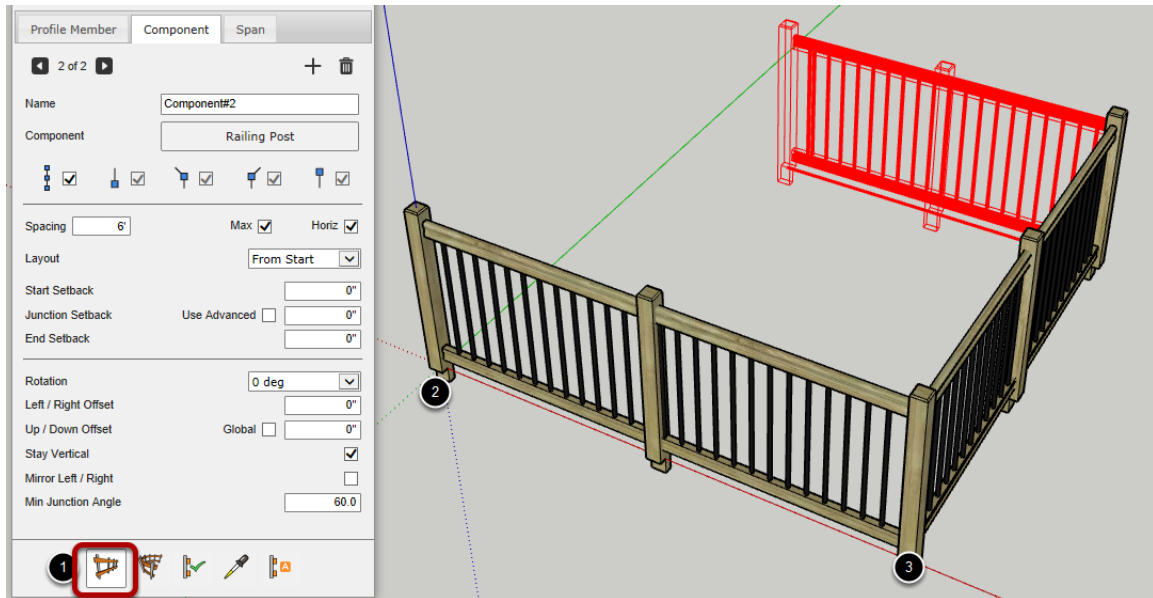
Finding an Assembly



1. Click the Open Library button to choose a new library folder.
2. The current folder is shown at the top left.
3. Enter text to filter results.
4. Click an assembly thumbnail to load it.

Build Assembly Tool

Build an Assembly



Now that you have created an assembly, it is time to build it!

1. Click the 'Build Assembly' Button.
2. Click a point in the model to define the start of the assembly.
3. Continue clicking points to define the path of the assembly.

To complete the assembly:

Press ESC or RETURN or ENTER

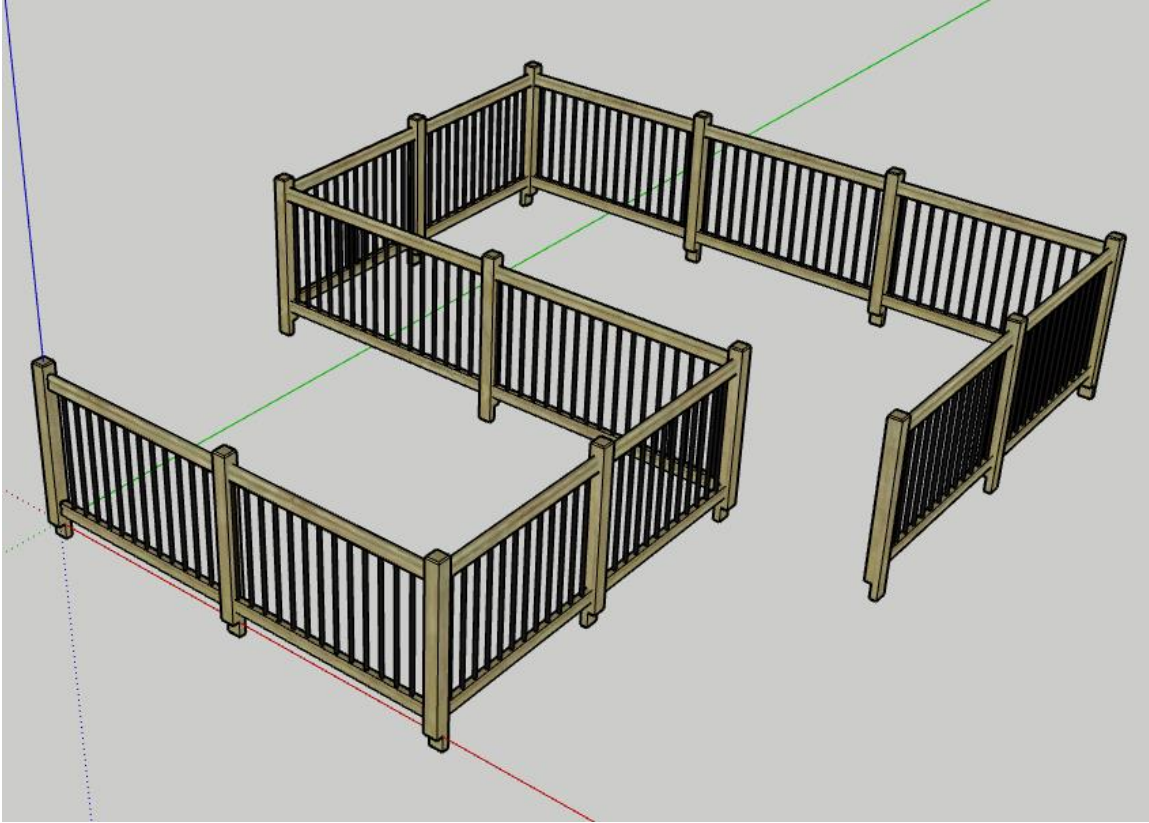
OR

Right-click and select 'Finish'

OR

Create a closed path for the Assembly.

Congratulations! You just built an Assembly!



Modifier Keys

Arrows Keys = Lock Axis

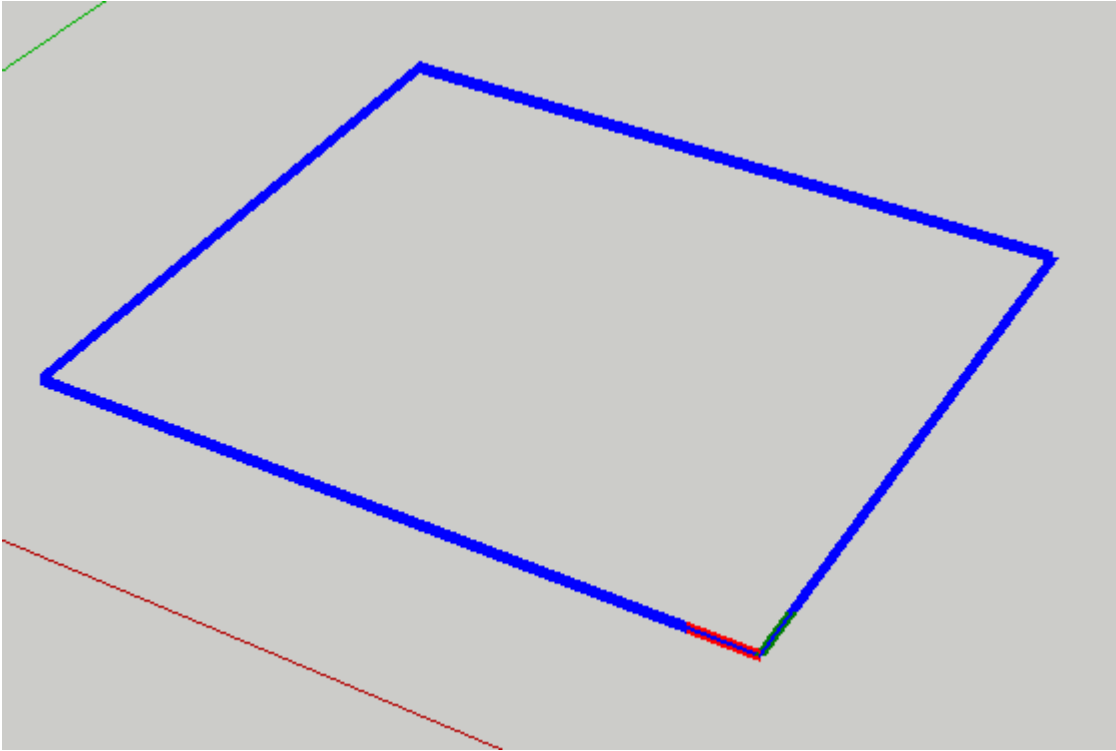
SHIFT = Lock Inference

BACKSPACE or DELETE = Undo the last path point

You can also enter values in the Measurements box to draw precisely.

Build Assembly along Path

Select a Path



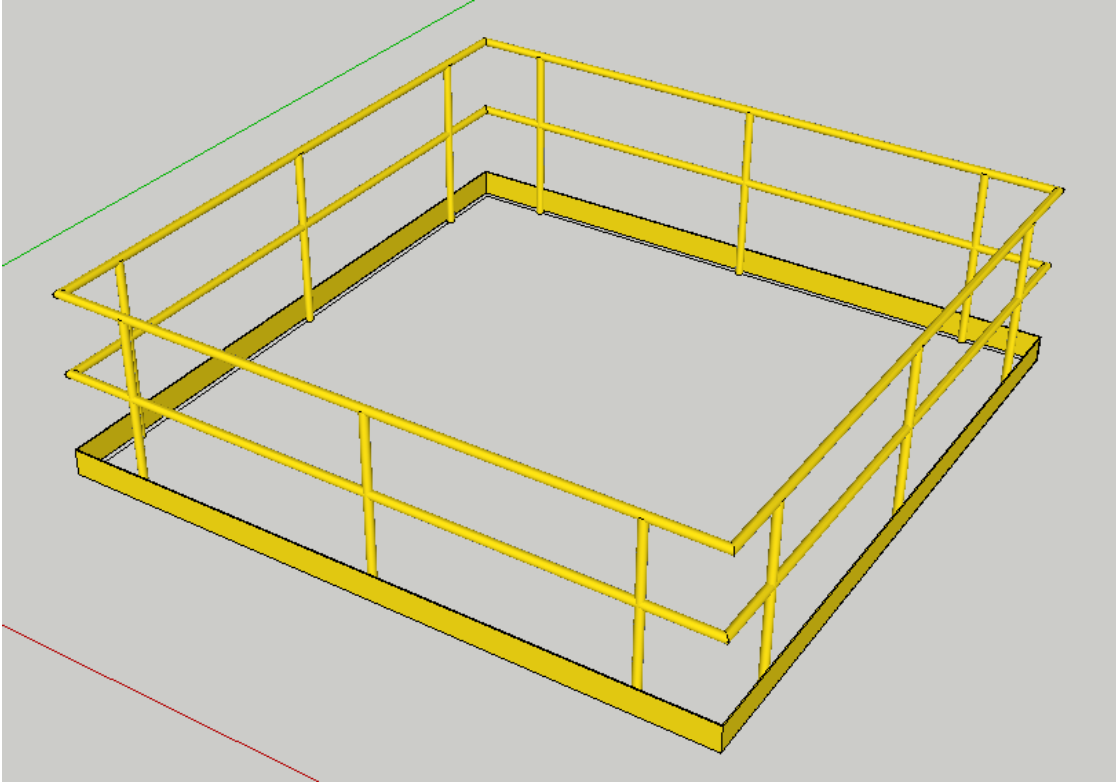
Select a series of connected edges, select a face, or use the [Smart-path select tool](#) to choose a path.

Build the Assembly



Click the 'Build Assembly Along Path' button.

Congratulations! You just built an Assembly along a Path!



You can also build along multiple paths in one operation if you have more than one path selected before launching the tool.

Apply Attributes to Assembly

Edit the Assembly Attributes

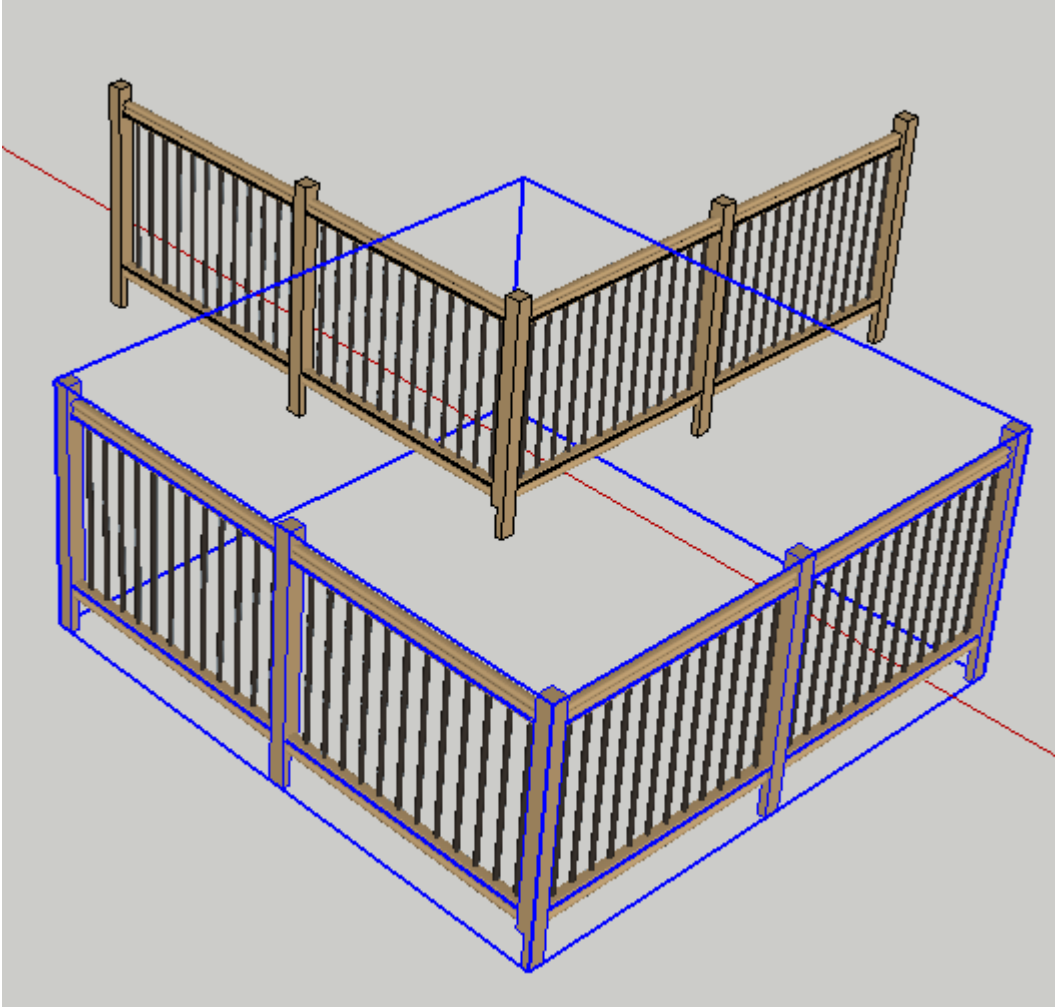
The screenshot shows a software interface with three tabs: 'Profile Member', 'Component', and 'Span'. The 'Component' tab is active. At the top, there are navigation icons and a '2 of 2' indicator. Below this, the 'Name' field contains 'Component#2' and the 'Component' field contains 'Railing Post'. A row of six icons with checkboxes follows. The 'Spacing' field is set to '4'' and is highlighted with a red rectangle. To its right, 'Max' and 'Horiz' are both checked. The 'Layout' dropdown menu is set to 'From Middle' and is also highlighted with a red rectangle. Below these are fields for 'Start Setback' (0'), 'Junction Setback' (0"), and 'End Setback' (0"). A 'Use Advanced' checkbox is located between the 'Junction Setback' and 'End Setback' fields.

Attribute	Value
Name	Component#2
Component	Railing Post
Spacing	4'
Layout	From Middle
Start Setback	0'
Junction Setback	0"
End Setback	0"

Change the attributes of the assembly.

In this example, the spacing of the posts was changed from 6' to 4'.
and the Layout was changed to 'From Middle'.

Select an Assembly in your Model



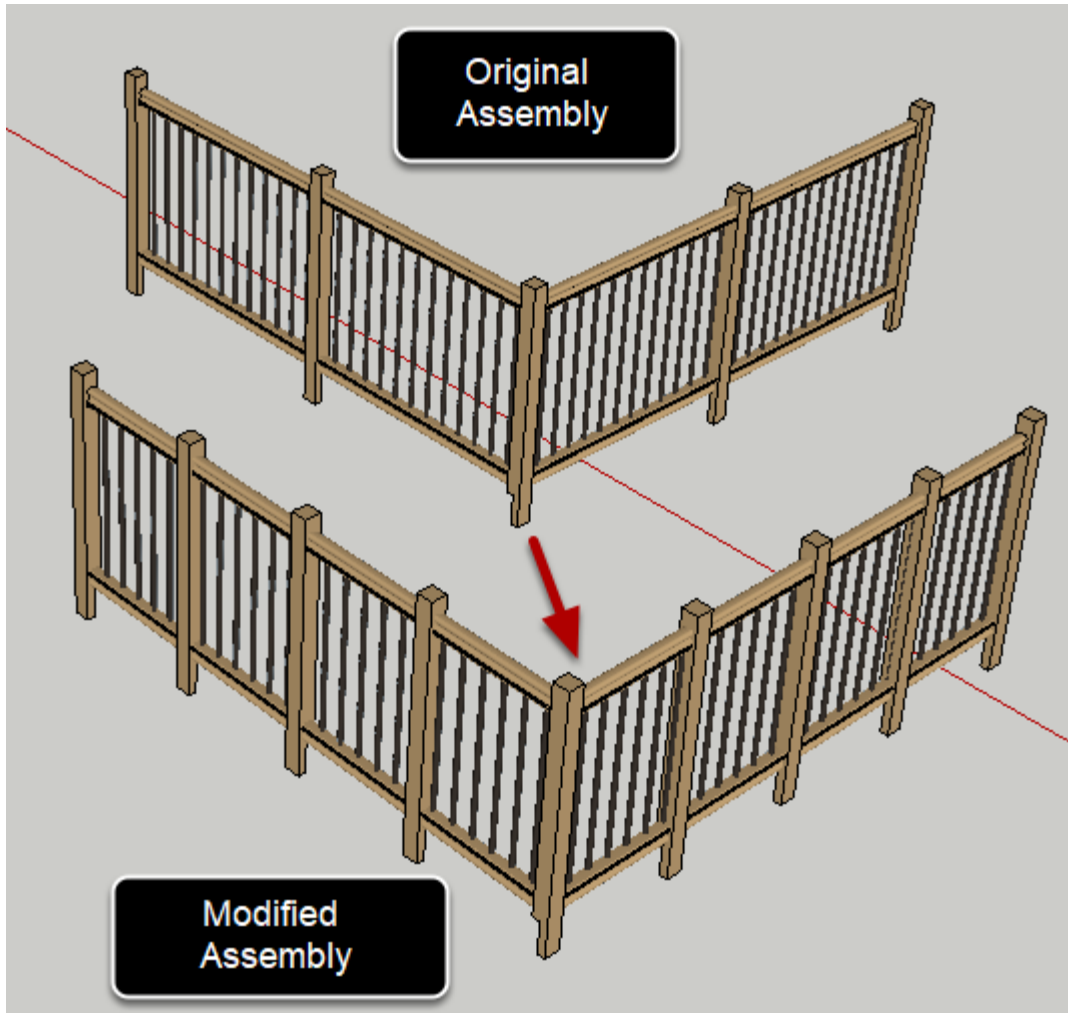
You can also select multiple assemblies to apply the attributes to all of the selected assemblies at once. You can also apply assembly settings to any Profile Members that are selected.

Click the 'Apply Assembly Attributes' button



Click the button to apply the current assembly settings to all selected assemblies.

Congratulations! You just applied attributes to an Assembly!

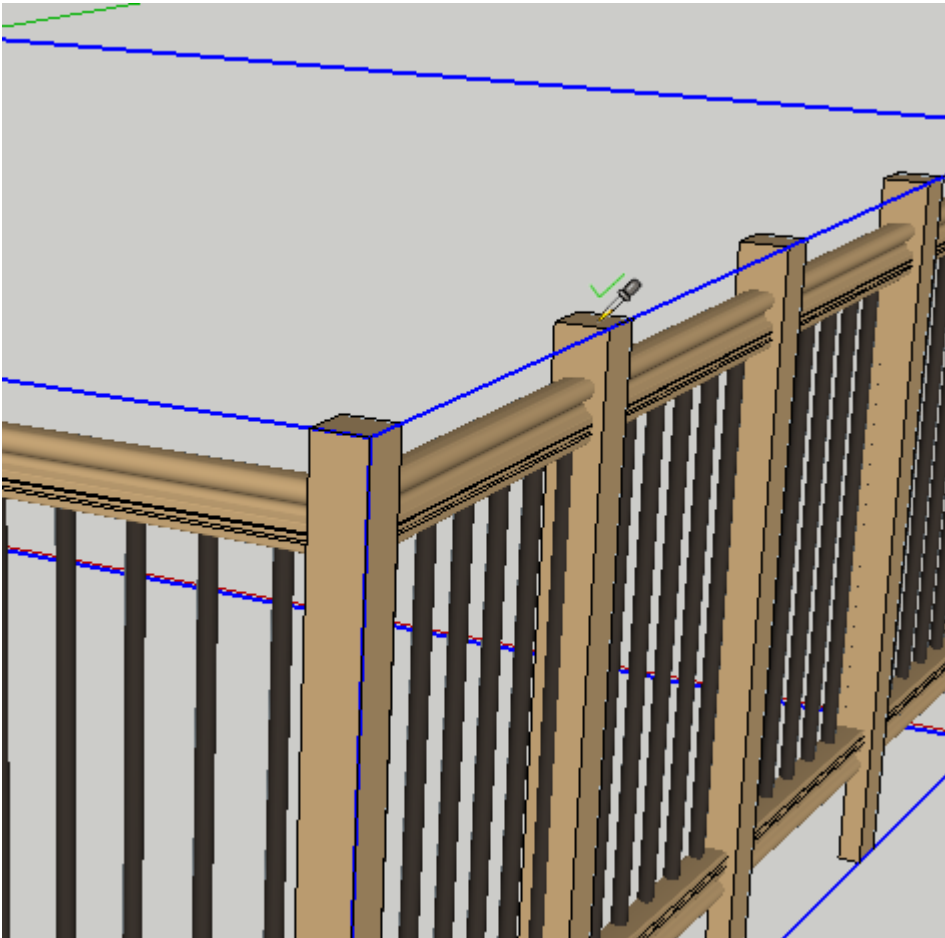


Get Attributes from Assembly

Click the 'Get Assembly Attributes' button

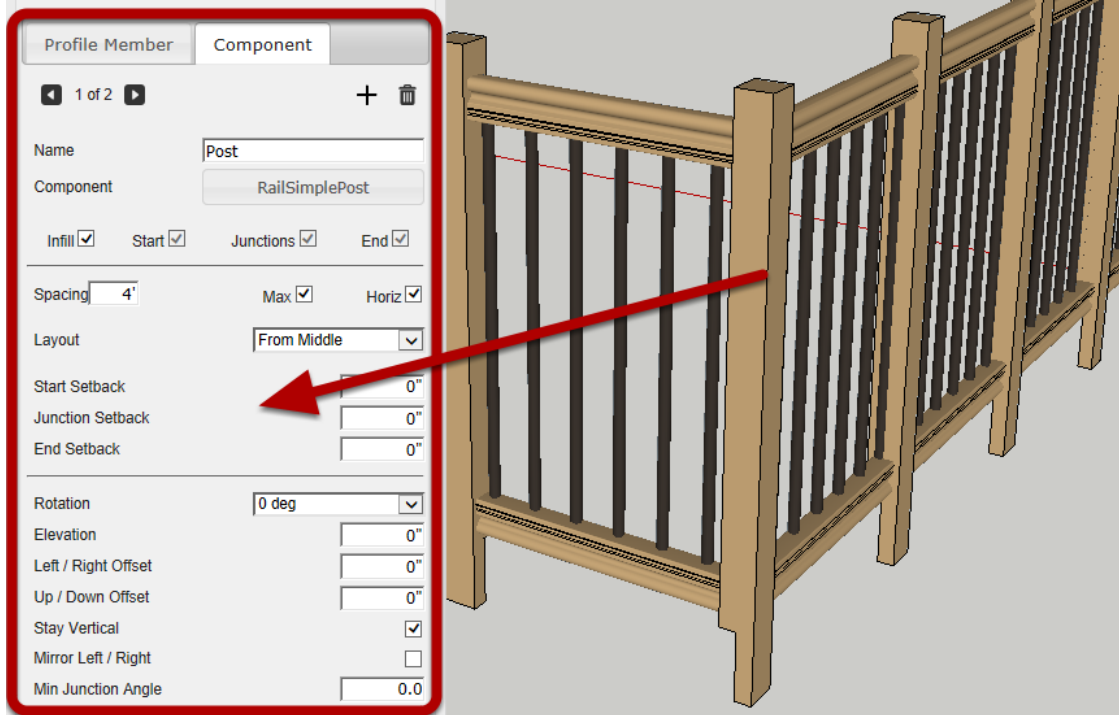


Click an Assembly in your Model



If there is an Assembly below the cursor (in the active context), it will become highlighted and the cursor will show a checkmark. Click the Assembly to load the attributes into the Assembler Dialog.

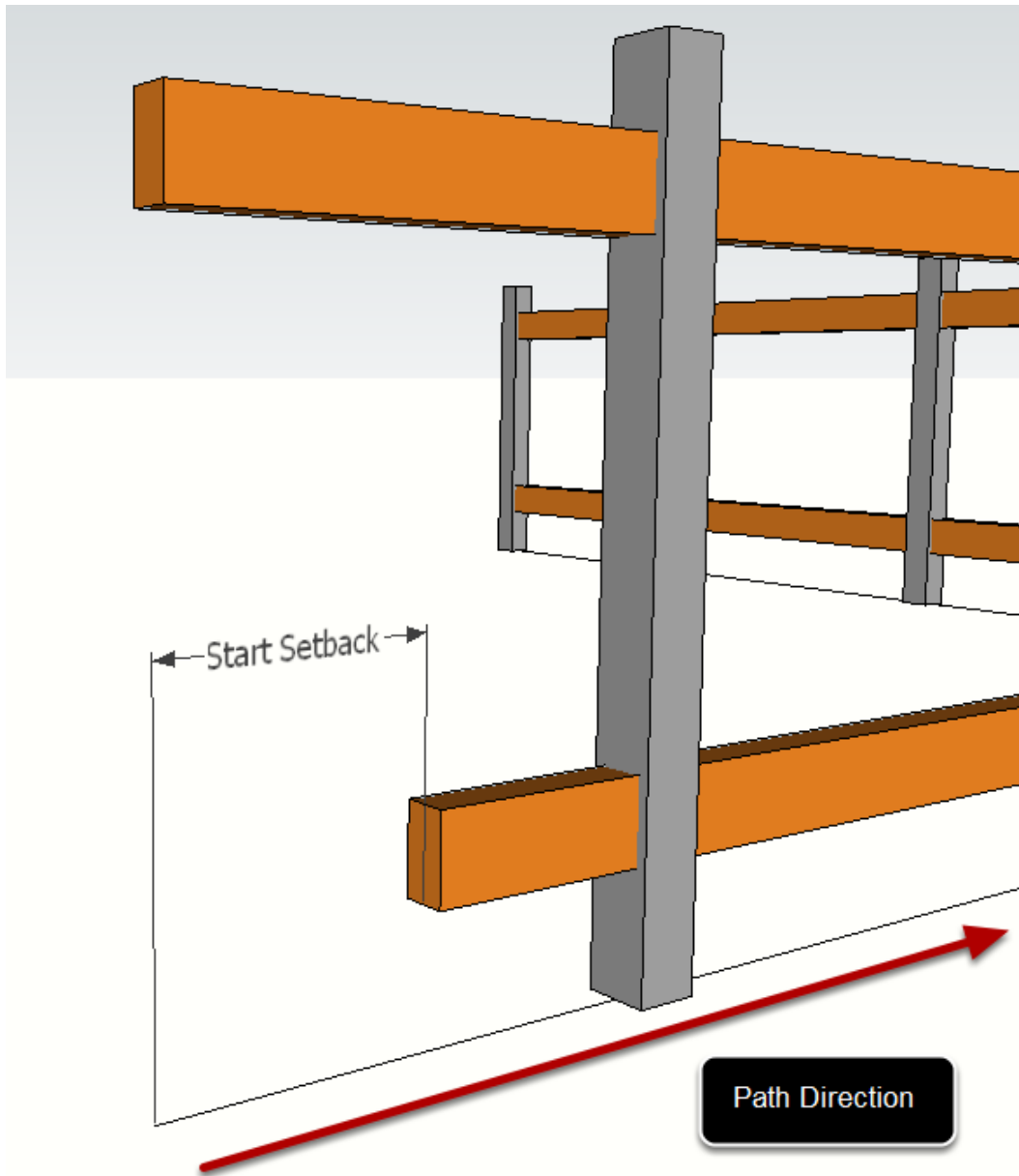
Congratulations! You just got the Attributes of an Assembly!



Typically before editing the attributes of an existing assembly, you would first load the existing attributes as shown above. Then, you can make changes and use the 'Apply Attributes' button to modify the assembly.

