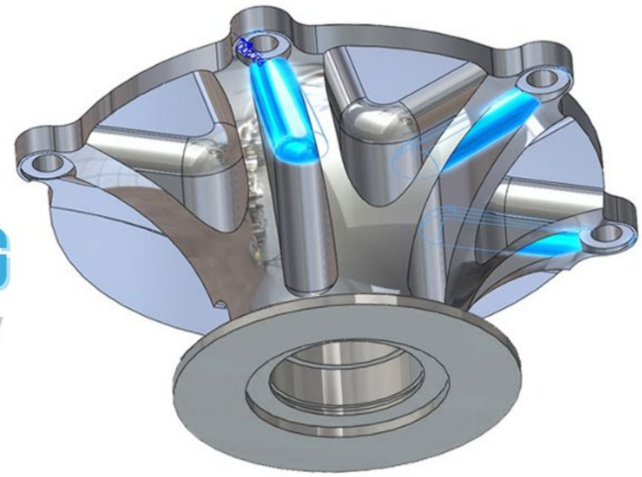
SOLIDWORKS

ADVANCED PART MODELING TAKE-AWAY



Lessons

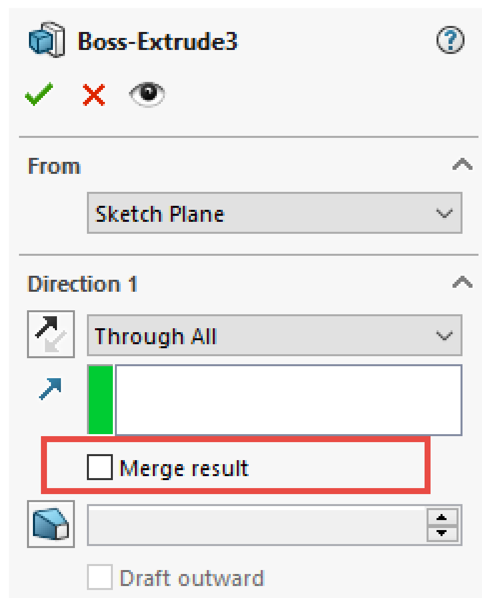
Disclaimer: This document is a comprehensive summary of critical key takeaways from lessons within SOLIDWORKS Advanced Part Modeling offered by GoEngineer. This document should not be considered a substitute for an official SOLIDWORKS training course.

Lesson 1: SOLIDWORKS Multibody Design Techniques

Multibody parts have two or more continuous solids in the same part file. Usually used to end up with a single body design, they can also be used in place of an assembly.

Set to 'show' to always see the folder, or 'automatic' to show the folder only when two or more bodies are present. You can also access this by right-clicking the top of the FeatureManager Design Tree to find 'Hide/Show tree items'.

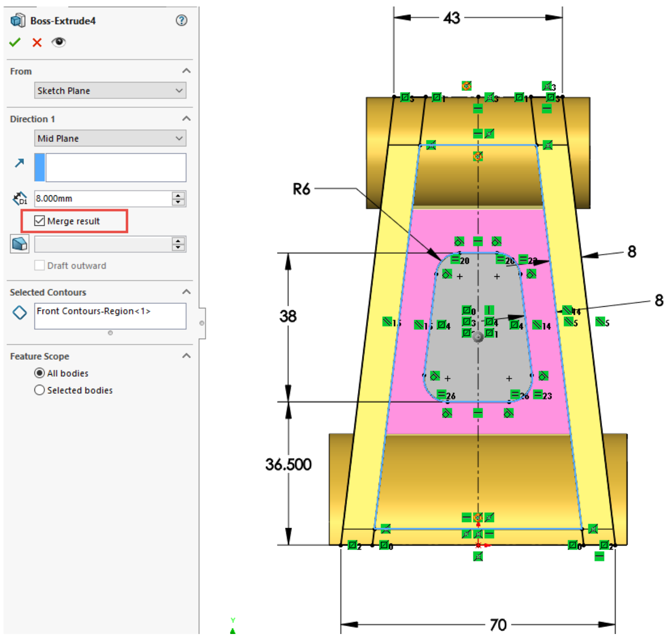
Creating Multibodies



Here are some common ways multibodies can be created:

- Creating bosses from disjoint features
- Creating a boss that is separated by a distance from other bodies
- Creating a boss with the 'merge result' option cleared
- Creating a cut that results in multiple bodies

Bridging



When Boss Extrudes share a volume and the 'merge result' option is on, all bodies that touch will become one.

The center extrusion allows the other four bodies to all merge at once.

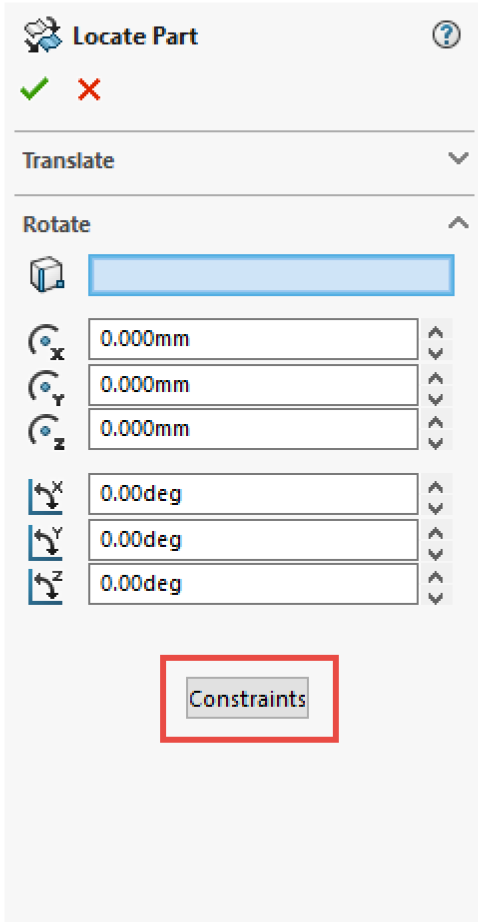
Tool Body or Insert Part

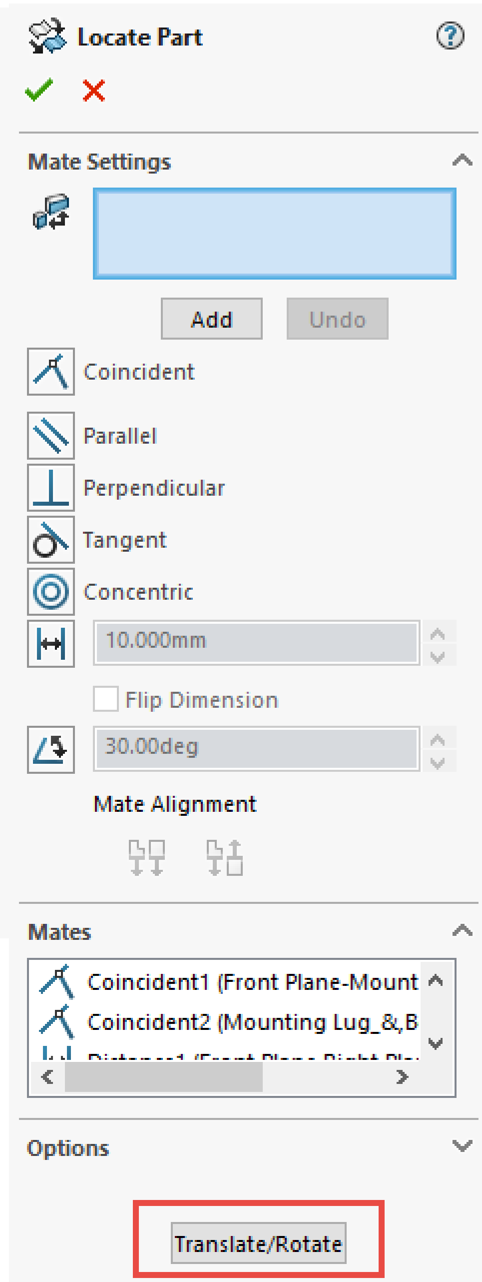
- When inserting a part into another part, it will act as a solid body. You can also choose whether or not to maintain the link (i.e., external reference) to the original file.
- Click OK to align the inserted part's origin to the active part's origin. Otherwise, click to manually place it where you desire.
- Choose the entities to transfer in with the body.
- Optionally, check the box to locate the body afterward using a Move/Copy feature.

Move/Copy Bodies

You can manually move the body by rotating or translating on the translate/rotate tab. You can use mates to align the bodies on the constraints tab.

Click the button at the bottom of the command to switch between these options.

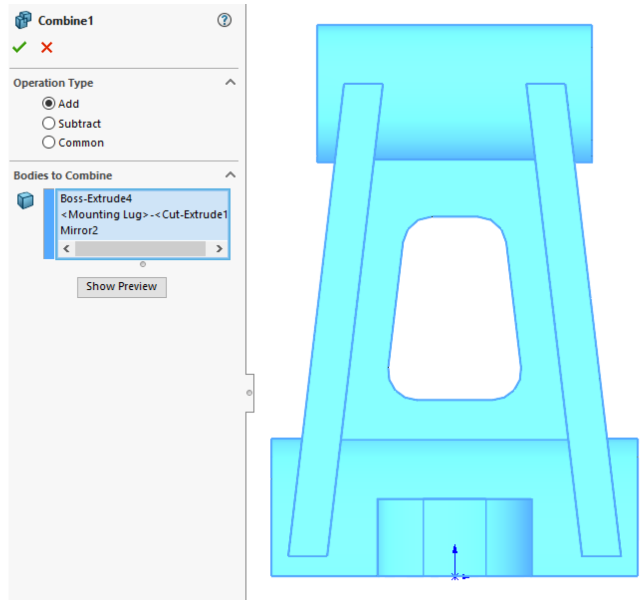


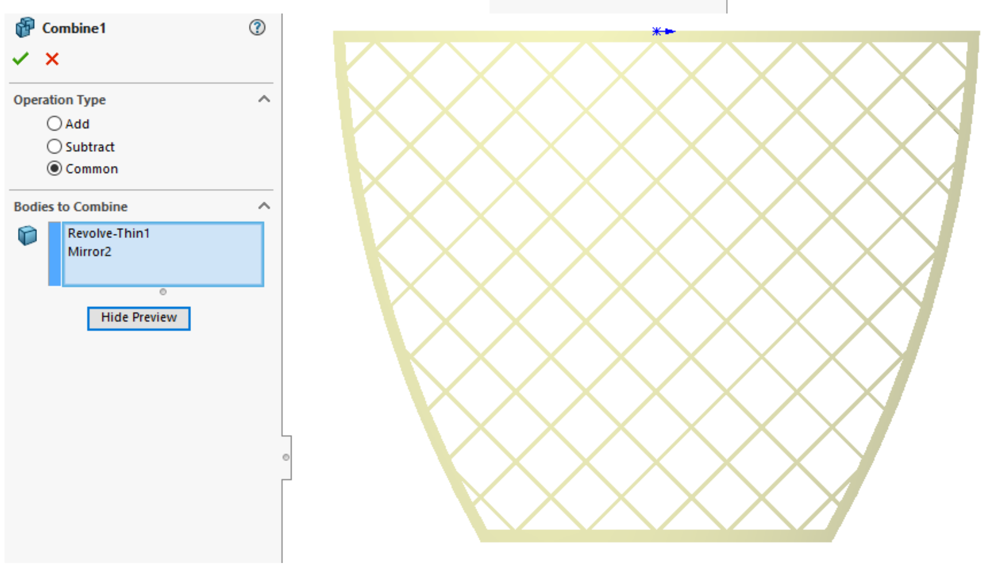


Combining Bodies Using Combine

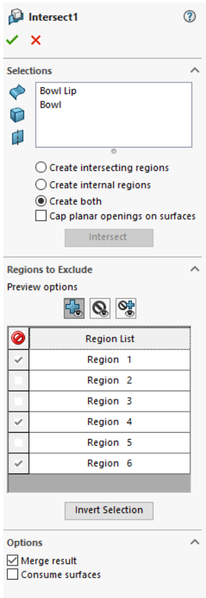
When we have bodies that we need to join, we can use the Combine command and choose from three options:

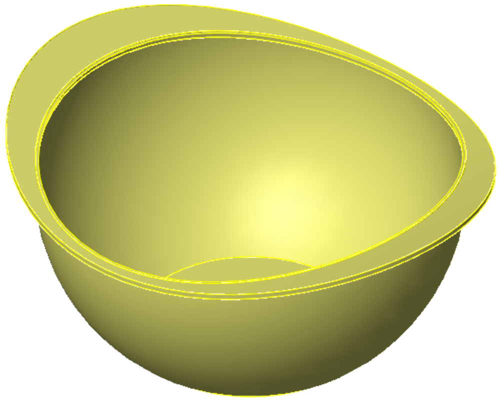
- **Add** - The selected bodies will form one solid by combining the volumes.
- **Subtract** - Removes selected body volumes from a 'main' body.
- **Common** - Uses only the volumes common to all the selected bodies.





Intersect



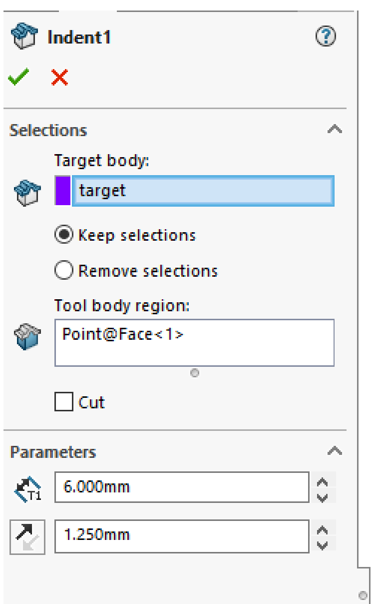


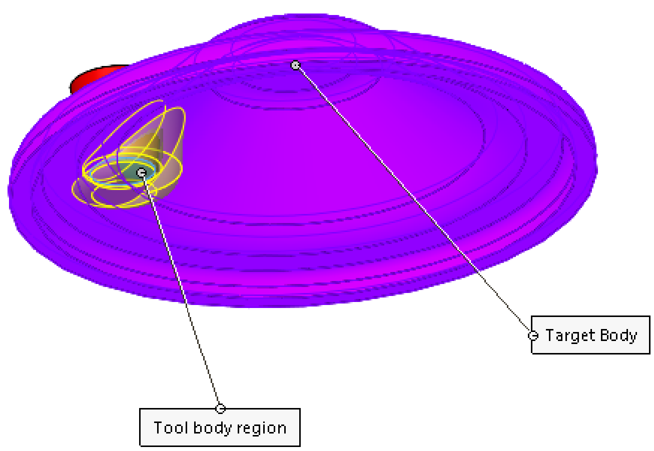
Creating new bodies or merging bodies can also be done using Intersect. This tool allows you to use Solid Bodies, Planes, and Surface Bodies to create your end result.

