SOLIDWORKS 2016
Learn by doing-Part 2

Tutorial Books

SOLIDWORKS 2016: Learn by Doing-Part 2

Tutorial Books

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Table of Contents

Chapter 1: Surface Design

- TUTORIAL 1 (Extruded Surfaces)
- TUTORIAL 2 (Revolved Surfaces)
- TUTORIAL 3 (Swept Surfaces)
- TUTORIAL 4 (Lofted Surfaces)
- Tutorial 5 (Planar Surfaces)
- TUTORIAL 6 (Creating a Ruled Surface using the Tangent to Surface option)
- TUTORIAL 7 (Creating a Ruled Surface using the Normal to Surface option)
- TUTORIAL 8 (Creating a Ruled Surface using the Tapered to Vector option)
- TUTORIAL 9 (Creating a Ruled Surface using the Perpendicular to Vector option)
- TUTORIAL 10 (Creating a Ruled Surface using the Sweep option)
- TUTORIAL 11 (Offset Surface)
- TUTORIAL 12 (Knitting Surfaces)
- Creating a Solid by Knitting Surfaces
- TUTORIAL 13 (Trimming Surfaces)
  - Trimming a Surface using the Standard option
  - Trimming Surfaces using the Mutual option
- TUTORIAL 14 (Extending Surfaces)
- TUTORIAL 15 (Untrimming a Surface)
- TUTORIAL 16(Deleting Holes)

- TUTORIAL 17 (Filled Surface)
- TUTORIAL 18
- TUTORIAL 19 (Converting a Surface to Solid)
- TUTORIAL 20 (Thickening the Surface)
- Tutorial 21 (Deleting Faces)
- TUTORIAL 22 (Replacing Faces)
- TUTORIAL 23 (Cutting with Surfaces)
- TUTORIAL 24 (Thickened Cut)
- TUTORIAL 25 (Freeform Surfaces)
- TUTORIAL 26 (Boundary Surfaces)
- TUTORIAL 27 (Flatten Surface)

Chapter 2: Mold Tools

- TUTORIAL 1
  - Performing Draft Analysis
  - Applying Shrinkage allowance
  - Inserting Mold Folders
  - Creating a Parting Line
  - Creating Shut-off Surfaces
  - Creating Parting Surfaces
  - Creating the Tooling Split
  - Performing the Undercut analysis
  - Creating side cores
  - Creating your own surfaces

Chapter 3: Weldments

- TUTORIAL 1
- TUTORIAL 2
  - Adding Structural members
  - Trimming the Structural Members
  - Creating Gussets
  - Creating Base Plates
  - Mirroring Gussets and Base plates

Creating Fillet Beads
Creating Weld Beads
TUTORIAL 3 (Creating End Caps)
TUTORIAL 4 (Working with Cut lists)
Adding Cut list to the Weldment Drawing
Adding Columns to the Cut list table
Creating Bounding box
Adding a Weld table to the Weldment Drawing
Adding a Weld Symbols
Creating Sub Weldments
TUTORIAL 5 (Creating Custom Profiles for structural members)

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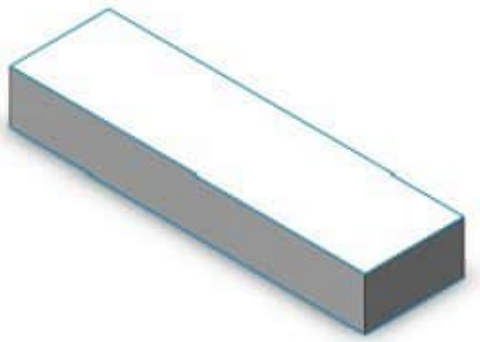
Chapter 1: Surface Design

SOLIDWORKS Surface modeling tools can be used to create complex geometries that are very difficult to create using solid modeling tools. In addition, you can also use these tools to fix broken imported parts. In this chapter, you will learn the basics of surfacing tools that are mostly used. The surfacing tools are available on the **Surfaces** Command Manager. You can also find these tools on the menu bar (click **Insert > Surface** on the Menu bar).

If the **Surfaces** CommandManager is not displayed by default, you can customize it. Right-click on any of the tabs of the CommandManager and select **Surface** from the shortcut menu.

SOLIDWORKS offers a robust set of surface design tools. A surface is an infinitely thin piece of geometry. For example, consider a box shown in Figure. It has six faces. Each of these faces is a surface, an infinitely thin piece of geometry that acts as a boundary in 3D space. Surfaces can be simple or complex shapes.



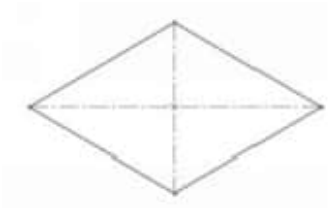
In solid modeling, when you create solid features such as an Extruded Boss or a Revolved Boss, SOLIDWORKS creates a set of surfaces forming a closed volume. This airtight closed volume is considered as a solid body. Although, you can design a geometry using solid modeling tools, but the surface modeling tools give you more flexibility.

Creating Basic Surfaces

In this section, you will learn to create basic surfaces using the **Extruded Surface**, **Revolved Surface**, **Swept Surface**, and **Lofted Surface** tools. These tools are similar to that available in solid modeling.

TUTORIAL 1 (Extruded Surfaces)

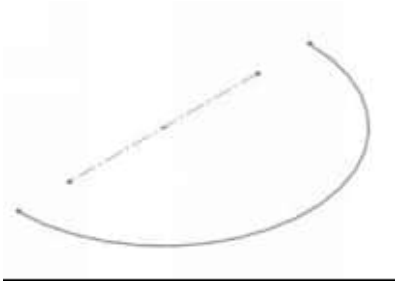
1. To create an extruded surface, first create an open or closed sketch.



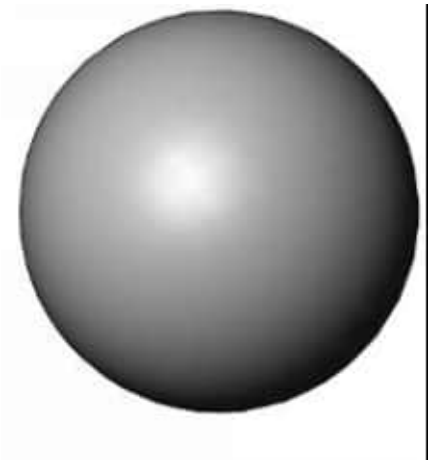
2. Click the **Extruded Surface** button on the **Surfaces** Command Manager.
3. Select the sketch. The **Surface-Extrude** Property Manager appears and is similar to the **Boss-Extruded** Property Manager.
4. Enter a value in the **Depth** box available in the **Direction 1** section.
5. Click **OK** on Property Manager to create the extruded surface. You will notice that the extrusion was not capped at the ends. You can check the **Cap end** option to cap the ends

TUTORIAL 2 (Revolved Surfaces)

1. To create a revolved surface, first create an open or closed profile and the axis of revolution.



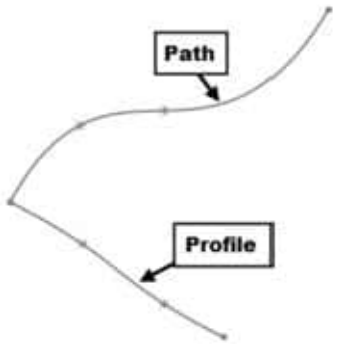
2. Click the **Revolved Surface** button on the **Surfaces** Command Manager.
3. Select the sketch. The preview of the revolved surface appears.
4. Enter the angle of revolution in the **Direction 1 Angle** box and click **OK**.



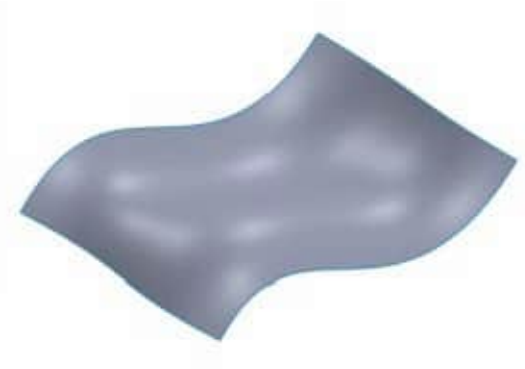
Even if you create an enclosed surface, SOLIDWORKS will not recognize it as a solid body. You will notice that the **Mass Properties** button is not available in the **Evaluate** Command Manager. This means that there exists no solid body. You will learn to convert a surface body into a solid later in this chapter.

TUTORIAL 3 (Swept Surfaces)

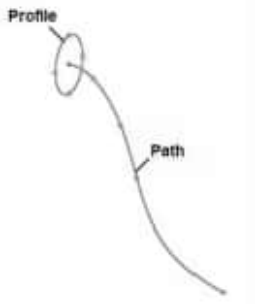
1. To create a swept surface, create a sweep profile and a path.

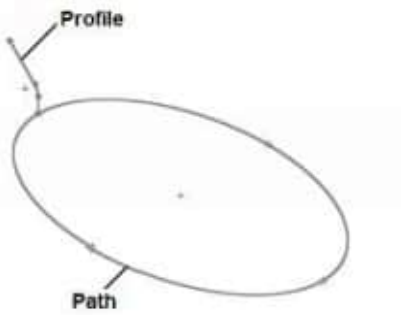


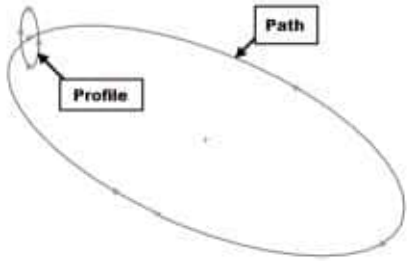
2. Click the **Swept Surface** button on the **Surfaces** Command Manager (or) click **Insert > Surface > Sweep**.
3. Select the sweep profile and then the path from the graphics window.
4. Click **OK** on the Property Manager.

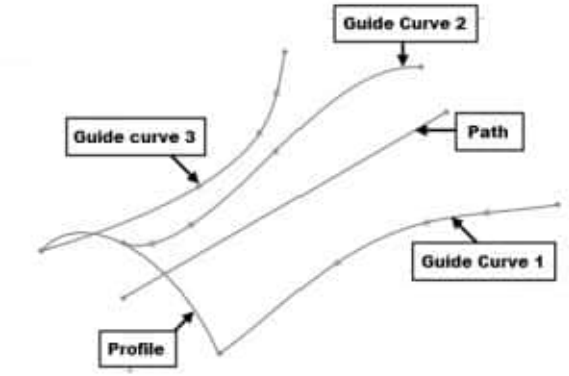


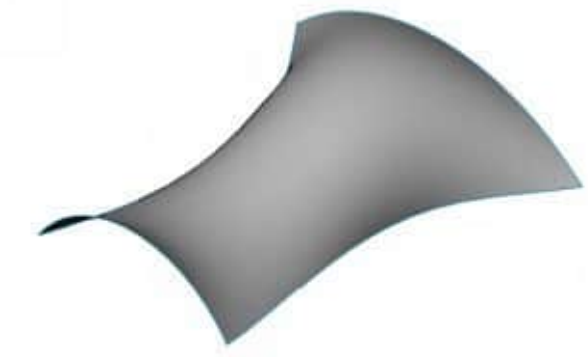
Various ways of creating swept surfaces are given next.