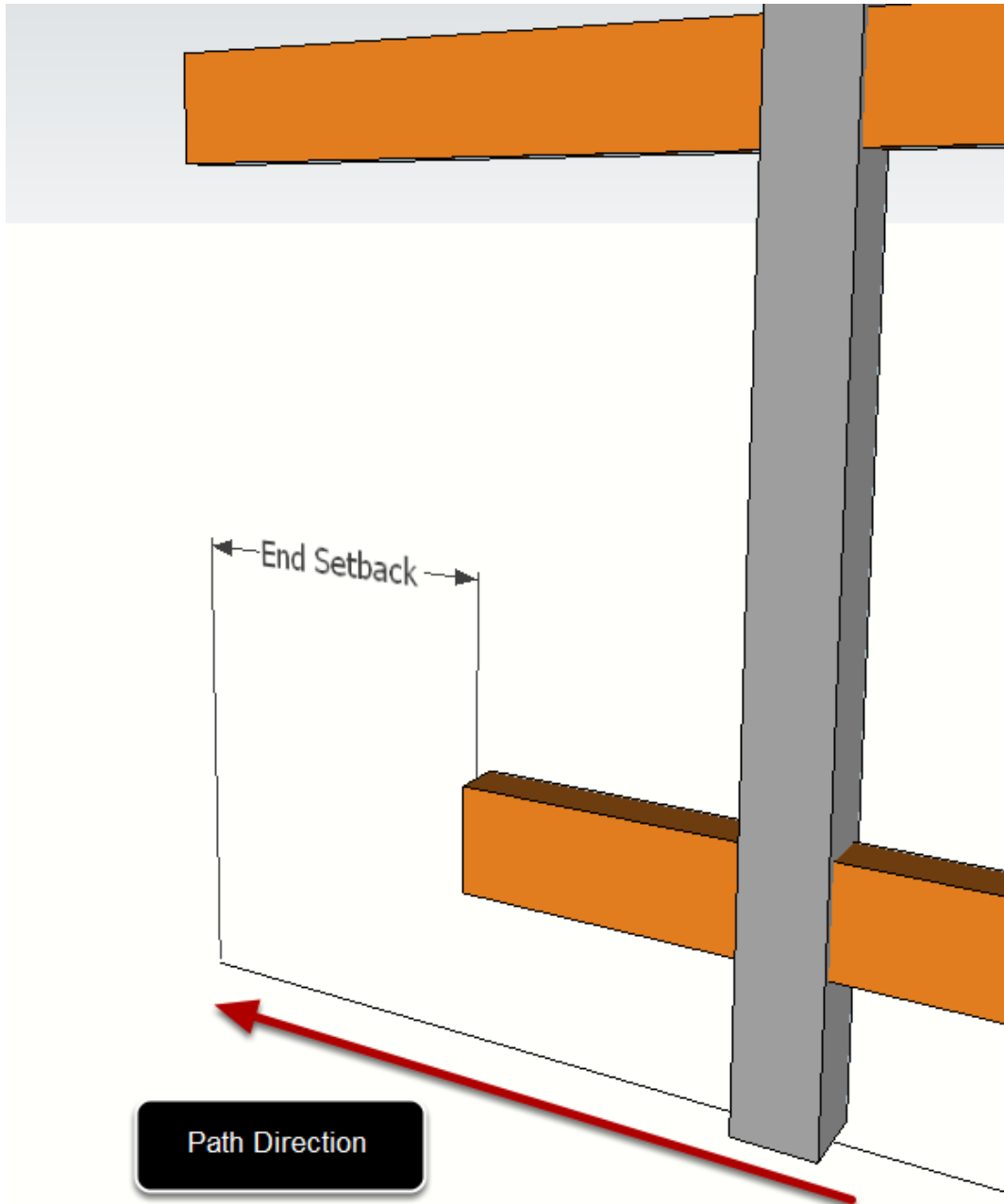
Assembly Attributes Reference

Profile Member - Start Setback



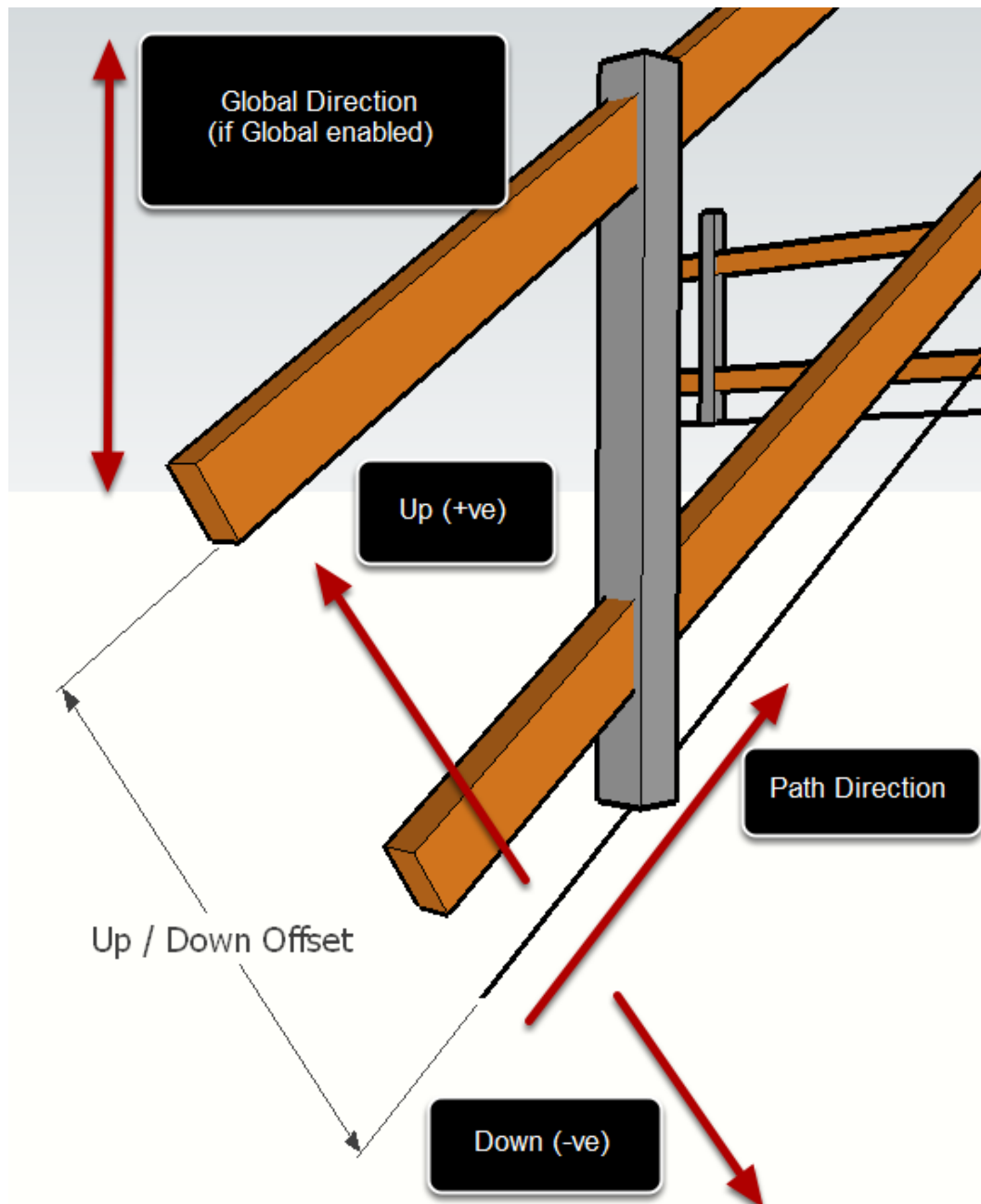
Start Setback defines the distance from the start of the assembly path that the Profile Member starts.

Profile Member - End Setback



End Setback defines the distance from the end of the path that the Profile Member ends.

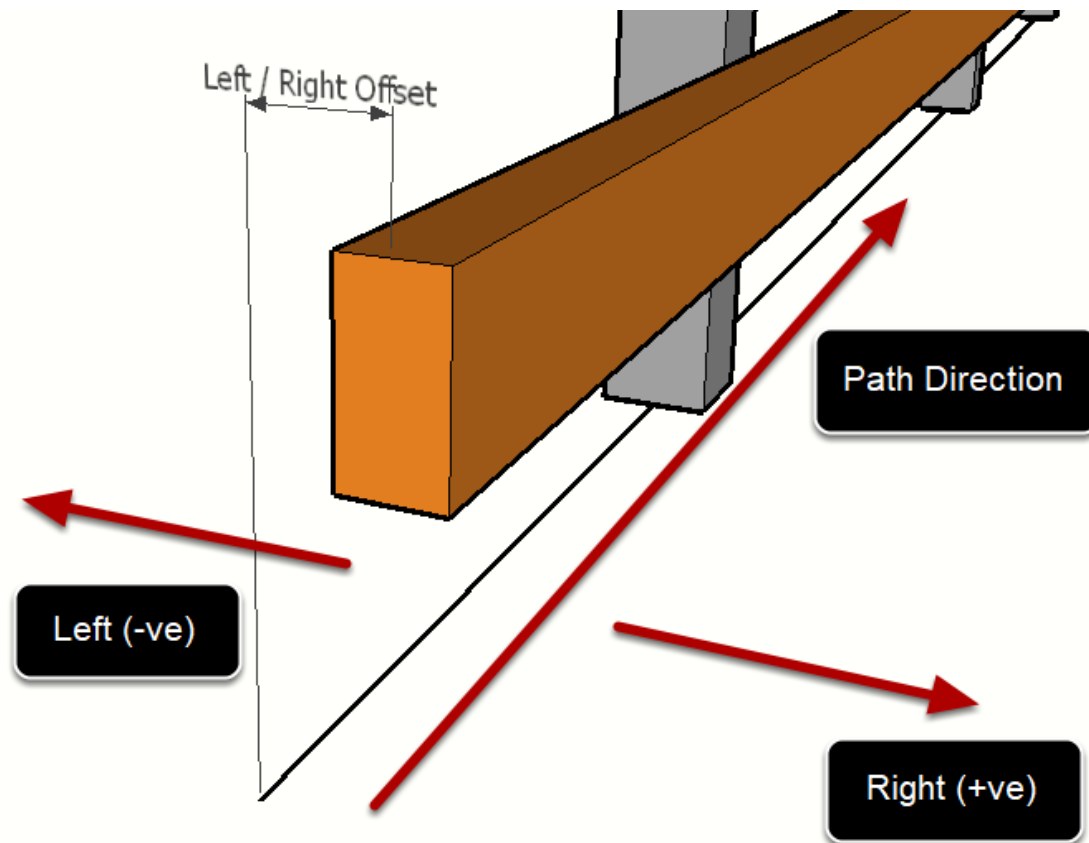
Profile Member - Up / Down Offset



Up / Down Offset defines the vertical offset of the Profile Member path relative to the Assembly path. Enter a positive value to offset up and negative value to offset down.

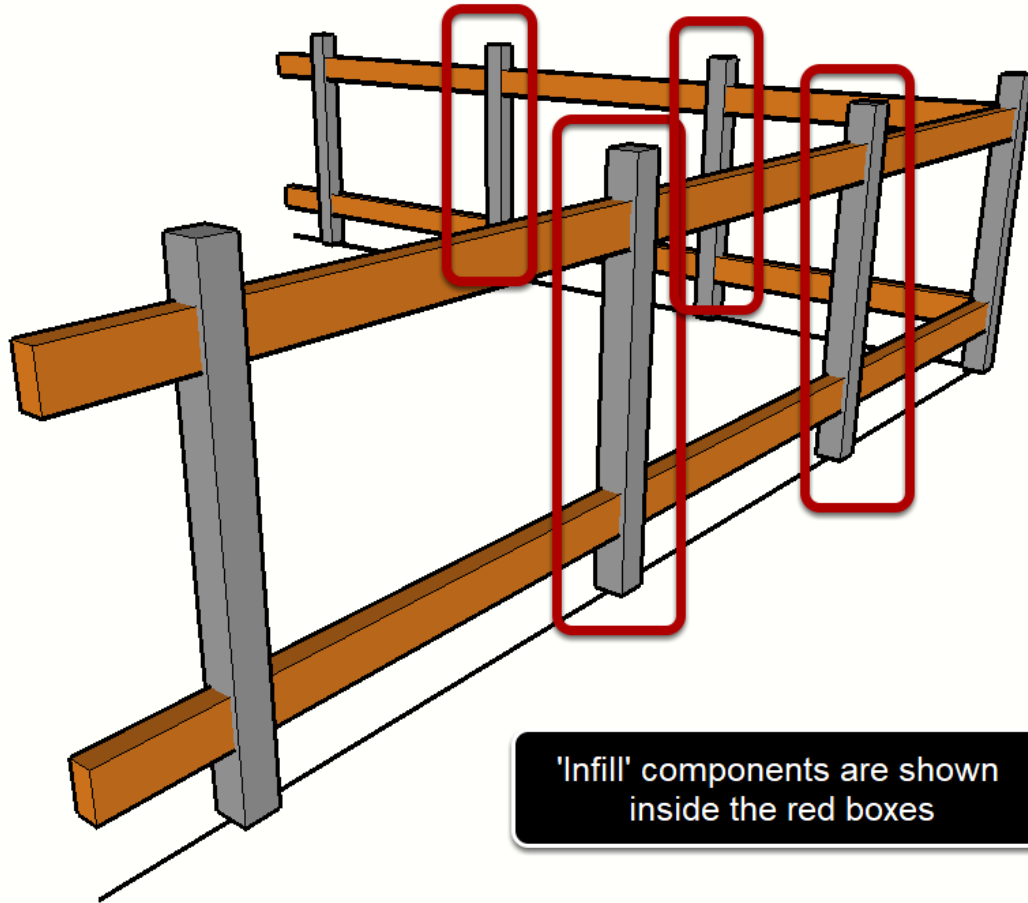
Use the Global checkbox to force the offset direction to be along the model blue axis. Otherwise, the offset will be along the local up / down axis which may vary if the assembly path is not horizontal.

Profile Member - Left / Right Offset



Left / Right Offset defines the horizontal offset of the Profile Member path relative to the Assembly path. Enter a positive value to offset to the right and negative value to offset to the left.

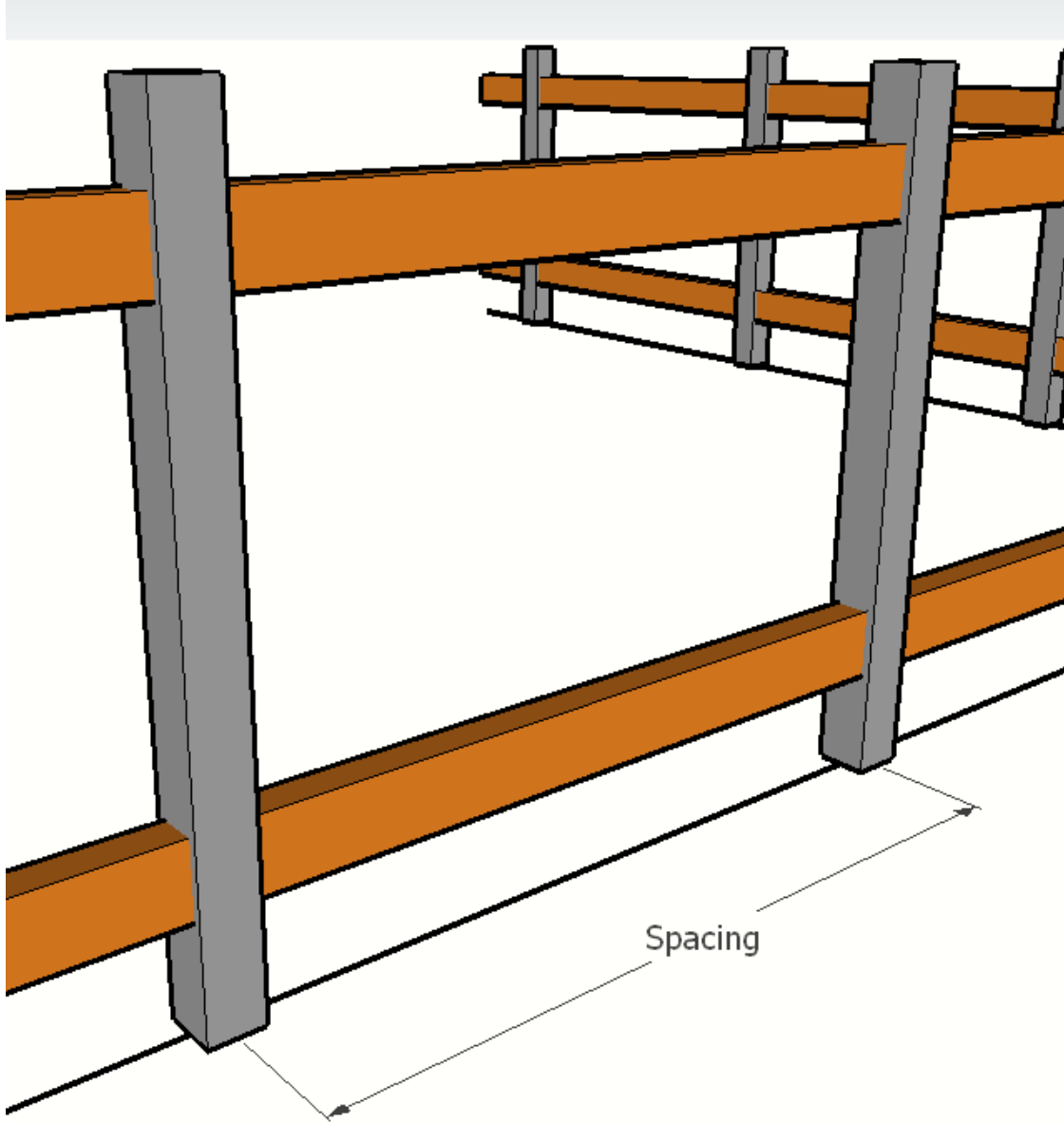
Component - Infill Checkbox



If the Infill box is checked, the component will be placed at regular intervals, defined by the following settings:

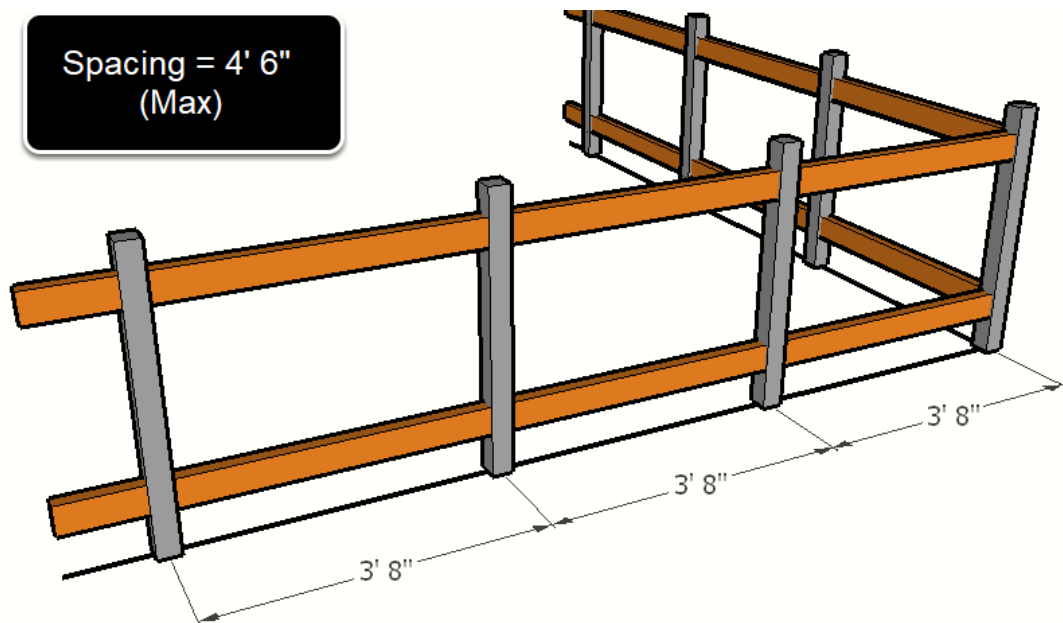
- Spacing
- Max (vs Fixed)
- Horizontal Distance (vs Distance along path)
- Layout (From Start or From Middle)

Component - Spacing



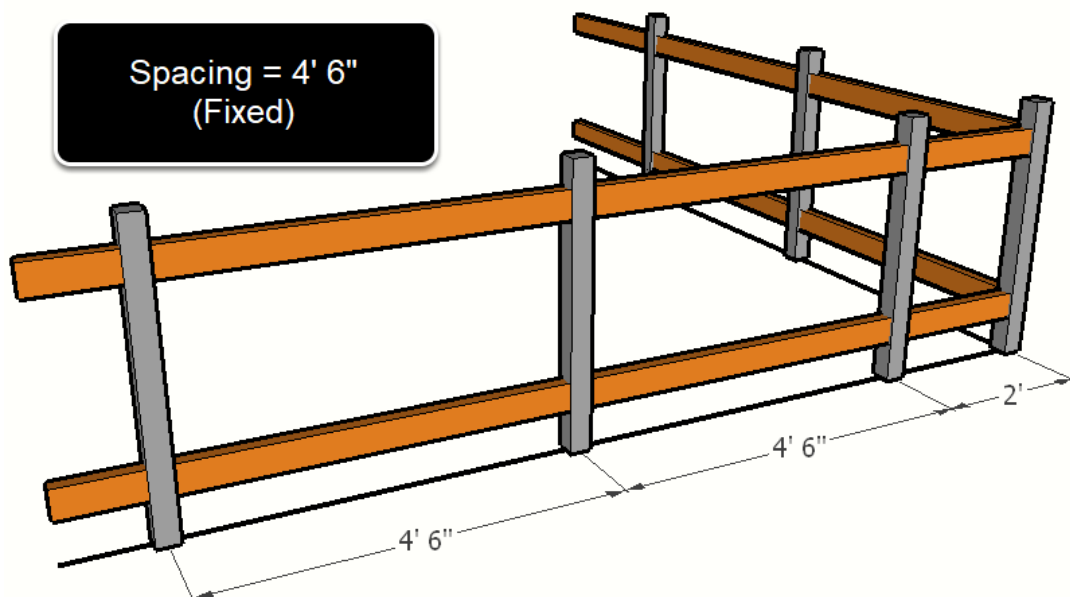
Spacing defines the distance between 'Infill' components.

Component - Spacing (Max)



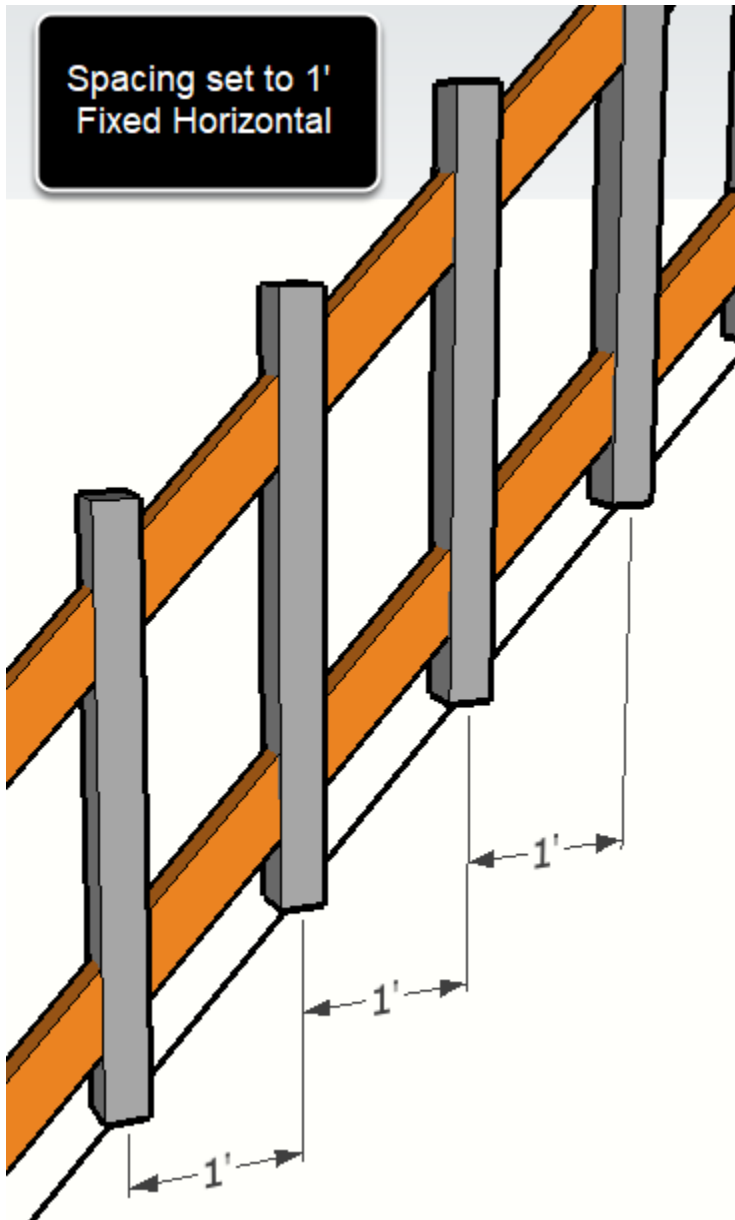
When 'Infill' components are set to 'Max' spacing, the spacing between infill components will not exceed the value entered for spacing and the components will be evenly distributed.

Component - Spacing (Fixed)



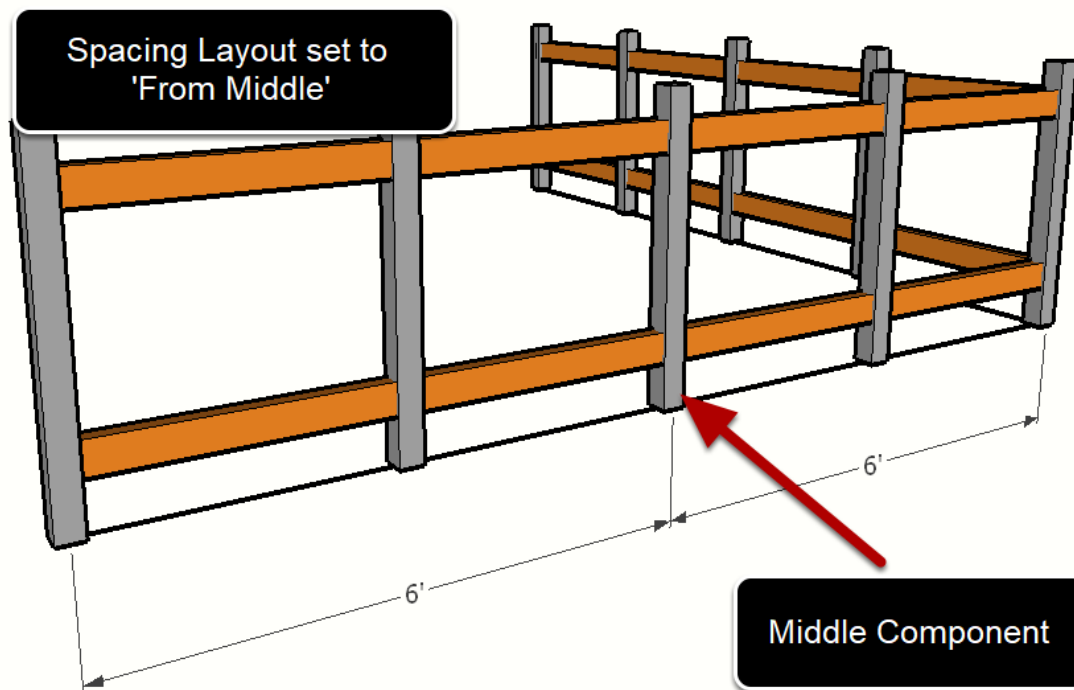
When 'Infill' components are set to 'Fixed' spacing (Max turned off) the spacing between infill components will be fixed to the value entered for spacing (where possible).

Component - Spacing (Horizontal)



When 'Infill' components are set to 'Horizontal' spacing, the spacing will be measured horizontally. If this setting is turned off, then the spacing will be measured along the path of the Assembly.

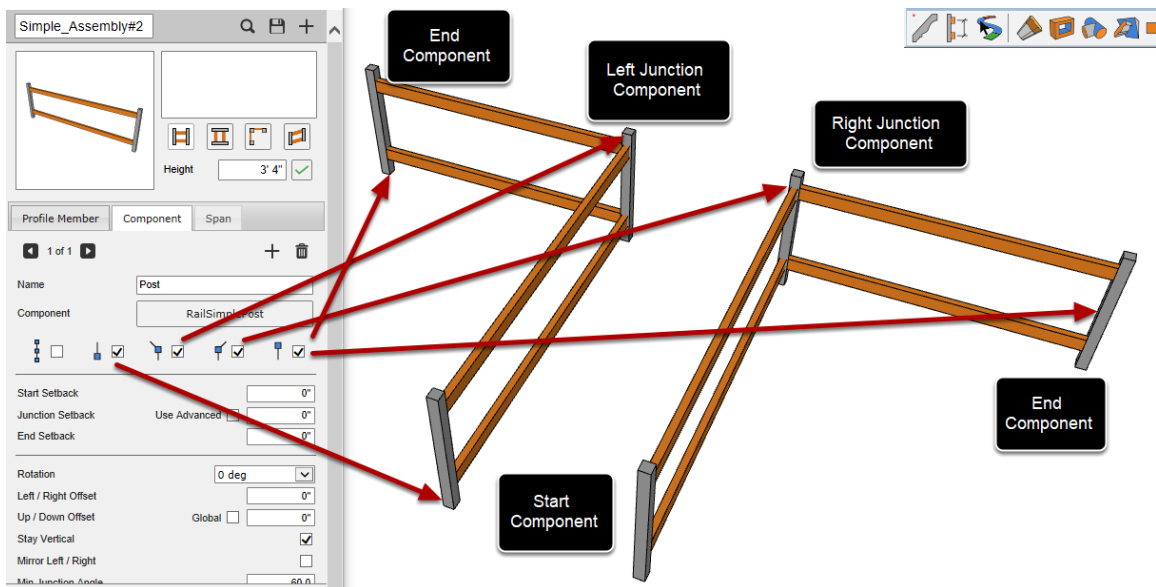
Component - Spacing Layout



When the Layout is set to 'From Middle', a component will be placed at the middle point between two junctions and the remaining components will be laid out relative to the middle component.

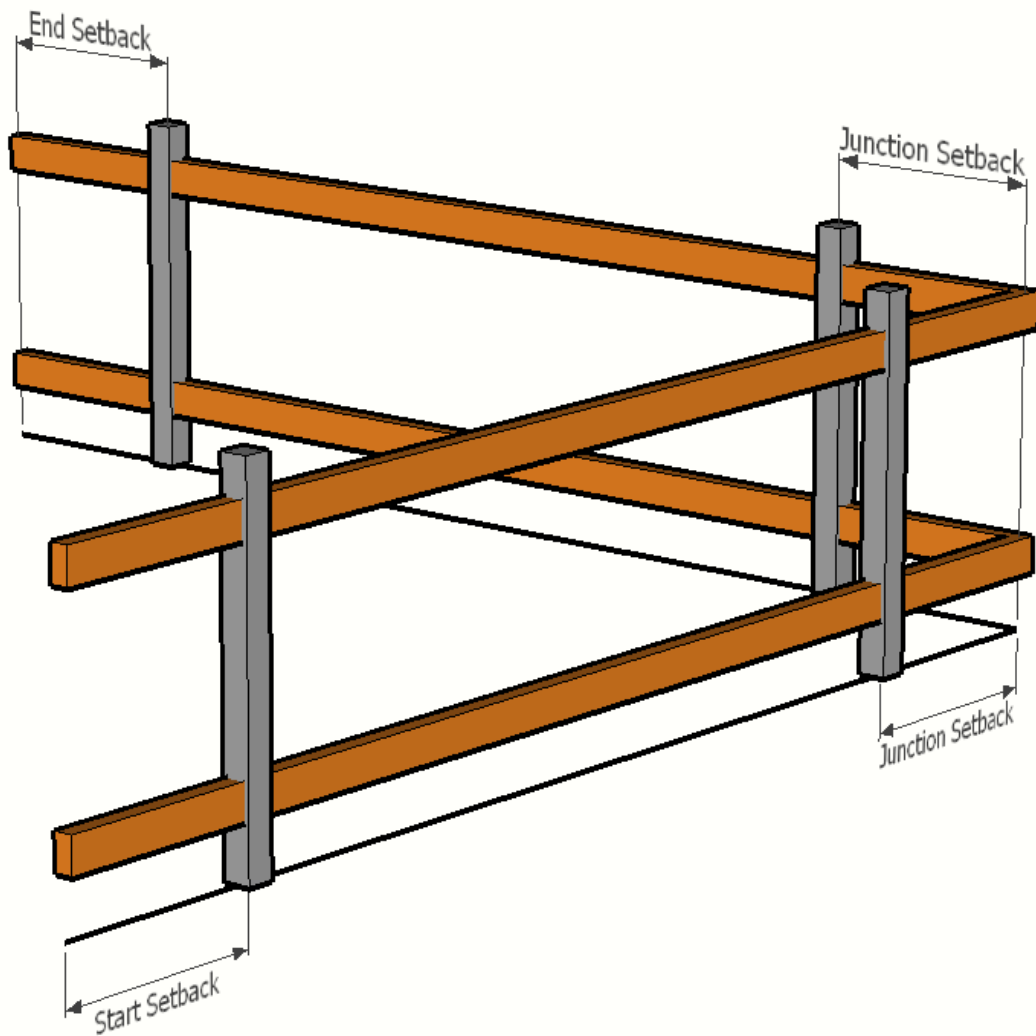
When the Layout is set to 'From Start', the components will be laid out relative to the start point between two junctions.