Select the regions to exclude, or select the regions to keep and invert the selection. The preview shows the desired end result.

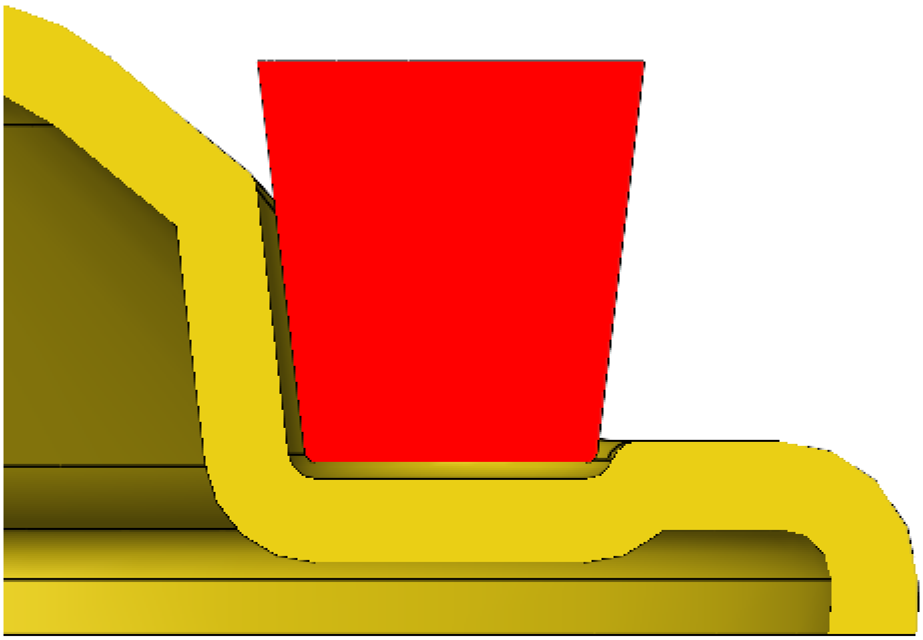
Indent

The Indent feature uses one body to reshape the walls of a target body.





The keep or remove selections will determine the direction of the wall offset. Set the wall thickness and any needed offset from the tool body to get the desired results.



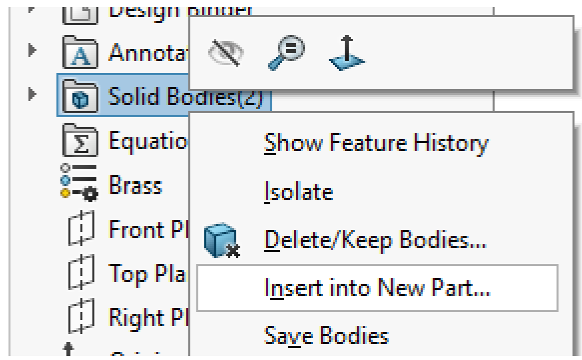
Finally, use the 'Delete Bodies' option to remove the tool body/bodies from the model, avoiding the need to delete it later with the Delete/Keep Bodies command.

Lesson 2: Saving Solid Bodies

When working with multibody parts, sometimes it is useful to turn a body into a separate part file. This can be for modeling convenience or for BOM accuracy.

Insert into New Part

The 'Insert into New Part' command allows you to create new parts from any or all bodies in the Solid Bodies folder. The command is in the Context Menu when you right-click the Solid Bodies folder or a body inside of it. The resulting files are linked to the original part file, so all changes there will propagate to the new part.

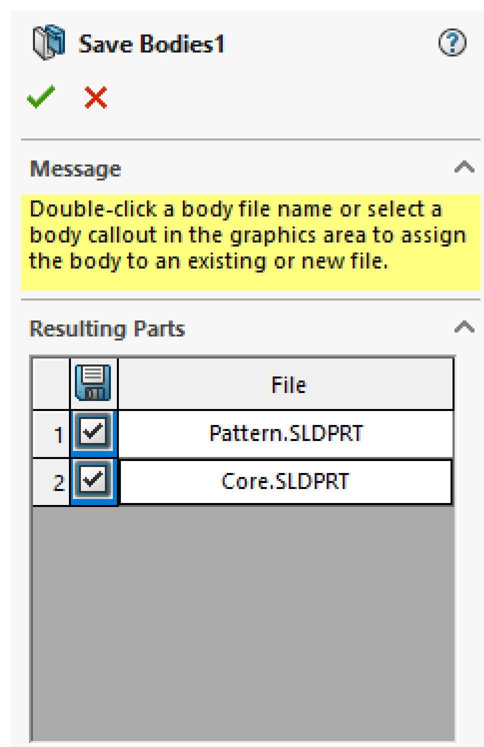


The new parts can then be used in assemblies, or anything else.

Save Bodies

Save Bodies also lets you save bodies as separate files, by right-clicking the Solid Bodies folder in the FeatureManager Design Tree.

Select the bodies from the list (or the graphics area) to mark them for save. Unlike 'Insert into New Part', Save Bodies creates a Feature in the tree, so any features below will not propagate to the new files.

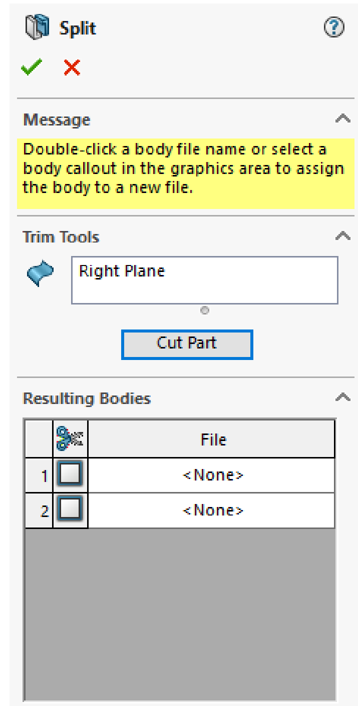


Split

The Split Feature allows a part to be cut into multiple bodies without taking any material away. The Split tool can be a Plane, Sketch, Face, or Surface Body.

After selecting the Trim Tool and clicking 'Cut Part', you can select the resulting bodies to be split and then saved externally or left as separate bodies in the current file.

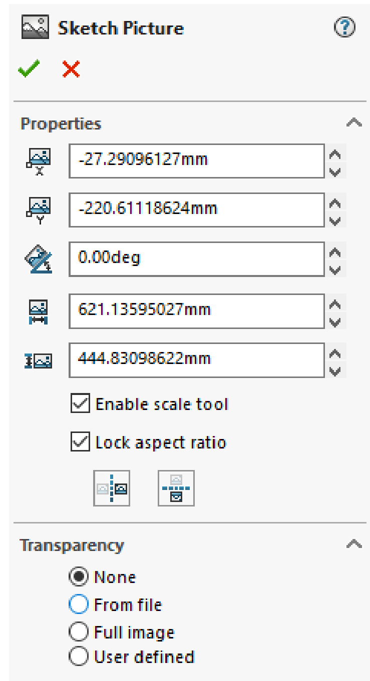
The resulting files are linked to the original as well. Since Split creates a Feature, any Features below that in the tree will not propagate to the new files.



Lesson 3: Sketching with Splines

Splines (also called curves) have a constantly changing curvature and come in a variety of types, each offering a unique way to create and manipulate them.

Using with Sketch Pictures



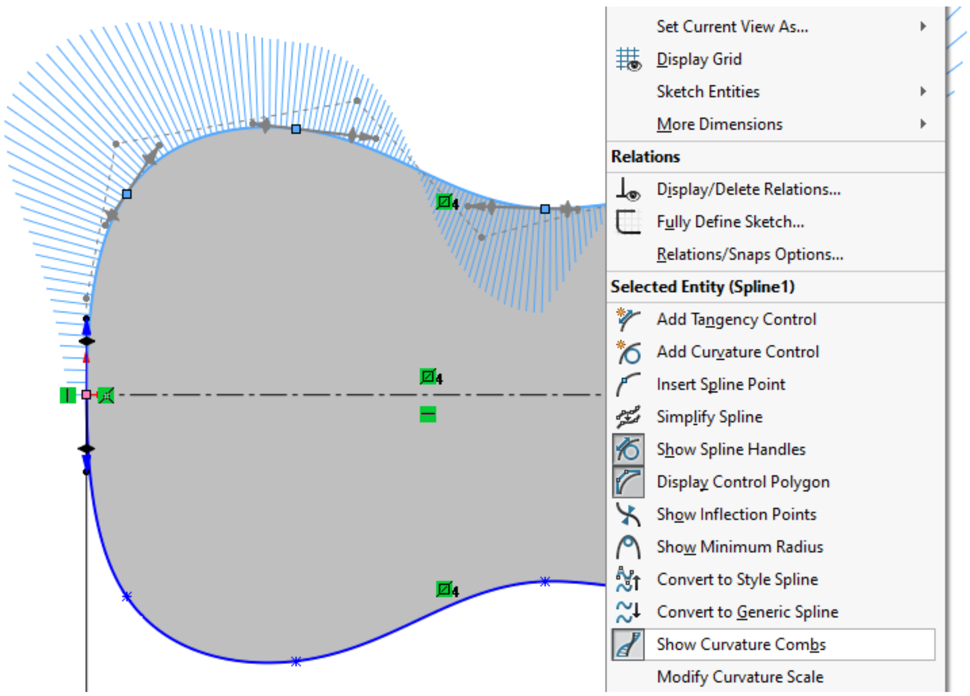
One really good way to reproduce something is to insert a picture of it into the Sketch, scale it to the right proportions, then trace the edges.

In a Sketch, use the Tools > Sketch Tools > Sketch Picture command and then scale using the X and Y dimension boxes in conjunction with the scale tool. Position by dragging and using the X and Y position boxes.

Transparency can also be used to help hide parts of a picture you don't want visible. The user-defined option allows you to select a color from the image, then set the percent of transparency.

Creating the Spline

When sketching Splines, it's best practice to use as few points as possible. This helps the spline curvature stay smoother and extra spline points can be inserted as needed. Usually, a new point is only needed for additional changes in curvature and to keep the spline where it needs to match the geometry.



Spline Manipulation from RMB Context Menu

- **Insert Spline Point** adds another point to the curve/polygon to help shape the spline
- **Show Spline Handles** allows manipulation of the spline by dragging the direction and magnitude of the shape influence
- **Display Control Polygon** allows shaping the spline and keeping it in the simplest form; drag the vertices to shape the spline
- **Show Curvature Combs** shows the curvature in lines that illustrate the smoothness of the curve

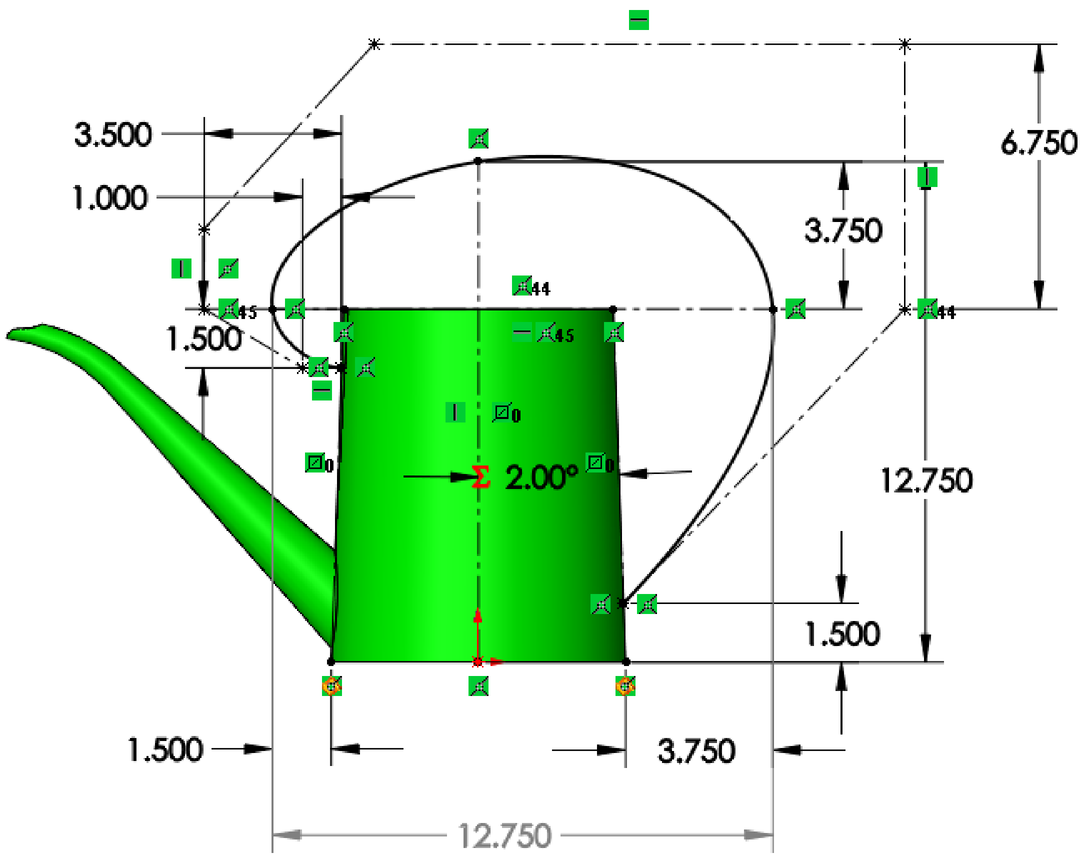
Spline Relations

Splines aren't the same as most sketch entities when it comes to adding relations, but they are compatible with Tangency, Equal Curvature, and Torsion Continuity. These relations help with blending curves together for a smoother transition.

Spline handles can be used to add relations, including Horizontal, Vertical, and others as vector-related relations. In the above image, the Vertical Relation (far left center) is a good example, but note that the curve itself may cross the vertical to maintain curvature as the Spline is adjusted elsewhere.