Profile
Path

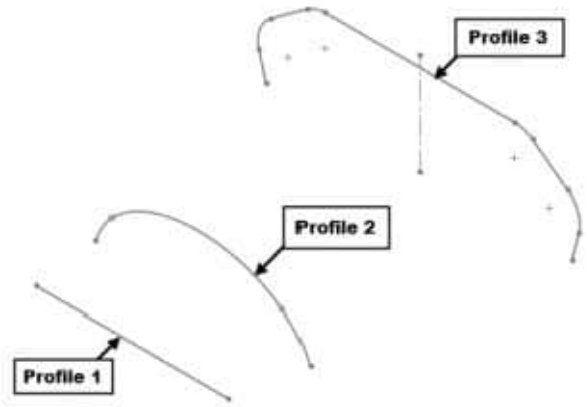

Path
Profile



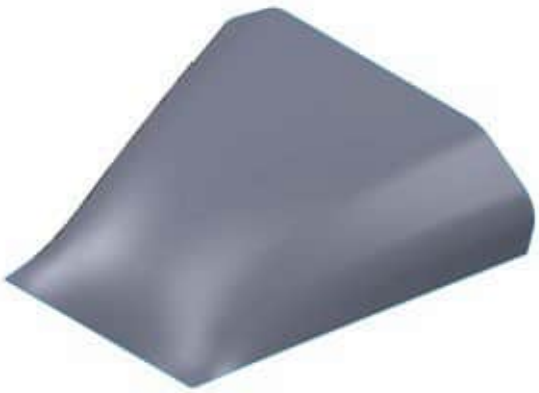


TUTORIAL 4 (Lofted Surfaces)

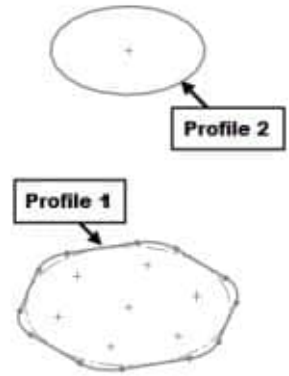
1. To create a swept surface, create two or more profiles.

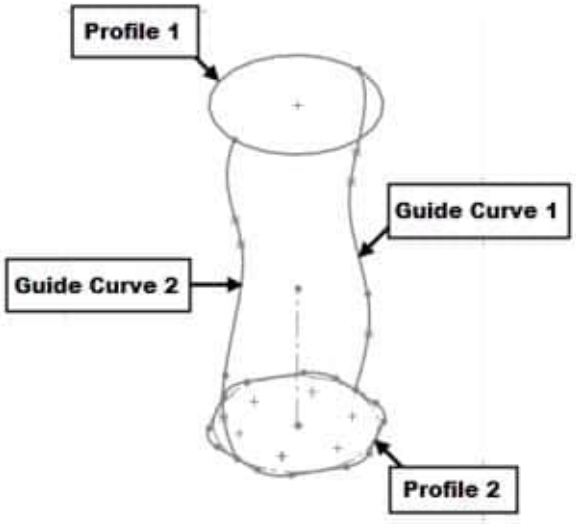


2. Click the **Lofted Surface** button on the **Surfaces** Command Manager (or) click **Insert > Surface > Loft** on the Menu bar.
3. Select the loft profiles from the graphics window.
4. Click **OK** on the Property Manager.



Various ways of creating lofted surfaces are given next.

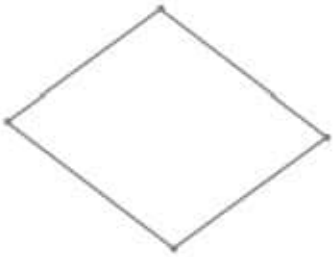


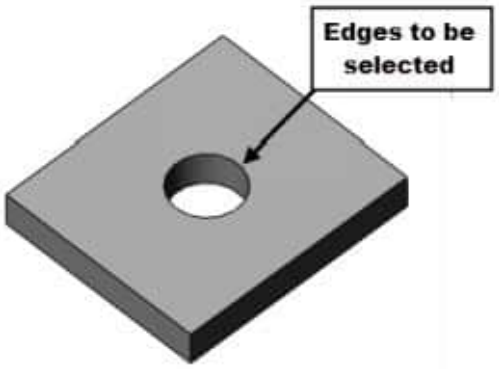


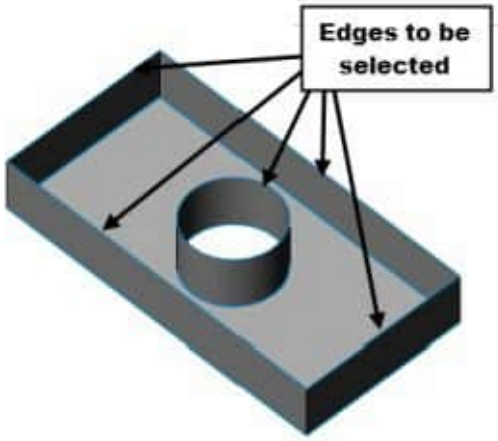
Tutorial 5 (Planar Surfaces)

1. To create a planar surface, click the **Planar Surface** button on the **Surfaces** Command Manager (or) click **Insert > Surface > Planar**.

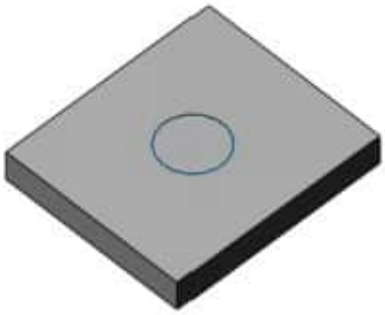
The PropertyManager prompts you to select the bounding entities. You can select a closed sketch or closed loop of edges.

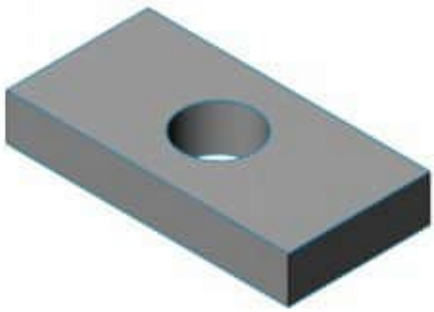






2. Select a closed sketch or a closed loop of edges.
3. Click **OK** on the Property Manager. The planar surface will be created.





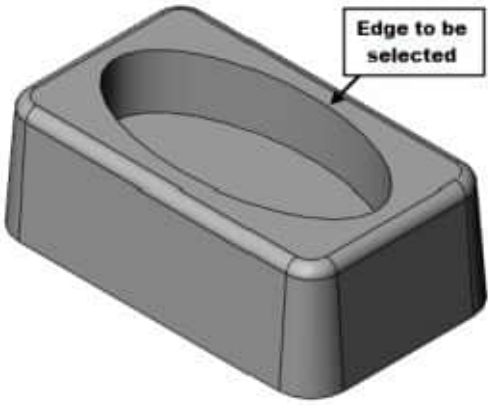
Creating Ruled Surfaces

The **Ruled Surface** tool allows you to create surfaces attached to the edges of an existing surface. You can find the **Ruled Surface** tool on the **Surfaces** Command Manager. You can also activate this tool from the menu bar by clicking **Insert > Surface > Ruled Surface**.

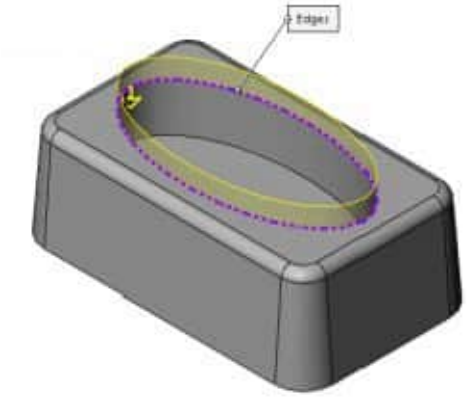
You can create five types of ruled surfaces using the options in the Property Manager. These five types of ruled surfaces are discussed next.

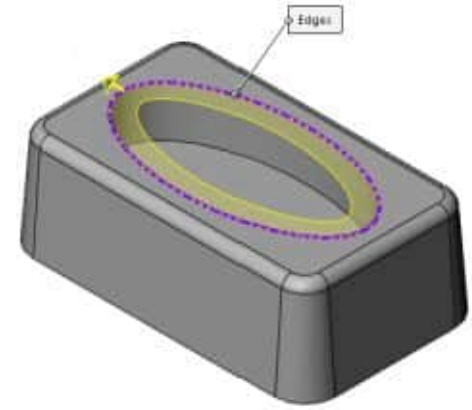
TUTORIAL 6 (Creating a Ruled Surface using the Tangent to Surface option)

1. To create a planar surface, click the **Ruled Surface** button on the **Surfaces** Command Manager (or) click **Insert > Surface > Ruled Surface**.
2. To create this type of ruled surface, select the **Tangent to Surface** option from the **Type** group of the **Ruled Surface** Property Manager.
3. Select an edge from the model.



Notice the preview of the ruled surface. The resultant surface will be tangent to the selected edge. In this case, the selected edge is associated with two reference surfaces (the vertical and the top surfaces). As a result, there will be two solutions created from the selected edge. Click the **Alternate Face** button from the **Edge Selection** group to view the alternate solution.





4. Enter a distance value in the **Distance** box of the **Distance/Direction** group.

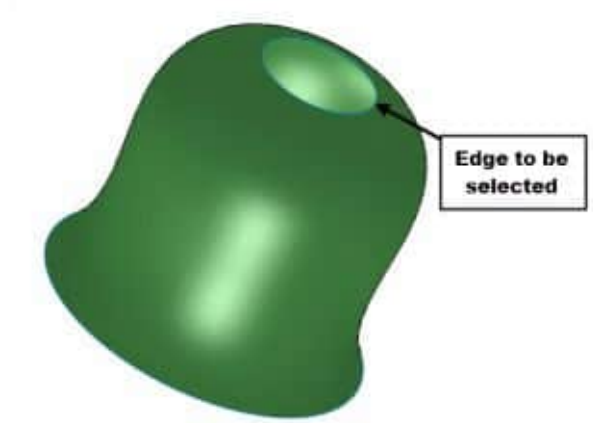
You will notice two options- **Trim and Knit** and **Connecting surface** options available in

the **Options** group. These options are useful while creating multiple ruled surfaces at a time. The **Trim and Knit** option, if enabled trims and knits the continuous ruled surfaces. The **Connecting surface** option creates a connecting surface at the sharp corners of the ruled surfaces.

5. Click **OK** to create the ruled surface.

TUTORIAL 7 (Creating a Ruled Surface using the Normal to Surface option)

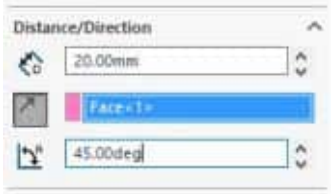
1. To create this type of ruled surface, select the **Normal to Surface** option from the **Type** group of the **Ruled Surface** Property Manager.
2. Select an edge from the model.



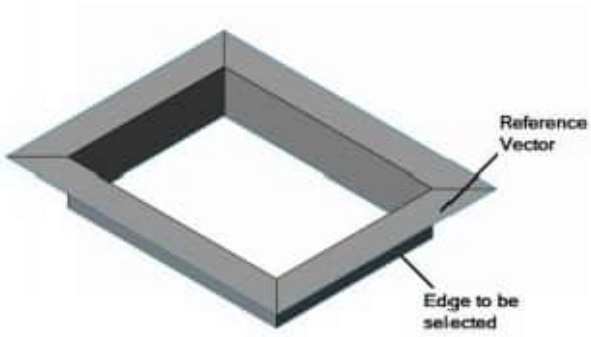
3. Enter a distance value in the **Distance** box of the **Distance/Direction** group.
4. Click the **Reverse Direction** button in the **Distance/Direction** group to reverse the direction, if required.
5. Click **OK** to create the ruled surface normal to the surface on which the selected edge lies.

TUTORIAL 8 (Creating a Ruled Surface using the Tapered to Vector option)

1. To create this type of ruled surface, select the **Tapered to Vector** option from the **Type** group of the **Ruled Surface** Property Manager; some additional options appear in the **Distance/Direction** section.

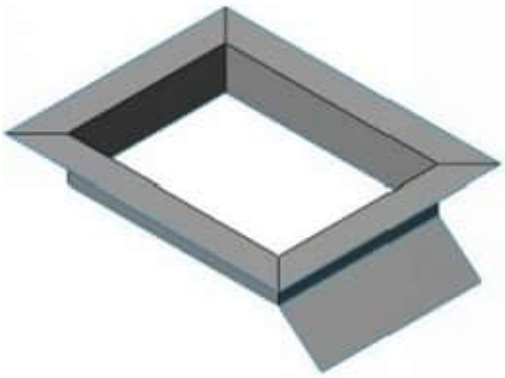


2. Select a face from the model to define the reference vector.
3. Click on the **Reference Vector** box available in the **Distance/Direction** group.
4. Select an edge from the model.