Component - Start, Junctions, and End Checkbox



The Start, Junctions, and End checkboxes define whether to place components at the path start, path junctions (left or right), and at the path end.

Component - Start Setback, Junction Setback, End Setback

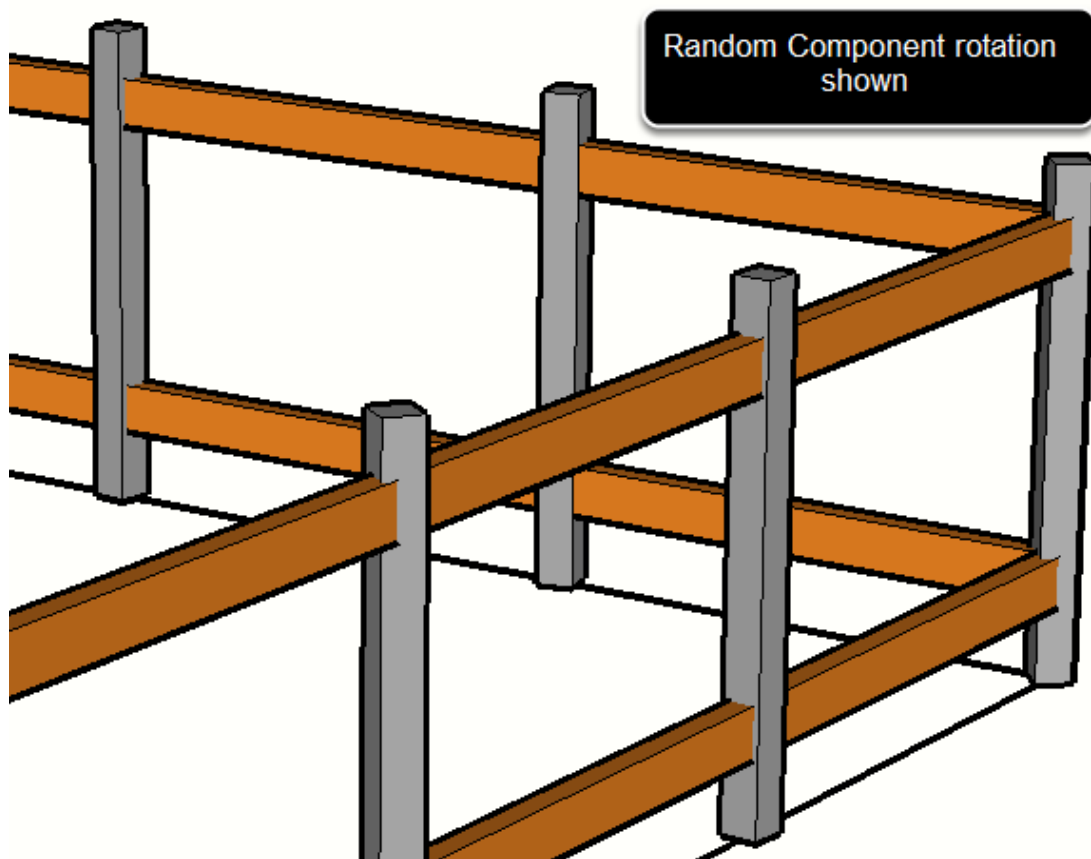


Start setback defines the distance from the path start to place the component.

Junction setback defines the distance from the path junctions to place the component.

End setback defines the distance from the path end to place the component.

Component - Rotation



The Rotation setting defines the rotation of the component about its Blue (up) axis. The options for rotation include:

- 0 deg (no rotation)
- 90 deg (1/4 turn)
- 180 deg (1/2 turn)
- 270 deg (3/4 turn)
- Average (no rotation except for at Junction locations. At junction locations, the component rotation will be averaged between the previous path edge and the next path edge)
- Smooth (This setting can be useful for curved or helical paths)
- Random (each component will be rotated randomly)

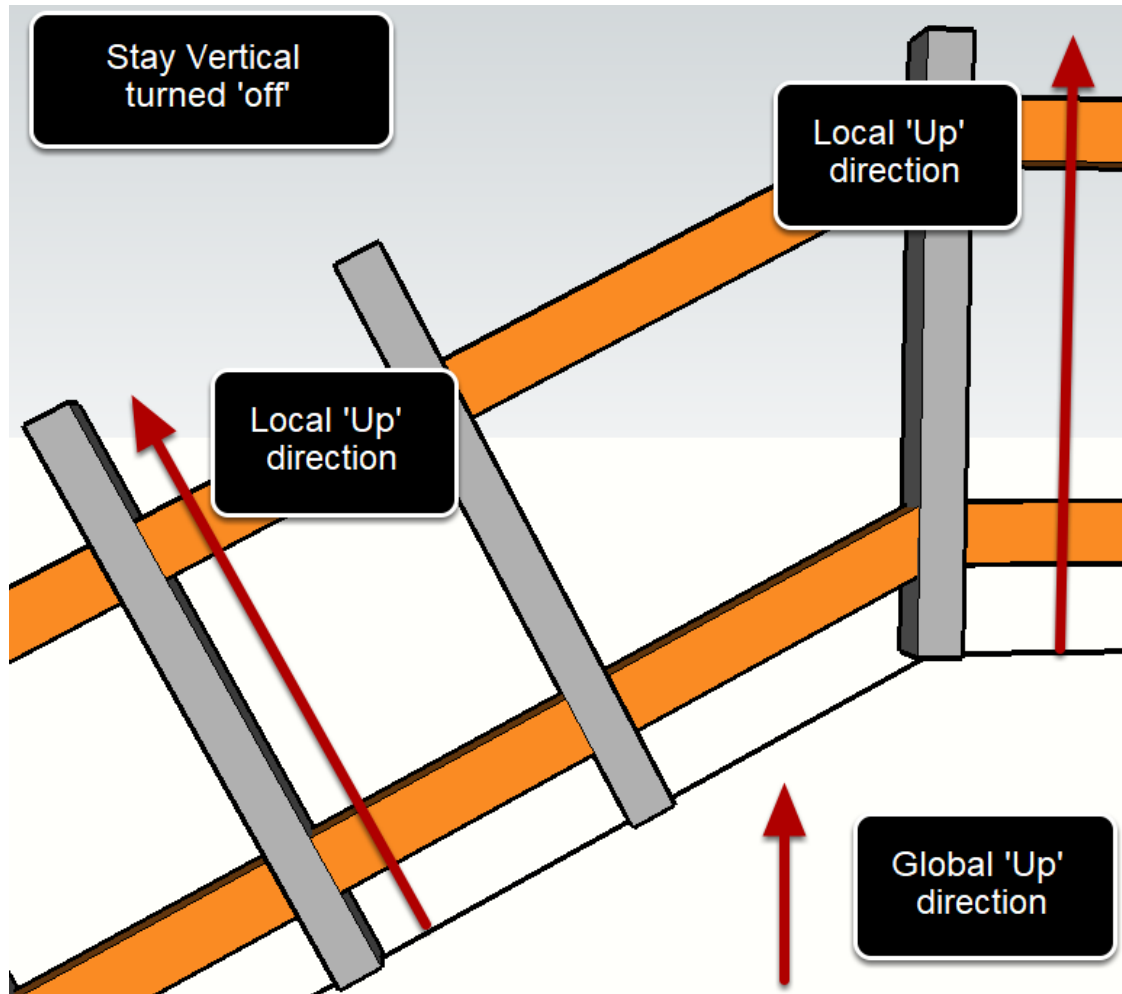
Component - Left / Right Offset

The Left / Right offset setting of a component is equivalent to the Left / Right offset setting of a Profile Member. See the Profile Member Assembly attributes for further details.

Component - Up / Down Offset

The Up / Down offset setting of a component is equivalent to the Up / Down offset setting of a Profile Member. See the Profile Member Assembly attributes for further details.

Component - Stay Vertical

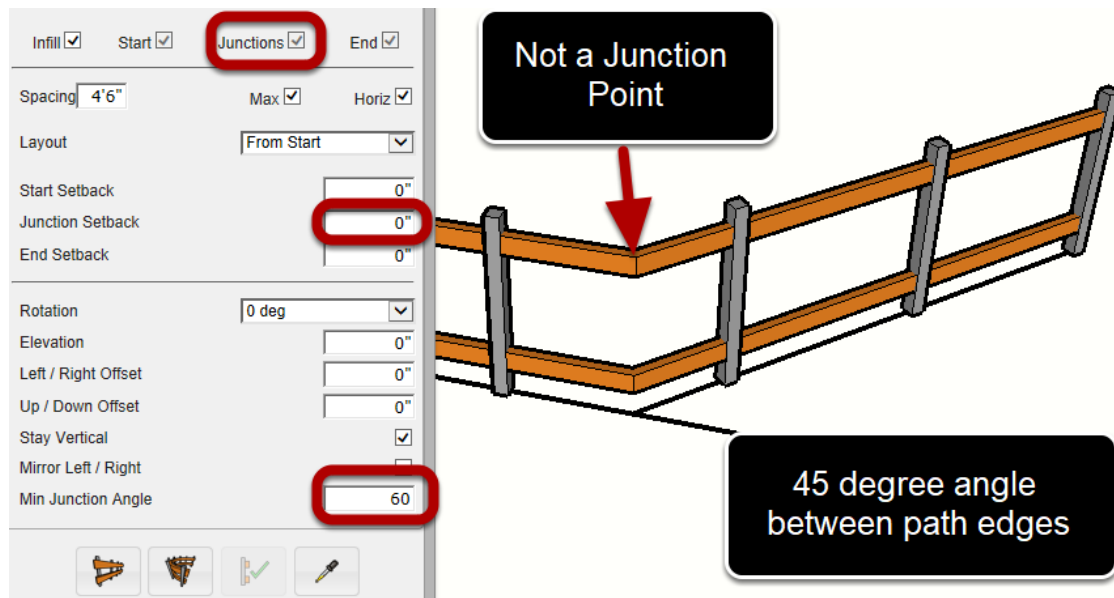


If the Component is set to 'Stay Vertical' then the Component's Blue axis will always point Up (globally), regardless of the path direction. If this setting is turned 'off', then the Blue axis will align with the local Up direction of the Assembly path.

Component - Mirror Left / Right

If this setting is enabled, the Component will be mirrored about the Up direction of the Assembly path. This is similar to the SketchUp command to Flip the component along it's Green axis.

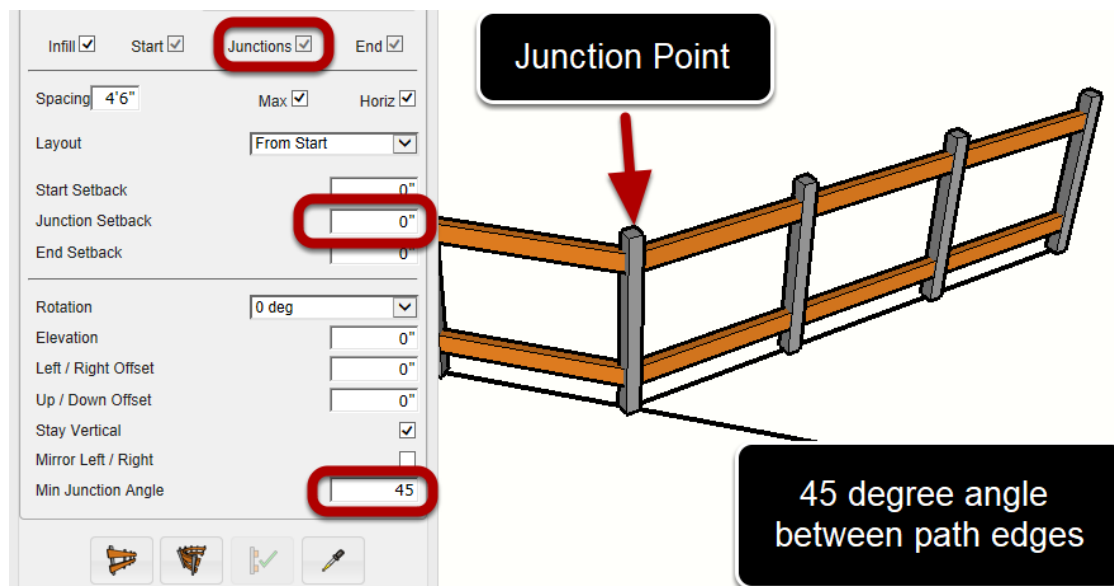
Component - Minimum Junction Angle



Minimum Junction Angle sets the angle between edges that is required to form a junction point. If the angle between path edges is greater than the minimum angle, the vertex will be treated as a junction point.

A junction point is a discontinuity in the layout and spacing of an infill component. Junction points divide an assembly path into segments.

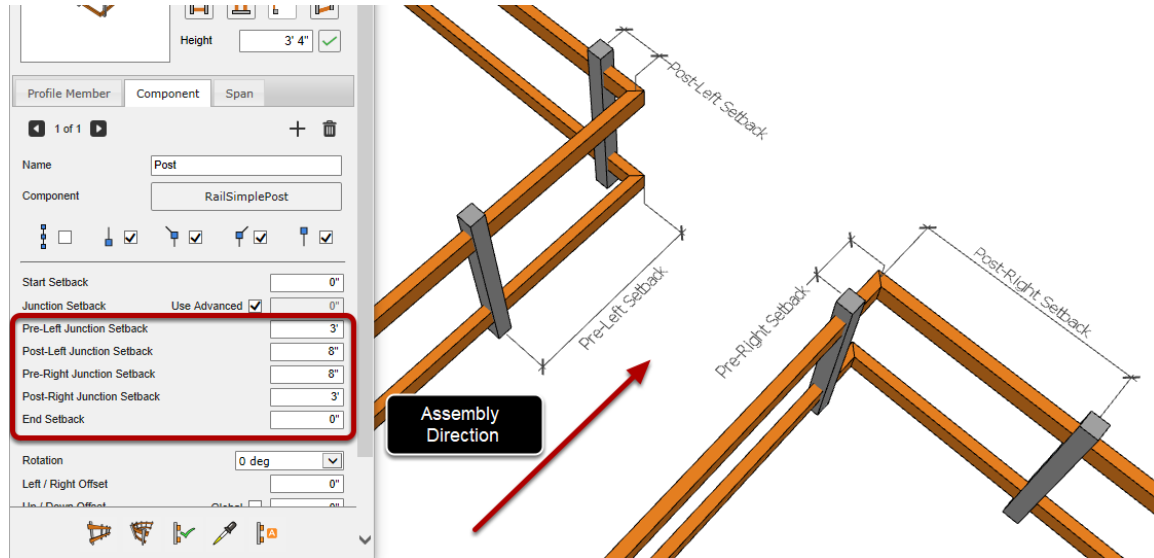
Components can be located directly over junction points if the junction setback distance is set to zero.



By decreasing the minimum junction angle to 45 degrees, a junction point will be created every time the assembly path changes direction by an angle of 45 degrees or more.

TIP: If you don't want your assembly path to ever create junction points for a component part, set the minimum junction angle to 180.

Advanced Junction Setbacks



Component parts may use advanced junction setbacks. This allows greater control of the position of Components before and after junctions.

Advanced Assemblies

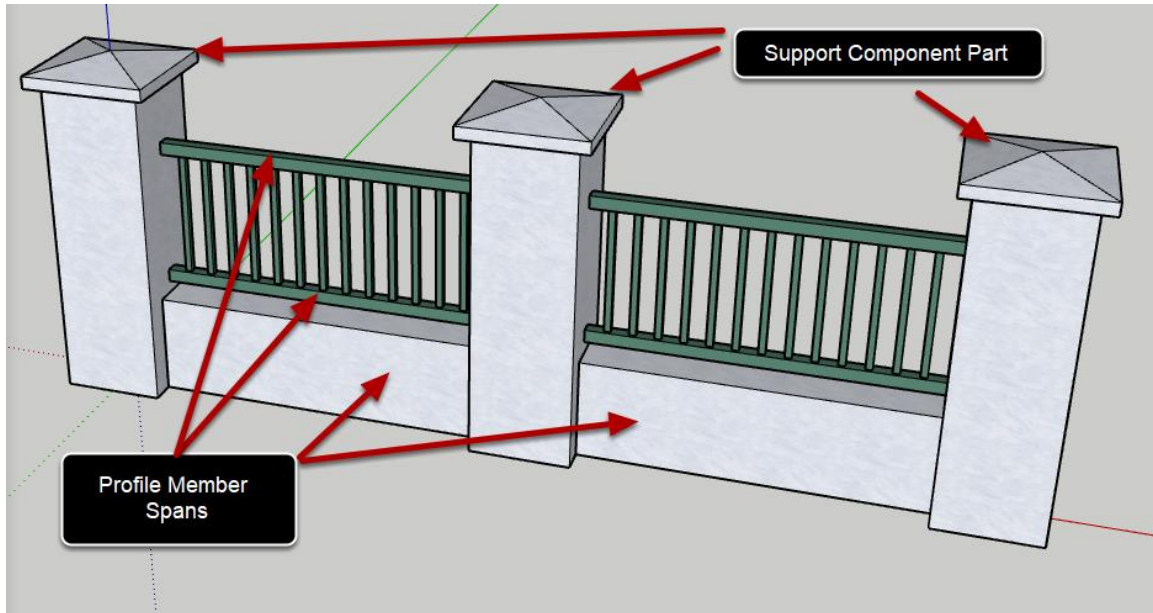
Spans

Spans are an incredibly powerful feature and can be used to create highly detailed complex assemblies. There are three kinds of span parts: Profile Member spans, Component spans, and Sub-Assembly spans.

The span types have some common attributes but they also have some attributes that only apply to a specific type of span.

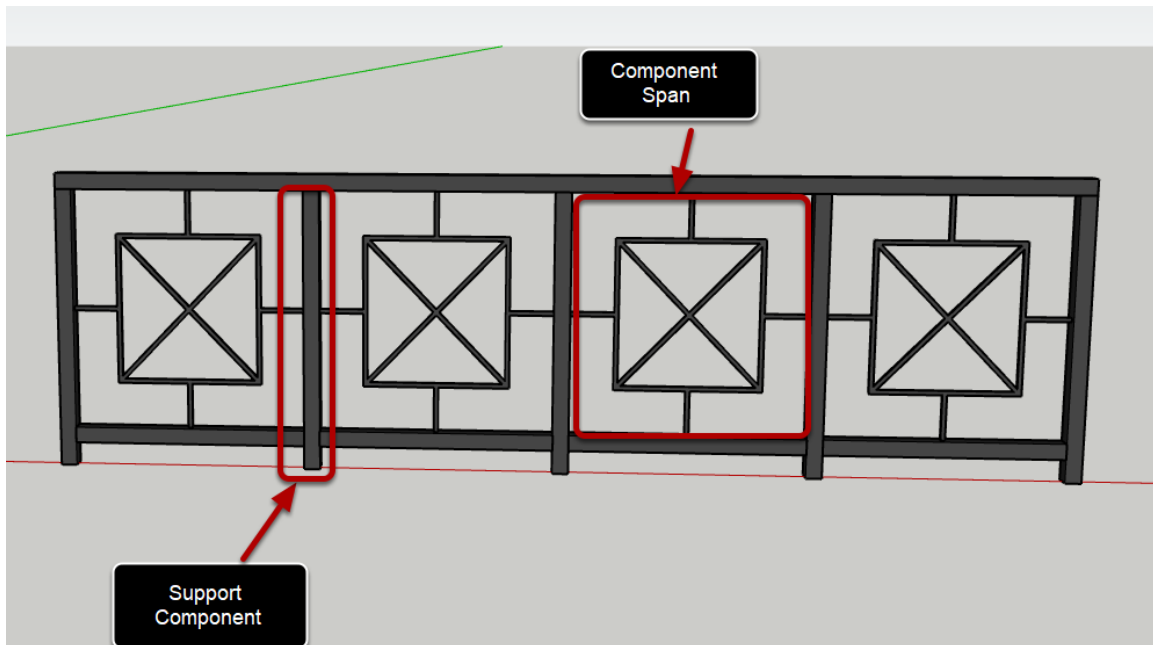
All spans are anchored by a 'Support' component part. The span parts will connect between two adjacent component parts.

Profile Member Spans



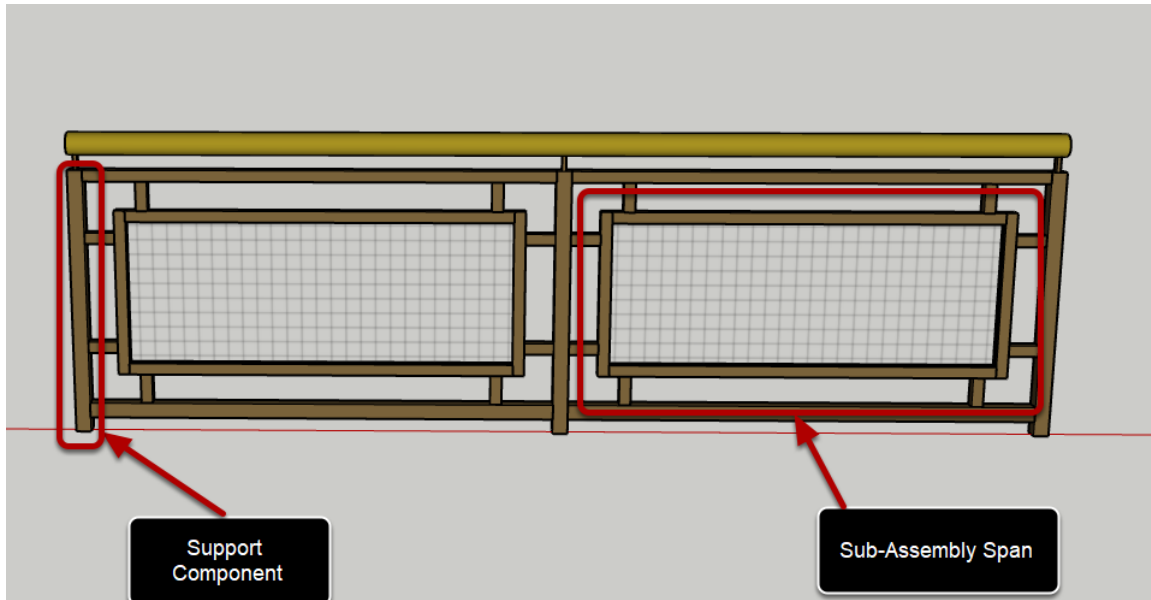
For this type, Profile Members will be created at each span location that connect between the adjacent support components.

Component Spans



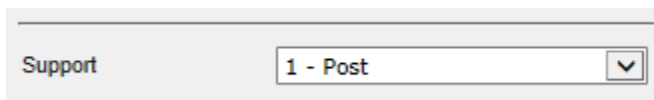
For this type, the specified component will be created at each span location.

Sub-Assembly Spans



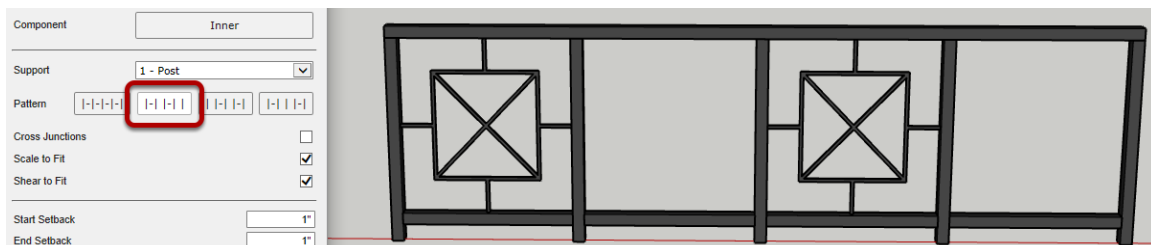
A sub-assembly is a nested assembly that will be created at each span location. Sub-assemblies can also be comprised of their own component, profile member, and span parts.

Common Span Attributes: Support



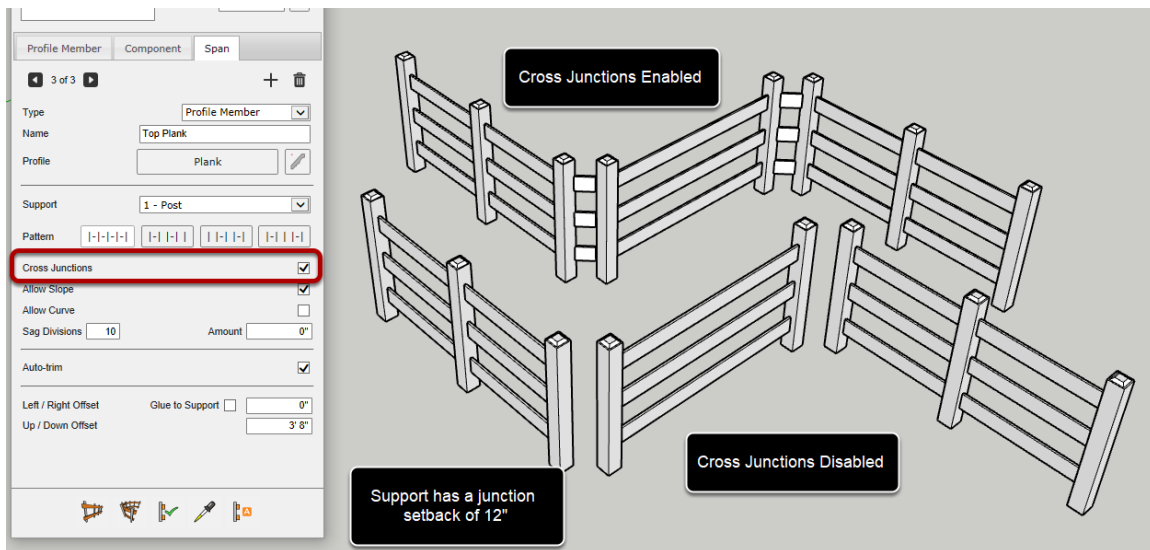
Each span part must have a valid support component part or it will not be created. To create component parts, select the 'Component' tab inside the Assembly Dialog. Then, choose the span support from the drop-down list.

Common Span Attributes: Pattern



By default, spans will be created between all support components. However, it is possible to select other layout patterns.

Common Span Attributes: Cross Junctions



Junctions are discontinuities in the layout of the support component. Junction locations are specified using the Junction Angle setting of the support component.

The support component must have a junction setback greater than zero for this setting to have an effect. Otherwise, the support will be located directly on top of the junction.

Common Span Attributes: Setbacks and Offsets

Start Setback	<input type="text" value="0"/>	
End Setback	<input type="text" value="0"/>	
Left / Right Offset	Glue to Support <input type="checkbox"/>	<input type="text" value="0"/>
Up / Down Offset	<input type="text" value="3' 8"/>	

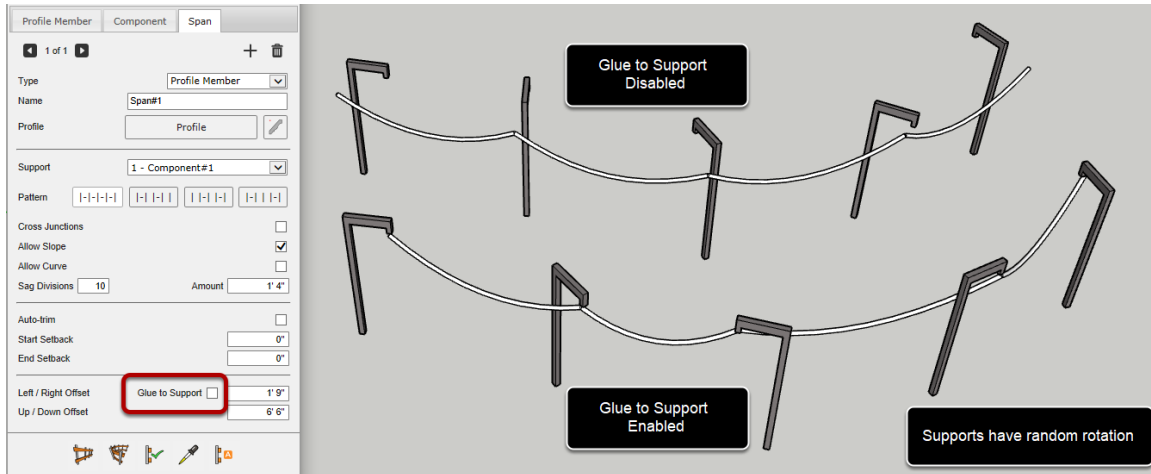
Start Setback: Distance from the axes of the start support to the start of the span (measured along the assembly path)

End Setback: Distance from the axes of the end support to the end of the span (measured along the assembly path)

Left / Right Offset: Distance from the axes of the support to the span (measured along the assembly local horizontal direction)

Up / Down Offset: Distance from the axes of the support to the span (measured along the support's blue axis)

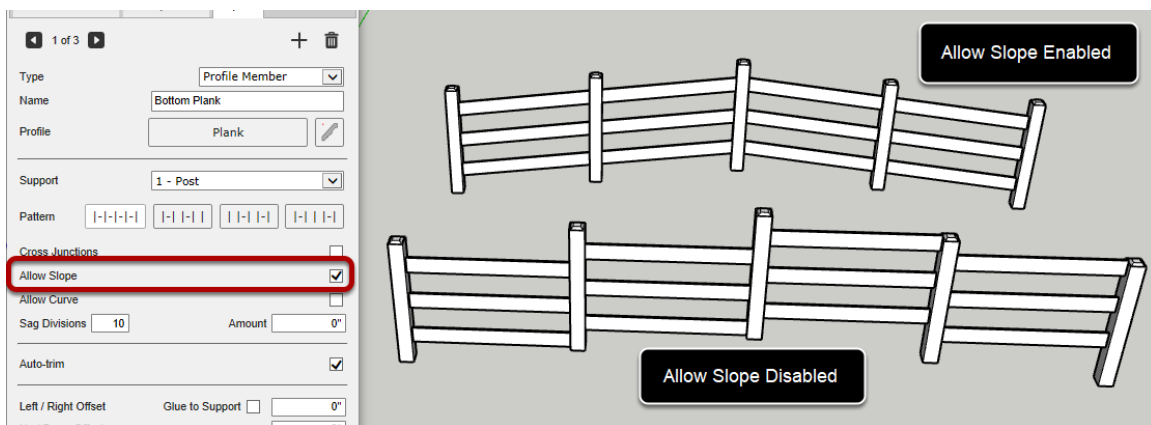
Common Span Attributes: Glue to Support



Glue to Support: If enabled, the Left / Right offset will be along the support's green axis and the setbacks will be along the support's red axis.