Style Spline

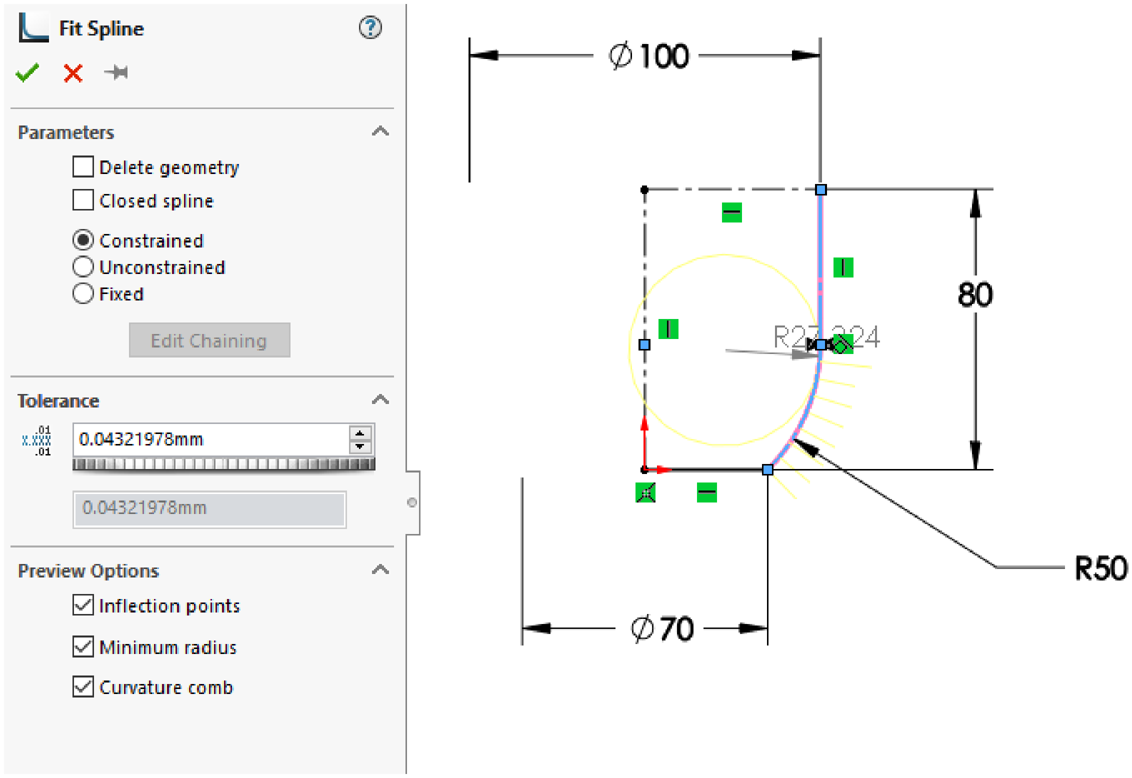
Style Splines are splines created by making points that surround the Spline, not points on the curve. This is very similar to the control polygon of a typical spline. The style spline then creates points and construction lines which can be used to add dimensions and relations to help define the spline and its curvature.



Fit Spline

Fit Spline is used to convert multiple entities to a single spline. It is sometimes needed so there is a single entity, but it is best used for creating a smooth transition between entities.

Fit Spline is also the easiest way to control a spline dimensionally.



Spline Creation and Manipulation Best Practice

There are many ways to create and then manipulate a spline to get the desired result:

- **Create Construction Geometry:** Centerlines and such are great tools to help you get your overall shape and size
- **Sketch your spline:** Remember to keep the points minimal at this step for smoothness
- **Add sketch relations:** Constrain the curve where needed by adding Relations to the Spline or the spline handles
- **Move the Spline** points and use the control polygon
- **Drag the Spline** handles to manipulate the shape (these last two steps may need to be repeated several times)
- If necessary to get the proper shape, **insert another spline** point and readjust

Lesson 4: Introduction to Sweeping

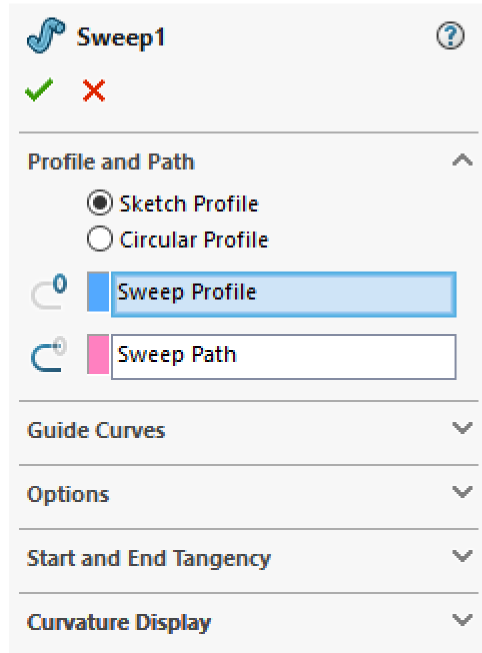
Sweeps are features that have a profile and path that can be either bosses or cuts and simple or complex.

The **path** can be 2D sketches, 3D sketches, curves, and model edges.

The 2D **profile** can be a sketch, a face of the model, or a circle with the diameter input inside the feature (the circular profile option doesn't require a sketch).

Sketched profiles require a single sketch. This sketch must be closed and have no self-intersecting contours. Multiple contours are OK; these can be nested or disjoint to create a variety of shapes.

Best practice is to sketch the path, then attach the profile sketch to it using a Pierce relation.



You have the option to use Guide Curves, Control Twist, and set the Start/End Tangency.

SOLIDWORKS then creates the Sweep by replicating the profile at various positions along the path and then blending them together.

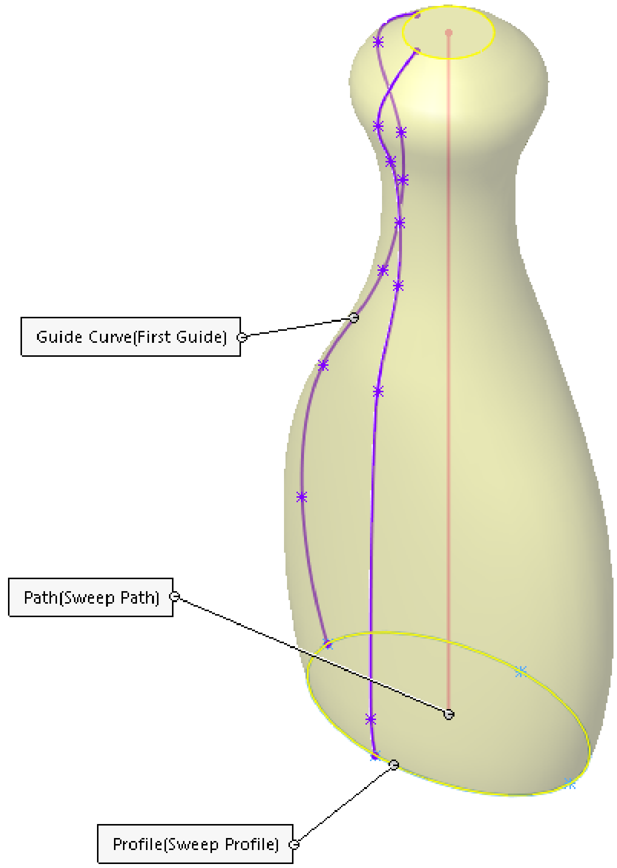
Guide Curves with Sweeps

In create more complex sweeps, or simply help with twist, Guide Curves can be created and used in the sweep.

For creating this bottle, for example, we used an ellipse as the profile. A straight line (vertical) was used as the path. The two guide curves were used to shape the sweep.

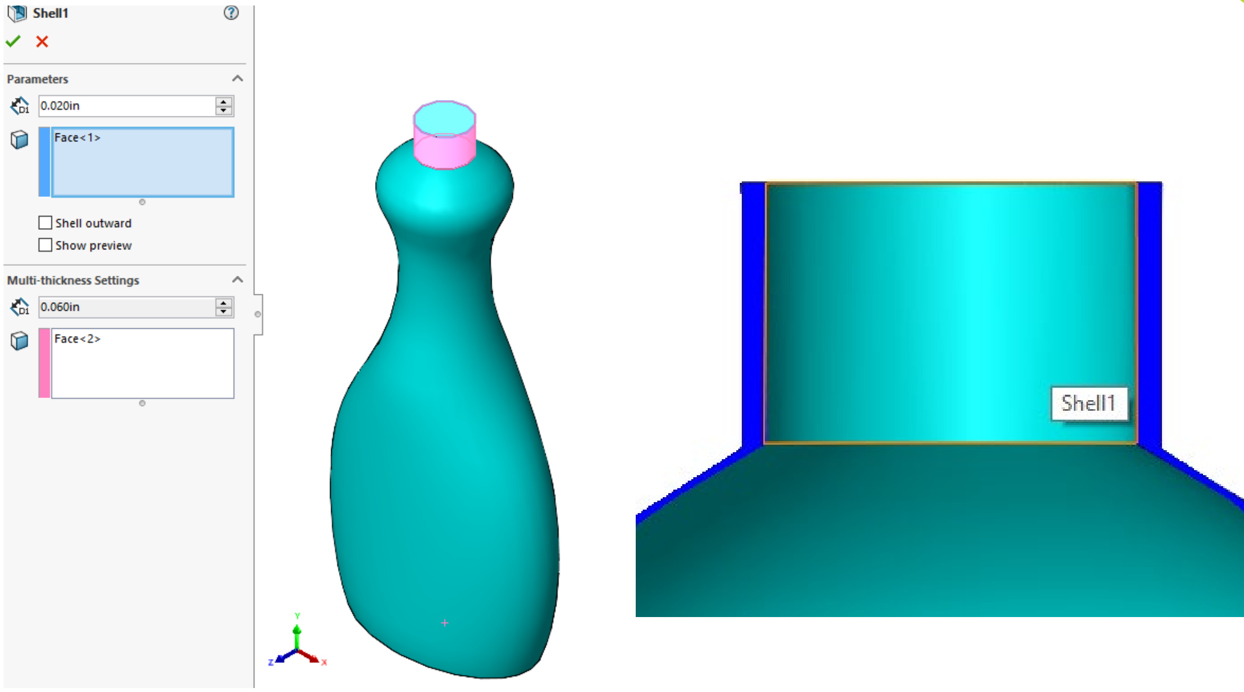
The result is a suitable bottle without creating many profiles or taking much time.

The top was also dimensioned to ensure the top would be a true circle so we could later use that shape to form the neck.



Multi-Thickness Shells

After defining the primary wall thickness, you can then pick specific faces that will require a different thickness. (Note: The command requires that multi-thickness faces meet at a sharp edge.)



Selection Manager

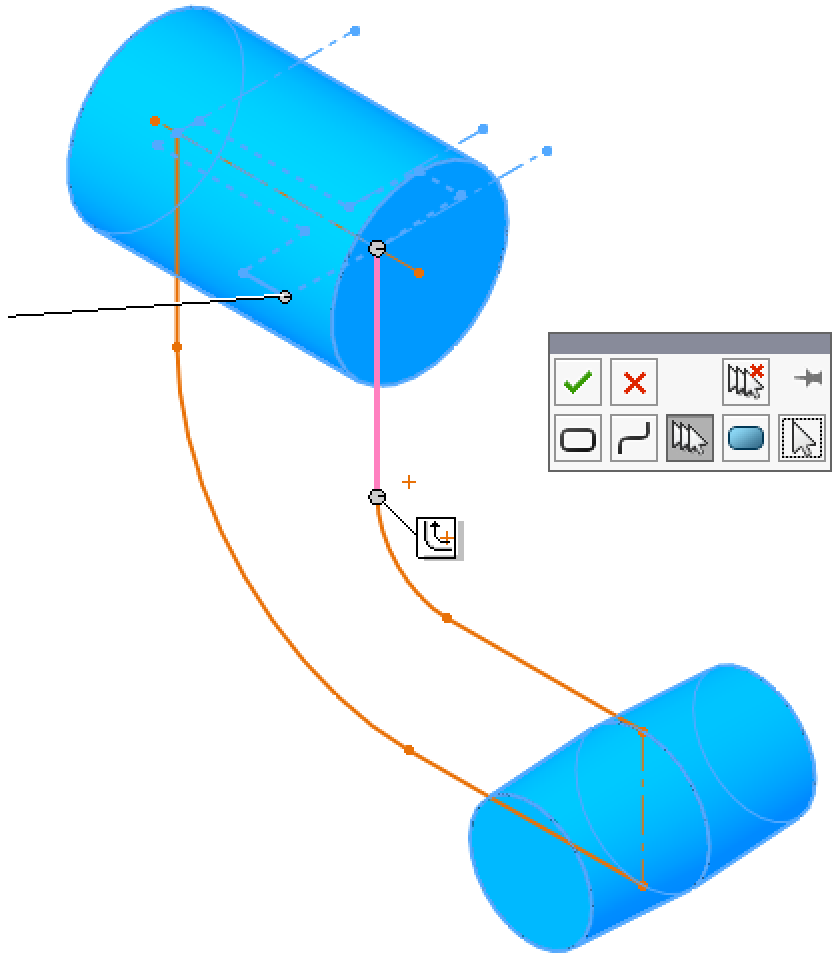
The Selection Manager allows portions of edges/sketches to be used/re-used for many features. In our case, we used only part of a sketch for the path. The other part of the sketch was then used as a guide curve.

Some selection sets will automatically open the selection manager, while others will require you to RMB to access it.

In the bottom row, you can choose from closed loop, open loop, group (a combination of picks), regions, and normal selection.

Once the desired selections are made , hit the ✓ to accept the
selection set. You can also cancel or clear the selection from the buttons in the top row.

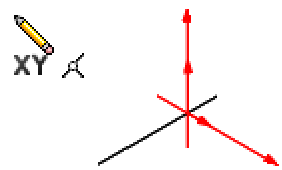
This allows the path and the guide curve to be all in one sketch and easier to manage and create. This is especially true if the geometry is similar.



Lesson 5: 3D Sketching and Curve Features

3D Sketching

3D Sketches are similar to 2D Sketches, but with a few noticeable differences. You will see the cursor change when the Tab key is pressed.