5. Specify the distance of the ruled surface in the **Distance** box.
6. Specify the taper angle of the ruled surface in **Angle** box available in the **Distance/ Direction** group.
7. Click the **Reverse Direction** button, if required.
8. Click **OK**.

The following figure shows the ruled surface created at a taper angle of 60 degrees.

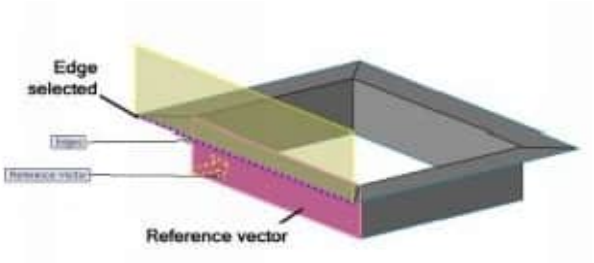


TUTORIAL 9 (Creating a Ruled Surface using the Perpendicular to Vector option)

1. To create this type of ruled surface, select the **Perpendicular to Vector** option from the

Type group of the **Ruled Surface** Property Manager.

2. Select an edge from the model.

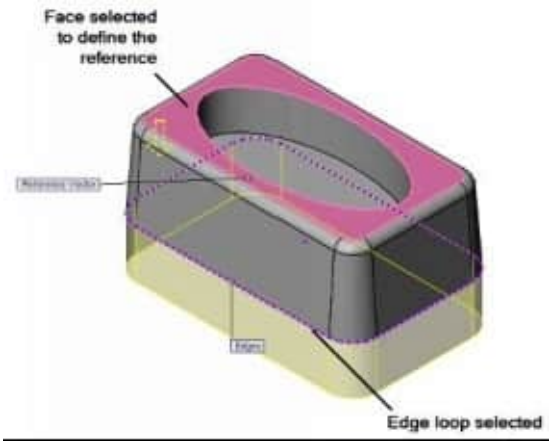


3. Under the **Distance/Direction** section, click in the **Reference Vector** box.
4. Select a face or an edge to define the reference vector.
5. Specify the distance of the ruled surface in the **Distance** box.
6. Click the **Reverse Direction** button, if required.
7. Click **OK**.

TUTORIAL 10 (Creating a Ruled Surface using the Sweep option)

This option creates a ruled surface by sweeping the selected edge along a reference vector.

1. To create this type of ruled surface, select the **Sweep** option from the **Type** group of the **Ruled Surface** Property Manager.
2. Select an edge or loop of edges from the model.



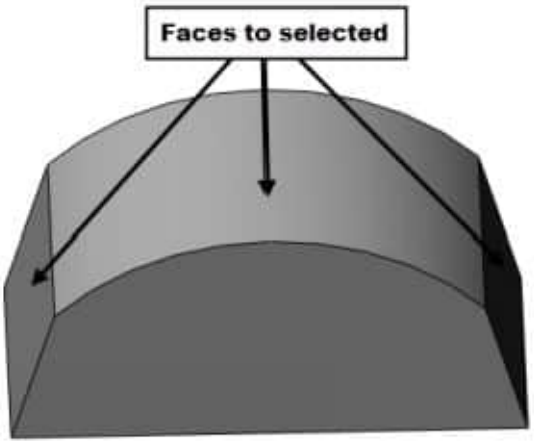
3. Specify the distance of the ruled surface in the **Distance** edit box.
4. Select a face or an edge to define the reference vector.

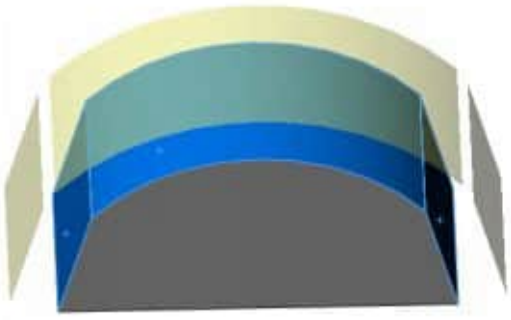
You can also use the **Coordinate input** option to define the reference vector.

5. Click the **Reverse Direction** button, if required.
6. Click **OK**.

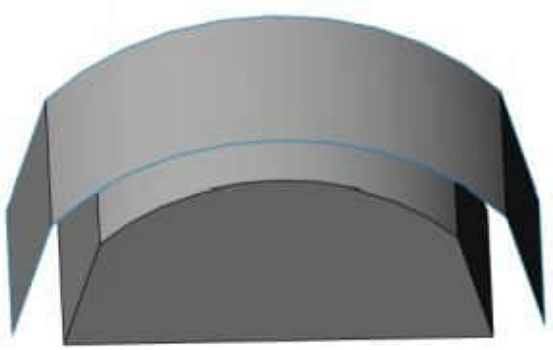
TUTORIAL 11 (Offset Surface)

1. To create a planar surface, click the **Offset Surface** button on the **Surfaces** Command Manager (or) click **Insert > Surface > Offset**.
2. Select the faces to offset.



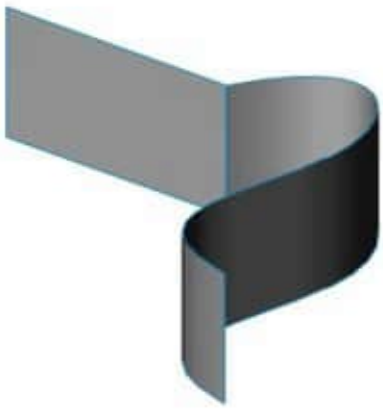


3. Specify the offset distance in the **Offset Distance** edit box.
4. Click **OK** on the Property Manager. The offset surface is created.



TUTORIAL 12 (Knitting Surfaces)

The surfaces which you create act as individual surfaces unless they are knitted together. The **Knit Surface** tool lets you to combine two or more surfaces to form a single surface.



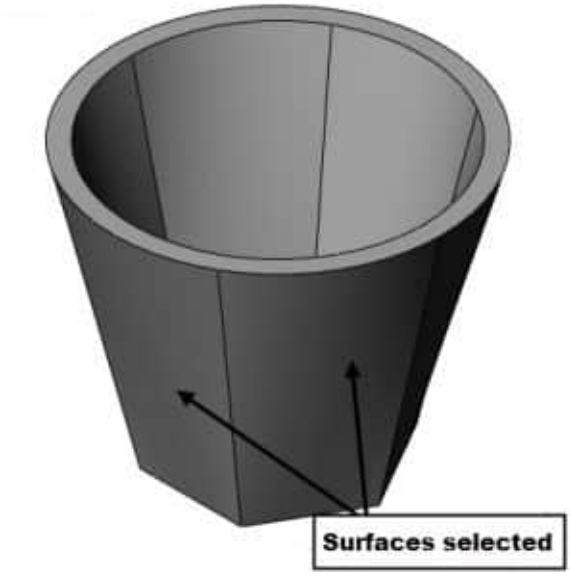
1. To knit surfaces, click the **Knit Surface** button on the **Surfaces** CommandManager (or) click **Insert > Surface > Knit**.
2. Select the surfaces to knit.

The **Merge entities** option available in the **Selections** section is used to merge the overlapping or redundant surfaces.

The options in the **Gap Control** group are used to knit the surfaces with small gaps. The **Knitting tolerance** box is used to define a tolerance value for the gaps. The gaps within the specified tolerance value will be knitted. You can also specify the gap range by using the sliders in the **Gap Control** group.

3. After specifying the required options, click the **OK** button to knit the surfaces.

In some cases, you may need to extract the surfaces of the solid body. You can use the **Knit Surface** tool to extract the surfaces of the solid body. Hide the solid body to see the extracted surfaces.



Creating a Solid by Knitting Surfaces

You can convert surfaces forming a closed enclosure into solid using the **Knit Surface** tool. Create surfaces forming a closed enclosure. Activate the **Knit Surface** tool and select the surfaces. On the PropertyManager, check the **Create Solid** option, and then click **OK** .

TUTORIAL 13 (Trimming Surfaces)

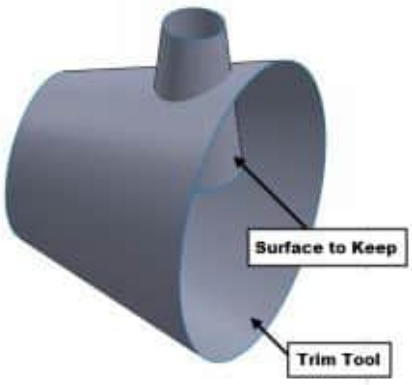
You can trim surfaces using the **Trim Surface** tool. The surfaces are trimmed using a trimming tool. The trimming tool can be a surface, plane or a sketched entity.

1. To trim a surface, click the **Trim Surface** button on the **Surfaces** Command Manager (or) click **Insert > Surface > Trim**; the **Trim Surface** Property Manager appears.

You can trim a surface using the **Standard** and **Mutual** options available in the **Trim Type** group. The two trimming methods are discussed next.

Trimming a Surface using the Standard option

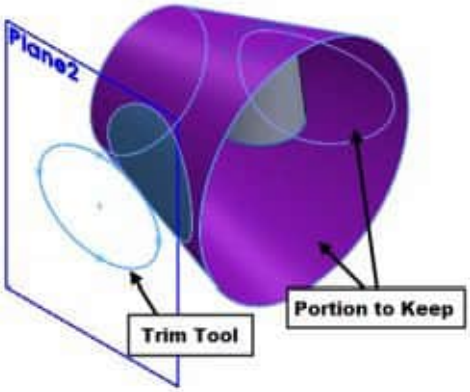
2. To trim a surface using this option, click the **Standard** option in the **Trim Types** group.
3. Select **Keep selections** on the PropertyManager.
4. Select the trim tool and surface to keep.

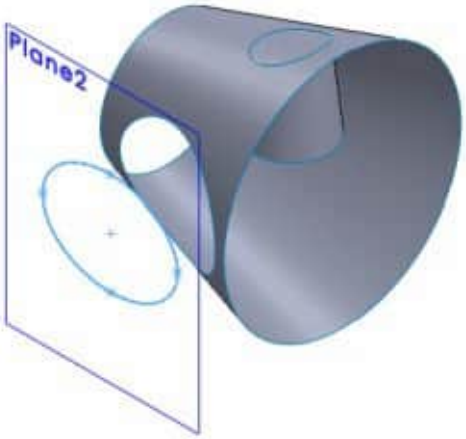


If you select the **Remove selections** option in the **Selections** group, you need to select the surface to remove.

5. Click **OK** to trim the surface.

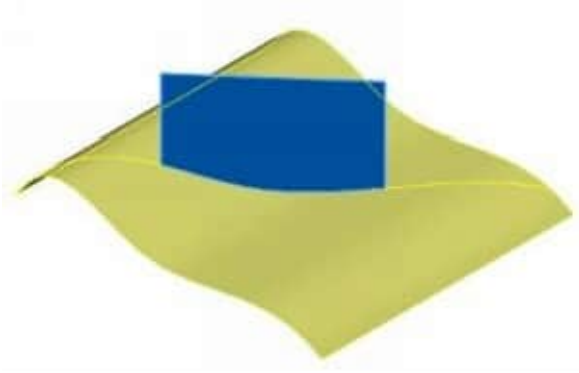
You can also trim a surface using a sketch.



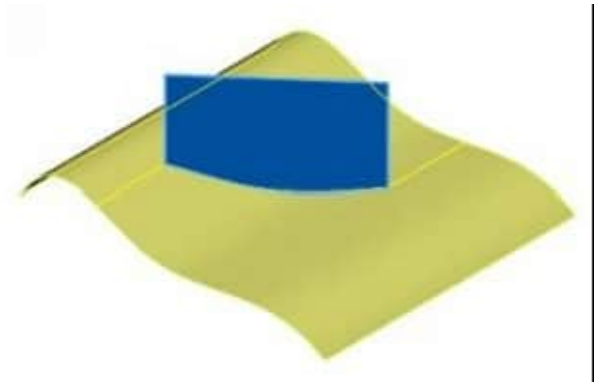


You can also use a surface as a trim tool that does not extend completely along the surface to be trimmed.