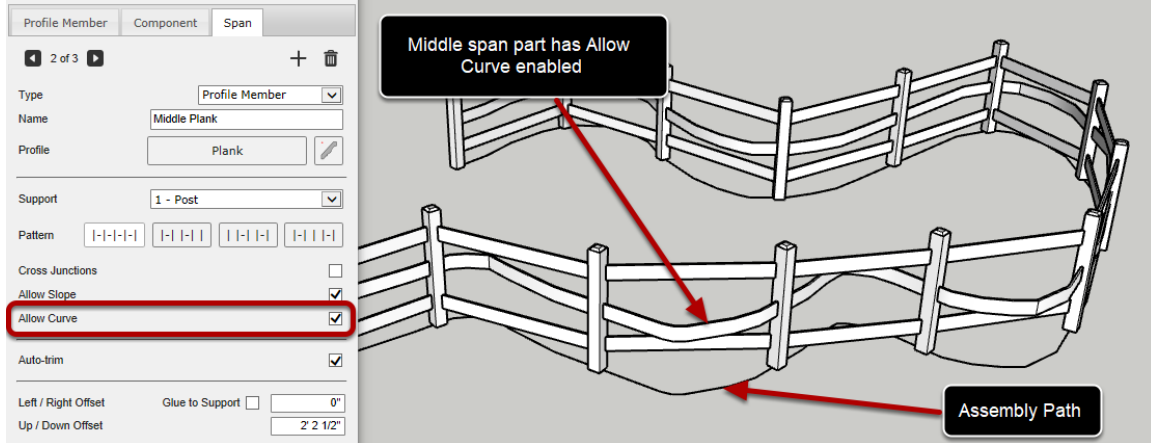
If you want the span to 'stick' to the same point on the support regardless of the rotation of the support, then enable this setting.

Allow Slope



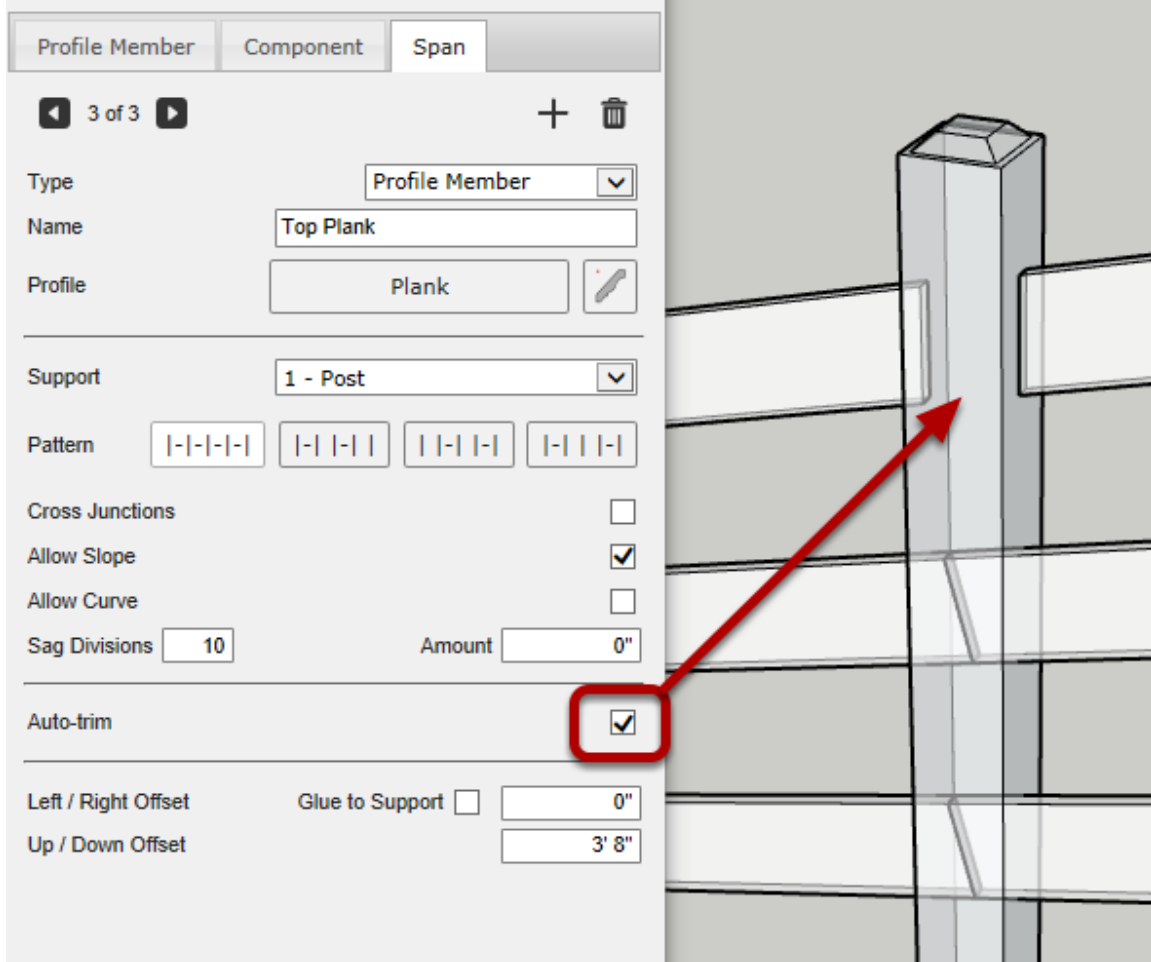
This attribute can be used for both Profile Member spans and Sub-Assembly spans. If disabled, the span will be forced to be horizontal.

Allow Curve



This attribute can be used for both Profile Member spans and Sub-Assembly spans. If enabled, the span will follow the curve (if any) of the assembly path. If disabled, the span will be created along a straight line between the supports.

Auto-Trim



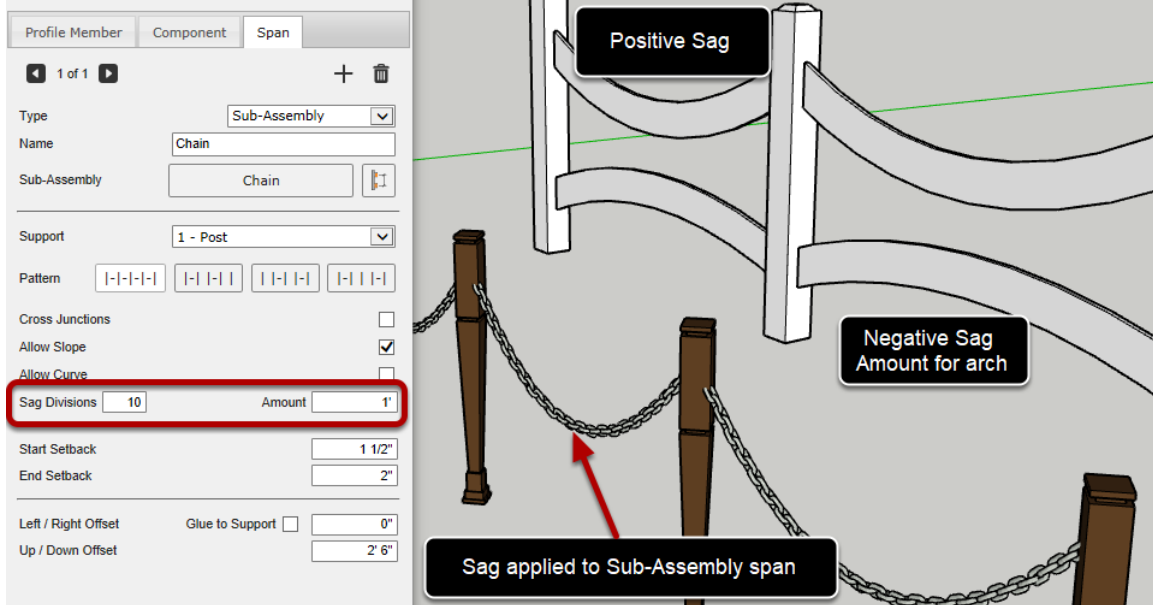
This attribute is only available for Profile Member spans. If enabled, each Profile Member span that is created will attempt to trim itself against the support components.

Note that only planar trims are supported for this feature.

If the Profile Member does not intersect the support, no trim will be applied and it will be as though the Profile Member was created with no trim or setback.

When auto-trim is enabled, the start and end setback settings are hidden since they are not used in this situation.

Sag

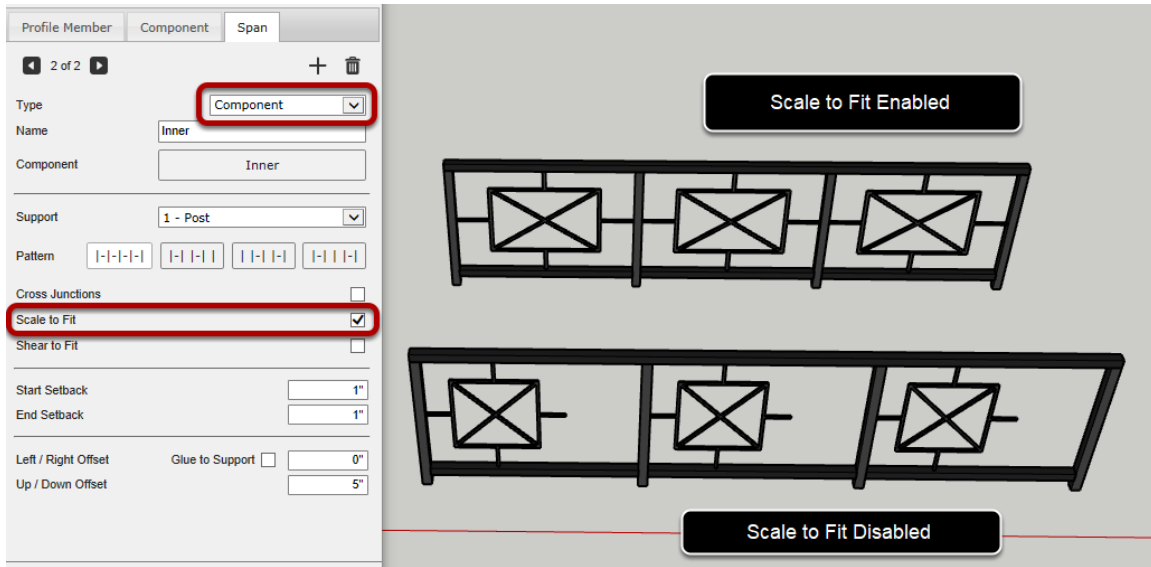


Sag can be applied to a Profile Member span or a Sub-Assembly span by entering a non-zero sag amount.

Sag Divisions control the number of segments that the span will be divided into before the sag is applied.

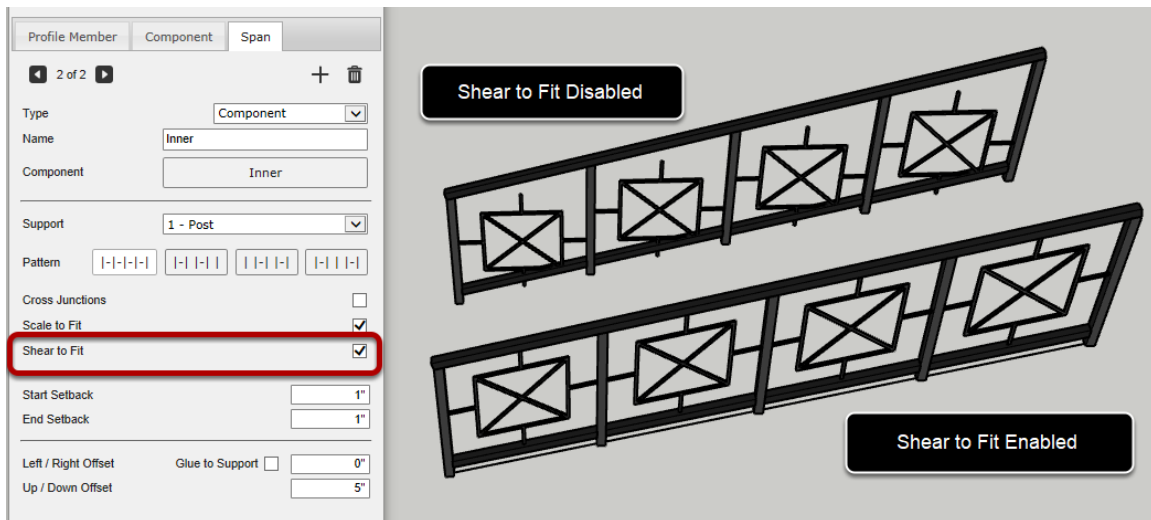
Sag Amount is the maximum distance for the sag to be applied at the middle of the span. Negative values can be used to create an arch effect.

Scale to Fit



This attribute can only be applied to Component spans. If enabled, each span will be scaled so that it fits exactly between the supports (taking into account the start and end setback settings).

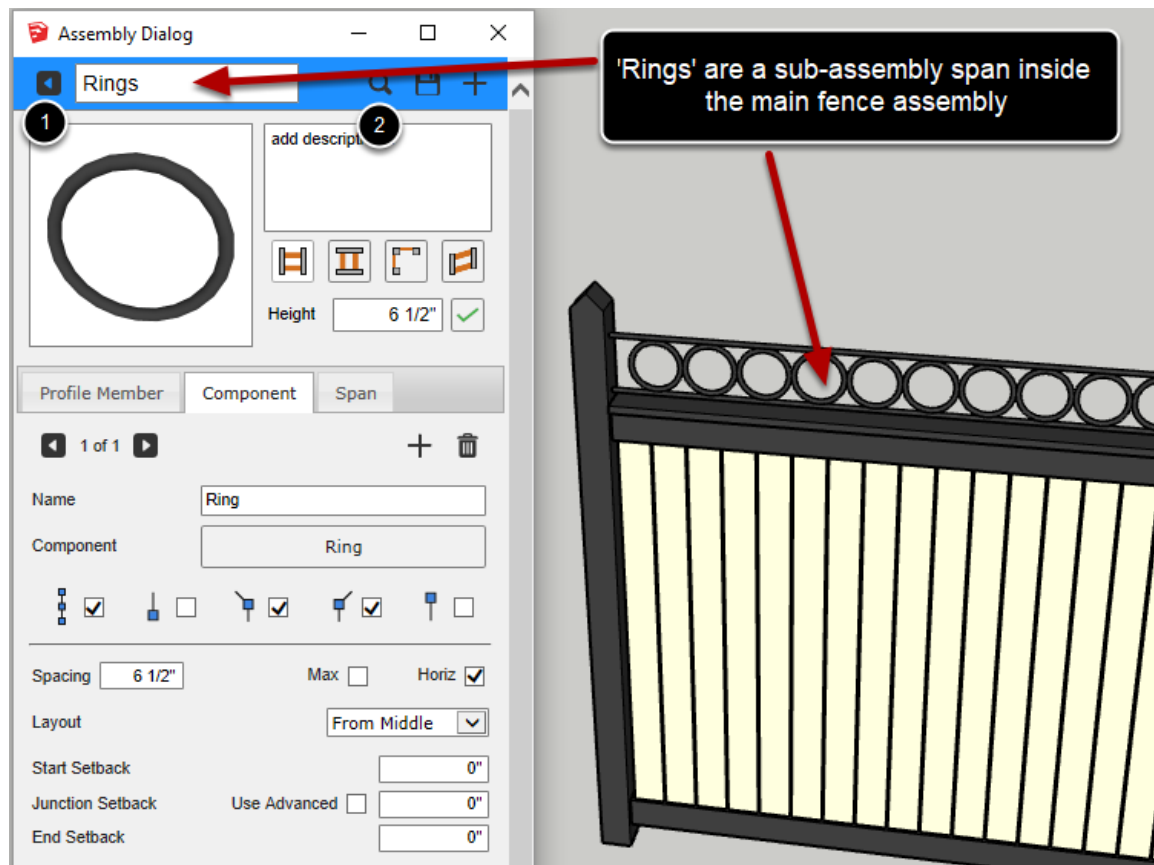
Shear to Fit



This attribute can only be applied to Component spans. If enabled, a shearing transformation will be applied if necessary so that the component span fits against the support.

This attribute is important if the assembly is built along a sloping path.

Editing Sub-Assemblies

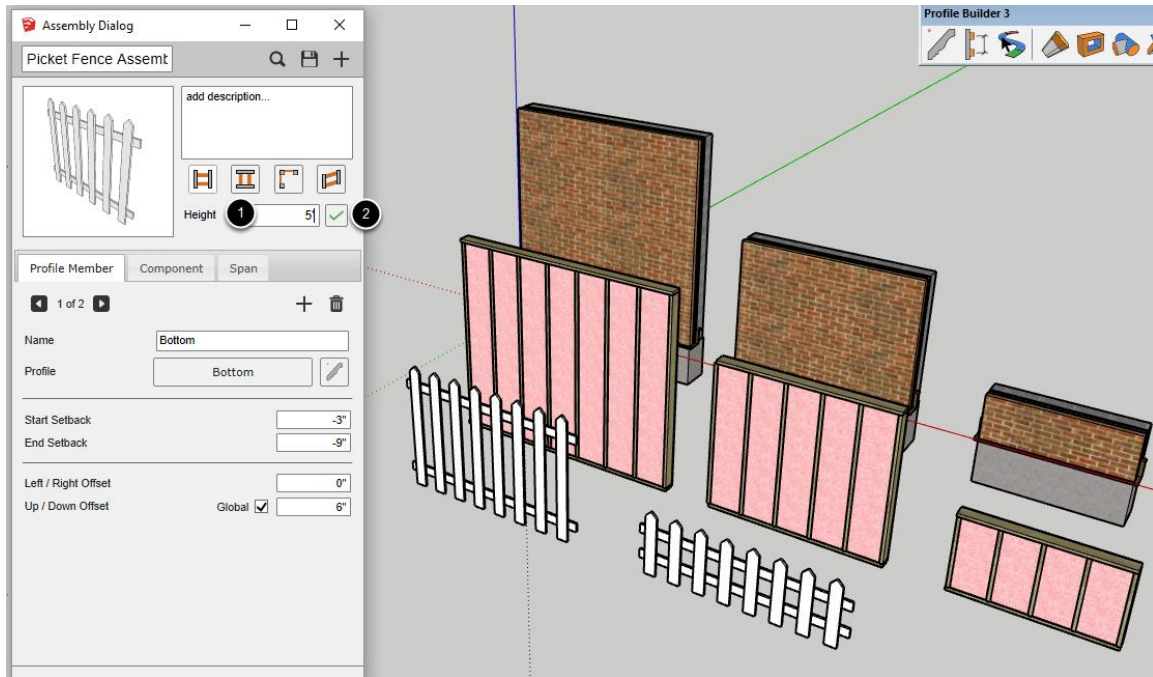


When editing a sub-assembly span, the top of the Assembly Dialog window will turn blue and a new 'back' button will be visible so that you can navigate back to the parent assembly span.

1. Click the 'back' button to go back to the parent assembly span.
2. It is possible to use the assembly browser to load a sub-assembly from a saved assembly SKP file.

Setting the Height of an Assembly

Set the Height



The height of an assembly can be easily modified.

1. Enter the desired total height of the assembly.
2. Click the 'Apply Height' button to change the height of the assembly.

Note that any selected assemblies in the model will NOT be modified. In order to apply the new height to an assembly in your model, you need to select the assembly with the Select tool and then click the 'Apply Assembly Attributes' button.

When setting the height of an assembly, the entire assembly will be stretched and several changes may occur:

- Profile Member parts may have their own 'height' attribute modified to suit the new assembly height
- Component parts may be made unique and have their internal geometry modified to suit the new assembly height
- Profile Member, Component, and Span parts may have their Up / Down offset attribute modified to suit the new assembly height

Currently, Sub-Assembly spans will not be automatically stretched when setting the height of an assembly. However, if you click the 'Edit Sub-Assembly' button, you can then adjust the height of the sub-assembly as needed to suit the new height of the parent assembly.

Note: Components parts will not be scaled by this operation. Rather, the internal geometry will be stretched.

Tools

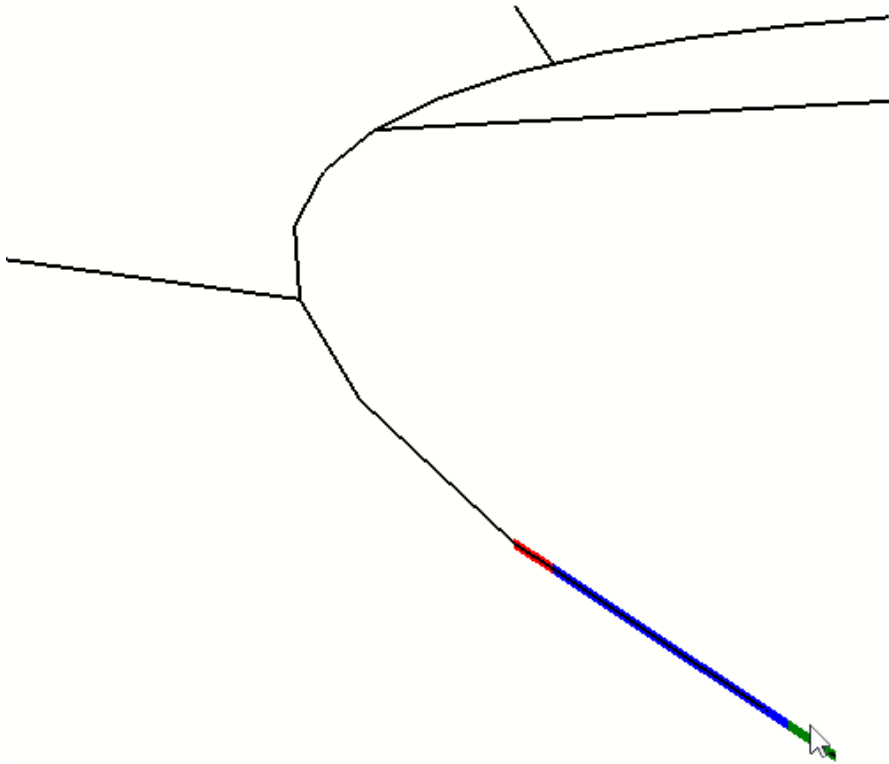
Smart-Path Select

Launch the Tool



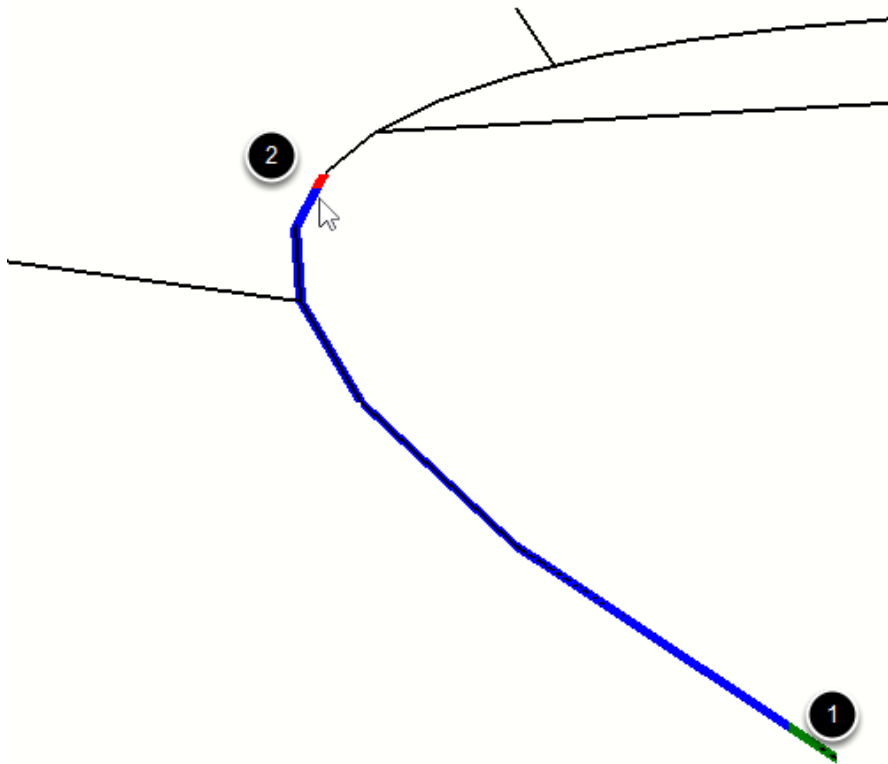
The Smart-Path Select tool makes it easy to select a complex series of connected edges.

Select the First Edge of the Path



Hover the mouse over an edge in your model. The vertex that the mouse is closest to will turn green representing the start position of the path. The end of the path will be colored red.

Click another Edge on the Path

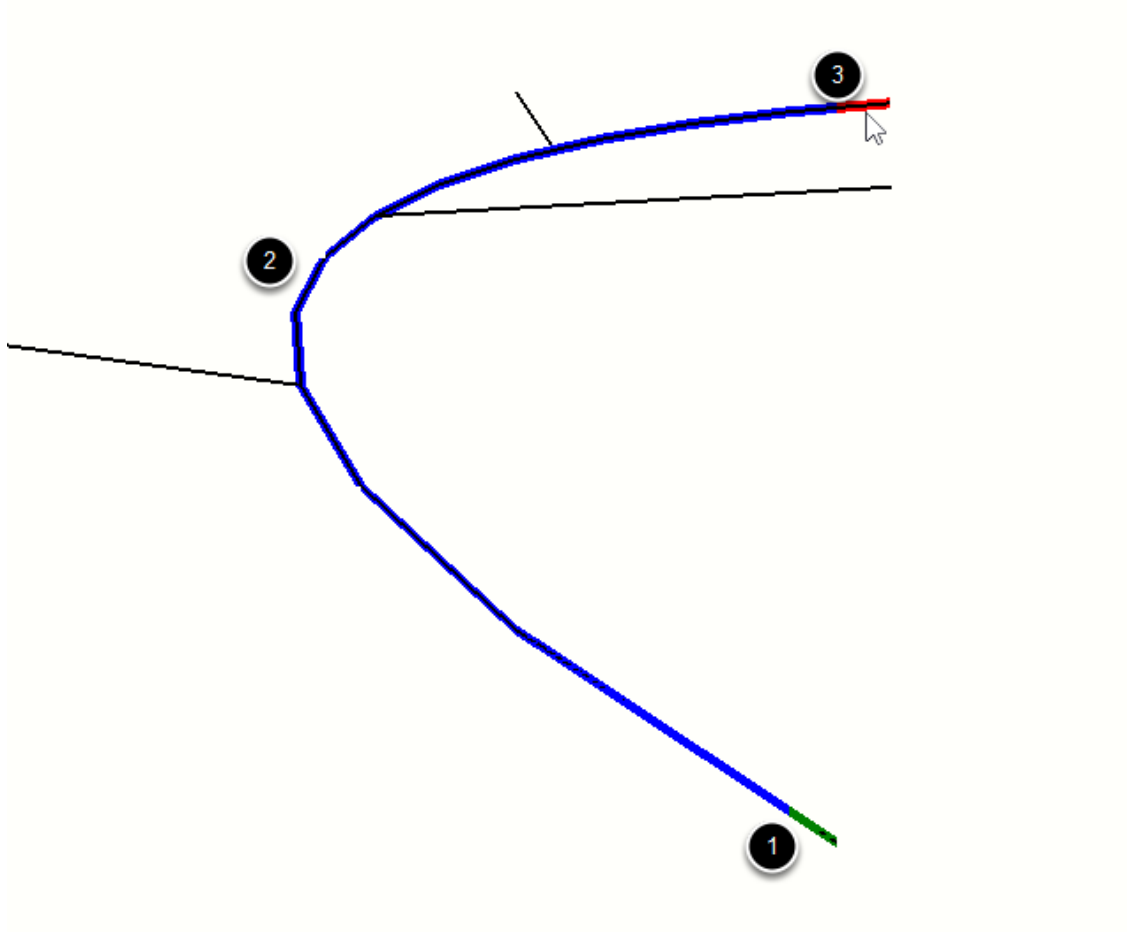


1. Click the first edge.
2. Click another edge along the path.

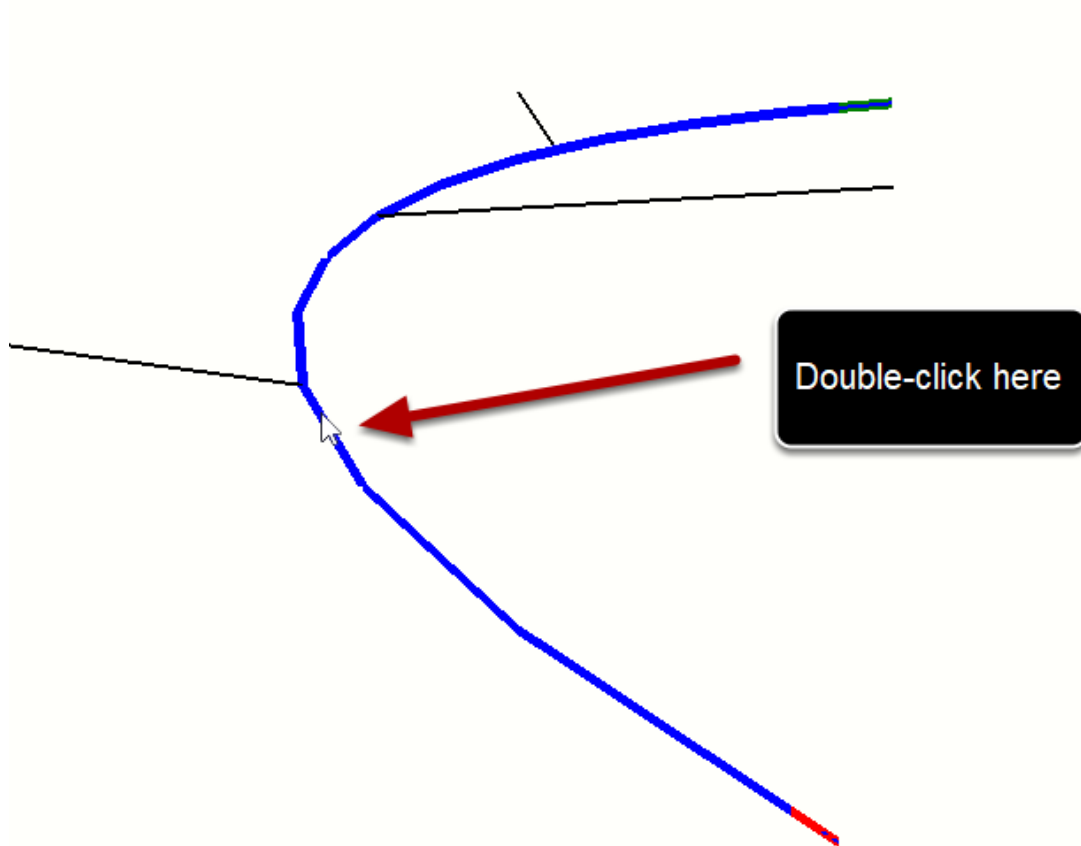
When you click on the next edge, all edges along the path between the two edges will also become selected.

TIP: If the first edge you click is a boundary edge, the tool will try to find other boundary edges along the path.

Continue clicking edges along the path



Double-click an edge to select a Smart-path



1. Double-clicking on a middle edge will select the edges as shown.

The selected path will generally be the one with the smallest angles between the edges.