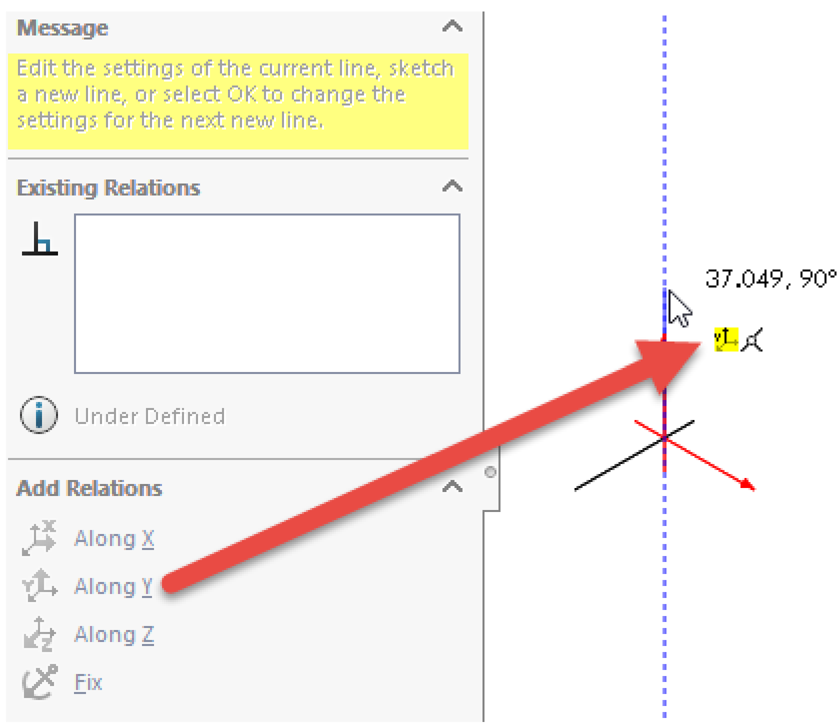
- XY – normal to the Front plane
- YZ – normal to the Right plane
- XZ – normal to the Top plane

Pressing the Tab key cycles through these.

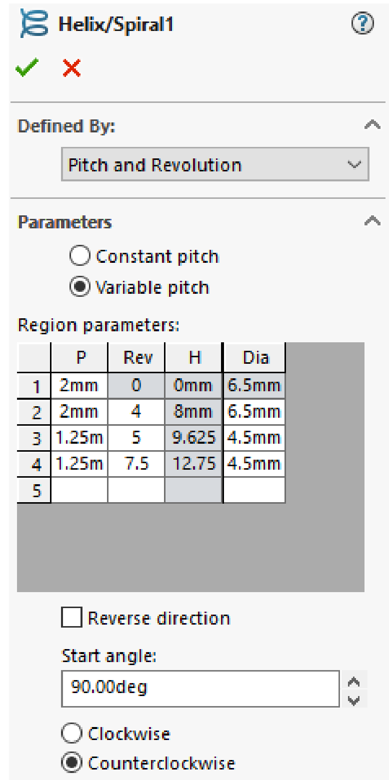
You can also create an Offset Plane by clicking a plane, holding down the CTRL key, and dragging the mouse. Double-clicking in space will release the Sketch plane, going back to 3D Sketch mode.

When not locked onto a face or plane you will also see the 'Along X' (or Y or Z) relation instead of Vertical and Horizontal.

Many familiar relations are slightly different in 3D Sketches. 'Parallel to plane', 'on plane', etc. are important to master in order to define your 3D Sketch accurately.



Helix/Spiral

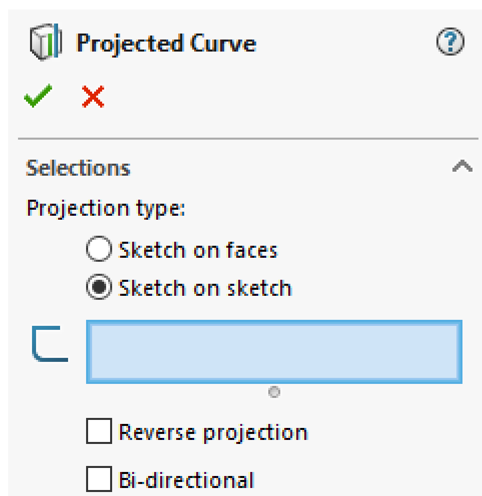


A Helix/Spiral uses a circle to determine the starting diameter and location. You can then set the options based on pitch, height, and number of revolutions.

Other options are reversing the direction, setting the start angle, and clockwise/counterclockwise. Once completed, you will have a 3D Helix or a 2D Spiral.

Many times, a Helix is used for cutting or adding threads, but there are many other uses for them.

Projected Curve

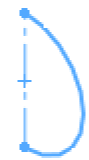


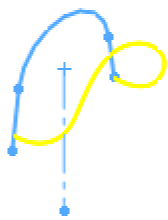
Projected Curves use Sketches to either project into another Sketch or onto the face/faces of the model.

'Sketch on Faces' will project the Sketch to the faces in the direction it's projected.

'Sketch on Sketch' will project the Sketches together, keeping only the geometry common to both, where they intersect.

The resulting curve might be very difficult if not impossible using normal 3D Sketch tools.





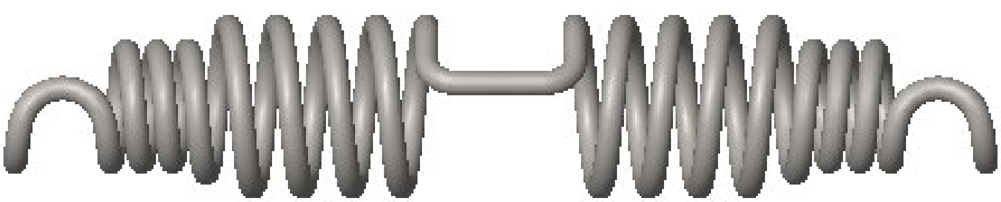
Composite Curve

Composite Curves combine Curves, Sketches, and/or model edges into a single curve. This can be very important, as some features will require a single entity to be used in the selection set.

To use, simply select the geometry desired and hit OK.

In this case, the sweep using the Composite Curve wasn’t exactly as smooth as could be. To correct that, use a 3D Sketch and convert entities to copy the curve. Then use Fit Spline to get a better transition between the 3D Sketch and the Helix.

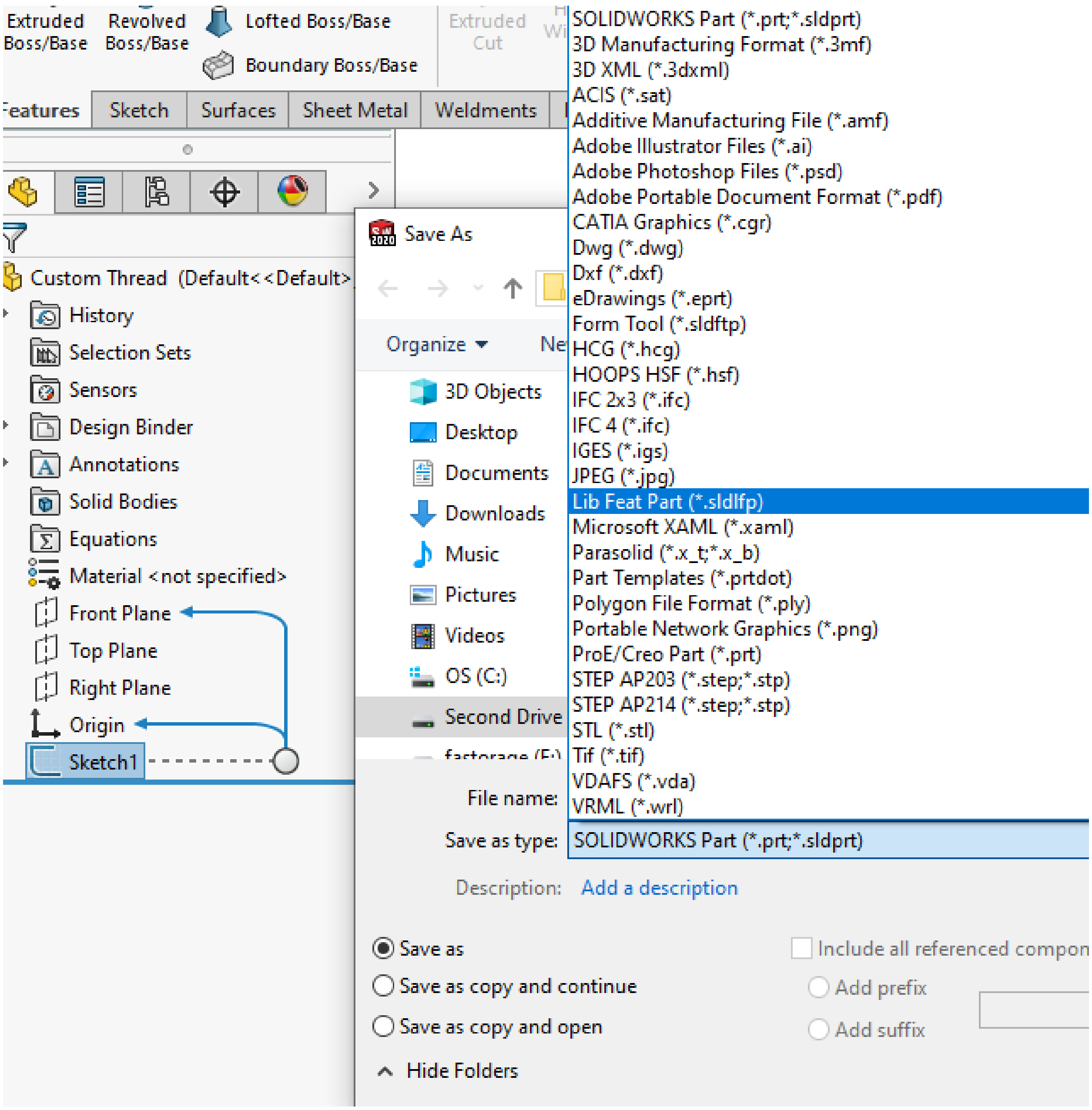
The part (or even the 3D Sketch) can be mirrored to complete the body. The profile for the sweep was set to 'circular' with the proper diameter.



Lesson 6: Threads and Library Feature Parts

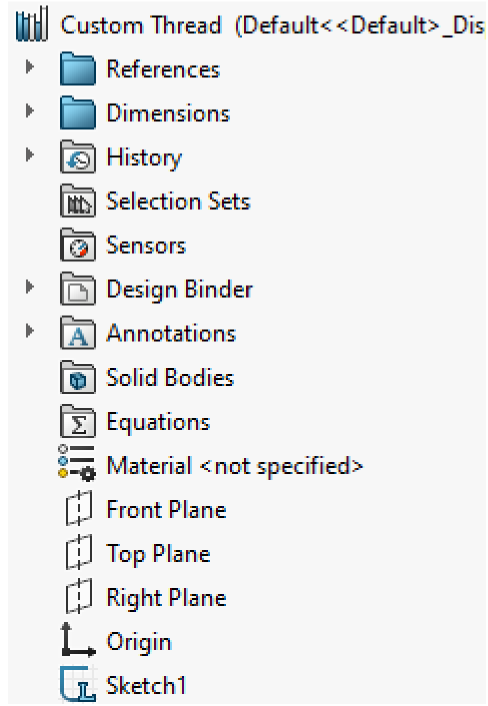
Library Feature Parts

Library Feature Parts are a great way to save and reuse data. They can be as simple as a Sketch for a profile, (e.g., weldments and threads) or a series of Features. To accomplish this, ensure the Sketch is selected, then Save As, and choose 'Lib Feat Part (.sldlfp)'.



If done correctly, the sketch should have an “L” symbol on it, as shown below. If not, RMB the Sketch and choose 'Add to Library'. If the Sketch is not set as the Library Feature, SOLIDWORKS will try to insert this as an entire Part, not just the Sketch. The same functionality applies if you are trying to use Features.

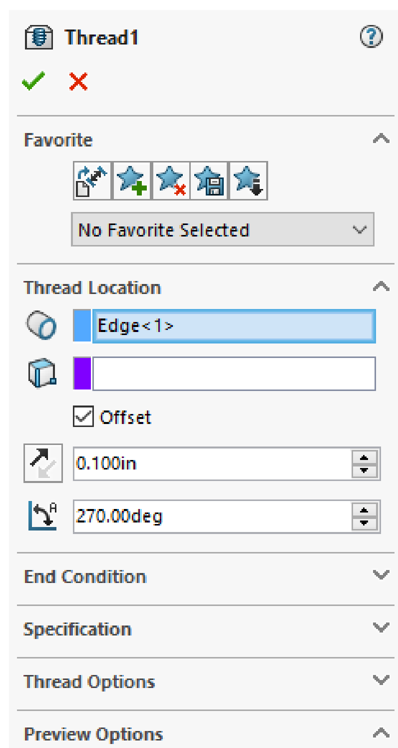
If you are making a thread profile or weldment profile, make sure you save in the proper location. You can find this under Options > System Options > File Locations and then selecting the proper type. Failure to save in the right location will result in not being able to use the file.



Threads

When making a thread, remember the default pierce point will be the origin. You can also set a default pitch by creating a horizontal centerline beginning at the origin.

When using the Thread Feature, you need to select the Thread Profile and a start circular edge. The edge will be used to create the Helix based on the other parameters you choose (e.g., pitch, revolutions, etc.). Use End Condition, Specification (size, pitch), and Thread Options to get the desired result. The profile can be rotated and mirrored as well.

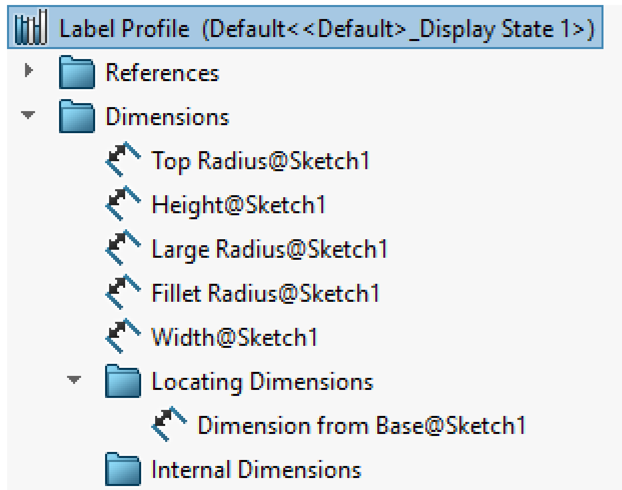


When inserting Library Features, we are often faced with a series of selections like planes and edges. We can understand these by looking at the Sketch and then understanding that Sketches always require a Sketch Plane.