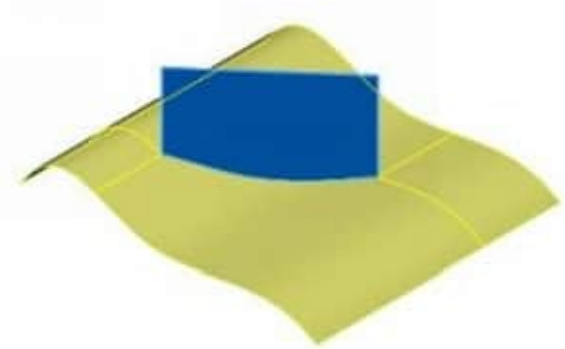
In this case, the **Surface Split Options** will be useful. If you use the **Natural** option, the split will occur following the natural extension of the trim tool.



If you select the **Linear** option, split will be a straight line to the nearest edge.



If you select the **Split All** option, multiple split lines will appear. You can decide which portion of the surface to keep or remove.



Trimming Surfaces using the Mutual option

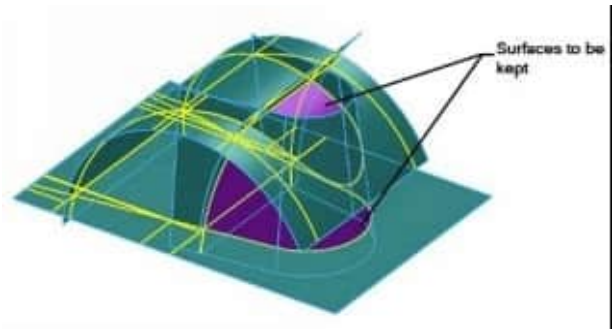
The **Mutual** option is used to trim multiple surfaces at a time. For example, you can use this option for the following case.

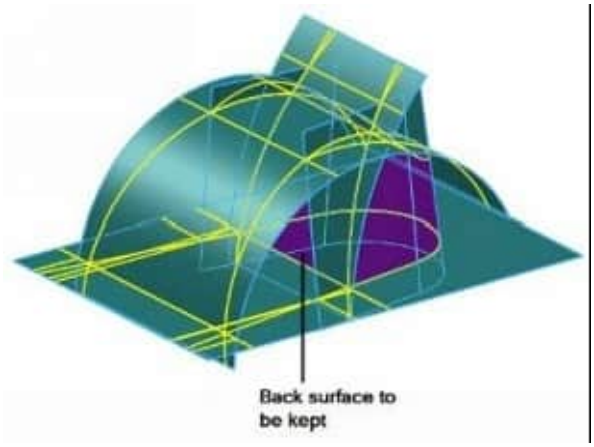
1. To trim surfaces using this option, select the **Mutual** option from the **Trim Type** group.

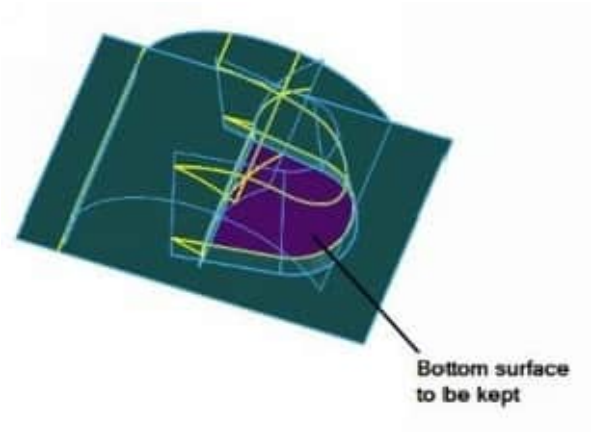
2. Select the required surfaces from the model.
3. After selecting the required surfaces, select the **Keep selections** or **Remove selections** option.

If you select the **Keep selections** option, you need to select the surfaces to be kept. If you select the **Remove selections** option, you need to select the surfaces to be removed. In this case, you select the **Keep selection** option.

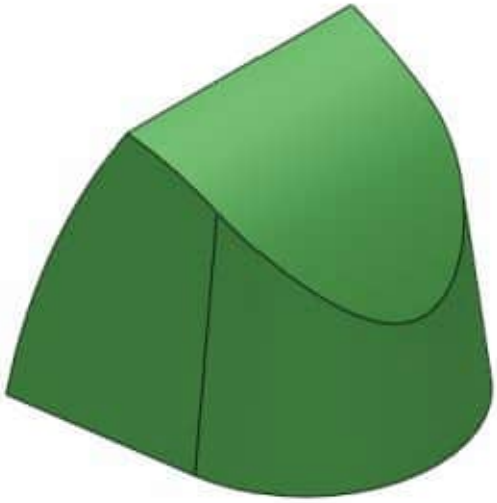
4. Click in the next selection box of the PropertyManager.
5. Select the surfaces to be kept.







6. Click OK to trim the surfaces.



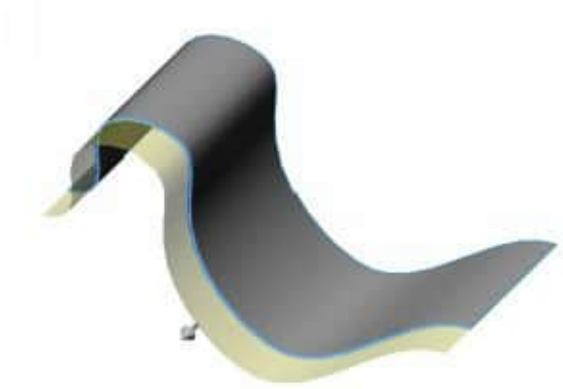
TUTORIAL 14 (Extending Surfaces)

During the design process, you may sometimes need to extend a surface. You can extend a surface using the **Extend Surface** tool.

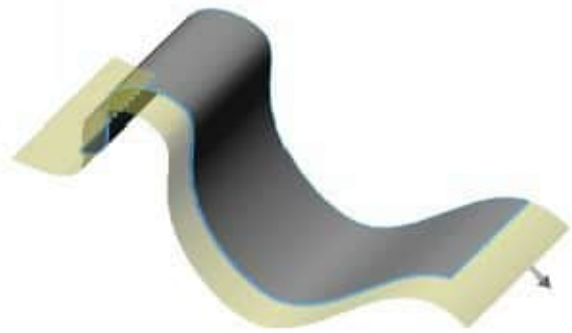
1. To invoke this tool, click the **Extend Surface** button on the **Surfaces** Command Manager (or) click **Insert > Surface > Extend**.

The parameters used to define an extended surface are available on this PropertyManager. You can select an edge, multiple edges or an entire surface to extend.

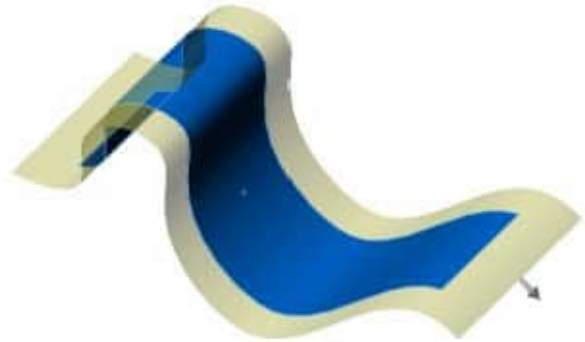
Single Edge



Multiple Edges



Entire Surface



2. Select an edge from the surface body.
3. Select the **Distance** option and type a value in the **Distance** box. If you select the **Up to point** option, you can define the distance by selecting a vertex or point. The **Up to surface** option is used to define the distance by selecting a surface.

When the surface you have selected is not planar, you can decide the type of extension by using the **Extension Type** options. Use the **Same Surface** option to extend the surface by maintaining the curvature of the original surface. If you select the **Linear** option, the extended surface will be created tangent to the original surface.

Same Surface Extension

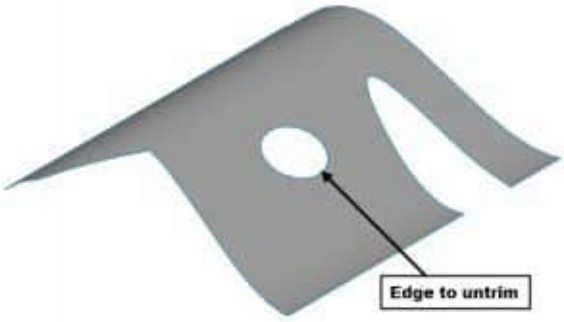
Linear Extension

4. Click **OK**.

TUTORIAL 15 (Untrimming a Surface)

You can untrim a trimmed surface using the **Untrim Surface** tool.

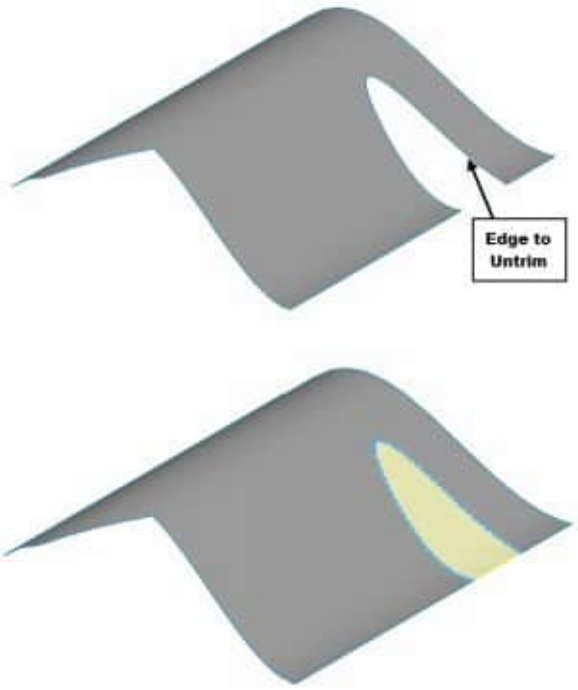
1. Click the **Untrim Surface** button on the **Surfaces** Command Manager (or) click **Insert > Surface > Untrim**.
2. Select the edges to be untrimmed.



3. Click **OK**.

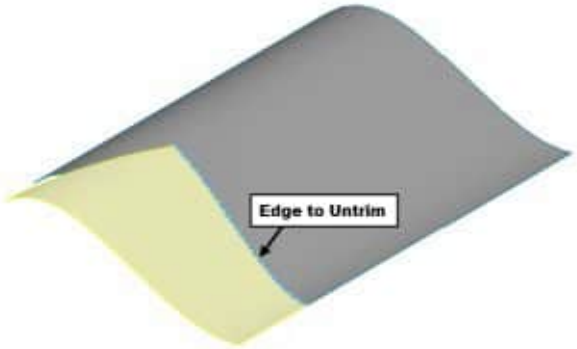
If you select the edge, which is intersecting the boundary of the surface, the untrim surface is created within the boundary. Make sure that the **Edge untrim type** is set to **Extend**

edges.

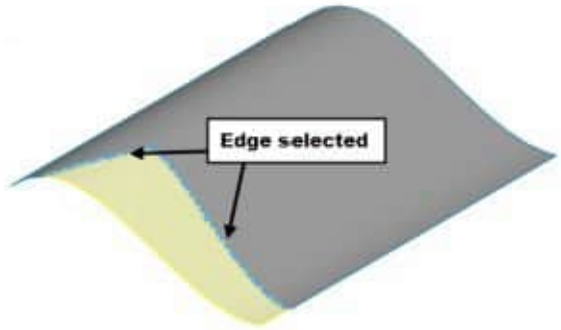


If you check the **Merge to original** option available in the **Options** group, the untrim surface will be merged with the original surface. If you uncheck this option, the untrim surface will be created as an individual surface.

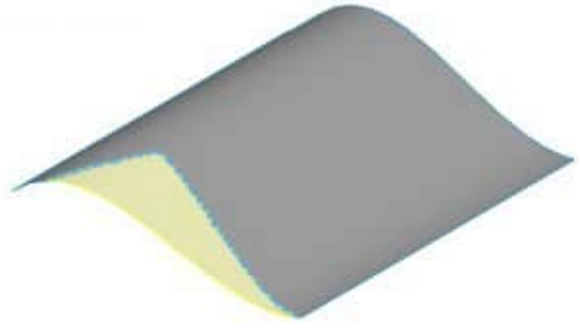
When you untrim the external edges, you have control over the surface created. You can extend the untrim surface beyond the boundary of the surface by entering the extension percentage on the PropertyManager.