Modifier Keys

ESC = clear selection

CTRL = add a new path to the selection (multiple paths may be selected)

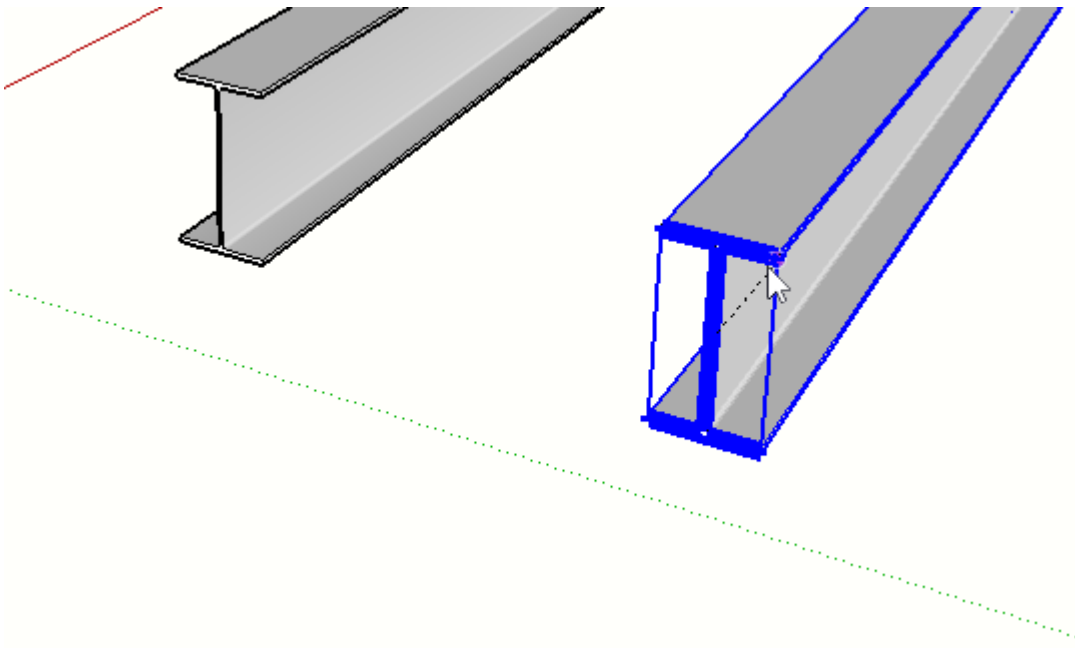
ENTER or RETURN = complete selection (optional)

Extend Profile Member

Launch the Tool

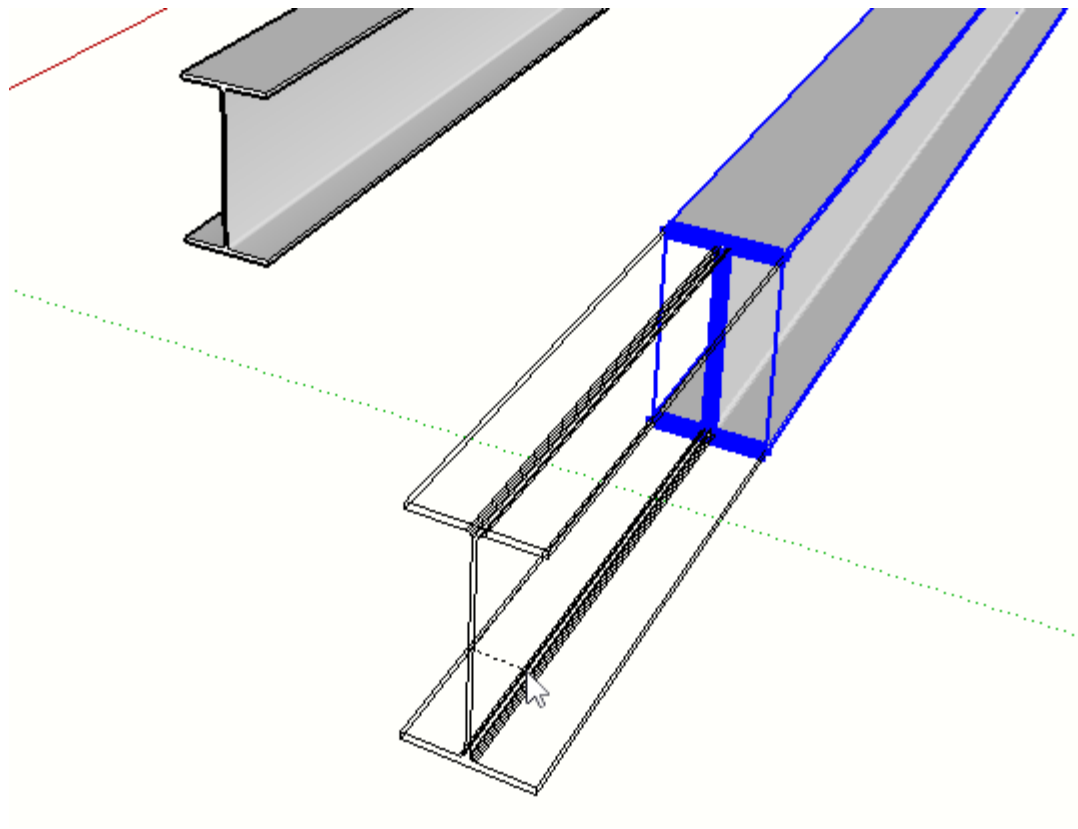


Click the end of a Profile Member to Modify

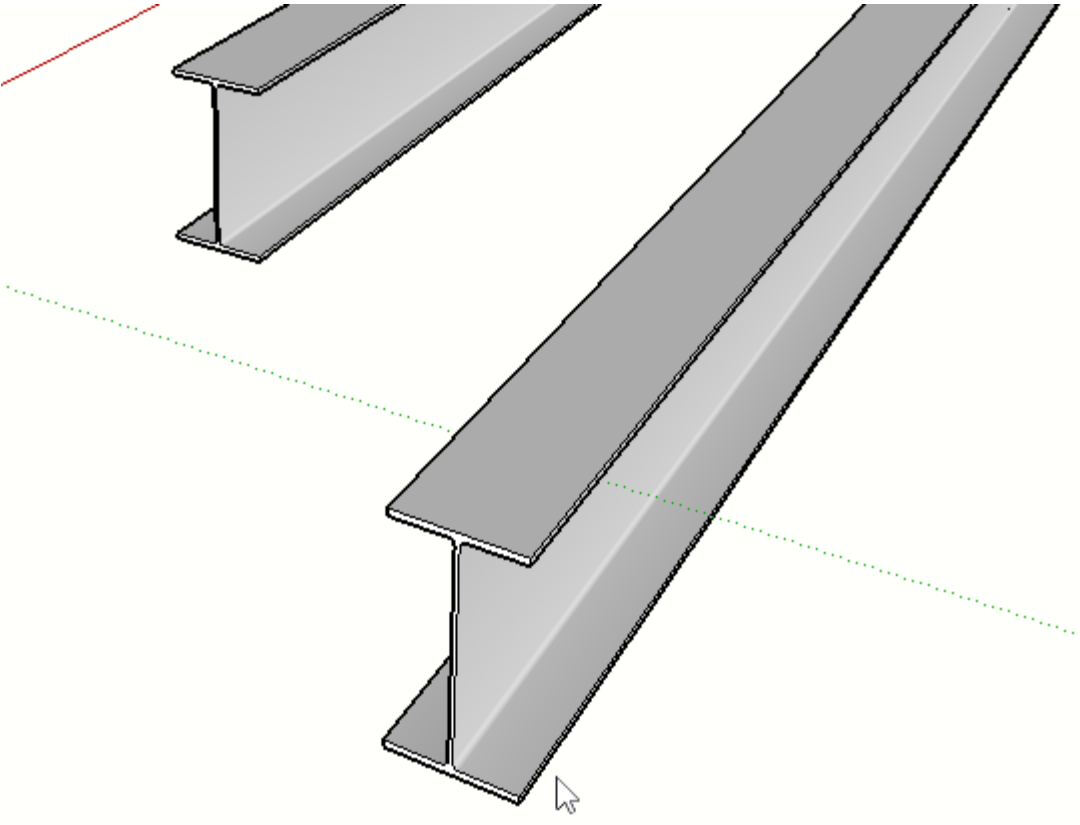


The selected end of the Profile Member becomes highlighted.

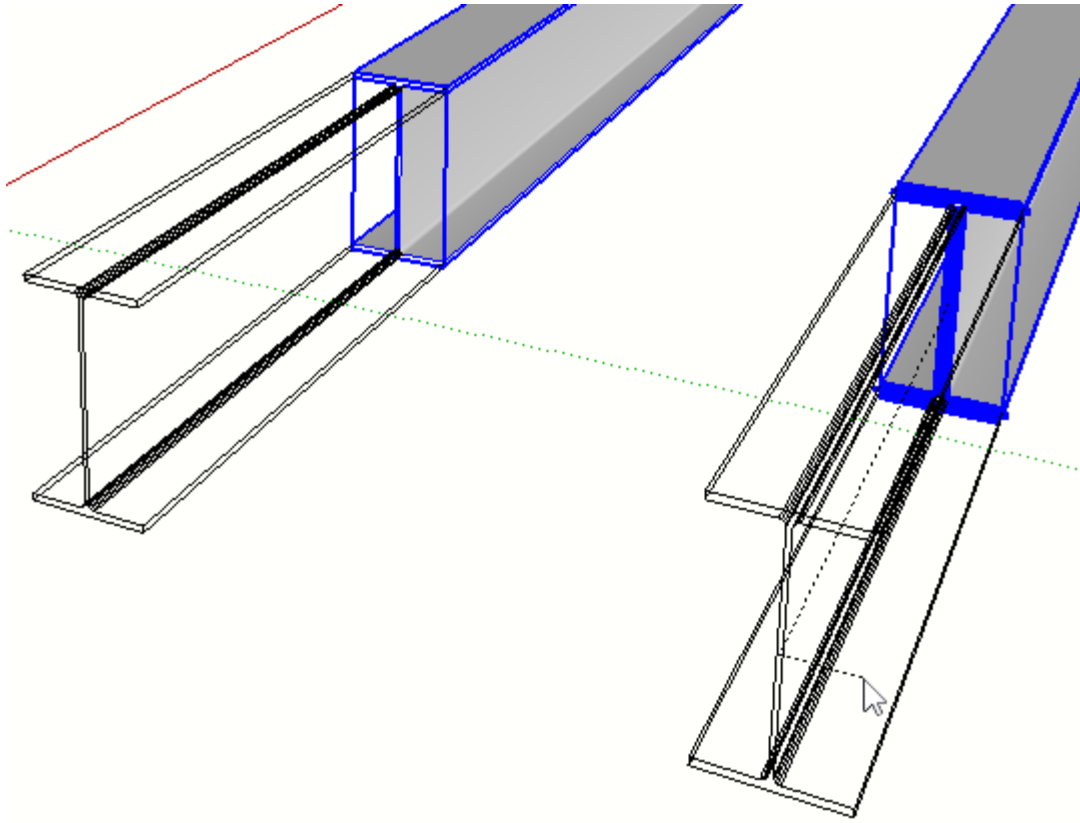
Move the mouse to define the extension distance



Click again to complete

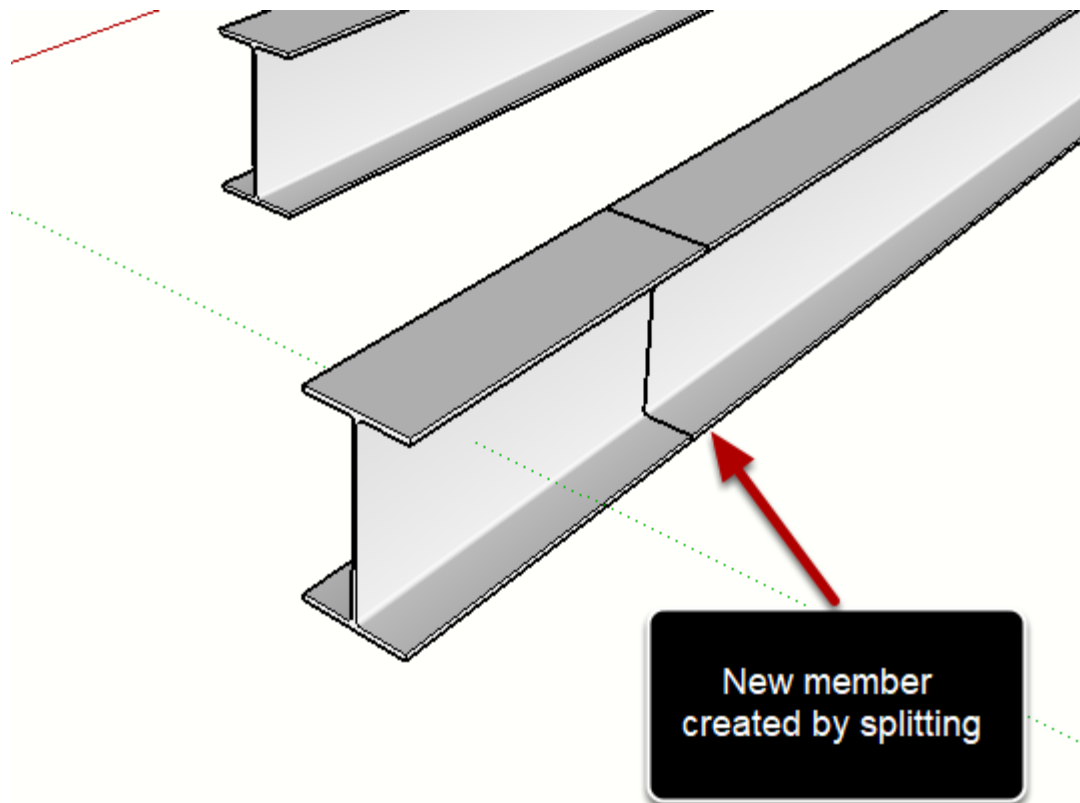


Extending Multiple Members



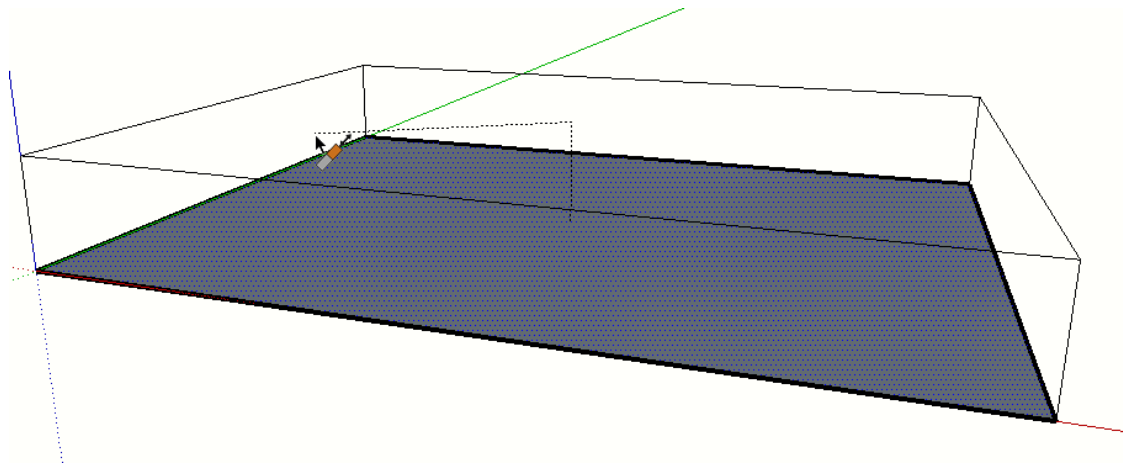
If you have multiple Profile Members selected prior to launching the tool, you can modify them all at the same time.

Splitting Members



Press CTRL / OPTION to toggle splitting mode.

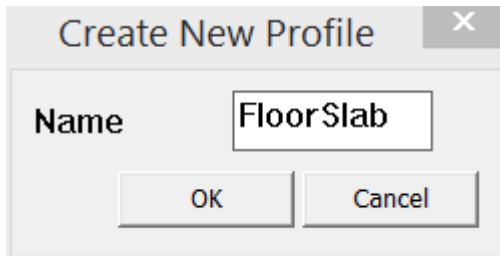
Thicken a Face into a Profile Member



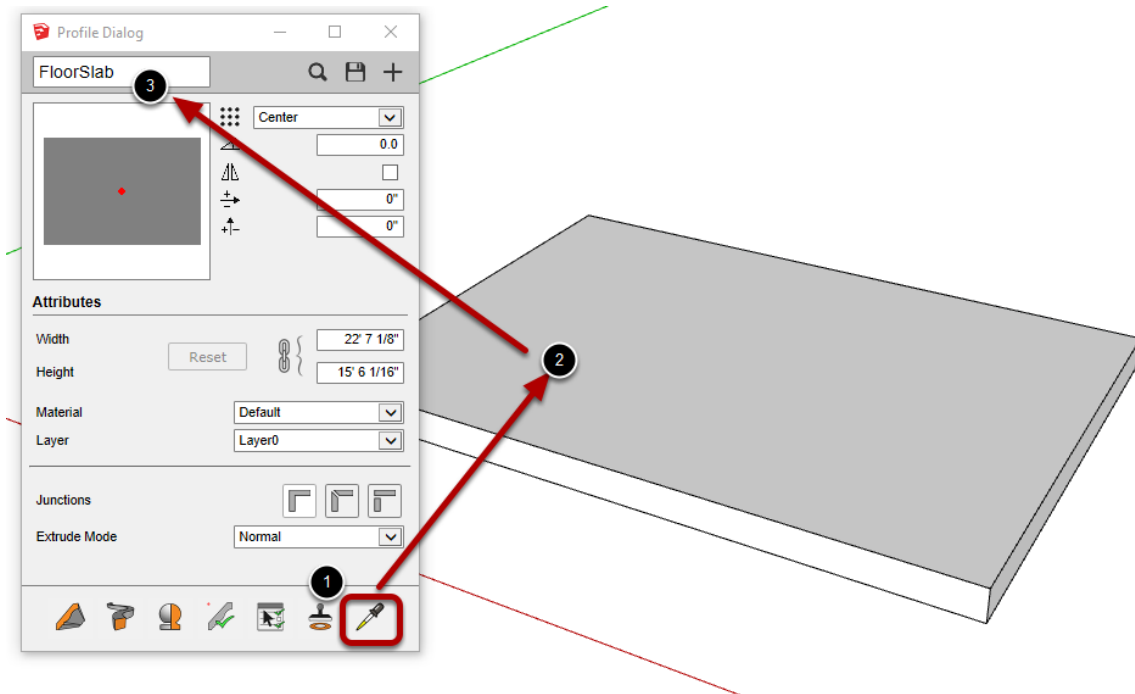
1. Draw a Face
2. Launch the tool
3. Click the face
4. Thicken or extrude the face using the tool.

This feature can be useful for creating solid slabs or panels.

Enter a Name for the Profile



The result will be a Profile Member

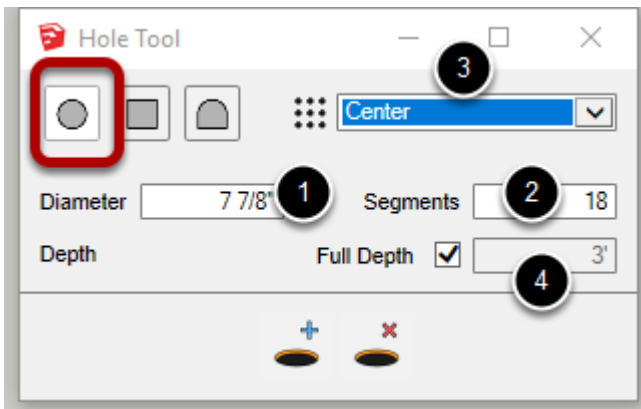


Hole Tool

Launch the Tool

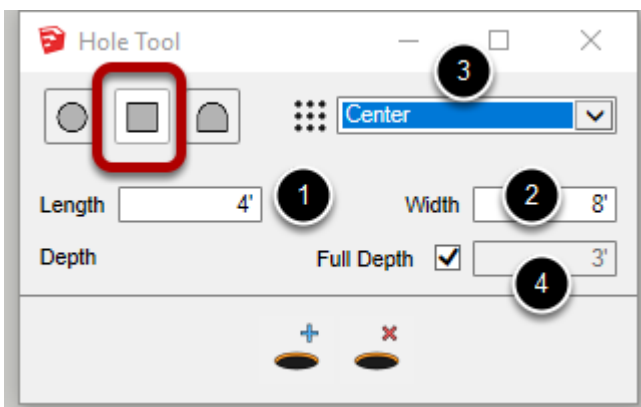


Circular Holes



1. Change the diameter to set the size of the hole
2. Segments is the number of divisions around the circular hole. More segments results in a smoother looking hole.
3. Adjust the placement point of the hole profile.
4. Set the depth of the hole or enable full depth to punch through the target object.

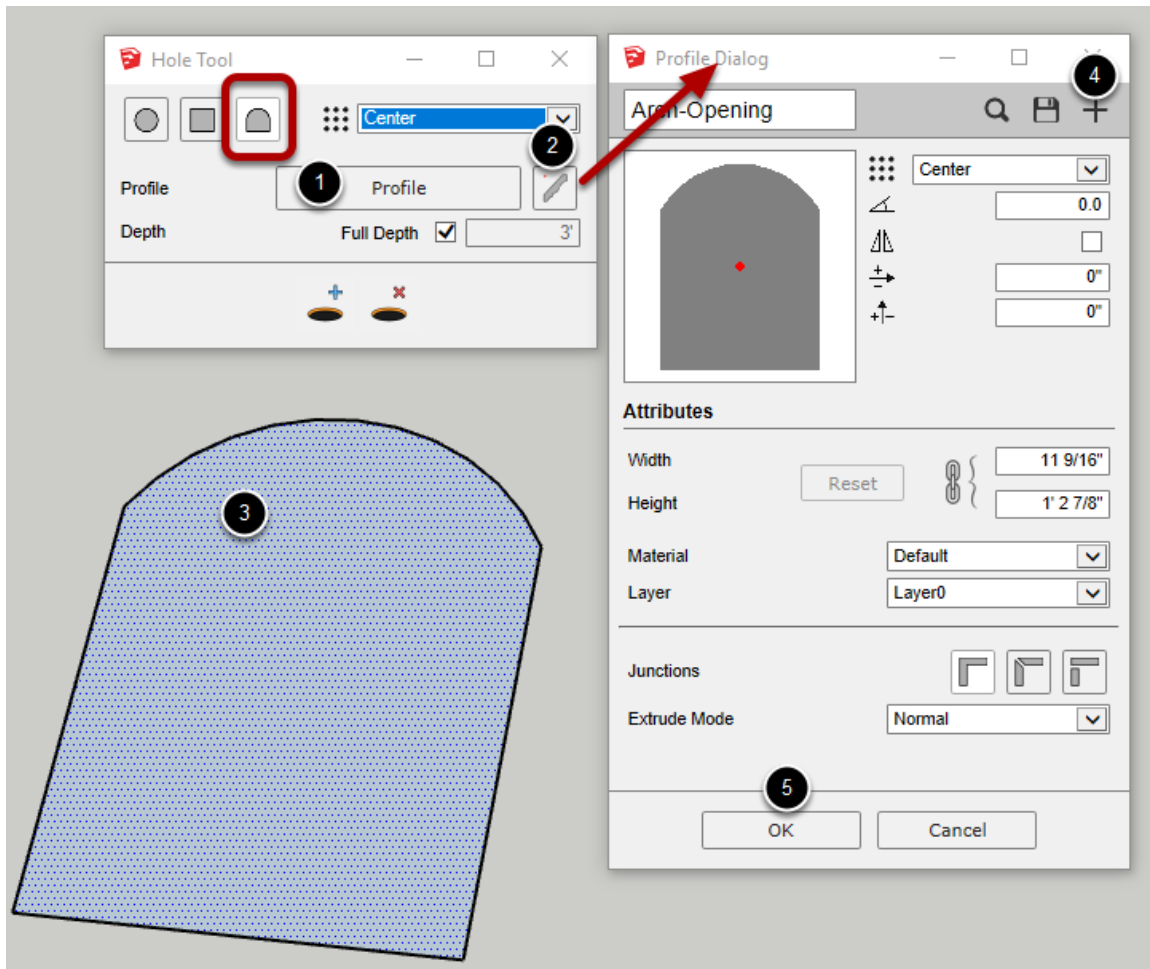
Rectangular Holes



1. Set the horizontal length of the hole.

2. Set the vertical width of the hole.
3. Adjust the placement point of the hole profile.
4. Set the depth of the hole or enable full depth to punch through the target object.

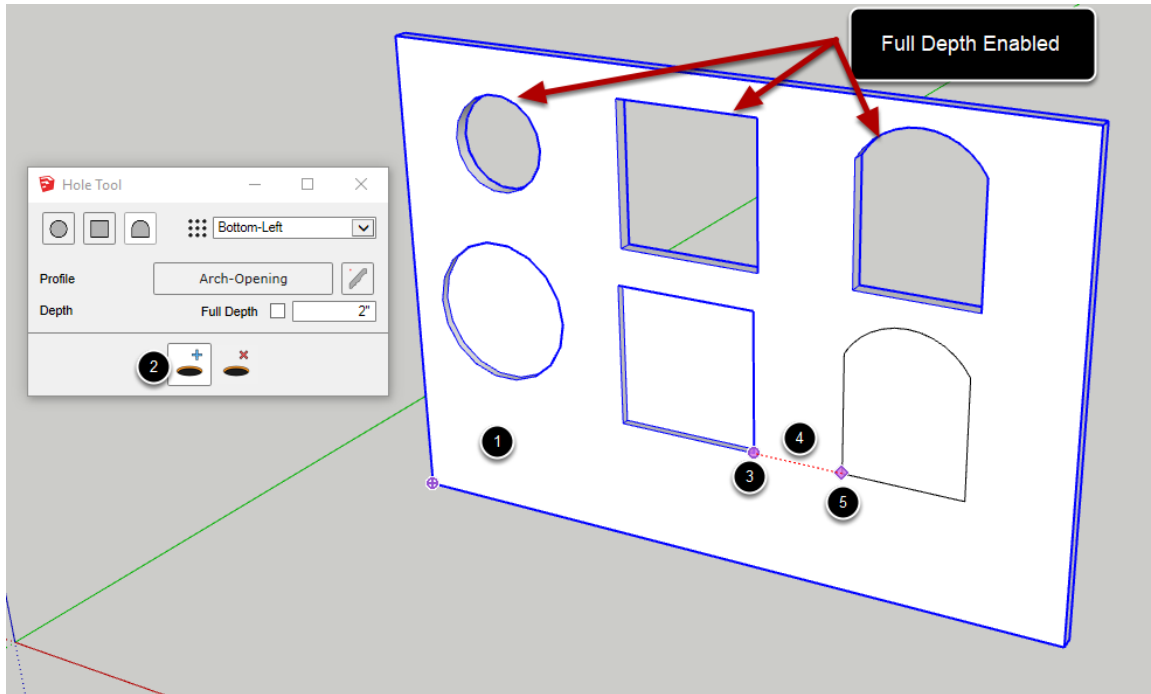
Custom Profile Holes



1. Click this button to use a hole Profile from an existing Profile Member in your model.
2. Click this button to create or edit a custom hole profile. The Profile dialog will open.
3. You can select a face in your model and use it as the hole Profile.
4. If you click the 'New Profile' button, you can use the selected face as the hole Profile.
5. Click the 'OK' button to finish editing the custom hole Profile.

Note: Material, Layer, Junctions, and Extrude Mode settings are not used for custom hole Profiles.

Adding Holes to Objects



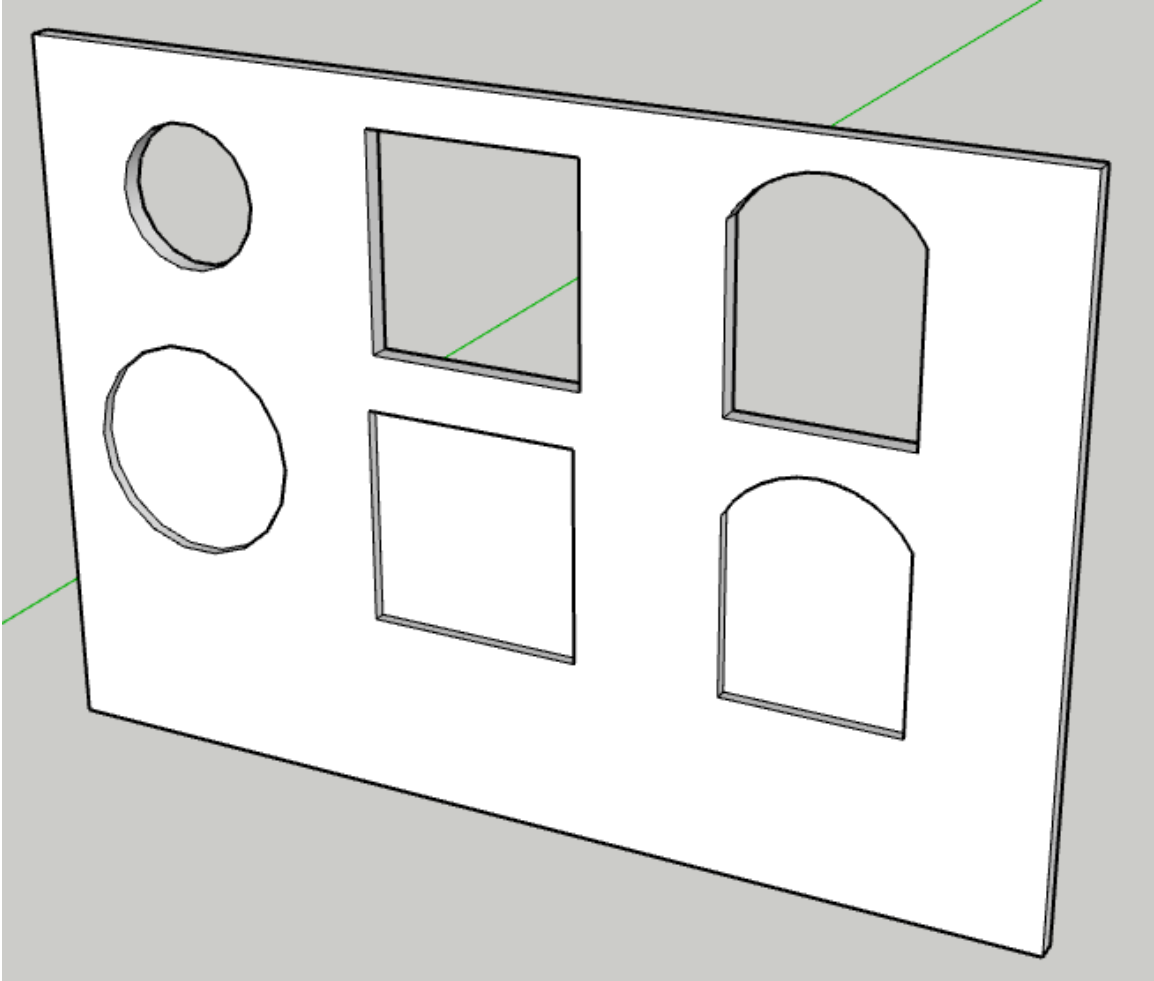
The hole tool will cut through the **selected group or component**. It is recommended (but not required) to first select the objects you want to cut before using this tool.

1. Select a group or component for cutting (recommended)
2. Click the 'Add Holes' button to launch the tool.
3. Move the mouse over a reference point and press the CTRL key to place a reference point (optional).
4. Set the position of the hole with the mouse or key-in a distance value from the reference point.
5. Click or press 'ENTER' or 'RETURN' to create the hole.