In Sketch Mode, choose display/delete relations. You can then change the filter to 'external'. This will let you know what entities in the Sketch refer to something outside the Sketch such as points, edges, and planes. Now you will know when you use the Sketch as a Library Feature what it will be looking for.

Once you have created the Library Feature you can also set up locating Dimensions. This will help with positioning and resizing the Library Feature without having to edit the Sketch(s).

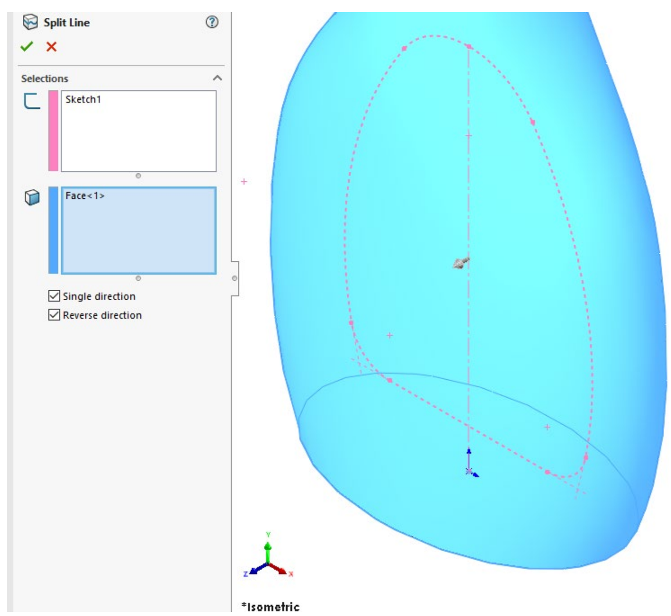
Drag any of the dimensions to the Locating Dimensions folder. You can also drag dimensions to the Internal Dimensions, so they are not available to modify when inserting the Library Feature.



Split Line

Split Line Features are curves that create edges or faces on the surface of a model. It will create edges but will not change the geometry of the part.

- **Silhouette** – projects the outside of the part to a face or plane
- **Intersection** – projects the outside of the part where a plane or surface body intersects
- **Projection** – projects a sketch or curve onto the face of the part



Lesson 7: Advanced Sweeping

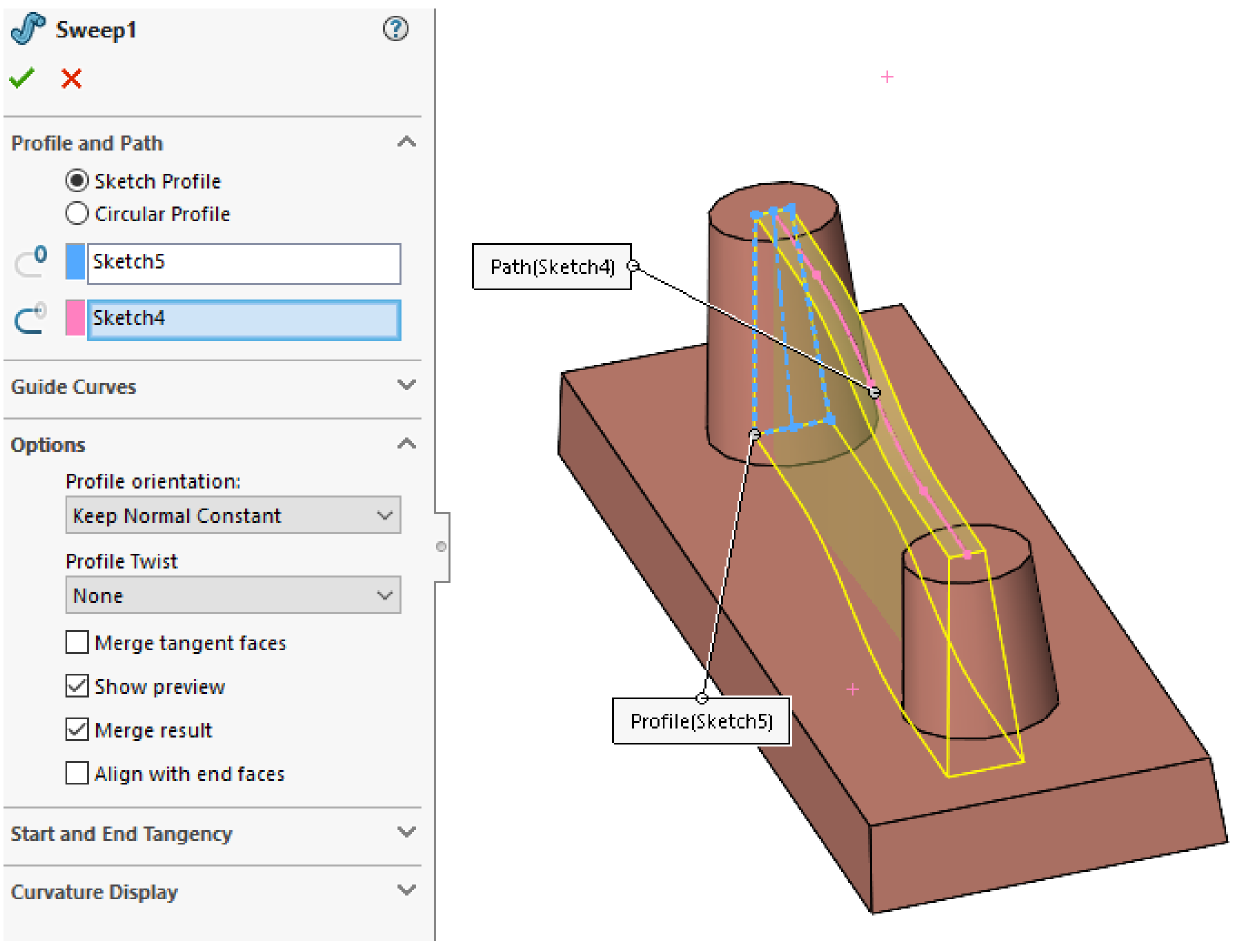
Profile Orientation

Since Sweeps are created by blending intermediate sections of a profile at various positions along the path, it is important to understand how the orientation of the intermediate profiles are controlled.

We can think of the profile orientation in terms of Pitch, Yaw and Roll. The Pitch and Yaw are defined by the orientation of the profile to the path. The Roll is the twist or spin of the profile around the path.

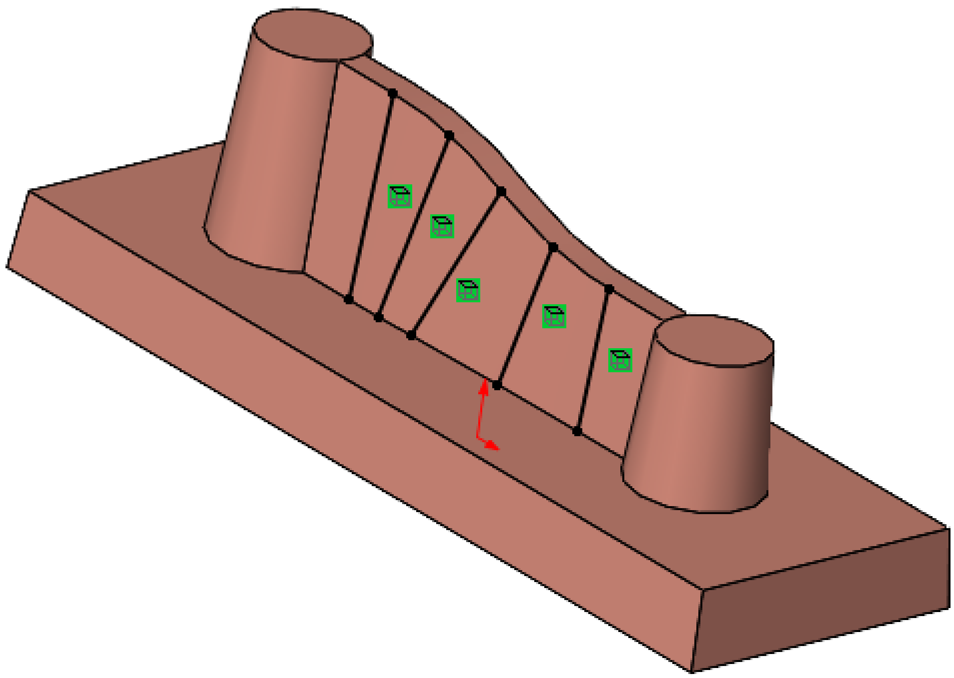
We can either use the Twist Options or Guide Curves to control the Roll.

In our case study, we found that using the default Profile Orientation ("Follow Path") did not maintain our design intent of 5.00° of draft.



Once we changed the Orientation to “Keep Normal Constant”, we had the desired effect. The reason for that is that Keep Normal Constant keeps the intermediate profiles parallel to the original. “Follow Path” allows the profile to Pitch/Yaw to match the original angle between the path and the profile. In this case, the profiles would remain perpendicular to the path.

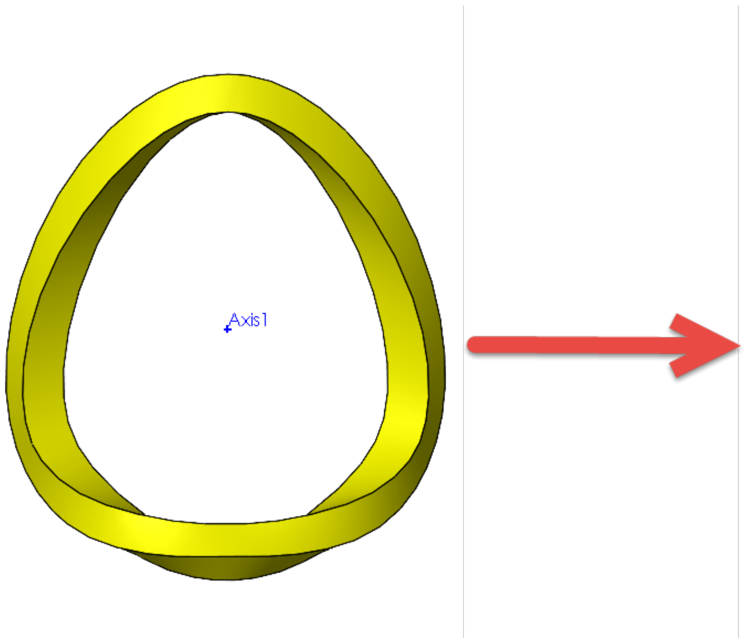
We were able to use Intersection Curves in a sketch to get the intersection of a plane and faces. We could then measure and found it to be < 5.00°.

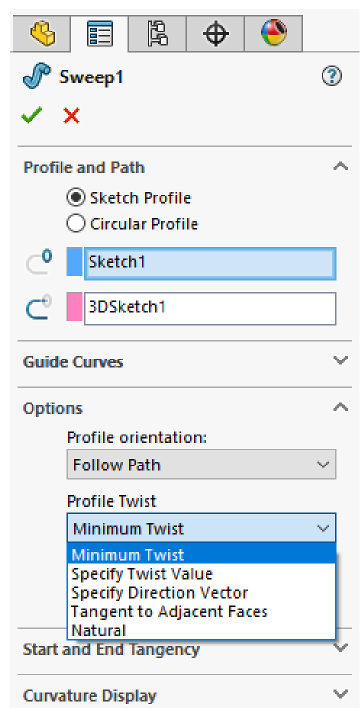


(Another way to preview how the intermediate profiles would rotate is by using face curves in a 3D Sketch: Start a 3D Sketch, turn on Face Curves and select a face.)

Controlling Twist

When a sweep twists by default, you can use the options to control or minimize the twist. For this part with a significant twist (left), we’ll minimize it (right).





Changing the Profile Twist to "Minimum" did not fix it, so we were able to use the Specify Direction Vector. In this case we used an Axis, but we could have used a plane instead.

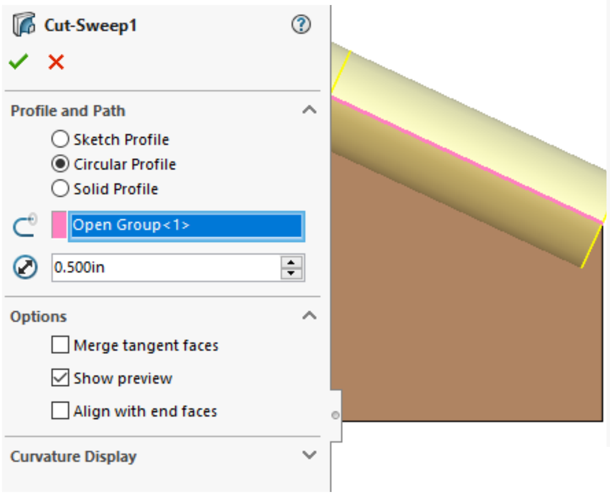
In other cases, you may want to specify the twist value, creating twist when there would have been none. This is a great way to create twisted items without having to create a helix first.

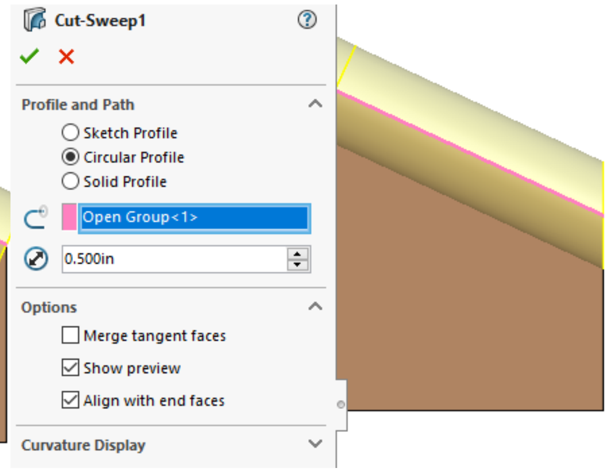
Undesired twist can often be caused by relations in the profile sketch; Horizontal and Vertical relations are the usual suspects. If you think about it, the orientation of "vertical" and "horizontal" may change as the sketch profile moves through space.

To keep this from happening, you can use pierce, parallel and perpendicular instead. If that is not achievable, you will need to create guide curves. These would be normal candidates for projected curves as well.