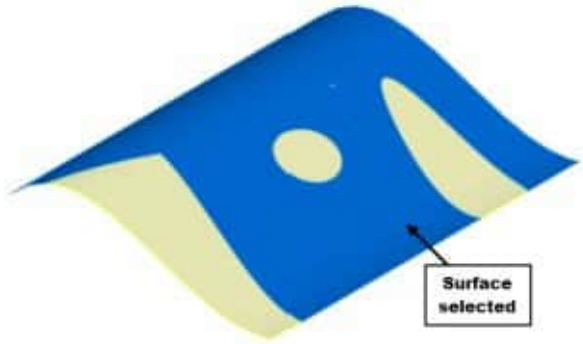
If you select the external edges adjacent to each other, the untrim surface will be bound by the selected edges.



If you select the **Connect endpoints** option available in the **Options** group, only the portion necessary to connect the two endpoints is created.

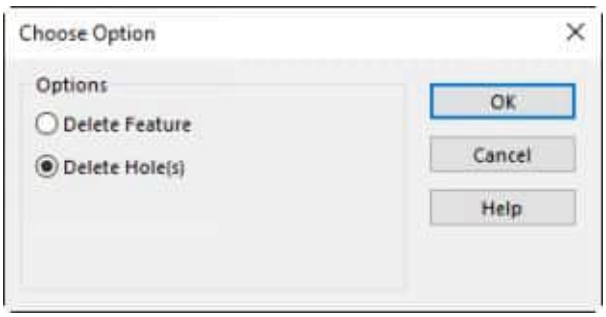


You can use the **Untrim Surface** tool to untrim an entire surface. In addition, the PropertyManager has options to untrim the internal, external, or all the edges of the surface.

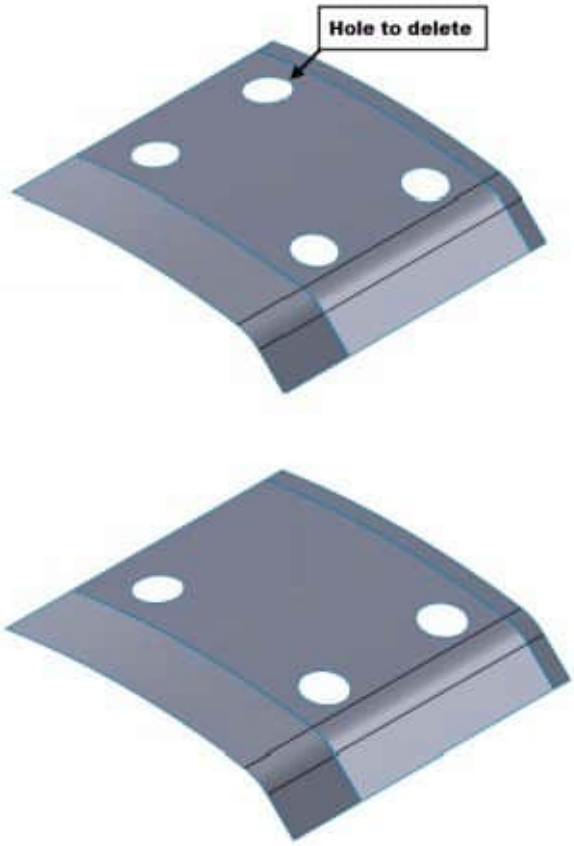


TUTORIAL 16(Deleting Holes)

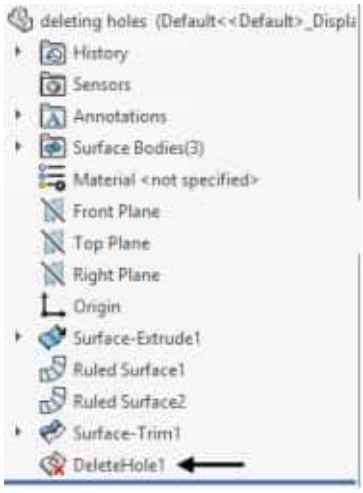
1. To delete a hole, select it from the model and press the **Delete** key. The **Choose Option** dialog appears.



2. Select the **Delete Hole(s)** option and click **OK**.



You will notice the **Delete Hole** feature in the FeatureManager Design Tree.

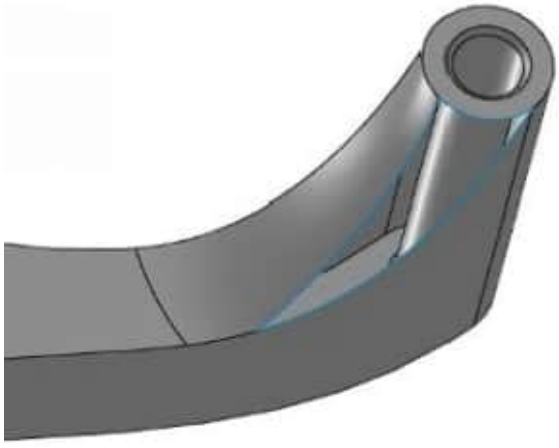


If you want the hole to be back, you can delete the **Delete Hole** feature from the FeatureManager Design Tree. You can also edit the **Delete Hole** feature. Click on the **Delete Hole** feature and click **Edit Feature**.

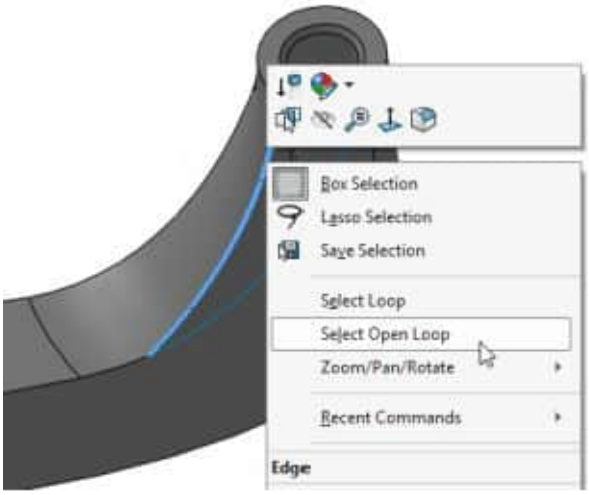
The **Delete Hole** dialog shows the list of holes that were deleted. You can add more holes to the list by selecting them from the model. You can also remove the hole from the list. Make sure that there is at least one hole in the list.

TUTORIAL 17 (Filled Surface)

The **Filled Surface** tool can be used either to patch holes in models or to create complex surfaces. While patching surfaces, the **Filled Surface** tool gives you more options than deleting holes or untrimming. It provides more control over the definition of the resultant patch. For example, consider the model shown in figure. You can see that a face is missing. In a case like this, both the **Delete Hole** and **Untrim Surface** tools fail to fill this gap. The **Filled Surface** tool will be used in this case.

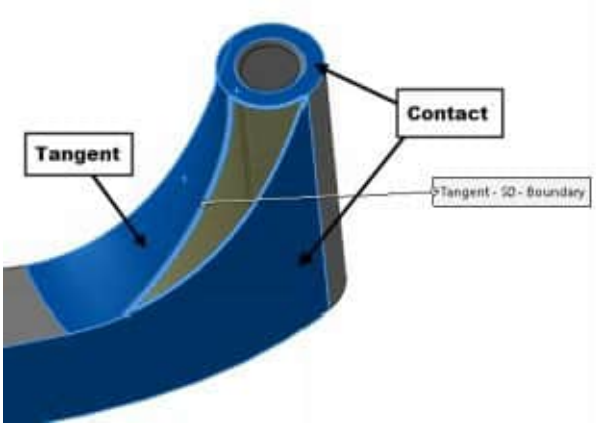


1. To create a filled surface, click the **Filled Surface** button on the **Surfaces** Command Manager (or) click **Insert > Surface > Fill** on the menu bar.
2. To select the patch boundaries, right-click on one of the gap edges and select the **Select Open Loop** option from the shortcut menu. All the gap edges are selected.



3. Select an option from the **Curvature Control** drop-down. This defines the curvature continuity of the fill surface.

The options in this drop-down are **Contact**, **Tangent**, and **Curvature**. Most of the gap edges should be tangent to the surrounding faces.



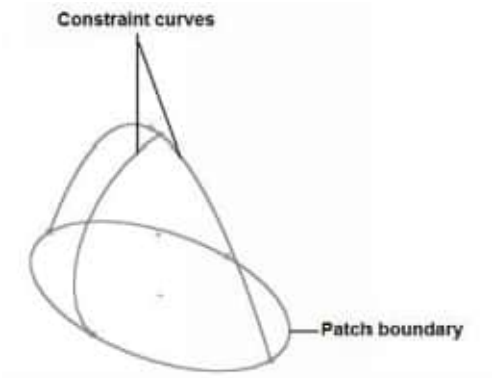
The **Apply to all edges** option is used to apply the curvature setting to all edges.

4. Click **OK** to create the filled surface.

TUTORIAL 18

You can also use the **Filled Surface** tool for creating a new surface.

1. Activate the **Filled Surface** tool and select the patch boundary.
2. Click in the **Constraint Curves** selection box and then select the constraint curves.



3. Specify the curvature setting (**Contact**, **Tangent**, or **Curvature**).

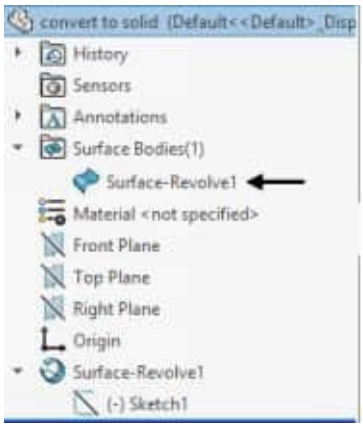
The **Optimize Surface** option available in the **Patch Boundary** group will create a simple surface very quickly. If you uncheck this option, the **Resolution Control** slider will appear. You can use this slider to define the resulting surface. You can remove wrinkles or other imperfections from the surface. However, the surface will be created very slowly. It

is recommended that you use the **Optimize Surface** option for quick results.

1. After specifying the required settings, click **OK** to create the filled surface.

TUTORIAL 19 (Converting a Surface to Solid)

Creating a solid from a surface can be accomplished by simply thickening a surface. You can also create a solid if a volume is enclosed by the surface. For example, consider the surface shown in figure. It encloses a volume. However, there is only one surface body listed in the FeatureManager Design Tree.

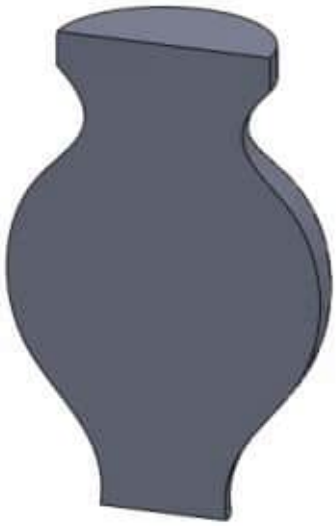


1. To make this surface into a solid, click the **Thicken** button on the **Surfaces** Command Manager.
2. Select the enclosed surface.

You will notice that the **Create solid from the enclosed volume** option is enabled. This option is available only when you select an enclosed surface.

3. Check the **Create solid from the enclosed volume** option.
4. Click **OK** to create a solid.

You can use the **Mass Properties** tool to check whether the surface is converted into a solid or not. You can also use the **Section View** tool.



TUTORIAL 20 (Thickening the Surface)

The **Thicken** tool can also be used to add a thickness to the surface. If the geometry contains more than one surface, than make sure that they are knit together using the **Knit Surface** tool.

1. Activate the **Thicken** command and select the surface to thicken.
2. Type-in the thickness value on the PropertyManager.
3. Define the thickness side using the **Thickness** buttons. Select the **Thicken Side1** button to add thickness outside the surface. The **Thicken Side2** button is used to add thickness inside the surface. The **Thicken Both Sides** button is used to add thickness to both sides of the surface.
4. Click **OK**.