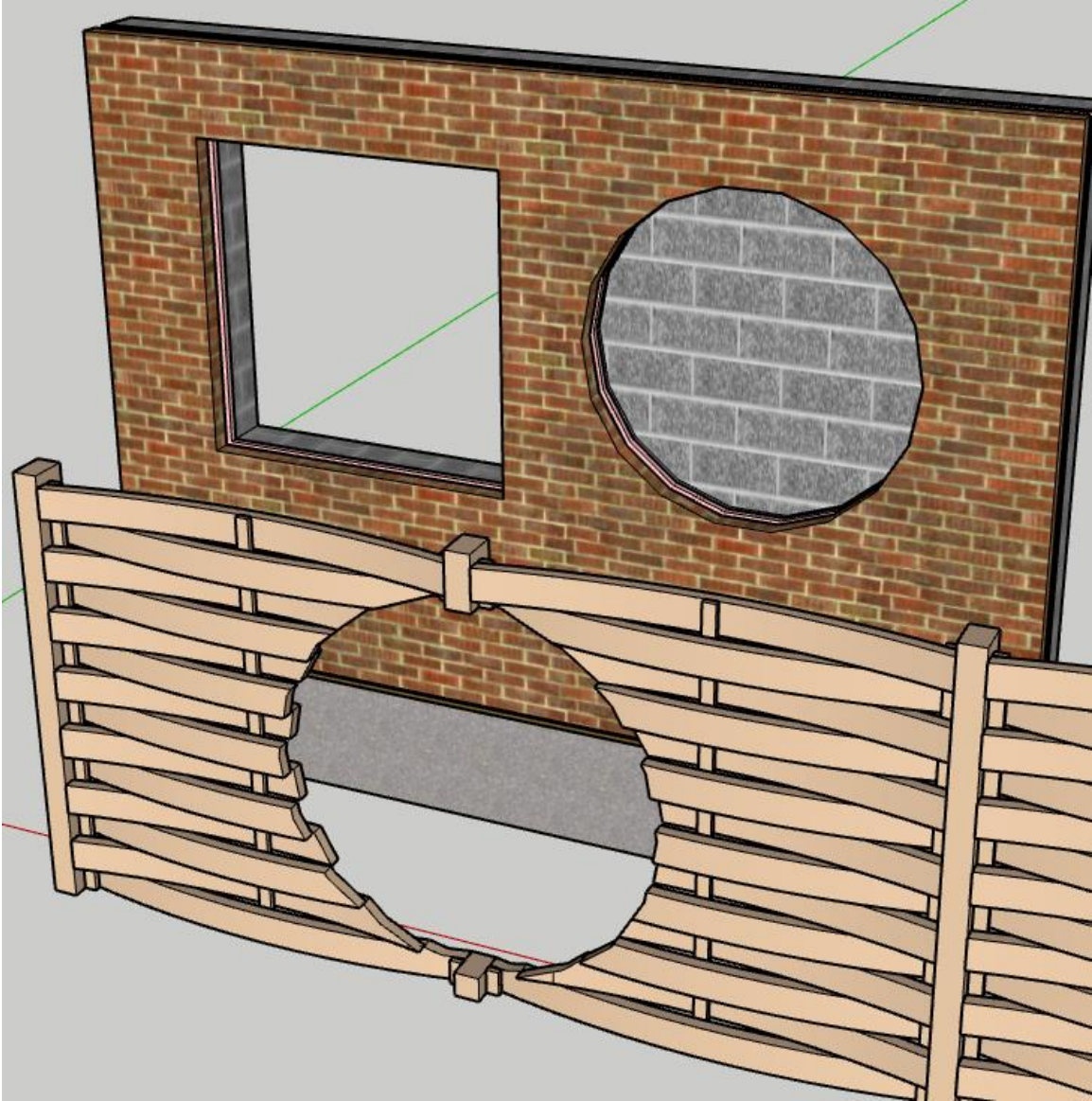
TIP: Use the SHIFT key to lock inferencing when positioning the hole.

If the object is not cut as expected, make sure that the object to be cut is selected. You might not realize that another object off-screen is actually selected and being cut.

Congratulations! You just created a hole!

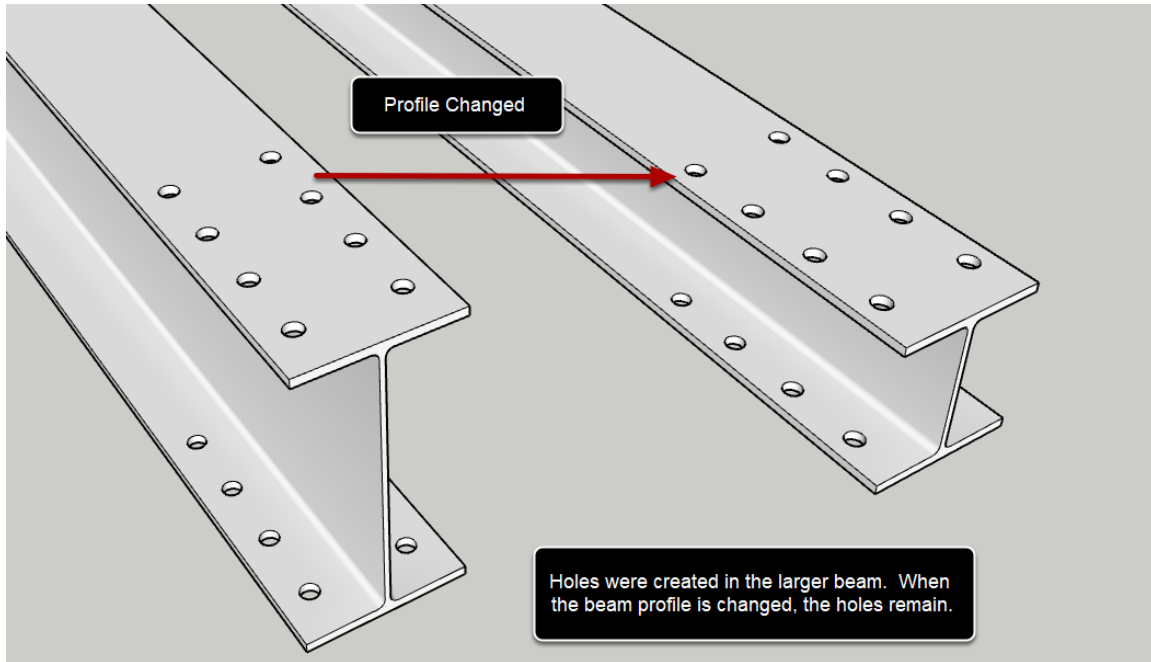


Cut through Nested Groups and Components



The hole tool can create holes through nested geometry (including assemblies).

Holes in Profile Members are Parametric

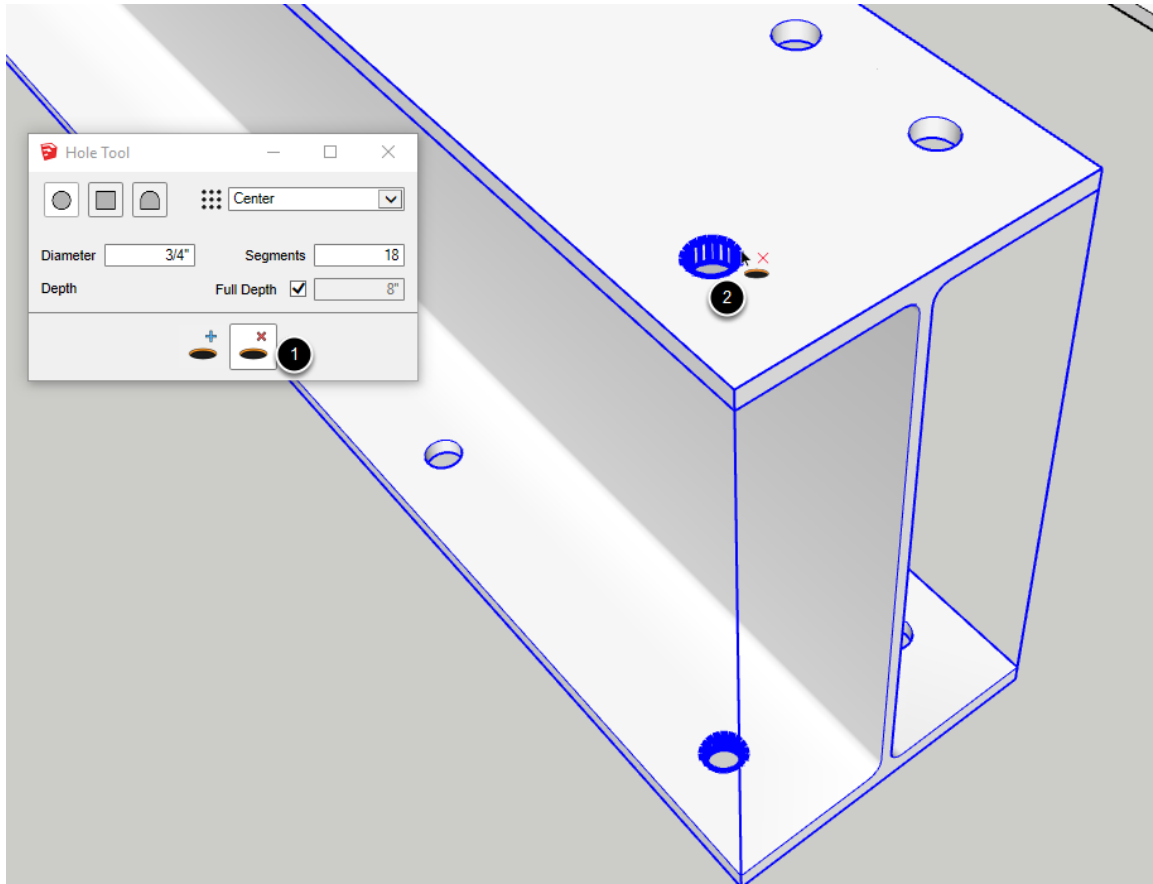


If you create a hole in a Profile Member object, the hole will be maintained even if the Profile Member is edited later with any of the Profile Builder tools.

Note: Profile Builder will attempt to maintain the holes when the Profile Member is edited, but it is not always possible. Sometimes, holes will need to be re-added after making edits.

Holes are only parametric for Profile Members (objects created using the Profile Builder dialog 'build' tools). Holes in basic Groups and Components are not parametric.

Removing Holes



Holes can be removed from Profile Members only. This is because Profile Members are parametric and Profile Builder always knows how to re-generate them if they have been edited with the Profile Builder tools.

1. Click the 'Remove Hole' button to launch the tool.
2. Click the edges of a hole in a Profile Member to remove the entire hole.

Note: You must be able to select the Profile Member with the SketchUp Select tool in order to remove a hole. If the Profile Member with the hole is contained within a parent group, first double-click the parent group and then you should be able to remove the holes from the Profile Member.

If you try to remove a hole from a non-Profile Member, you will see an error message.

Trim Profile Member to Solid

Launch the Tool

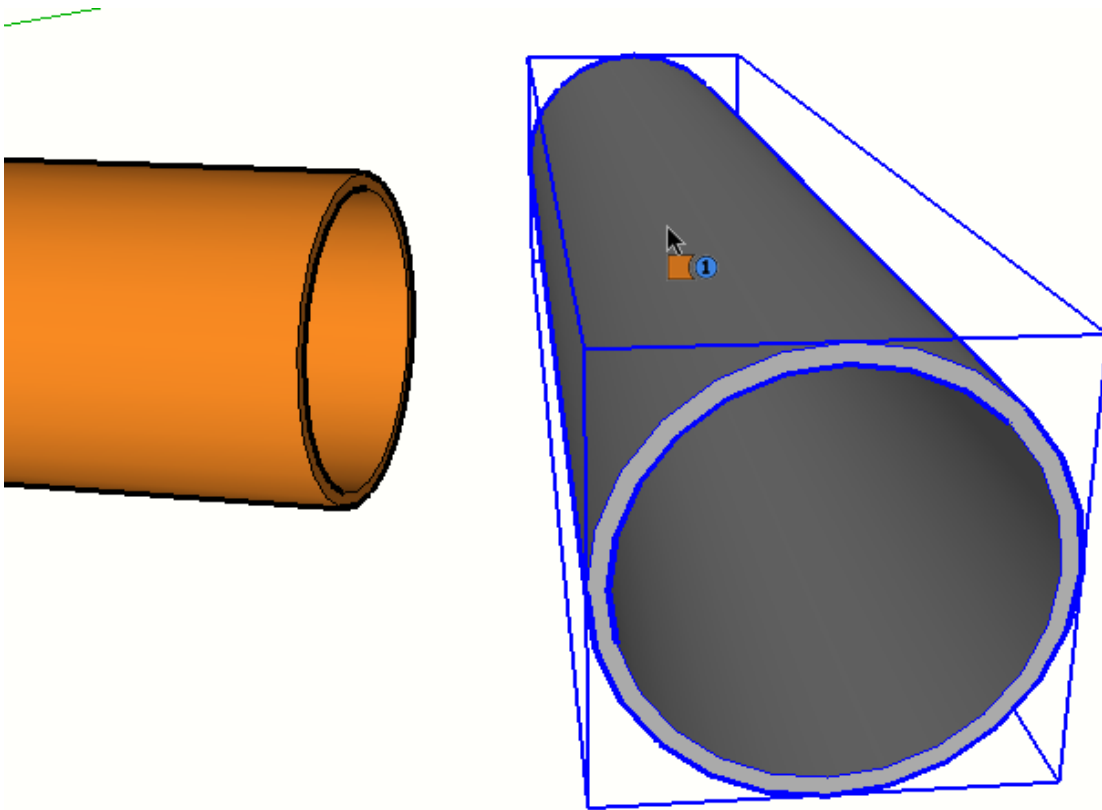


Ensure that Profile Members are Solids



The Profile Member must be a solid. Any Profile Member created from a 2D face (not a polyline) will automatically be a solid. However, it is highly recommended to use the Solid Inspector SketchUp extension (by ThomThom) to check for solid objects and to help fix any non-solid objects. This extension is available on the SketchUp Extension Warehouse.

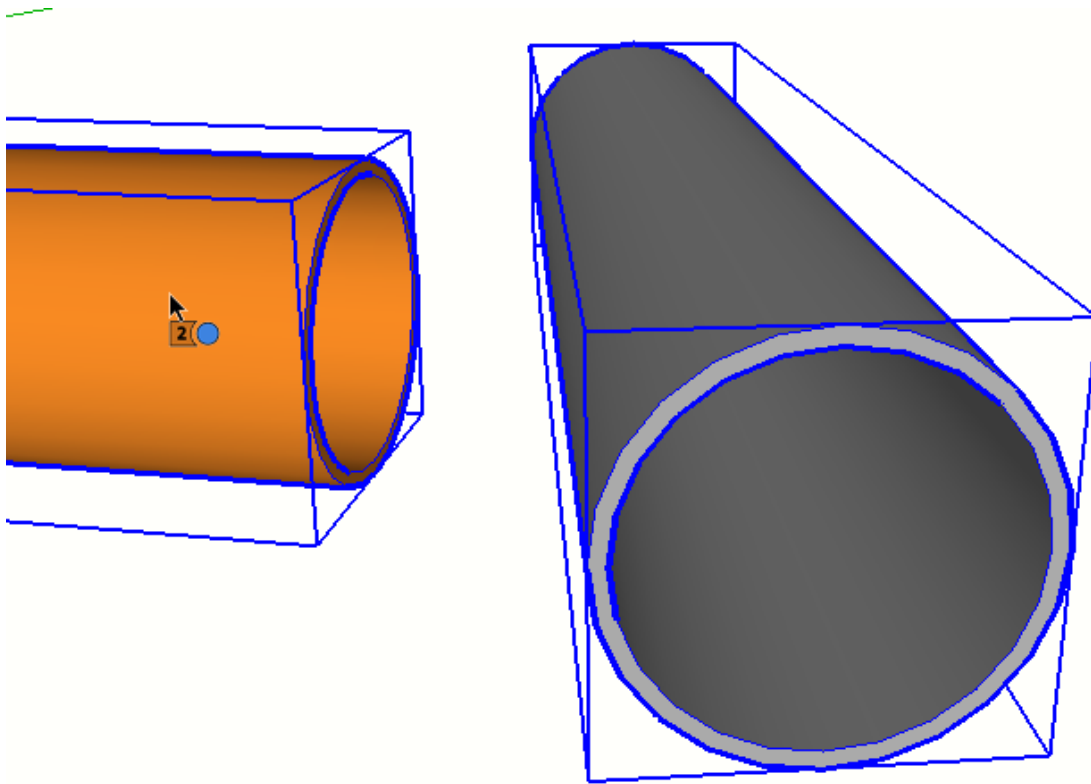
Select a Solid Object to trim to



The object does NOT have to be Profile Member but it must be a solid.

As you hover the mouse, the object will highlight to indicate that it is a valid solid.

Click the end of a Profile Member to Trim



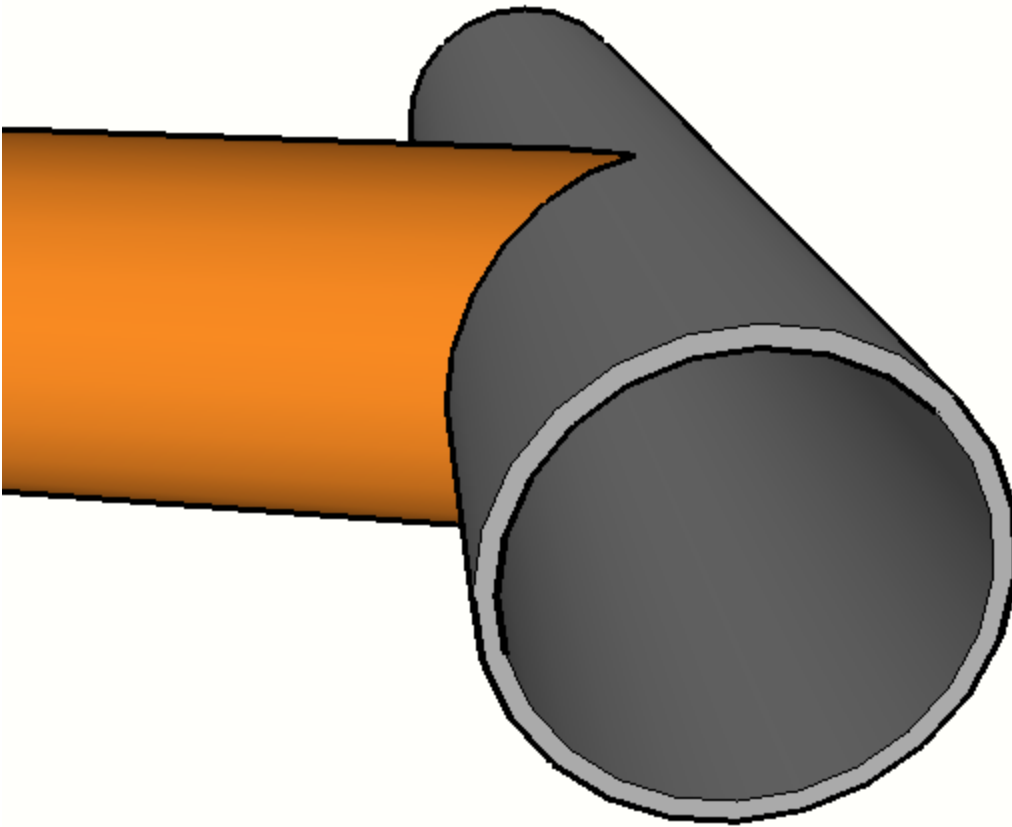
As you hover the mouse over a Profile Member, it will become highlighted indicating that it is a valid solid.

Click on the end of a Profile Member to select it.

Continue clicking other Profile Members ends for trimming if desired.

Press ESC to reset the tool.

Congratulations! You just trimmed a Profile Member to a solid!



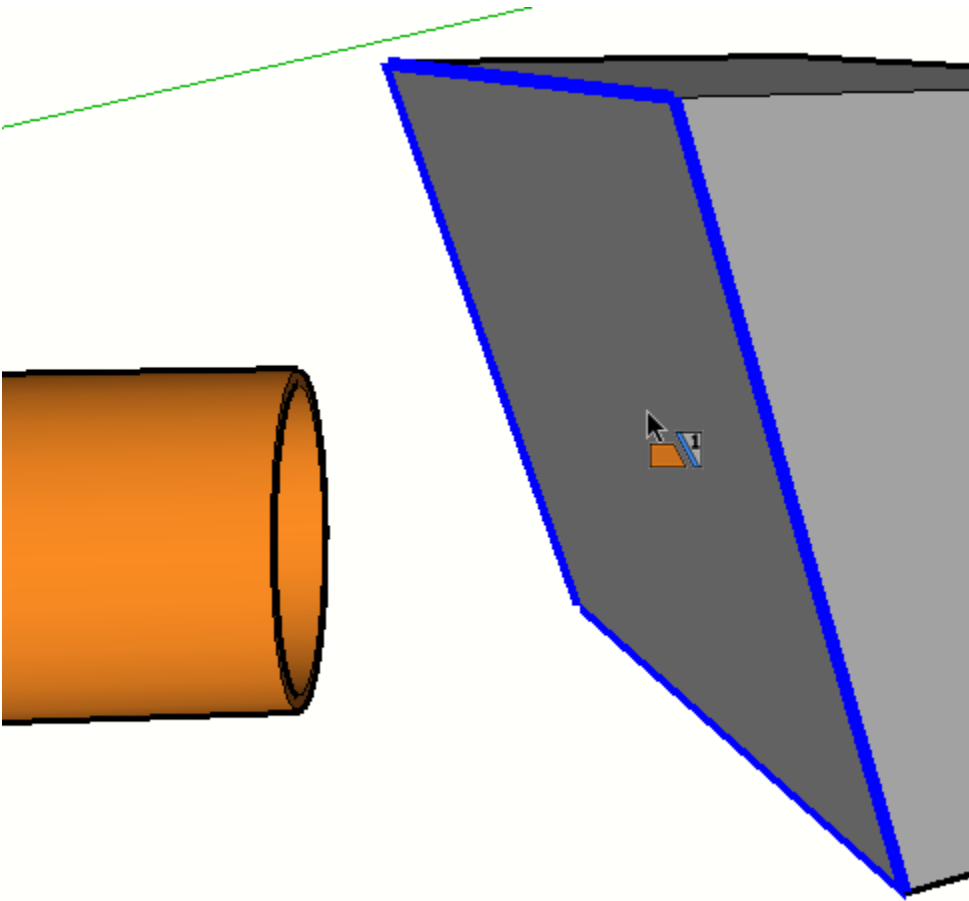
Note: Trims will be maintained even if the Profile Member attributes or path is edited.

Trim Profile Member to Face

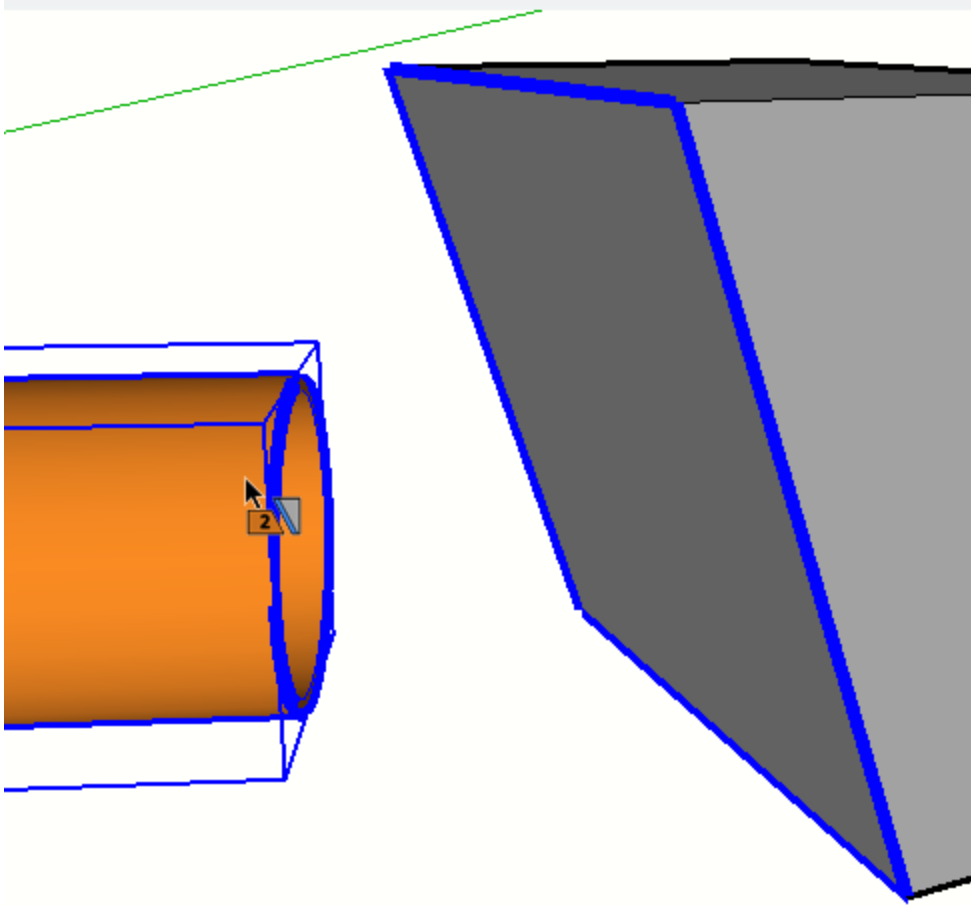
Launch the Tool



Click any Face in your model to trim to



Select the End of a Profile Member

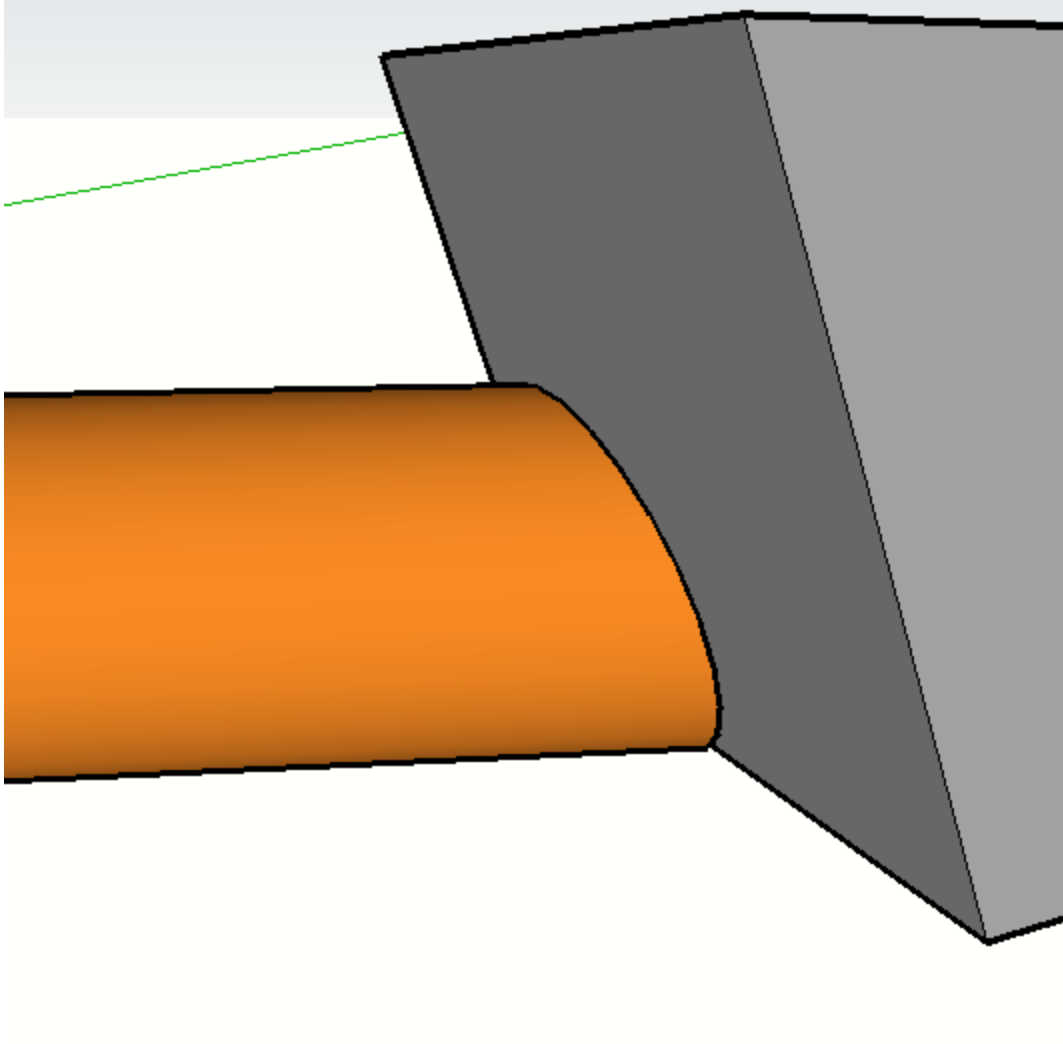


The Profile Member does not need to be a solid.
Click on the end of a Profile Member to select it.

Continue clicking additional Profile Member ends for trimming if desired.

Press ESC to reset the tool.

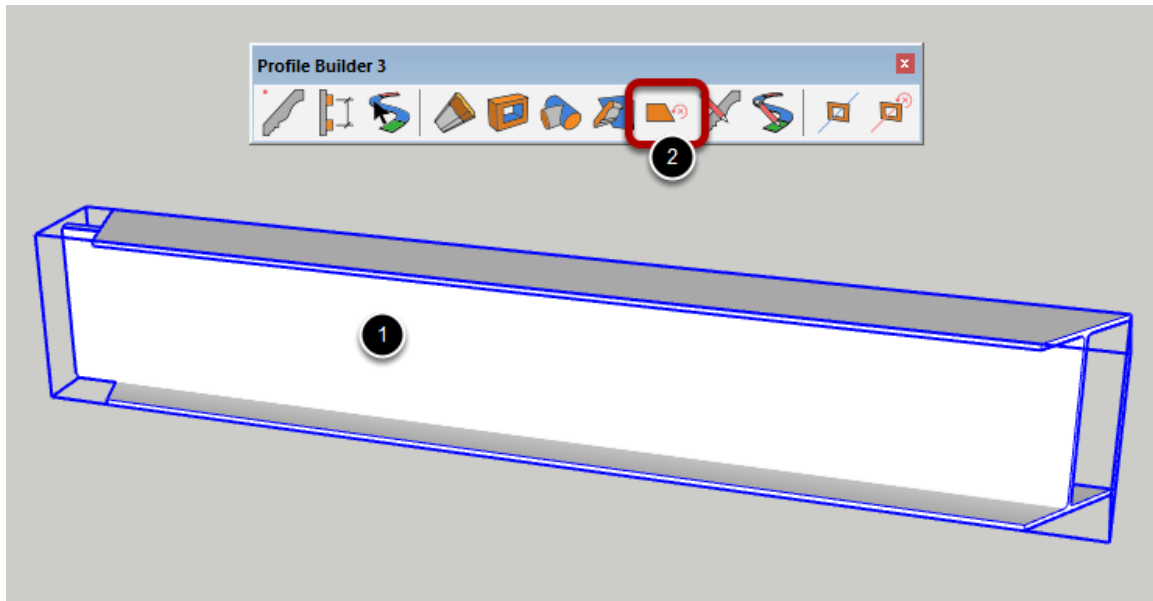
Congratulations! You just trimmed a Profile Member to a Face!