Align with End Faces and Model Edges Sweep Path

When going through the Sweep Options, you may notice “Align with end faces”. This is particularly useful when using keep follow path, which is the orientation for a circular profile.



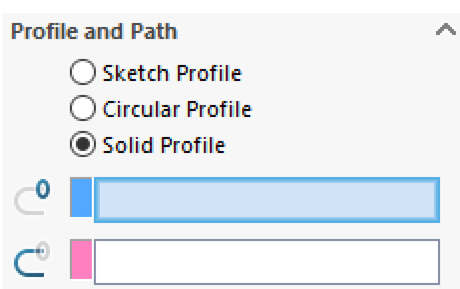


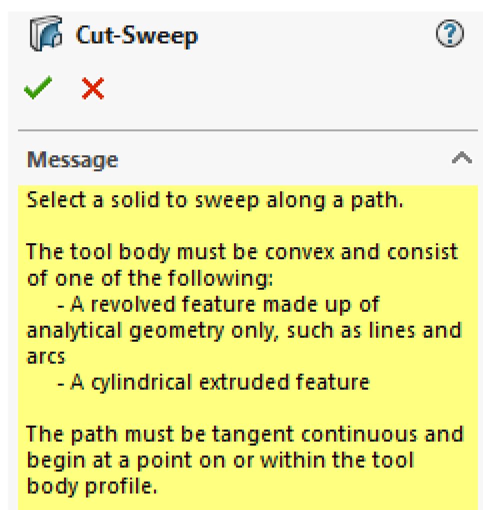
In this case “Align with end faces” extends the cut all the way to the end face.

If using model edges to define the path, make sure to use the Selection Manager, as the field will only pick the first edge you pick by default. This means you don’t need to recreate the edges as a sketch.

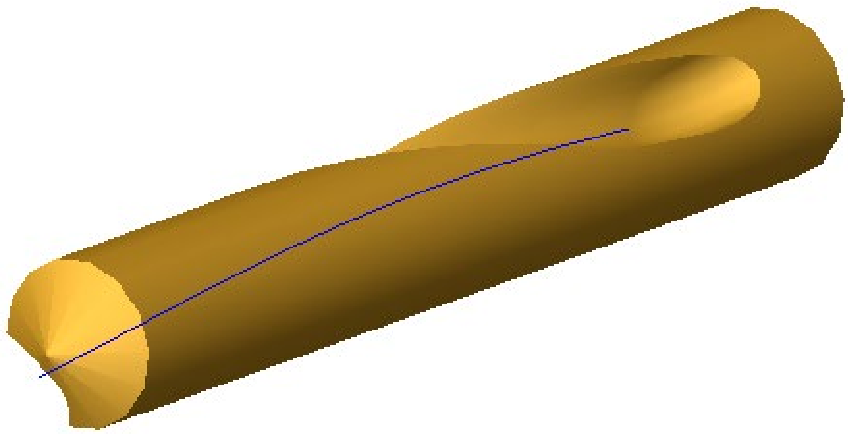
Sweeping with a Solid Profile

Instead of a sketch profile, you can use a separate body for a Swept Cut, then select the body and the path as usual.





The resultant cut would be much harder to create as a normal Swept Cut, particularly the end of the cut.

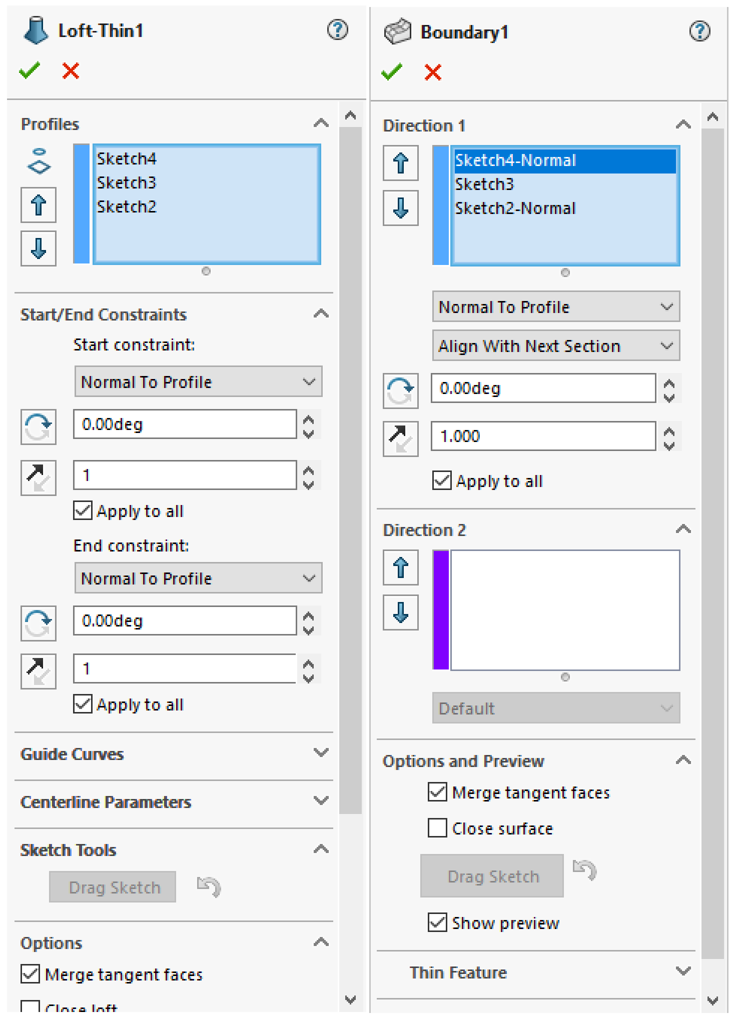


Lesson 8: Loft and Boundary Features

Loft vs Boundary Features

Lofts are used to blend multiple profiles of different shapes. They can use Guide Curves and even a Centerline. However, control is limited to the beginning and end profile orientations.

Boundary Features offer all the same options as a Loft, plus the ability to control intermediate profiles and the use of other directions (guide curves). However, it doesn't have centerline control and can lead to greater rebuild times.



Both connect points in the profiles to the coinciding point on the next profile. If you click near a point, it will snap to the closest point. If you don't have points, like a circle, it is difficult knowing where the connections will be. You can manually tweak the connectors. Remember to pick profiles in order or use the blue arrows to change the order.

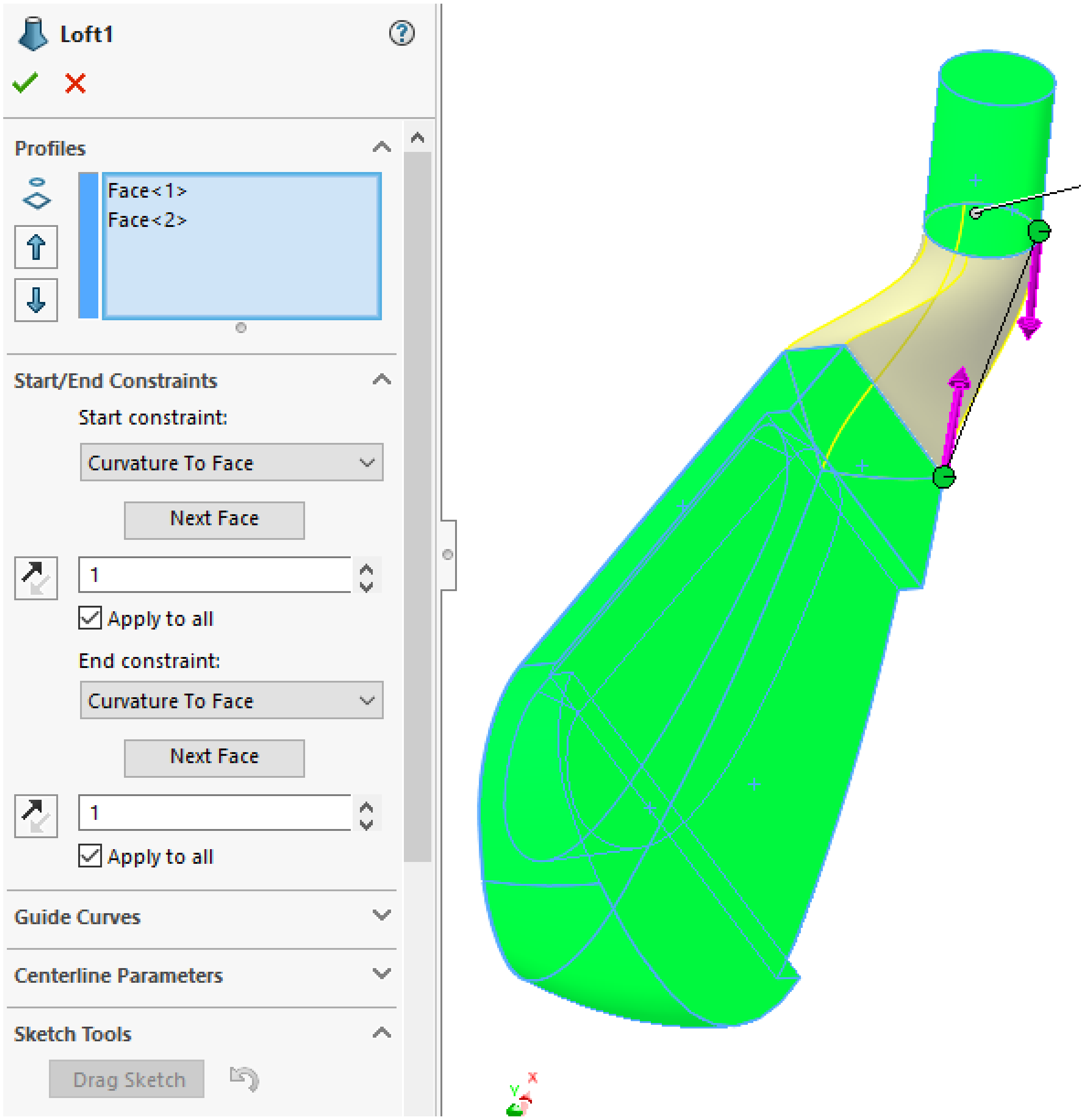
Other options include "Thin feature" and "Merge tangent faces".

It is more common to use Lofts, but as you can see, both Features are very similar.

Loft and Boundary Features on Solid Bodies

When creating a transition between faces on a model, both Features do a great job. Select the end faces, then set the constraints (you can even adjust the Start Tangent

Length by typing another number or dragging the arrows in the graphics area). Not only is this fast, but it easily does things that are difficult with normal lines, arcs, etc.



Derived and Copied Sketches

Copying and pasting works the same way with a Sketch as it does a file. The new Sketch is not linked to the original.

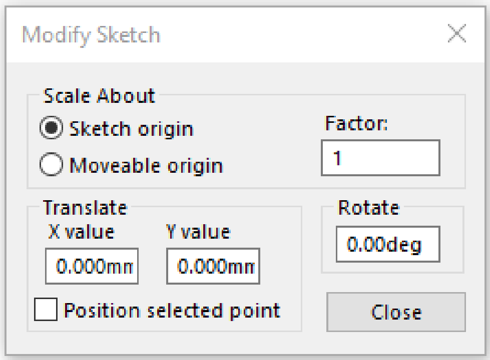
But with a Derived Sketch, you get an exact, linked, duplicate. You are able to locate and/or re-orient the sketch, but it won't have dimensions or relations to edit.

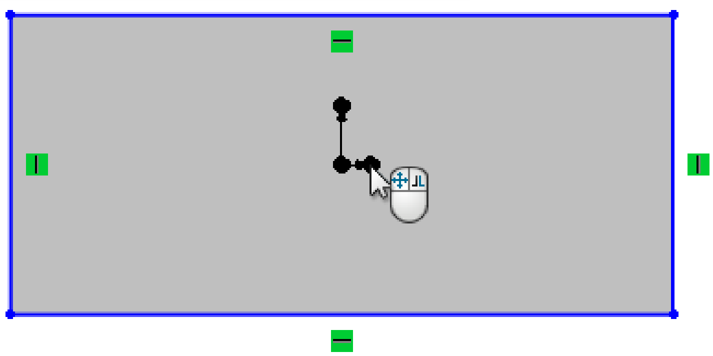
To create a Derived Sketch, select both the Sketch to copy, and the plane/face to paste to. Then click "Insert > Derived Sketch". You will now be editing the new Sketch.

Remember, any changes to the original will affect the Derived Sketch, but not a copied and pasted Sketch.

Modify Sketch

Modify Sketch allows you to move, rotate, scale, and mirror a sketch. You can use the dialog box or use the mouse to drag and move (LMB) or rotate (RMB). Using the black manipulator icon, you can also mirror by right-clicking on the spheres. The three spheres have different mirror options, and one should do exactly what you need.





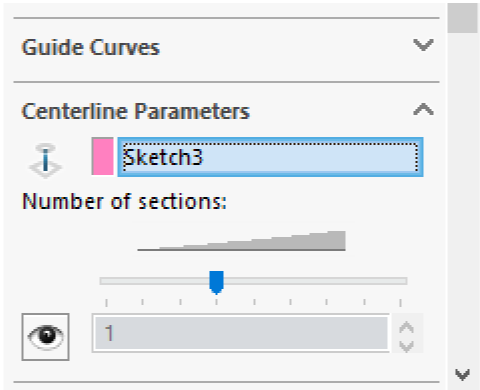
Lesson 9: Advanced Loft and Boundary Features

Using Secondary Curves

Lofts and Boundary Features both allow use of secondary curves. For Lofts, they're called "Guide Curves"; Boundary Features are added as "Direction 2".

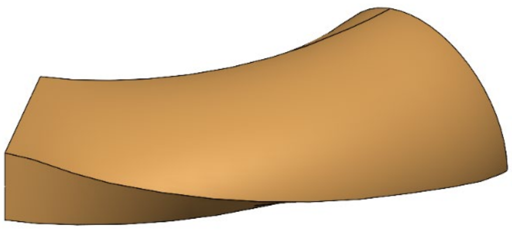
Secondary curves have more influence on the overall shape of Lofts, whereas with Boundary Features, both direction 1 and 2 can have equal influence. This is part of how you can get different results between the two features.

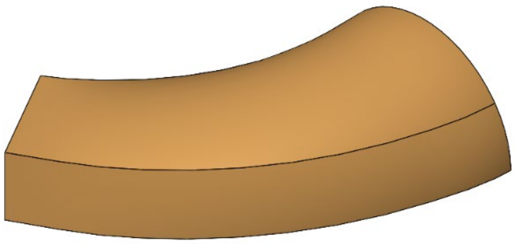
Lofts, however, also allow for the use of a centerline. In our example, we used this option to control how the intermediate profiles were blended.