Thickness Side1

Thickness Both Sides

Thickness Sides2

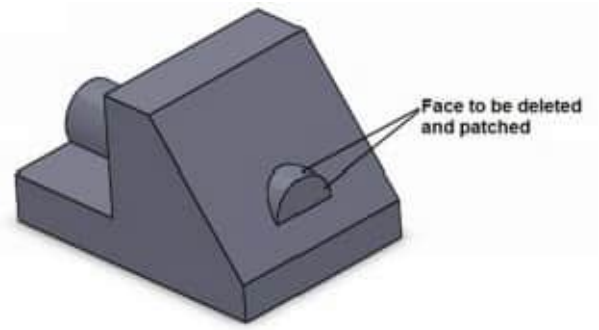
Tutorial 21 (Deleting Faces)

You can delete unwanted faces from a model. In case of a solid, you can delete a face and make it a surface model.

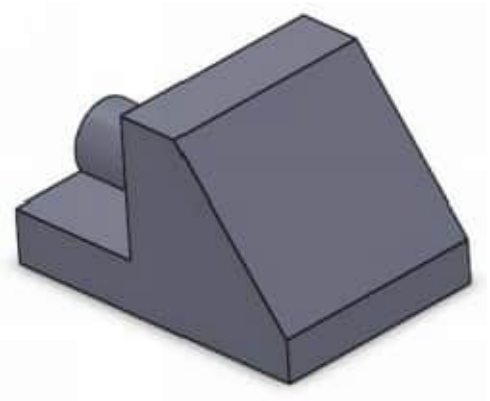
1. To delete a face, click the **Delete Face** button from the **Surfaces** Command Manager.
2. Make sure that the **Delete** option is selected in the **Options** group.
3. Select the face to be deleted.

4. Click **OK** to delete the face.

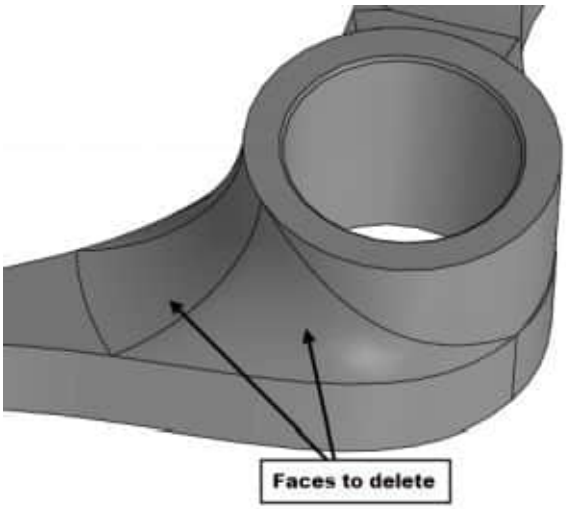
You can also use the **Delete and Patch** option available in the **Options** group. This option will be useful for the following type of cases.



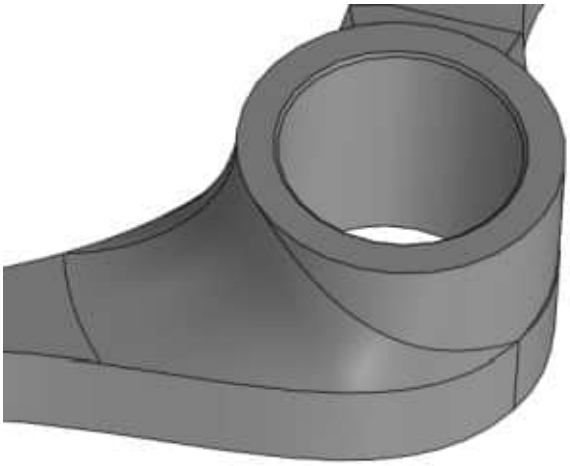
1. Select this option from the **Options** group and then select the faces to be deleted.
2. Click **OK**.



The **Delete and Fill** option is used for the following type of cases.



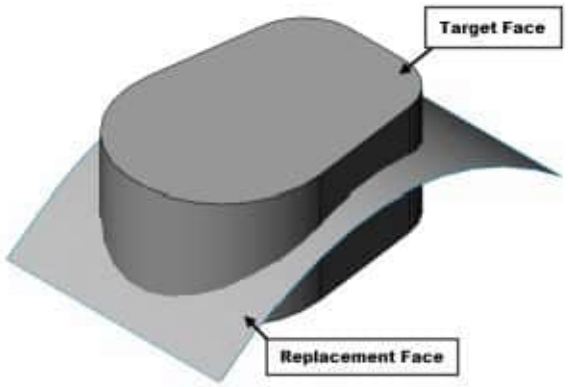
Resulting face is shown next.



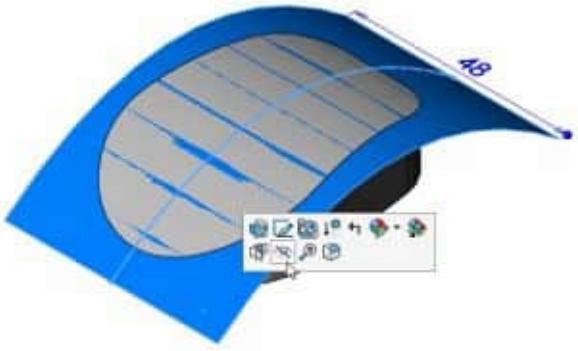
TUTORIAL 22 (Replacing Faces)

The **Replace Face** tool replaces a face or group of faces with another face or group of faces.

1. To replace a face, click the **Replace Face** button on the Command Manager (or) click **Insert > Face > Replace** on the Menu bar.
2. Select the target faces for replacement.
3. Click in the **Replacement Surface** selection box and select the replacement surface.

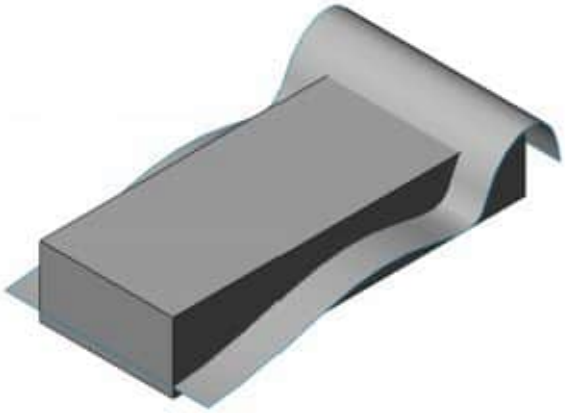


4. Click **OK** to replace the surface.

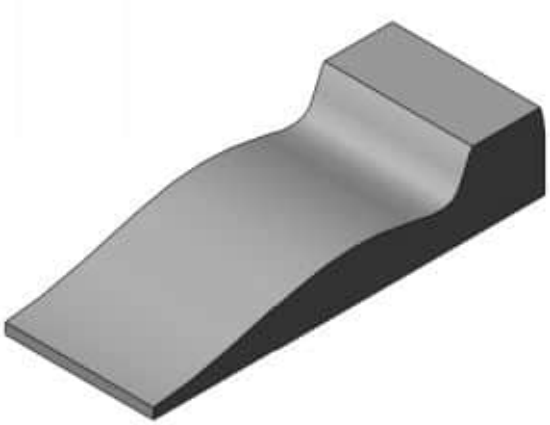


TUTORIAL 23 (Cutting with Surfaces)

You can cut a solid with a surface using the **Cut with Surface** tool. For example, you can cut the solid shown in figure using a surface, and create a complex face on the top.



1. To cut a solid using a surface, click the **Cut with Surface** button on the **Surfaces** Command Manager (or) click **Insert > Cut > With Surface** from the Menu bar.
2. Select the cutting surface.
3. Specify the direction of the cut using the arrow that appears on the surface.
4. Click **OK**.

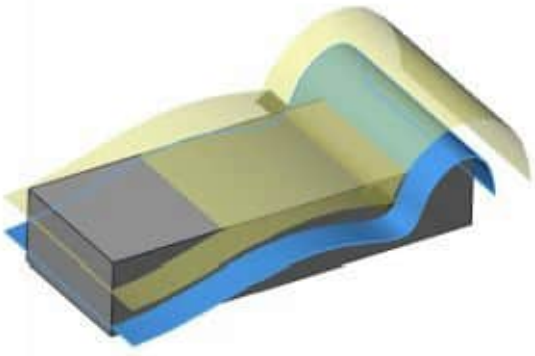


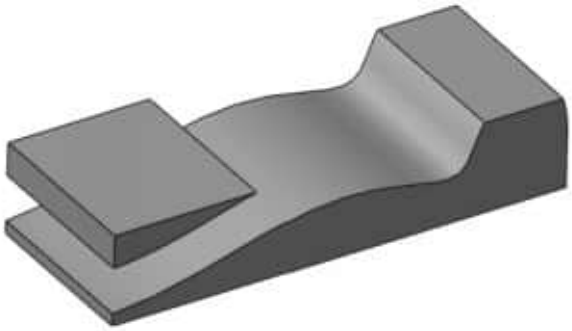
The **Bodies to Keep** dialog appears if the cut results in more than one body. Select the **All bodies** option (or) the **Selected Bodies** option and select the bodies to keep from the list. Click **OK** to cut the solid.

TUTORIAL 24 (Thickened Cut)

You can also use the **Thickened Cut** tool to cut a solid with the surface.

1. Click the **Thickened Cut** button on the **Surfaces** CommandManager (or) click **Insert > Cut > Thicken**.
2. Select the cutting surface.
3. Specify the thickness side.
4. Specify the thickness value.
5. Click **OK**.

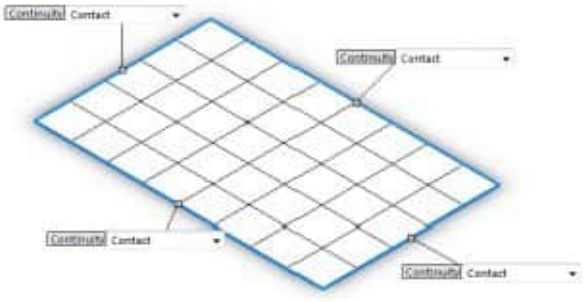




TUTORIAL 25 (Freeform Surfaces)

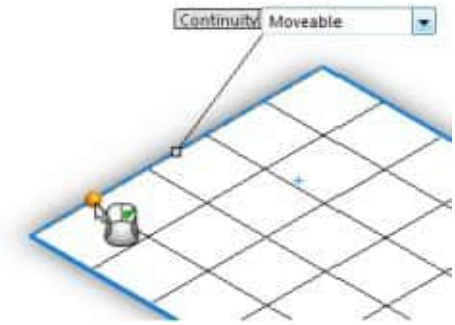
The **Freeform** tool is a powerful tool that is used to create ergonomic shapes. This tool allows you to push and pull a surface to create complex shapes easily. A freeform shape can be created using a face of the solid body or surface.

1. To create a freeform surface, click the **Freeform** button on the **Surfaces** Command Manager (or) click **Insert > Surface > Freeform** on the Menu bar.
2. Select a face to modify. A mesh appears on the selected surface.

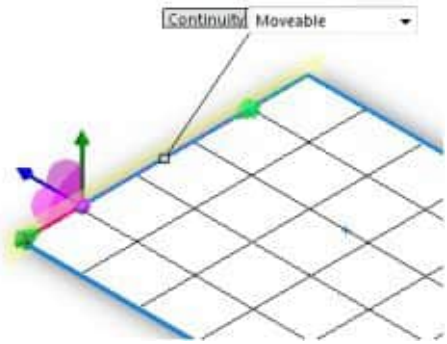


In addition, **Continuity** callouts are attached to the surface boundaries. These callouts are used to control the boundaries of the face. The options in the callouts are **Contact**, **Tangent**, **Curvature**, **Moveable**, and **Moveable/Tangent**.

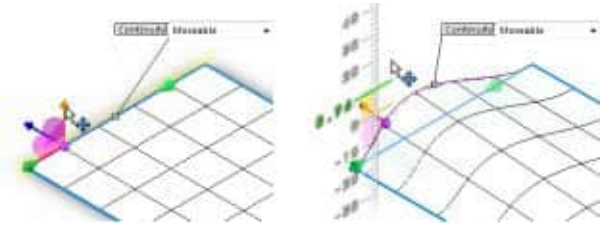
3. Select **Moveable** from the **Continuity** drop-down. The boundary acts as a control curve.
4. On the PropertyManager, click the **Add Points** button in the **Control Points** group and add control points to the boundary.