Note: Trims will be maintained even if the Profile Member attributes or path is edited.

Remove Trims

Remove the Trims from a Profile Member

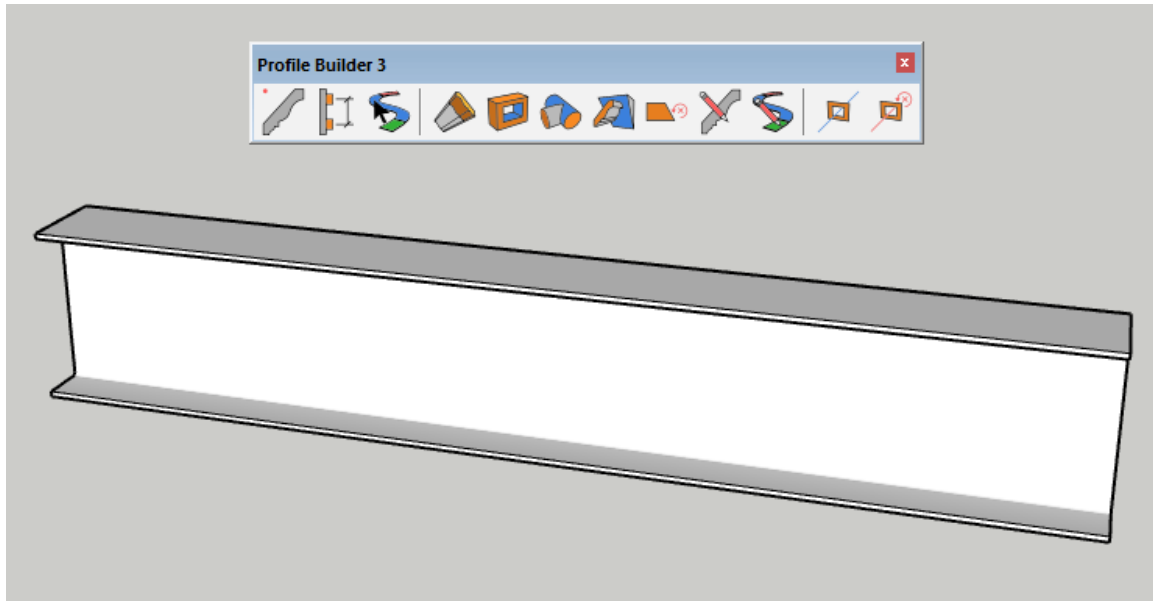


When applying a trim operation to a Profile Member, the trim will be maintained even if the Profile Member is edited using the other Profile Builder tools.

To remove trims:

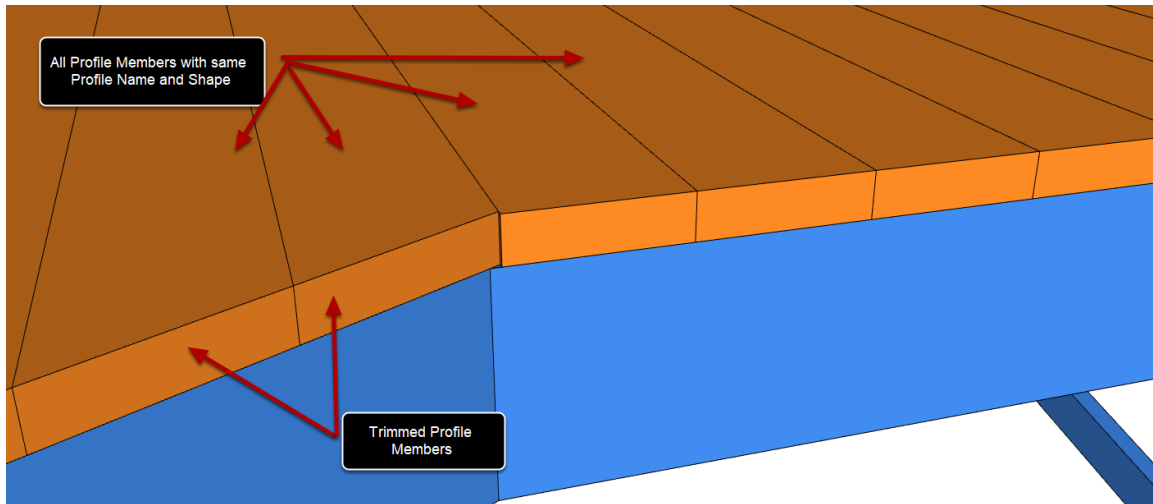
1. Select Profile Members
2. Click the Remove Trim button.

Congratulations! You removed the trims from a Profile Member!



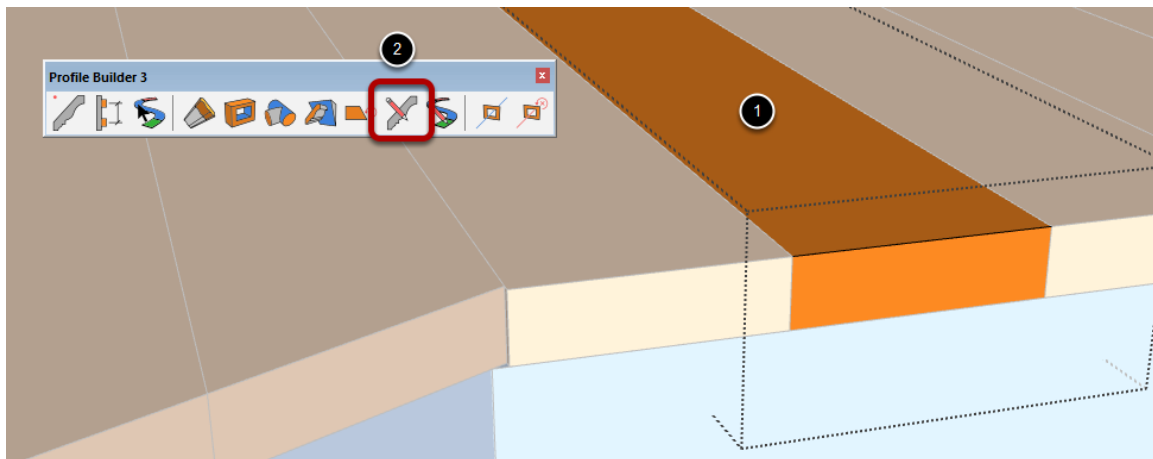
Edit Profile Shape Tool

Editing the Profile of a Profile Member



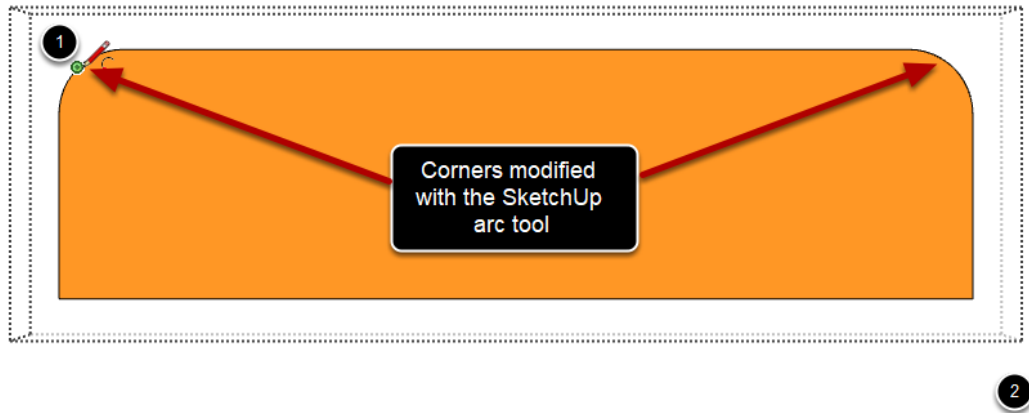
You can edit the Profile Shape of an existing Profile Member in your model.

Select the Profile Member to be Edited



1. Double-click the Profile Member to be edited with the SketchUp select tool.
2. Click the 'Edit Profile' button in the toolbar.

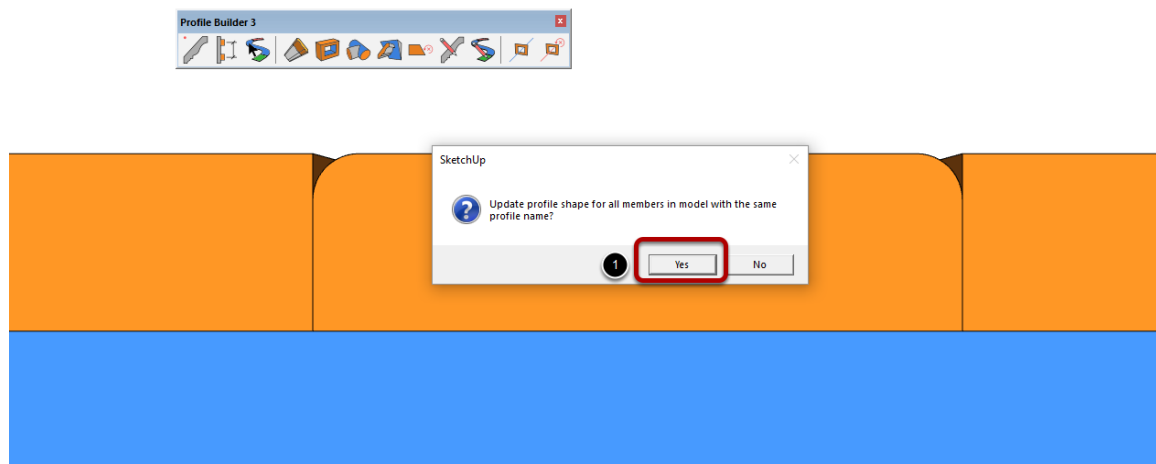
Edit the Profile



After clicking the 'Edit Profile' button, the profile shape will be redrawn and the camera will be positioned so that you can easily make changes to the profile shape.

1. Make changes to the Profile shape using the SketchUp built-in drawing tools (line, arc, circle, erase, etc.)
2. With the SketchUp select tool active, click outside the object to exit profile editing mode.

Choose which Profile Members to Update

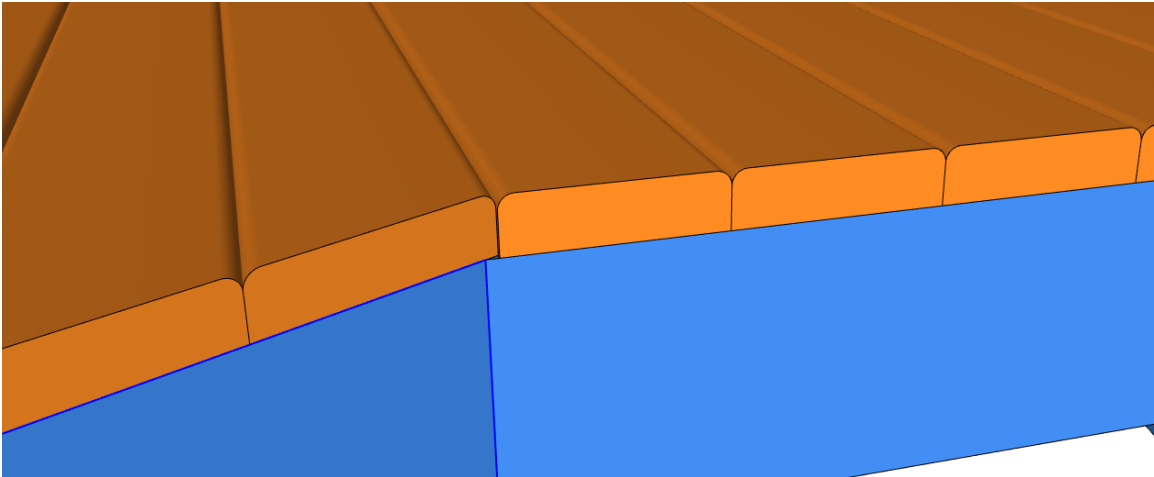


You can update only the Profile Member you selected or you can update all Profile Members that have the same profile name as the one you are editing.

1. Click 'Yes' to update all Profile Members in your model with the same profile name.

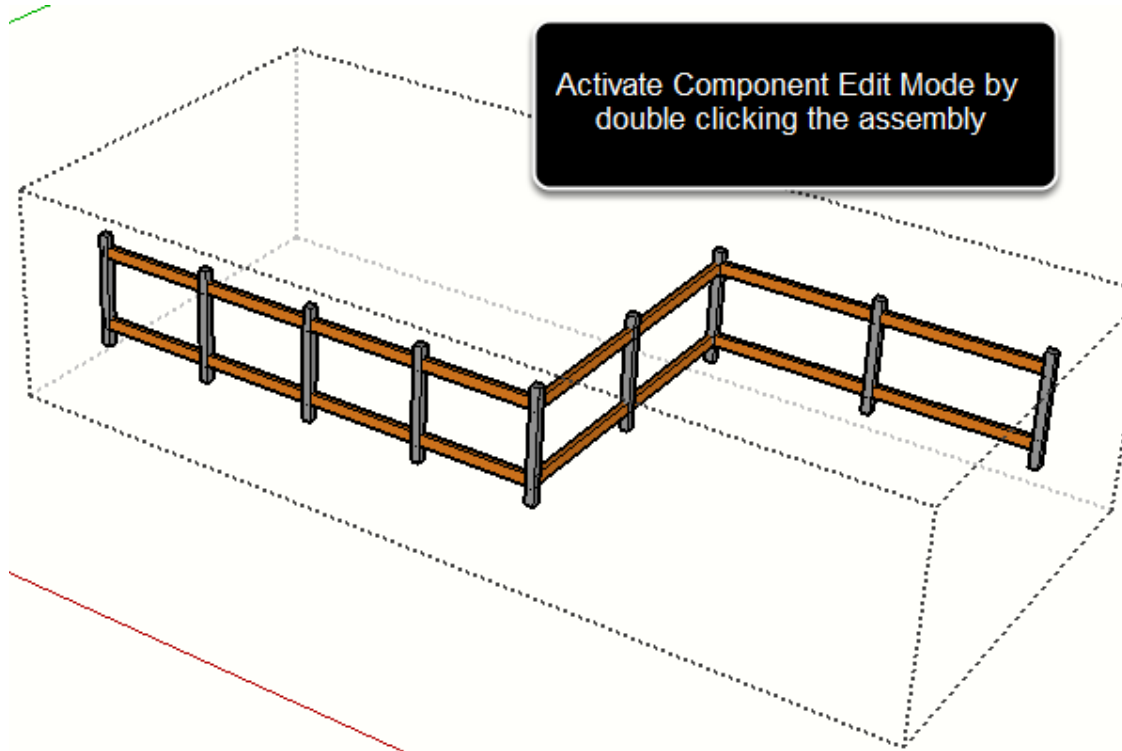
Note: The profile name is not always the same as the Group or Component Name. To check the profile name of an object, click the object using the 'Get Attributes' tool inside the Profile Dialog.

Congratulations! You just edited the Profile Shape of a Profile Member!



Edit Path Tool

Double-click a Profile Member or Assembly

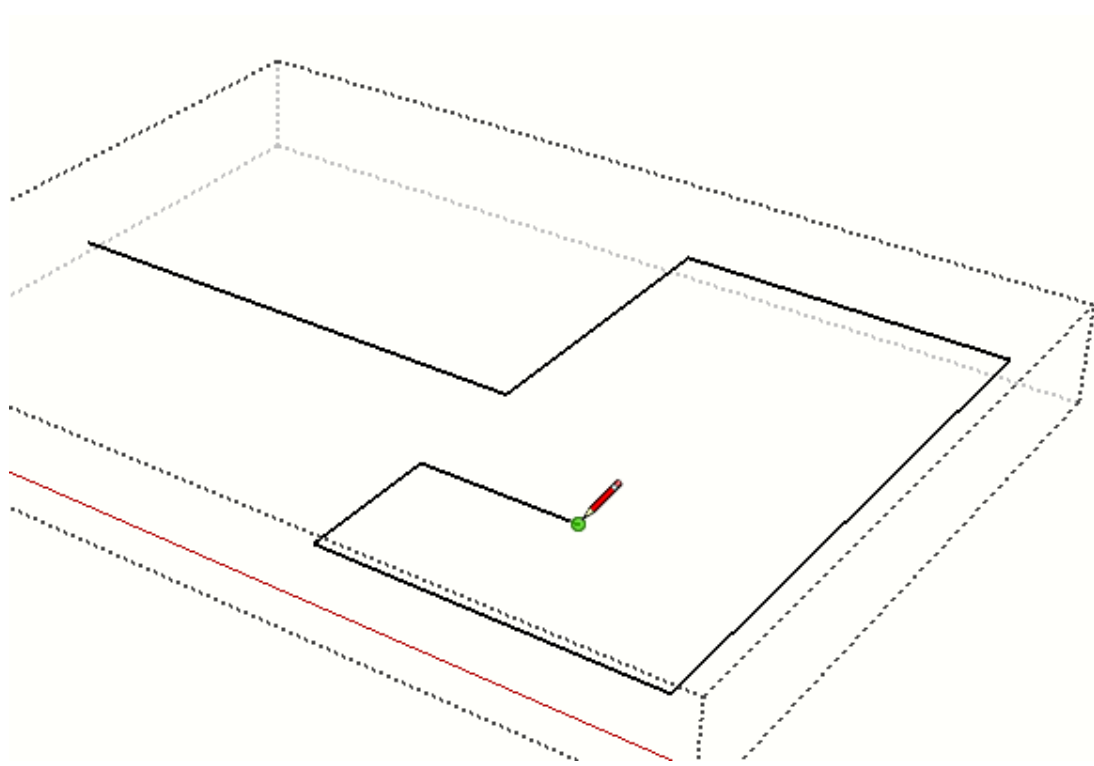


Launch the Tool



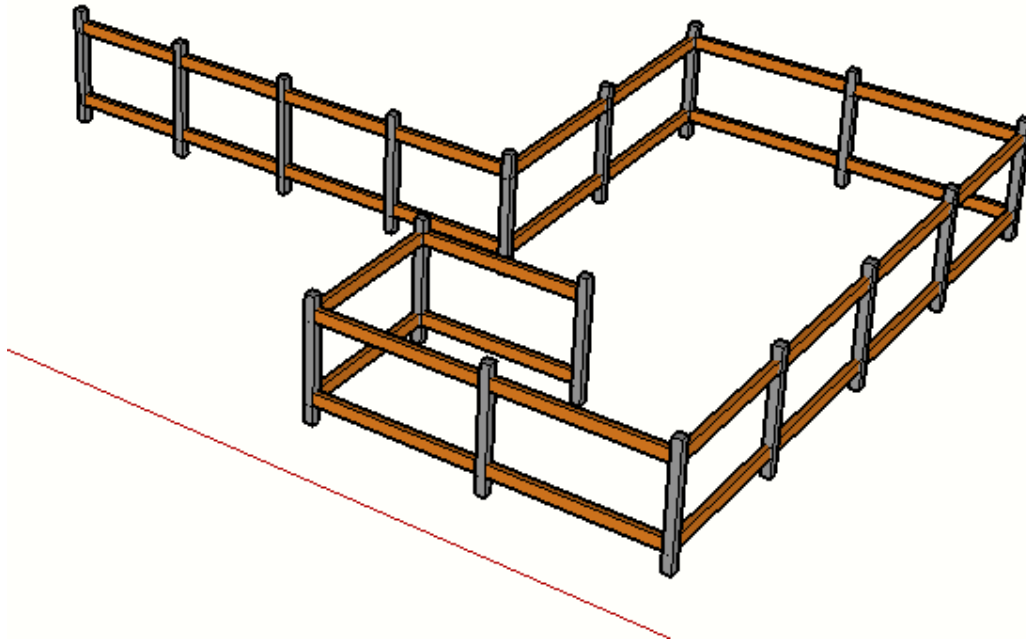
You must be inside 'Component / Group' edit mode prior to using this feature.

Edit the Path of the Profile Member or Assembly



Use the various drawing tools in SketchUp to edit the path.

Close Component Edit Mode to Update the Profile Member or Assembly



The Profile Member or Assembly will be re-built to follow the new path.

Path Mode

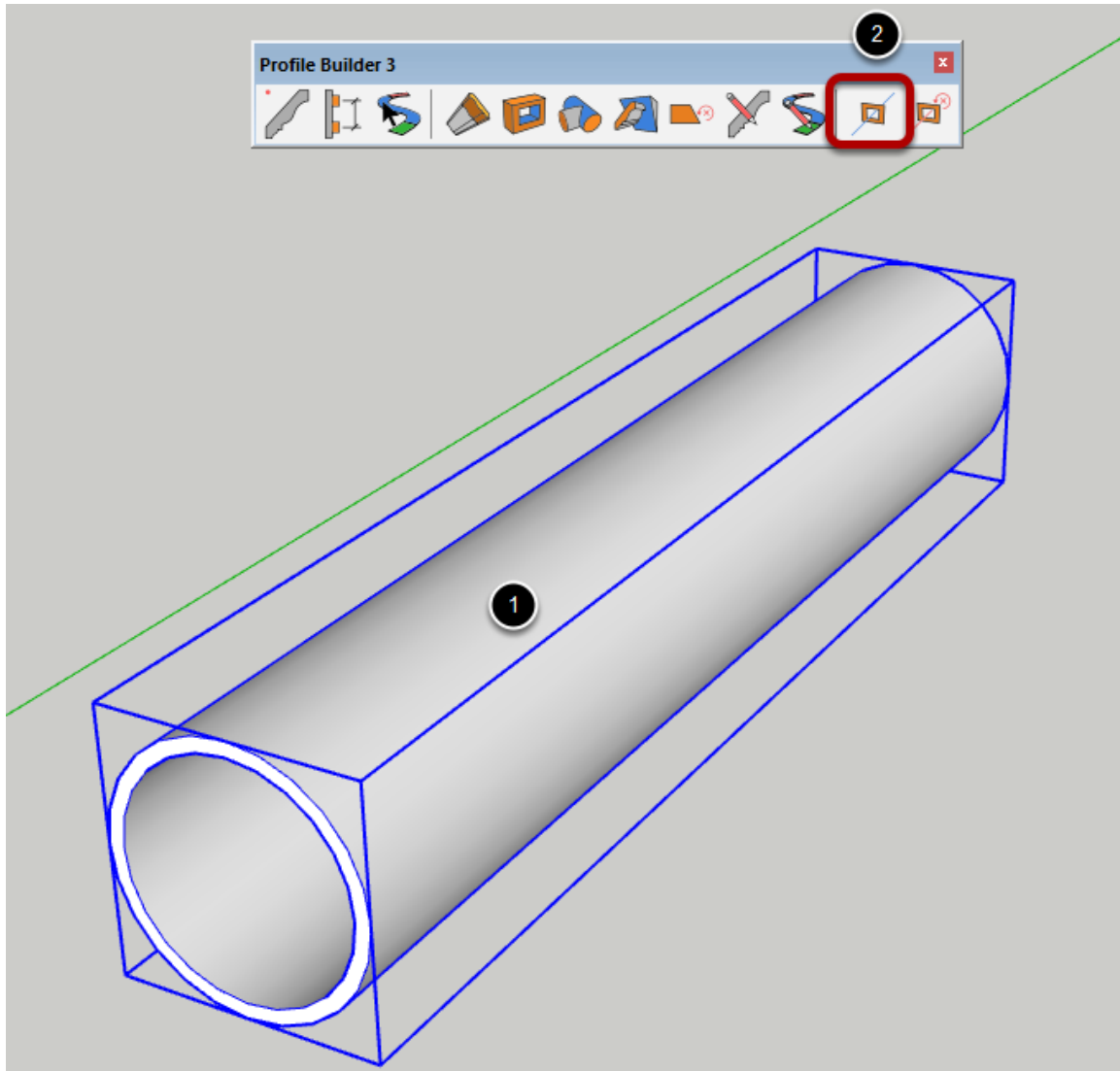
Using Path Mode

Path Mode lets you convert a 3D Profile Member to a simplified representation. While in Path Mode, only the internal path lines are drawn as well as a face that represents the profile of the object.

In Path Mode, all of the Profile Member attributes are maintained even though the visual display is different.

The biggest advantage of using Path Mode is that it allows you to easily inference to the Path of the Profile Member which makes editing operations easier, quicker, and more accurate.

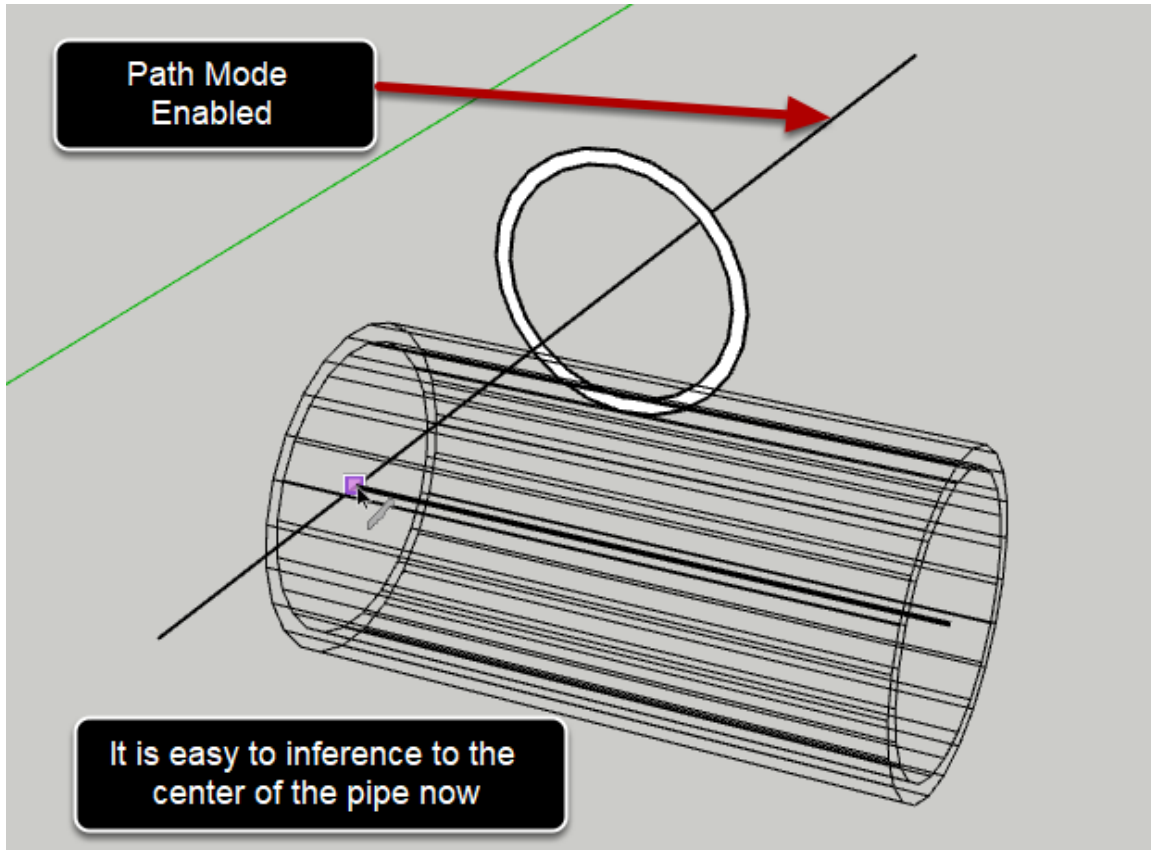
Enable Path Mode



To enable path mode:

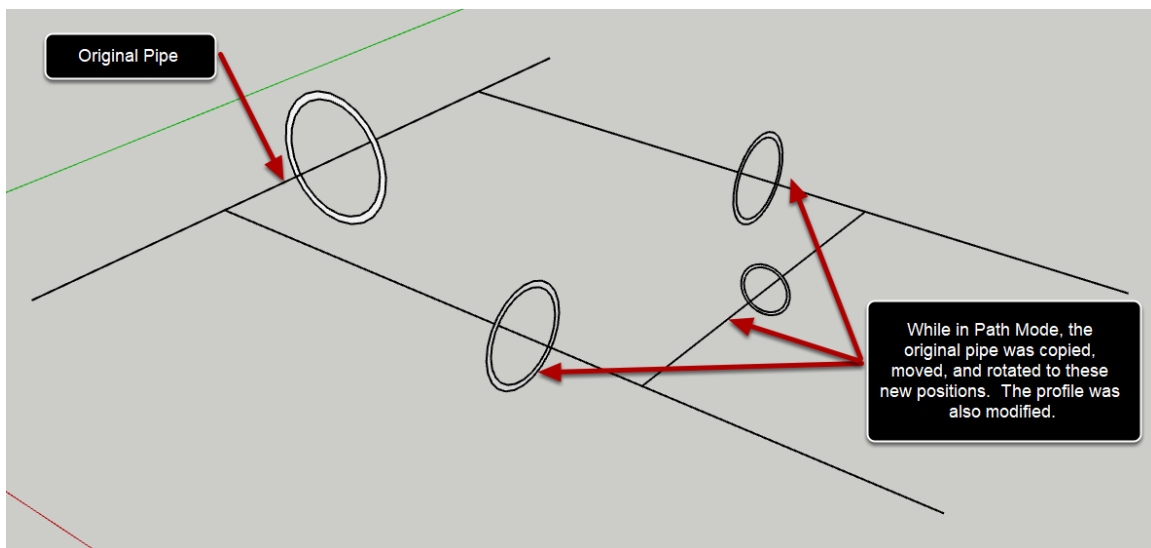
1. Select Profile Members with the Select Tool.
2. Click the Path Mode button in the toolbar.

Continue Modeling while in Path Mode



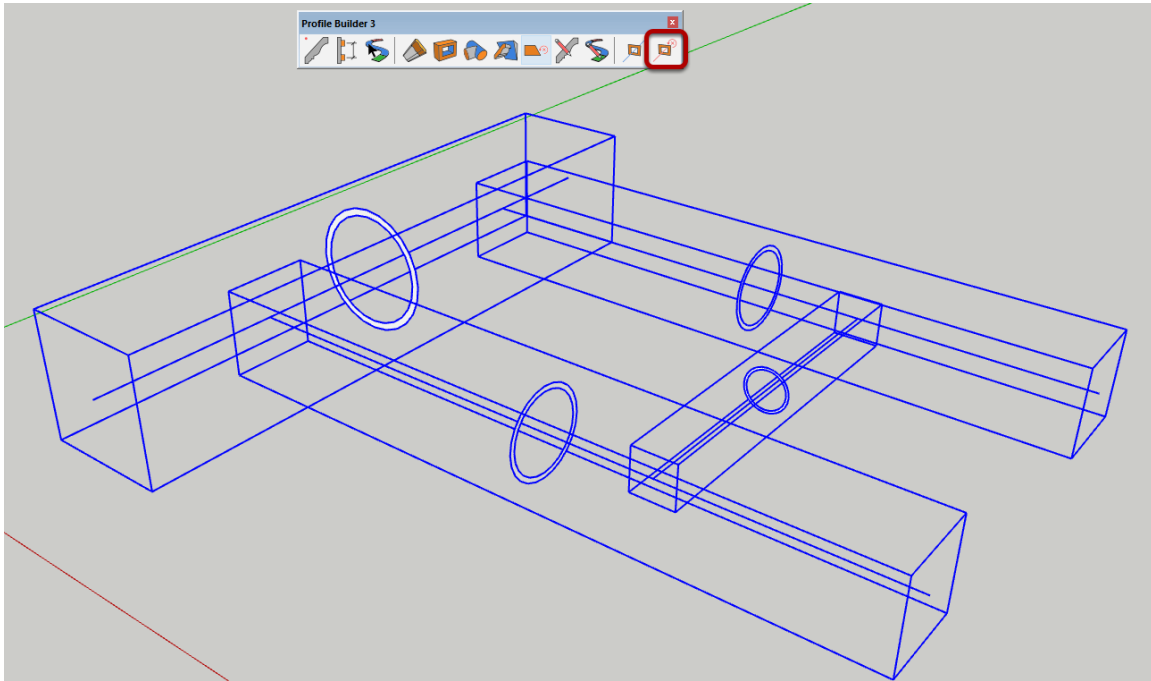
While in Path Mode, it is easy to inference to the path of the Profile Member.