Profiles and Segments

When using Loft or Boundary Features, the way the profiles line up is determined by the points you click (on or near) while selecting them. When profiles have different numbers of segments/vertices there may be undesired results. Creating a new profile with the same number of segments will fix this. You can use the Split Entities sketch tool to split profiles such as arcs to get the total number of segments (and thus points) to be the same. Note the difference when the number of segments is equal as shown below.



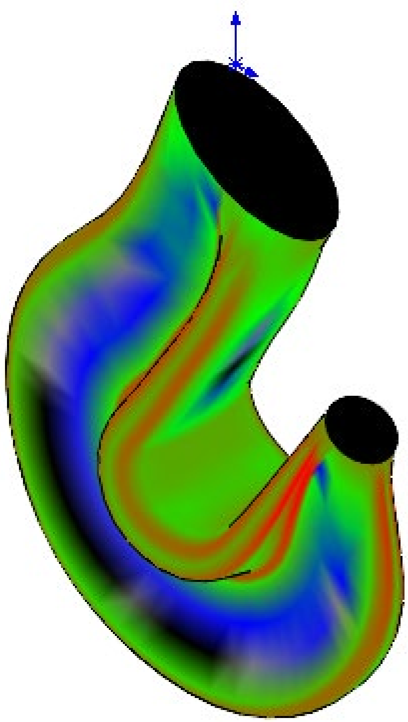


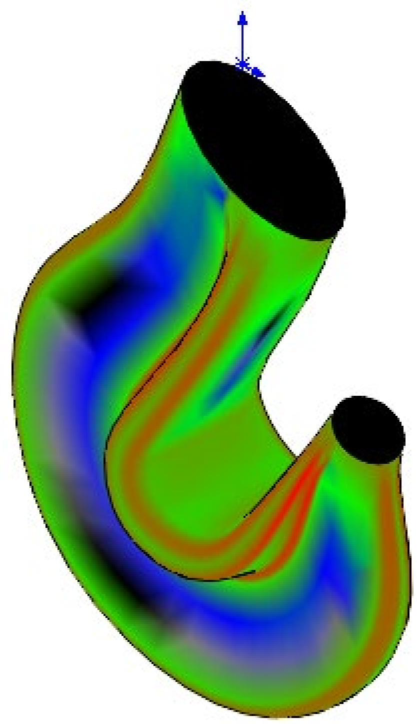
You can also manually adjust the way the profiles come together by using "Show all connectors" from the context menu, then dragging them around. Be aware this isn't as precise but is more of a drag-and-drop-as-close-as-you-can sort of tool.

Curve Influence

In this example, we used an ellipse in each profile Sketch and splines in the middle to model a hook two ways. For this part, one of the big differences between the Loft and Boundary was the Curve Influence.

The Loft Feature has limited options for how we can apply tangency control to the guide curves, while Boundary has all the same options for Direction 2. This can result in some noticeable differences between the two features. Which to use in practice will depend on your part's geometry. The differences are more noticeable when the "Show Curvature" option is enabled, to get a color representation for changing curvature.



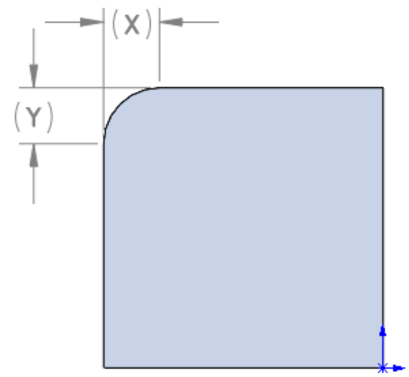


Lesson 10: Advanced Filleting and Other Features

Fillet Method and Fillet Type

There are four Fillet Types: Constant Size, Variable Size, Face Fillet, and Full Round. Though the Constant Size with "Symmetric" method is most common, each of these fit certain applications, and thus there is no one-size-fits-all fillet.

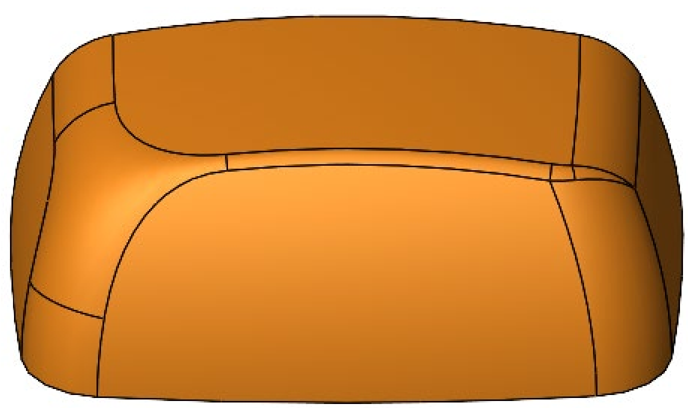
Fillet methods are Symmetric, Asymmetric, Chord Width, and Hold Line. Symmetric means that the X and Y distances will be equal. Asymmetric, naturally, means X and Y are not equal. Chord Width refers to the distance between the ends in a straight line. Hold Line uses model edges to determine the radius and tangencies.



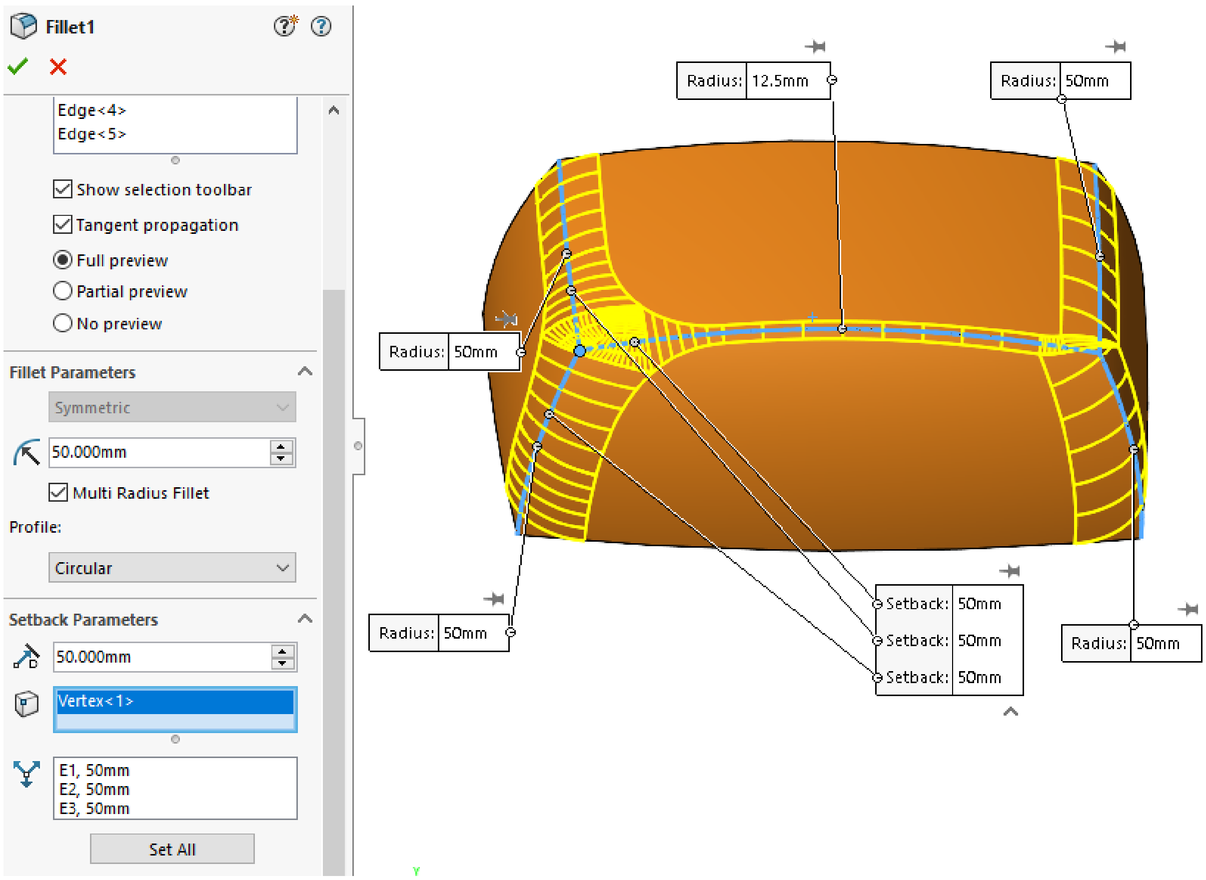
- The "**Symmetric**" method is available for Constant Size, Variable Size, Face Fillet and Full Round
- **Asymmetric** is available for Constant Size, Variable Size, and Face Fillet
- **Chord Width** and **Hold Line** are only available for Face Fillets

Setback Parameters and Multiple Radius

Setback Parameters are used to define how far away from a common vertex three or more fillets begin to blend. This can result in a much different look despite using the same radius.



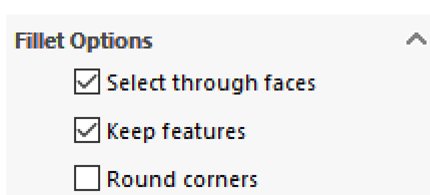
You can also set multiple radius values in the same fillet command. These options are only available for Symmetric fillets however.



In our example, a Delete Face Feature (with “Delete and fill,” and “Tangent fill” options enabled) was used to smooth the altered fillet.

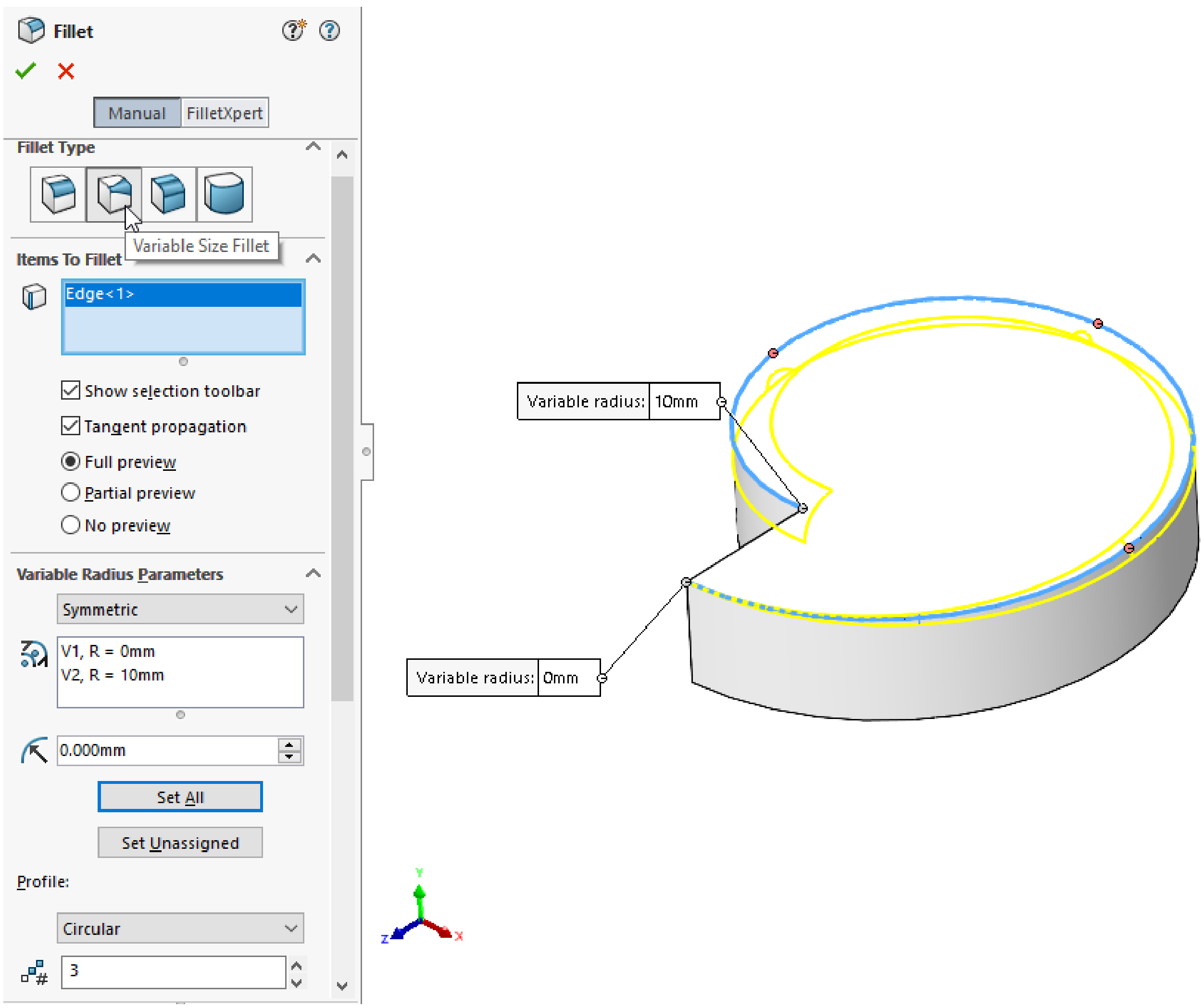
Keep Features

Another Fillet Option for Constant Size Fillets is “Keep Feature”. This defines whether or not existing Features in the model should remain if the new Fillet completely surrounds them.