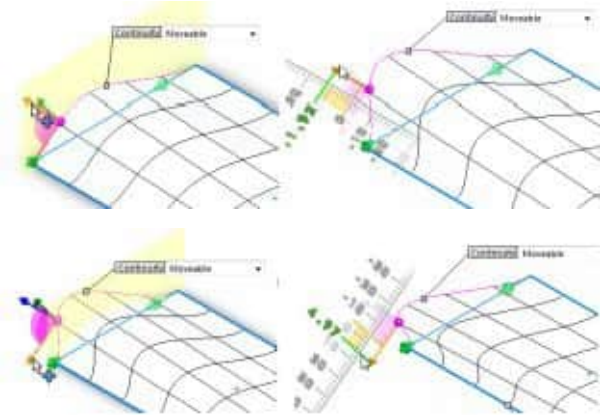
5. Click the **Add Points** button again or press ESC.
6. Select a control point on the boundary. A reference triad appears, as shown.



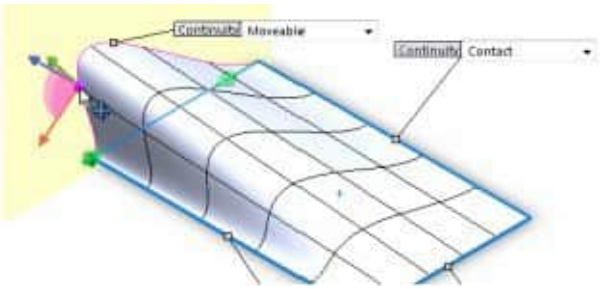
7. Click and drag the Y-axis of the triad and notice the surface. You can use the scale to move the control point precisely.



8. Likewise, drag the Z and X axes to modify the surface.

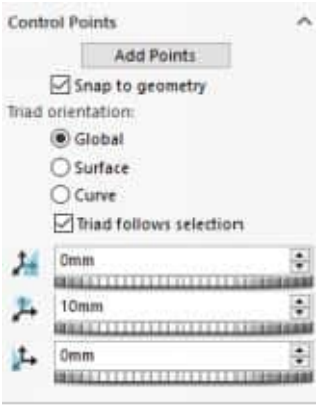


9. Click and drag the origin of the triad.



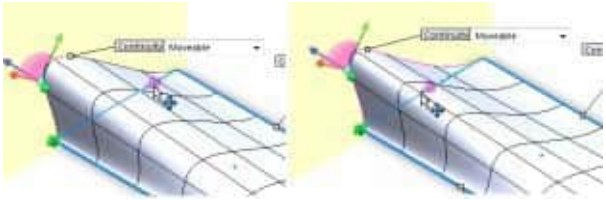
Click **Undo** on the PropertyManager, if you want to return to the previous state of the surface.

The behavior of the triad can also be controlled using the options in the Property Manager.



You can change the orientation of the triad using the **Global**, **Surface**, and **Curve** options. The **Triad follows selection** option will move the triad along the control point.

0. Select the control curve and notice the tangency control arrows.
1. Drag the tangency control arrows to modify the curvature.

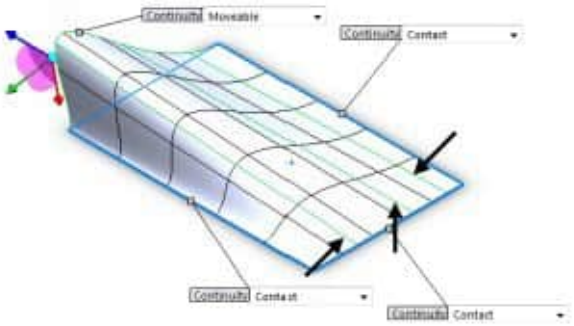


After modifying the surface, if you have by mistake changed the option in the **Continuity** callout, the modifications will be lost.

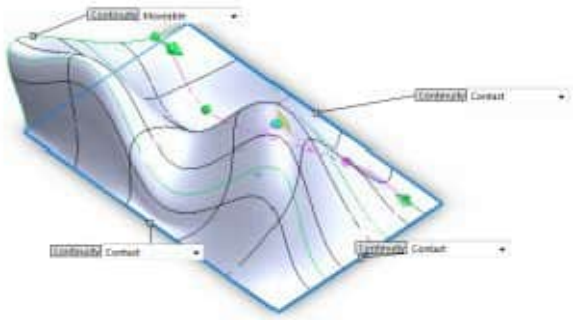
2. Click the **Add Curves** button in the **Control Curves** group to add control curves on the surface.

You can create two types of control curves- **Through points** and **Control polygon**. The **Through Points** curve is formed joining the various points. The **Control polygon** is similar to that used while creating a spline.

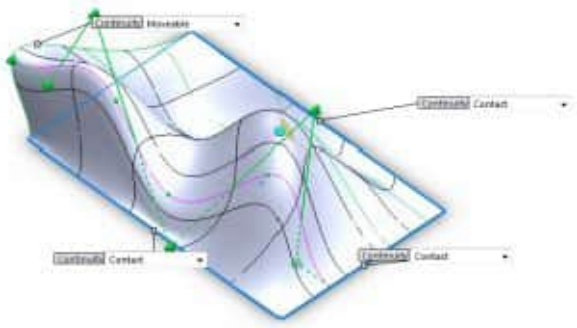
3. Select the required curve type (**Through points** or **Control polygon**).
4. Move the pointer on the surface. You will notice that a green curve appears on the surface. If you click the **Flip Direction** button in the Property Manager (or) press Tab on your keyboard, the curve will appear in the other direction. You can also switch the direction by pressing the TAB key.
5. Move the pointer and click to specify the location of the control curves. You can create as many curves as possible by clicking at the required locations.



6. Click the **Add Curves** button again or press ESC.
7. Add control points on the control curves by using the **Add Points** button.
8. Select the required control curve; the control points appear on it. Push and pull the control points to modify the surface.

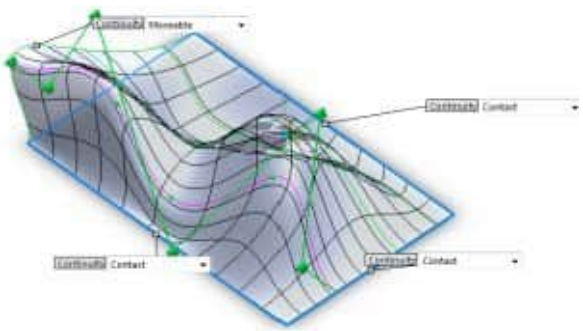


While modifying the surface if you want to change the control curve type (for this example **Though points**), you can select the required option from the **Control Curves** section. The control curve type will be changed.

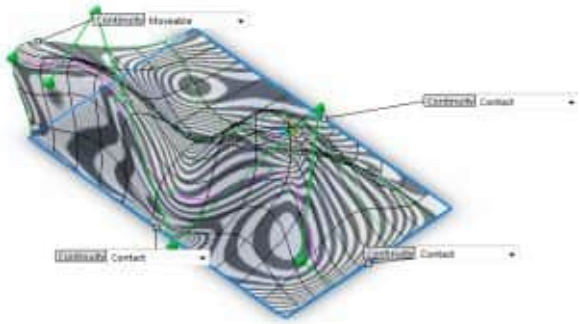


The **Display** section in the Property Manager can be used to modify the display.

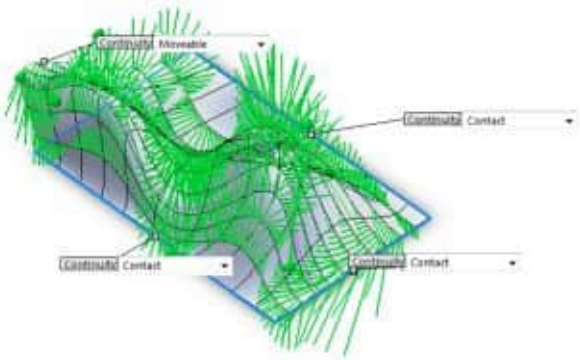
9. Drag the **Face transparency** slider to change the transparency of the surface.
0. Drag the **Mesh density** slider to refine the mesh.



1. Display zebra stripes by checking the **Zebra stripes** option.

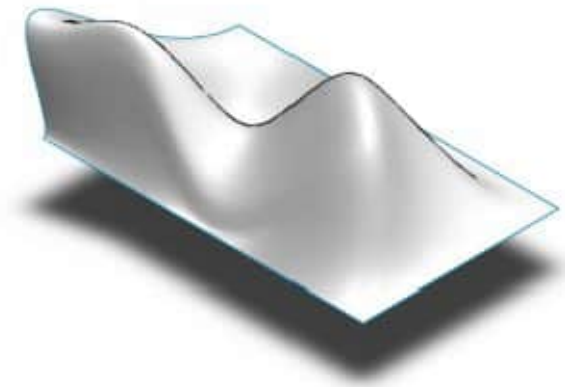


2. Uncheck the **Zebra stripes** option.
3. Check the **Curvature combs** and **Direction 1** options. The curvature combs appear in the first direction.



You can set the curvature type, scale and density of the curvature combs

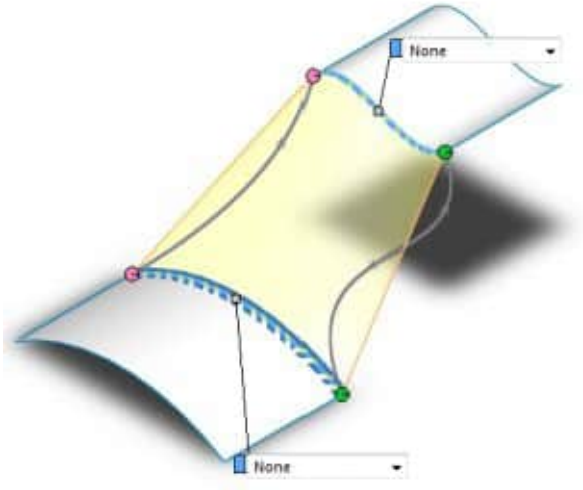
4. Click **OK** on the Property Manager.



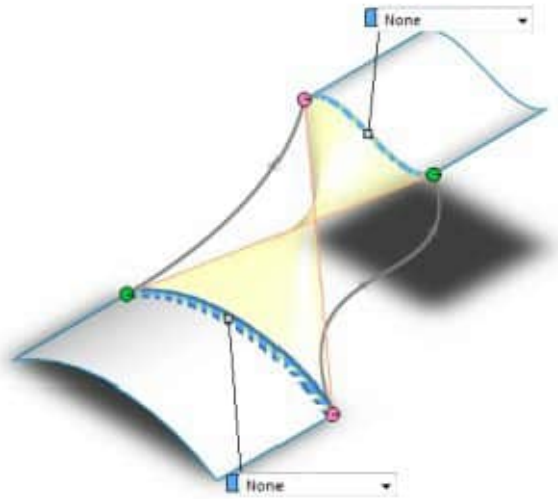
TUTORIAL 26 (Boundary Surfaces)

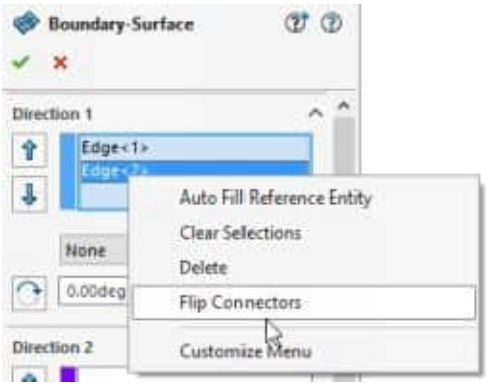
The **Boundary Surface** tool is very similar to the **Lofted Surface** tool but allows you to create surfaces that can be tangent, or curvature, continuous in both the directions.

1. To create a boundary surface, click the **Boundary Surface** button on the **Surfaces** Command Manager (or) click **Insert > Surface > Boundary Surface** on the Menu bar.
2. Select the curves to define the **Direction 1** of the boundary surface.