Profile Members in Path Mode can be Edited



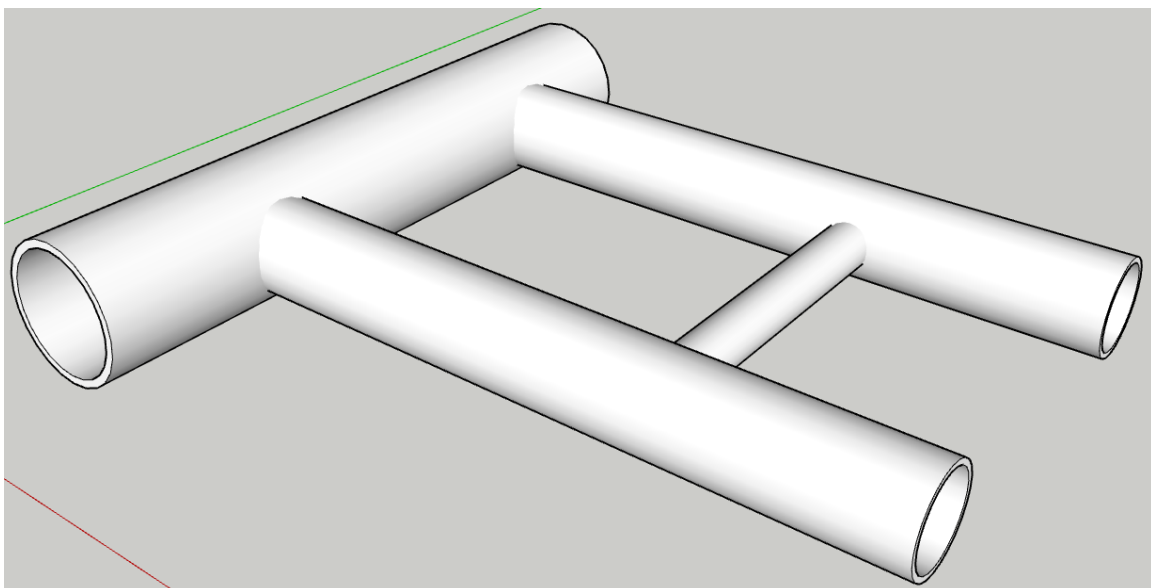
While in path mode, Profile Members can be moved, rotated, and edited with most of the Profile Builder Tools. However, the trim tools cannot be used for a Profile Member that has Path Mode enabled.

Disabling Path Mode

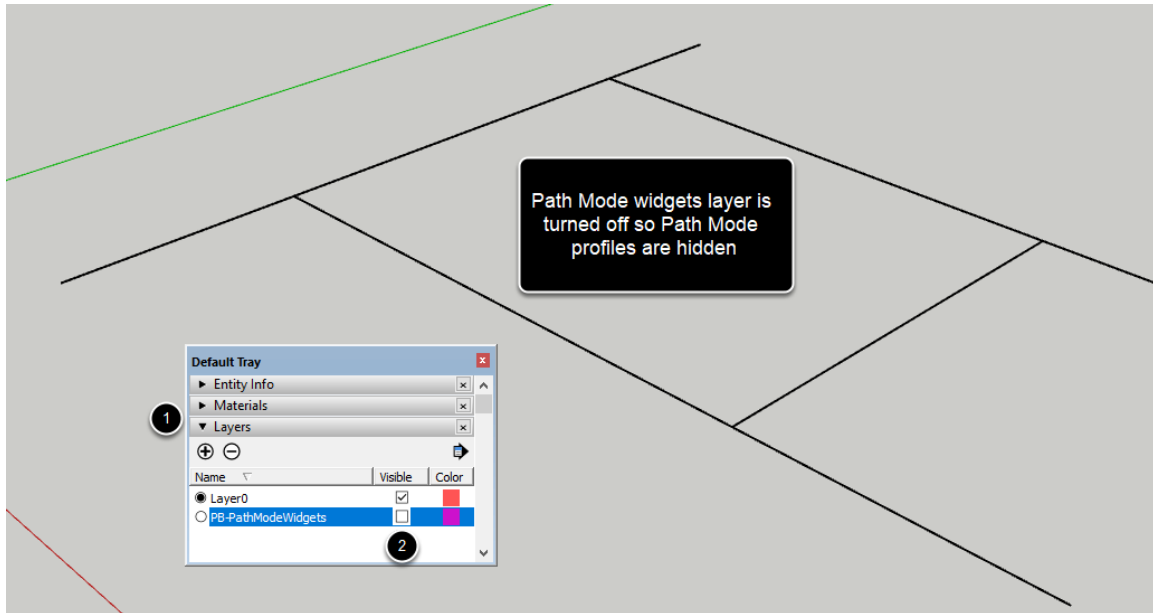


To disable path mode, select the Profile Members that are in Path Mode and then click the 'Revert Path Mode' button in the main toolbar.

Path Mode Reverted



Hiding the Path Mode Profiles



The Path Mode Profiles or 'widgets' are drawn on a layer named 'PB-PathModeWidgets'

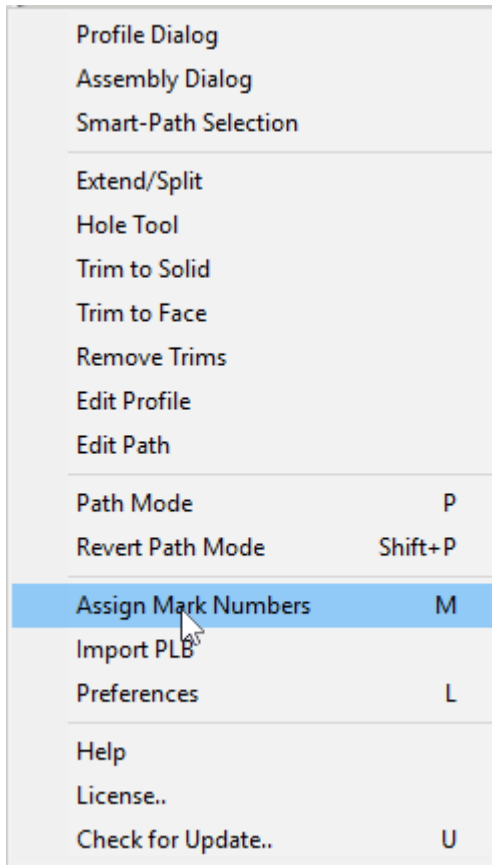
To hide the widgets:

1. Open the Layers panel.
2. Turn off the 'PB-PathModeWidgets' layer.

Assign Mark Numbers (Batch Rename)

Assign Mark Numbers to Profile Members

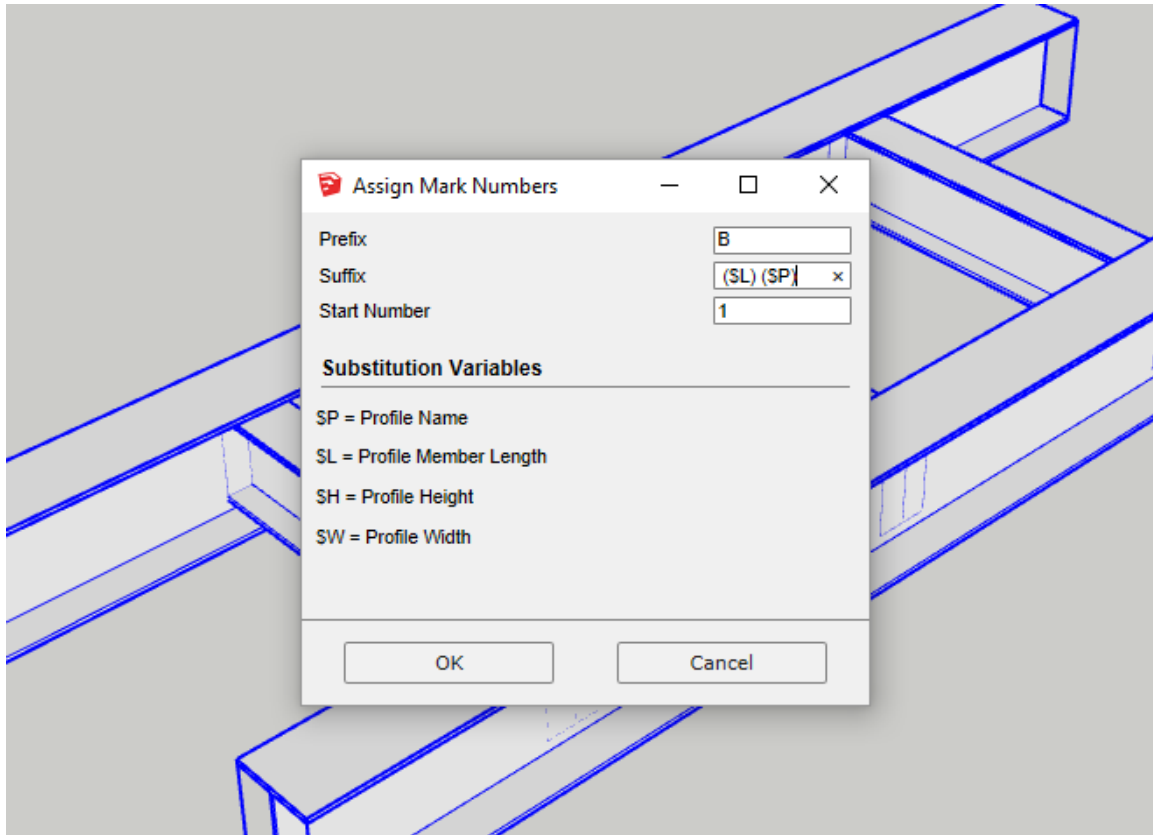
Launching the Tool



First select Profile Members within your model and then launch the tool.

Select the tool from the Profile Builder menu.

Input Naming Parameters

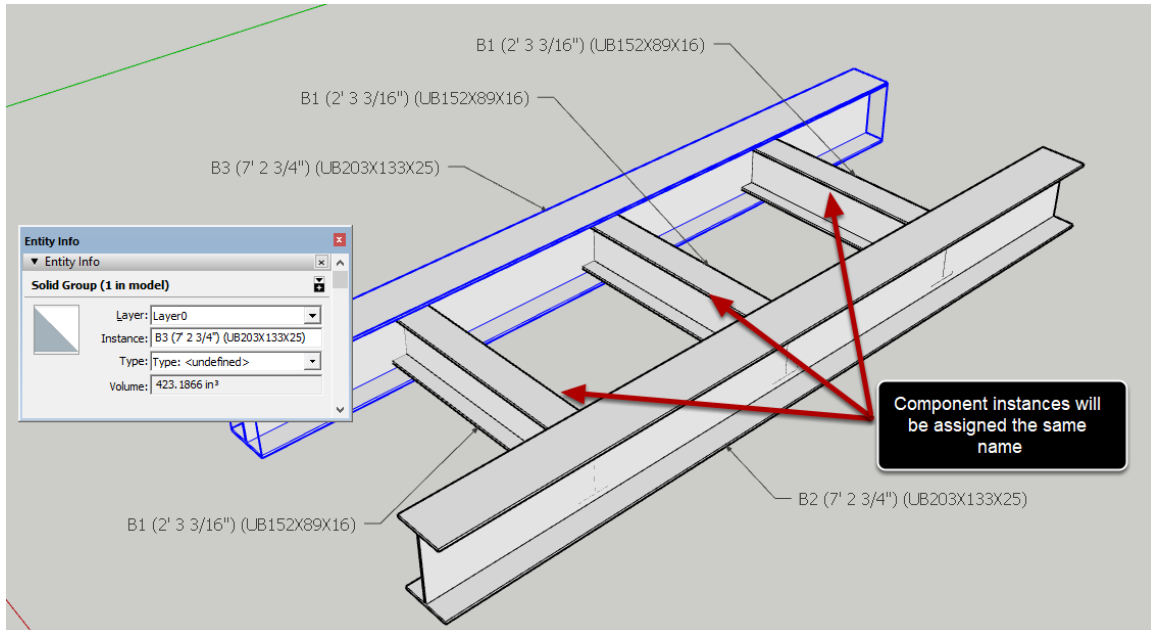


Prefix = first part of name
Suffix = last part of name
Start Number = the first number

You can use the following variables and substitute the actual value into the name.

\$P = Profile Name
\$L = Profile Member Length
\$H = Profile Height
\$W = Profile Width

Congratulations! You just assigned mark numbers to Profile Members!

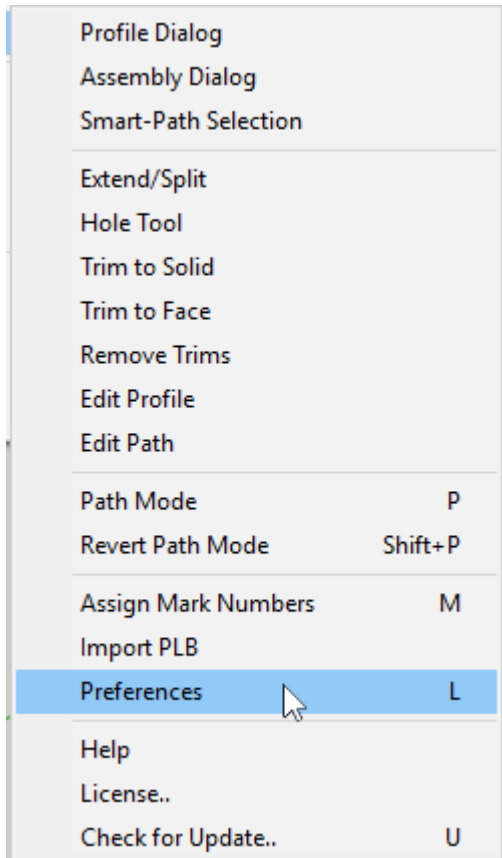


The SketchUp text tool or Entity Info window can be used to reveal the new names for the objects.

Preferences

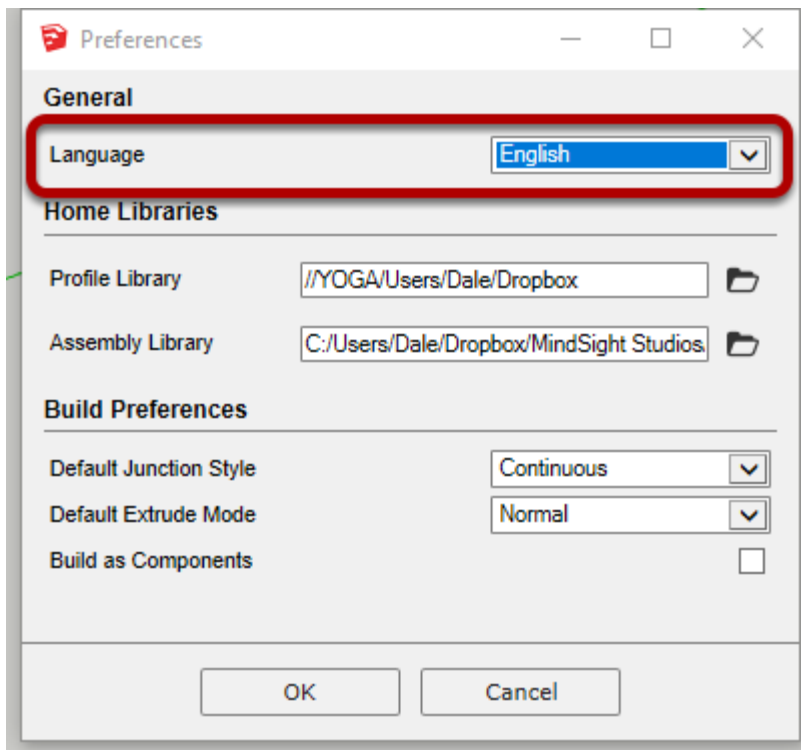
Preferences and Defaults

Open Preferences



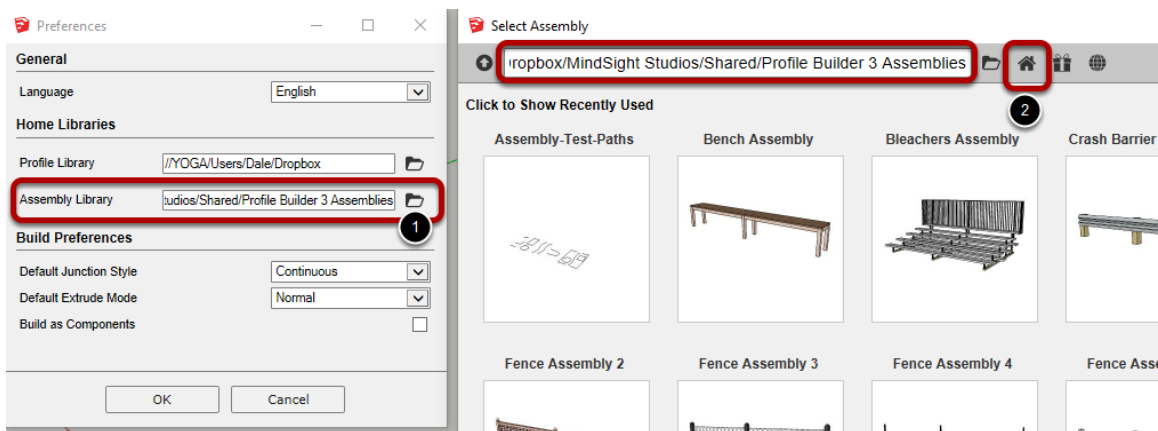
Open the menu item Extensions -> Profile Builder -> Preferences

Language



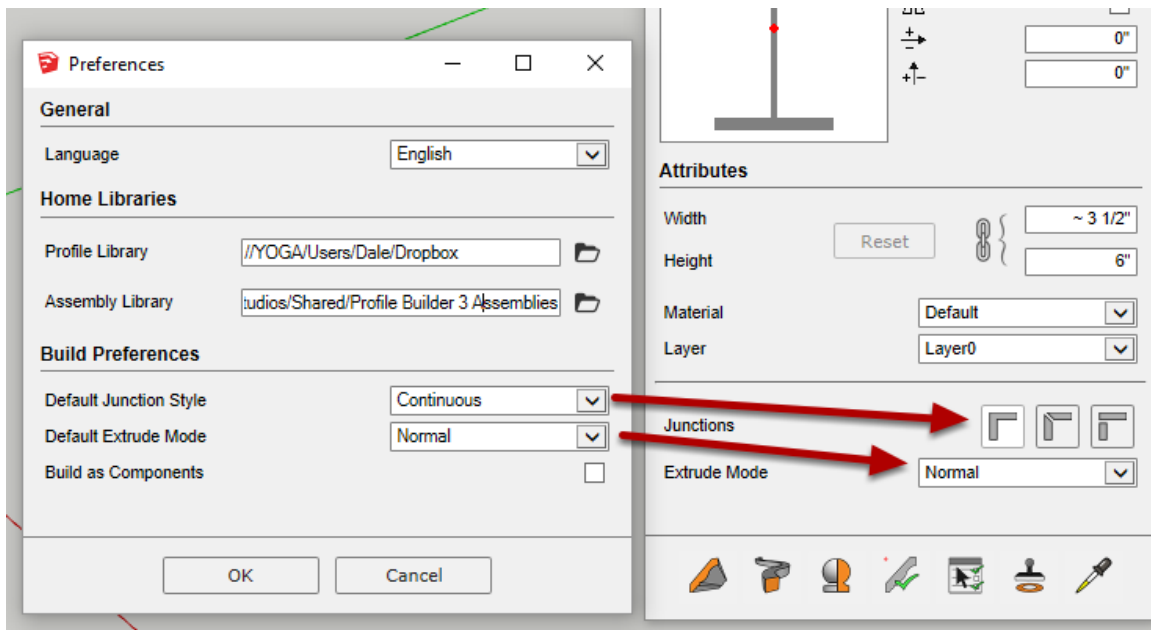
Choose your preferred language from the drop-down list. SketchUp must be restarted for the new language settings to take effect.

Home Libraries



1. Select your default 'home' profile and assembly library folders by clicking the folder icons. Network paths are permitted.
2. These home library folders can then be easily accessed by clicking the 'home' button in the Profile and Assembly browser windows.

Build Preferences



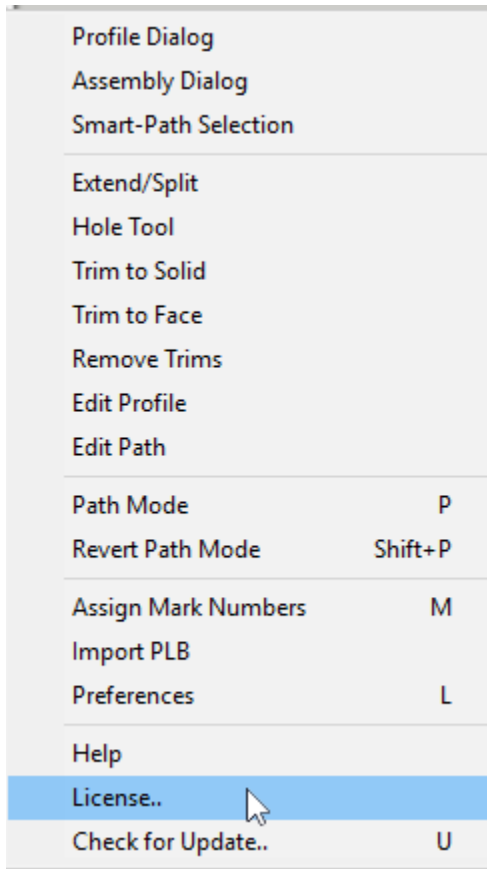
The build preferences can be used to set your preferred junction style, extrude mode, and whether to create Profile Members and Assemblies as Components instead of Groups.

Note: The default junction style and extrude mode will only be used if the profile does not currently have these settings saved. For example, if you save a profile as a SKP file with the extrude mode set to 'Follow Me', when you load the profile, it will still use the 'Follow Me' extrude mode even though the default extrude might be set to Normal. Saved settings will over-ride the default preferences.

Licensing

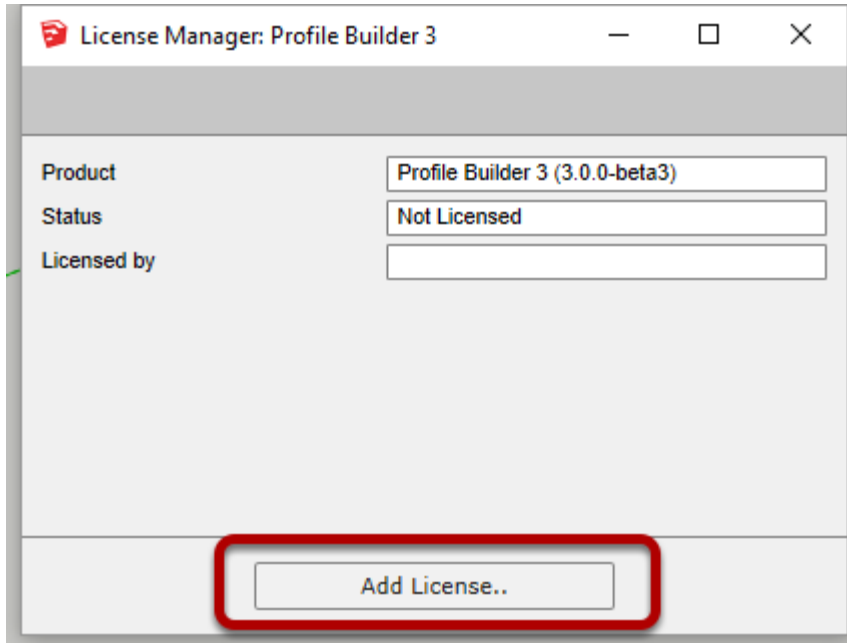
License Manager

Open the License Manager



The License Manager is used to activate and remove your Profile Builder license. Open the License Manager from the Profile Builder menu.

Activate License

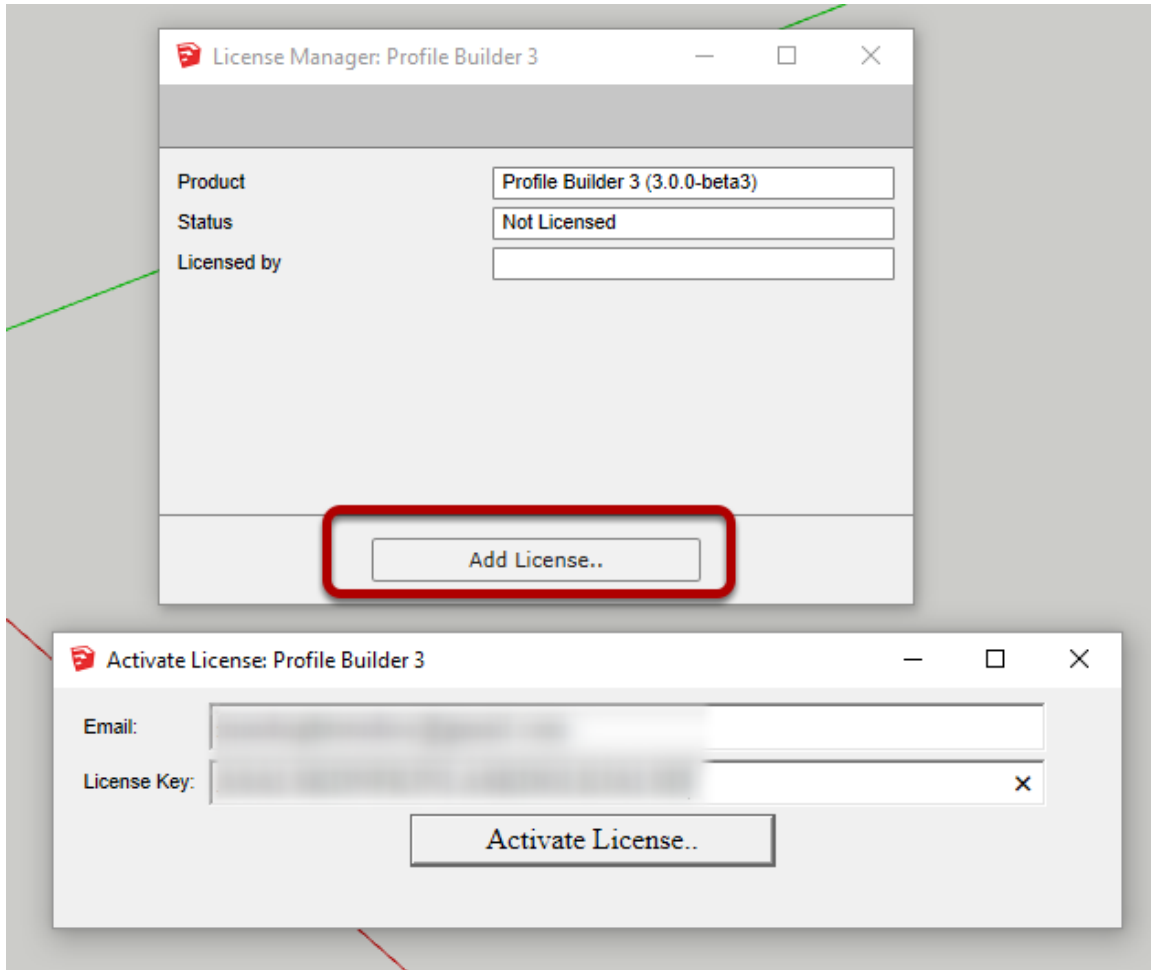


The image shows a Windows-style window titled "License Manager: Profile Builder 3". It contains a form with three labels on the left: "Product", "Status", and "Licensed by". To the right of each label is a text input field. The "Product" field contains the text "Profile Builder 3 (3.0.0-beta3)". The "Status" field contains the text "Not Licensed". The "Licensed by" field is empty. At the bottom of the window, there is a button labeled "Add License..". This button is highlighted with a thick red rectangular border.

Product	Profile Builder 3 (3.0.0-beta3)
Status	Not Licensed
Licensed by	

Add License..

Activating a License

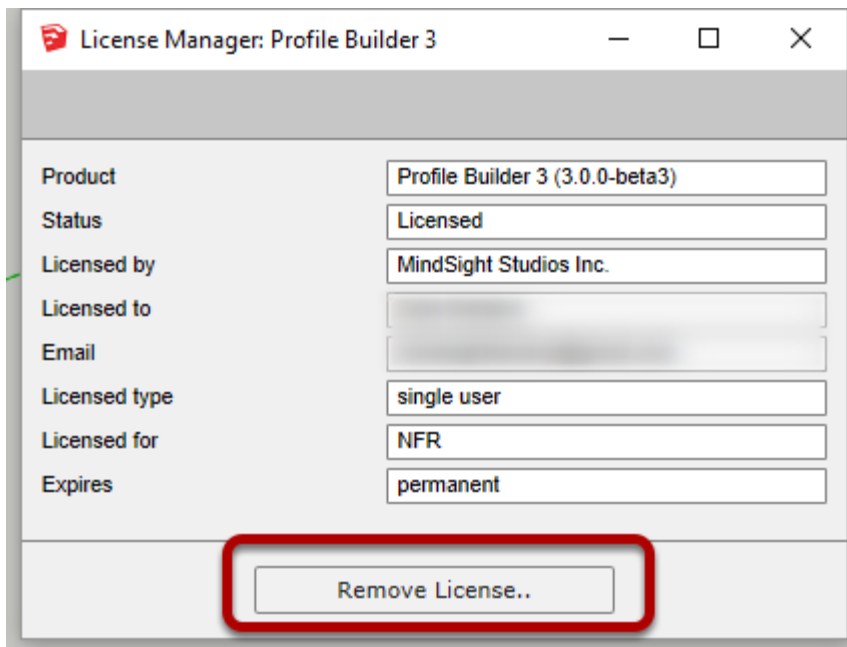


Click the 'Add License..' to activate your trial or full license.

Once you have ordered a license from the [Profile Builder website](#) or any of our [authorized resellers](#), you will receive a license key. Simply enter your email address and the license key to activate the license. Please use copy / paste (CTRL+C and CTRL+V) to enter the key. If the key is not accepted, please try again.

If you are unable to activate your license, please contact customer support.

Remove or Transfer a License



The screenshot shows a window titled "License Manager: Profile Builder 3". It contains a table of license information:

Product	Profile Builder 3 (3.0.0-beta3)
Status	Licensed
Licensed by	MindSight Studios Inc.
Licensed to	
Email	
Licensed type	single user
Licensed for	NFR
Expires	permanent

At the bottom of the window, there is a button labeled "Remove License.." which is highlighted with a red rectangle.

If you would like to remove your license, you can do so by clicking the 'Remove License..' button.

Use this feature if you want to remove and transfer your license to another computer.