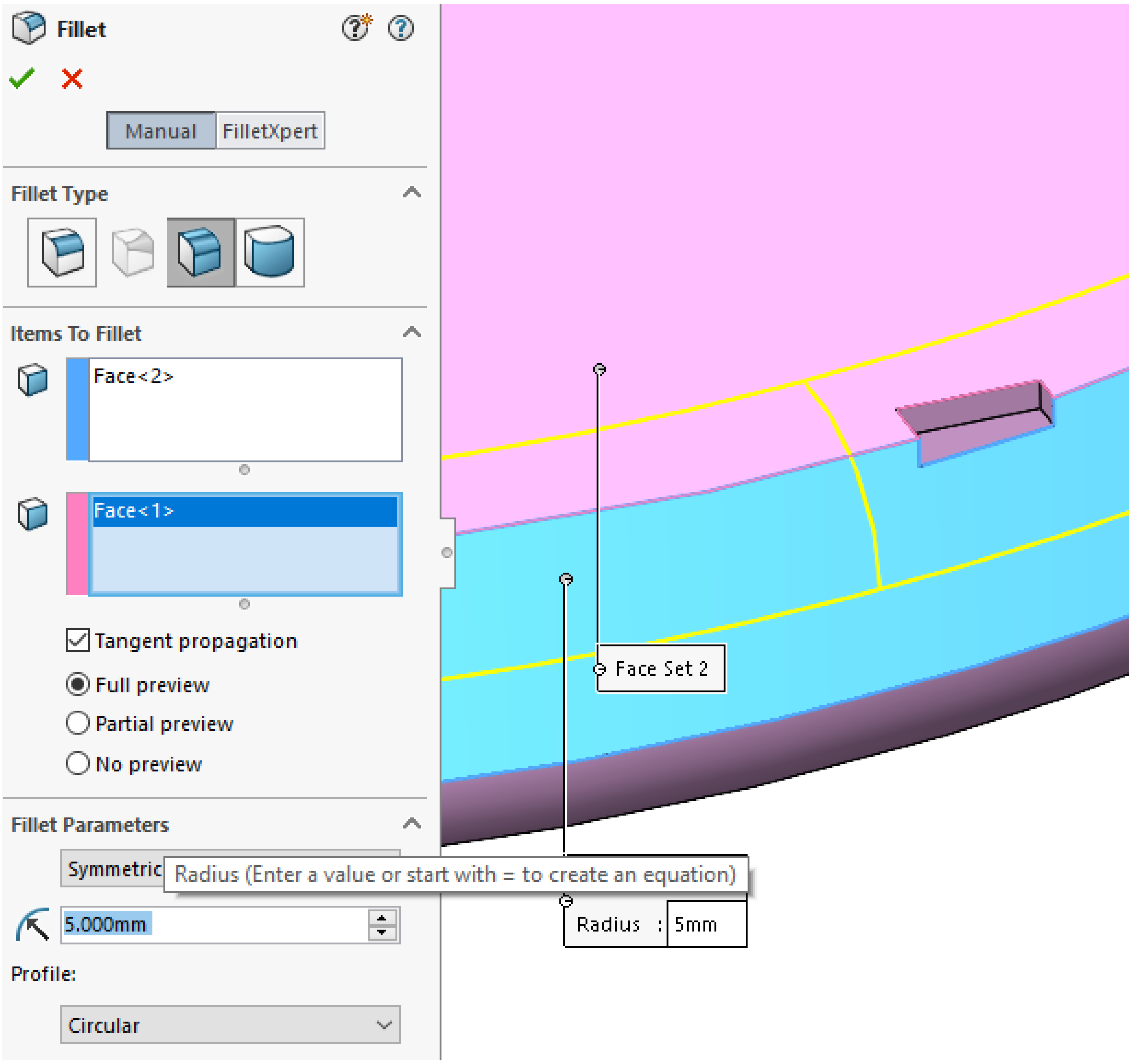
Variable Size Fillets

Variable Size Fillets allow you to create more complex fillets where the radius differs along the way. This example also shows that the Fillets radius can be zero. You can also choose to have Symmetric or Asymmetric fillets, as well as changing the profile shape (the default profile is Circular). The number of points is three by default, but that can change. The selectable points are shown along the edge, along with the size of each radius in the graphics area.

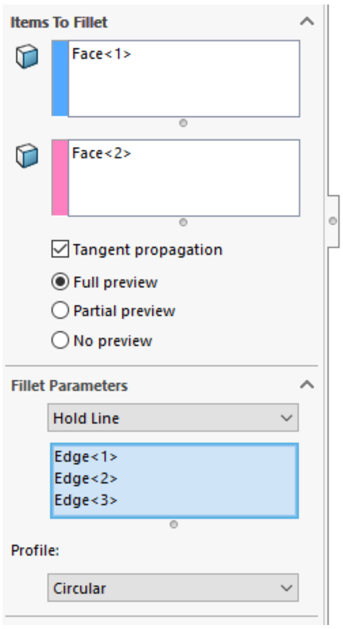


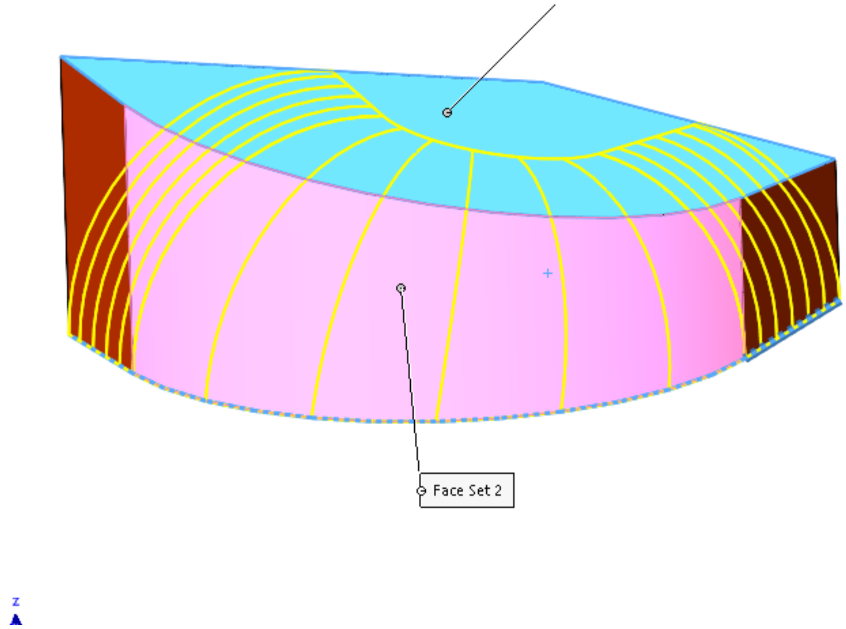
Face Fillets

Face Fillets are created between faces instead of edges. These are usually used when edges have problems, such as when the edges are broken by other features. They also can help remove faces from “dumb solids”.



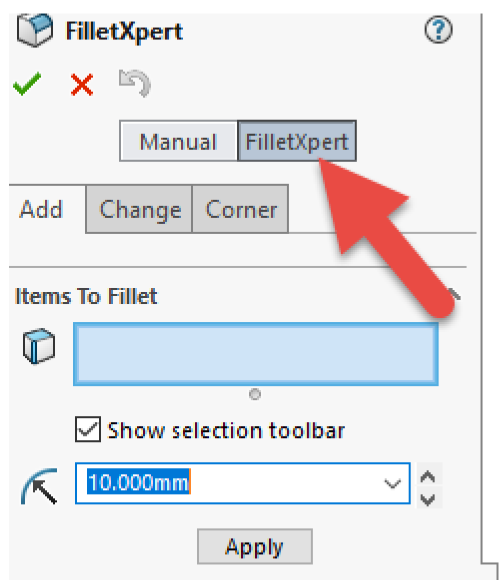
Face Fillets also allow the use of a “Hold Line”. This will create a fillet tangent to the selected faces and the radius will be automatically adjusted so the Fillet ends at the Hold Line edge(s).





FilletXpert

FilletXpert allows you to add multiple sizes of Constant Size Fillets, change sizes, remove existing Fillets, and modify the corners where fillets blend - all in one tool.



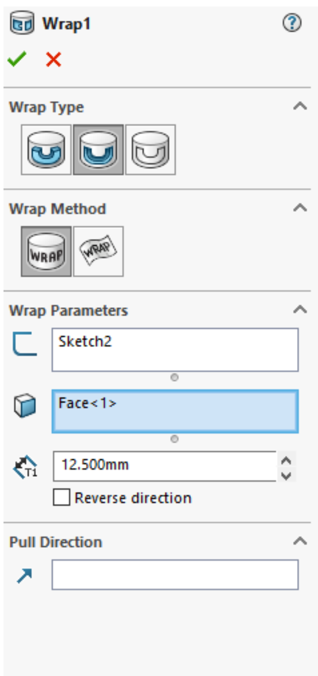
Use the “Add” tab add Fillets by picking an edge, using the Selection Toolbar to quickly select multiple related edges, and click Apply. Adjust the fillet size as needed, select more edges, and Apply again. Use the “Change” tab to modify the radius values of existing Fillets and remove any unneeded Fillets from edges.

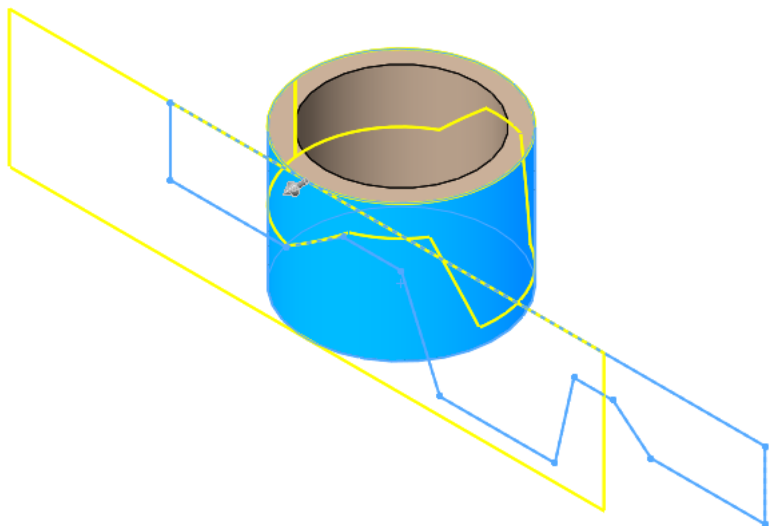
The “Corner” tab allows you to choose between different blend options where three Constant Size Fillets of mixed convexity come together.

Wrap

The Wrap Feature takes a sketch and wraps around a non-planar face. You can choose to add material, remove material, or split the face(s).

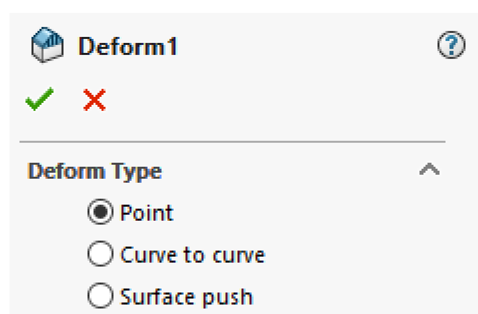
Select the proper Wrap Type, which specifies whether your face is round or a spline surface. Then pick the Sketch and the face to apply it to; set the depth of material to add or subtract and Pull Direction if desired.





Deform

The Deform tool lets you modify an existing body by using a “Point”, “Curve to curve”, or “Surface push”.

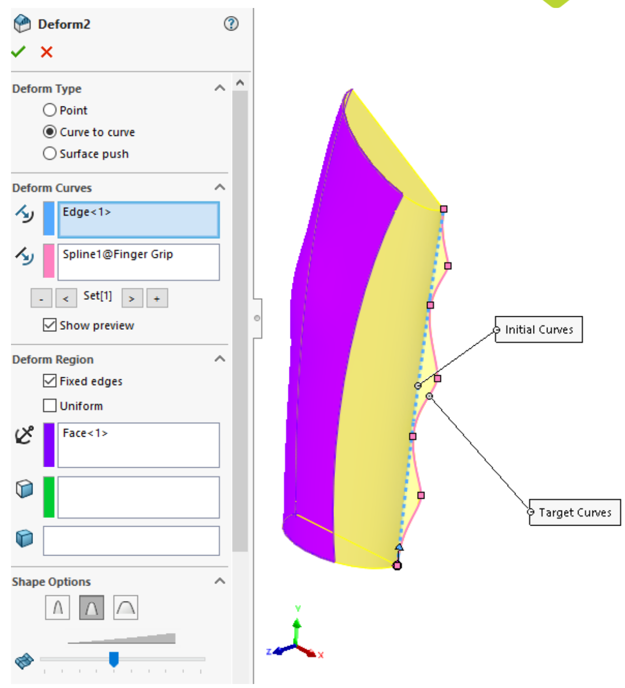


“Point” was used on our model in a 3D sketch. We set the Radius and Height and Deform Distance as well. The “Deform Region” option kept the shape change limited to the face the point was on. “Maintain Boundary” was also enabled to keep the edges of the face as they were.

"Curve to curve" was used to modify the edge of the model as well, to conform to the Sketch as shown.

The face in purple was set to "fixed" to ensure that the face didn't change due to the new deform. Note: This fixed entity is not allowed to be touching the initial curve.

You can also change the Shape Options accuracy slider if the result isn't accurate enough.

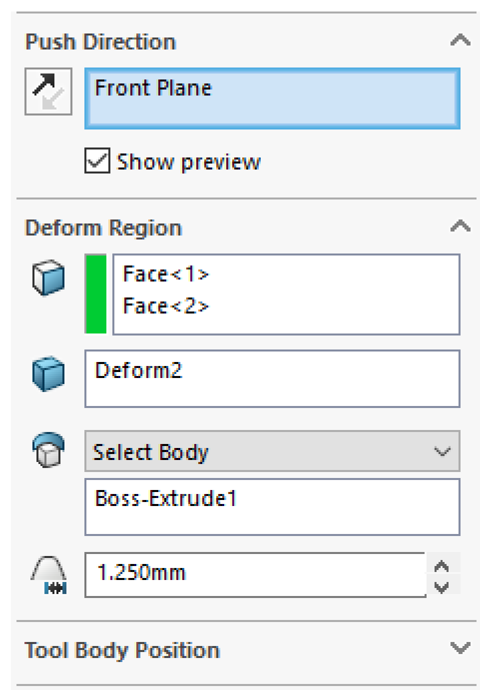


A "Surface Push" was used to create the last deform area.

After making a copy of the two top faces that merge (using the "Knit Surface" Feature), the other Sketch was extruded to the far side of the new surface with an Extruded Boss.

The "Merge result" checkbox was cleared to keep the new body separate.

The faces on the top were set as the Deform Region, ensuring only those faces would be modified. We used the "Select Body" option and picked that body, then set the deform deviation.



Move Face

Move Face can be used to modify files with or without Features. This can allow even a native SOLIDWORKS file to be modified with ease, no matter how many features were used to create the end result.

You can “Offset”, which has different results for planar vs cylindrical faces, “Translate”, or “Rotate” the selected face(s).

Either enter a distance value or drag the face(s) to the desired location.