To avoid the twist, make sure that you click on the same side of the two curves. If a twist is created, right-click on anyone of the edges in the **Direction 1** selection box and choose the **Flip Connectors** option.

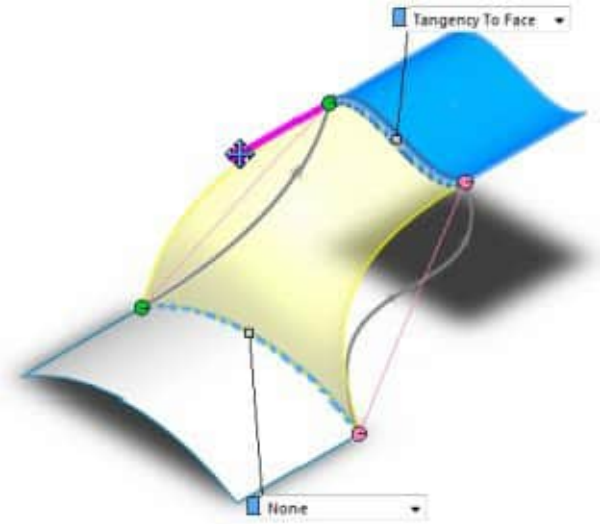




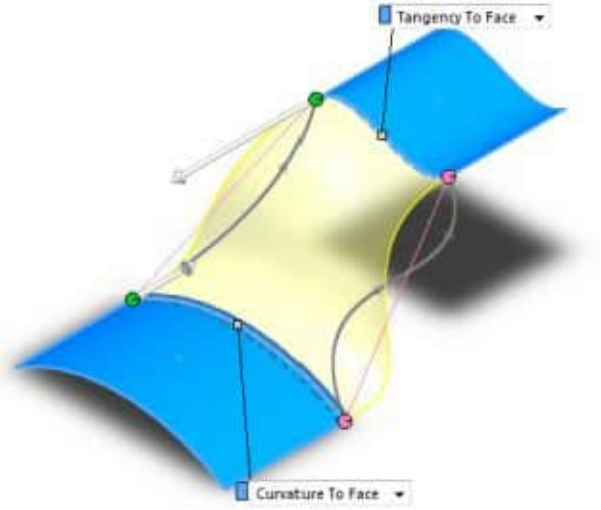
You can specify the way the boundary surface will be connected to the selected edges. To do so, select the required edge from the **Direction 1** selection box. Next, select the **Tangent type** option.

3. Select the **Edge 1** from the **Direction 1** selection box.
4. Select **Tangency To Face** option from the **Tangent type** drop-down (for this example).
5. Specify the tangent length in the **Tangent Length** box or by dragging the arrow

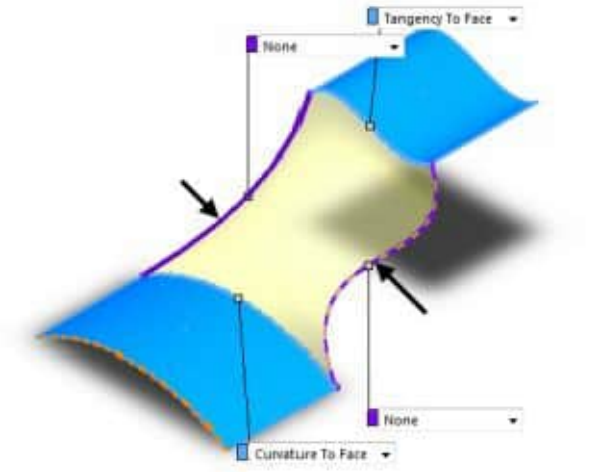
displayed on the curve.



5. Select the **Edge 2** from the **Direction 2** selection box.
7. Select **Curvature To Face** option from the **Tangent type** drop-down (for this example).



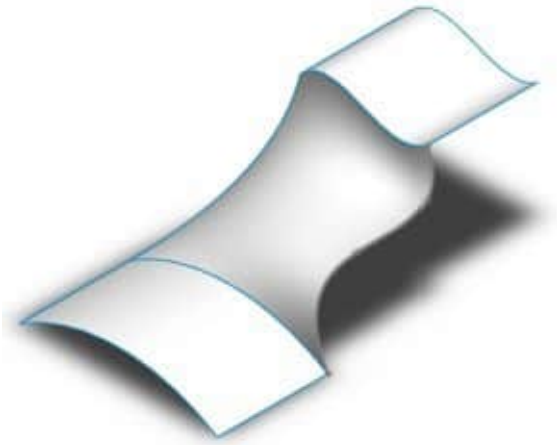
8. Click in the **Direction 2** selection box.
9. Select the guide curves.



You will notice that the **Tangent type** drop-down is replaced by the **Tangent influence** option in the **Direction 1** group. This is just another way to specify the tangent value.

In addition, the **Curve Influence Type** drop-down appears under both the **Direction 1** and **Direction 2** groups. Select the required option from this drop-down.

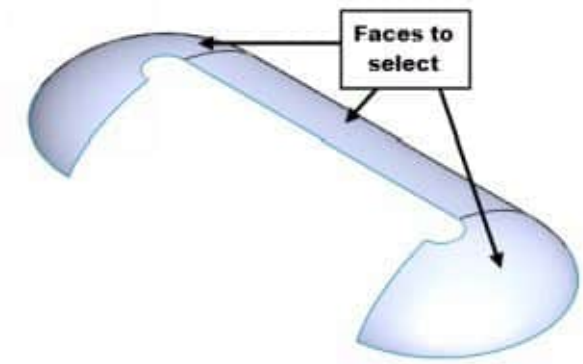
0. Click **OK** to create the boundary surface.



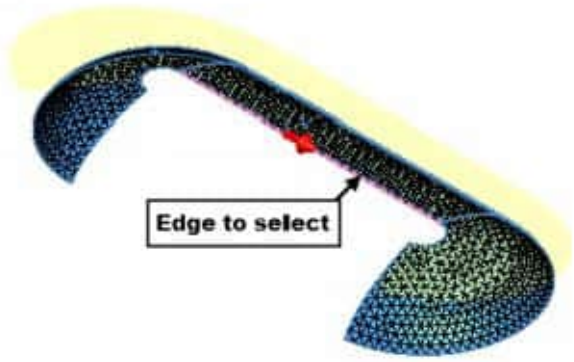
TUTORIAL 27 (Flatten Surface)

The **Flatten Surface** tool is flattens the surface model.

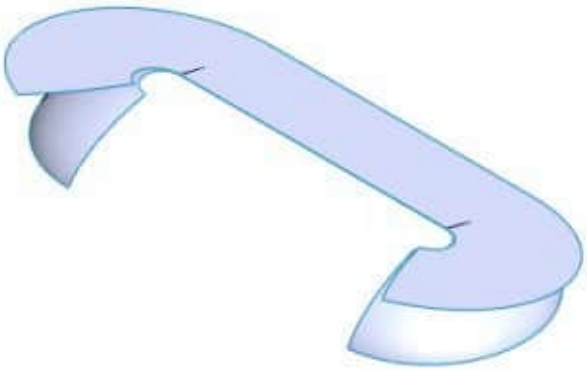
1. On the **Surface** CommandManager, click the **Flatten Surface** tool.
2. Select the faces from the surface model, as shown.



3. Click in the reference selection box.
4. Select the horizontal edge.



5. Click **OK** .



6. Right click on the flatten surface and select **Export to DXF / DWG**.
7. Select **Dxf** from the **Save as type**.
8. Type a name in the **File name** box and click **Save**.

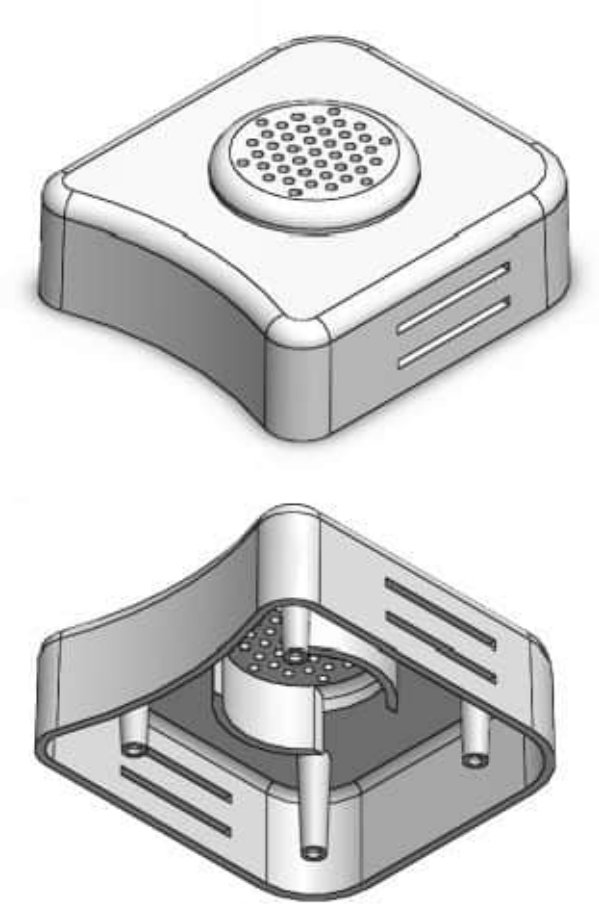
Chapter 2: Mold Tools

In this chapter, you will learn the basics of mold tools in SOLIDWORKS. The process of a creating mold is very complex but the in-built mold tools in SOLIDWORKS make it easy for you.

In SOLIDWORKS, mold tools are located in the **Mold Tools** CommandManager. You need to customize the Command manager to display this CommandManager. Click the right mouse button on any of the tab in the CommandManager and select **Mold Tools** to display the mold tools.

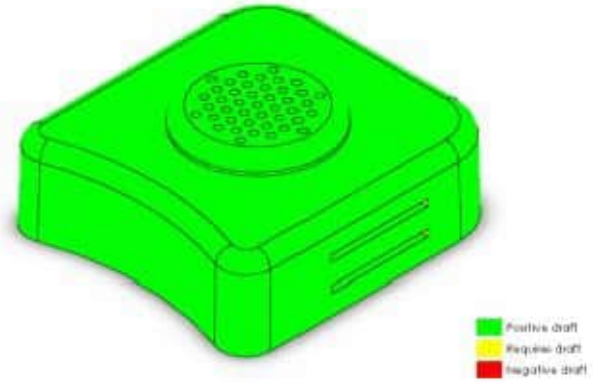
TUTORIAL 1

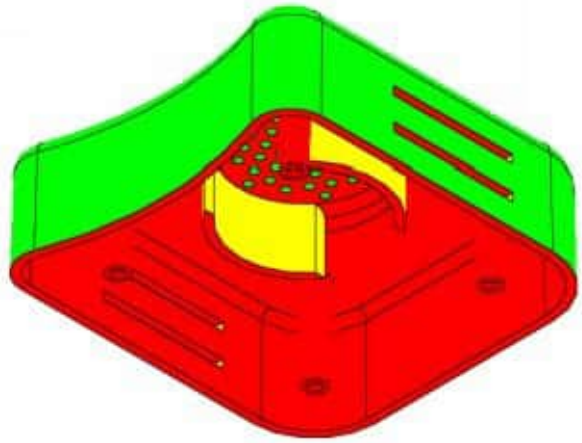
In this tutorial, you will design a mold for the part, as shown.



Performing Draft Analysis

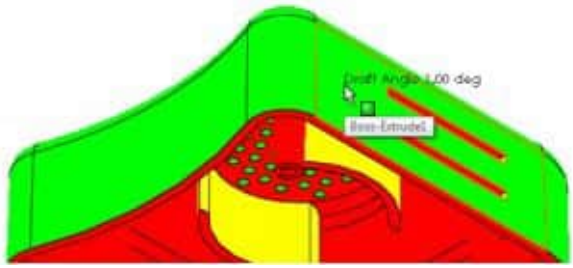
1. Download the plastic part from the companion website and open it in SOLIDWORKS.
2. Click the **Draft Analysis** button on the **Mold Tools** CommandManager.
3. Select the Top plane from the FeatureManager Design Tree to specify the Direction of Pull.
4. Set **Draft Angle** to 1 degree.
5. Click **Calculate**.



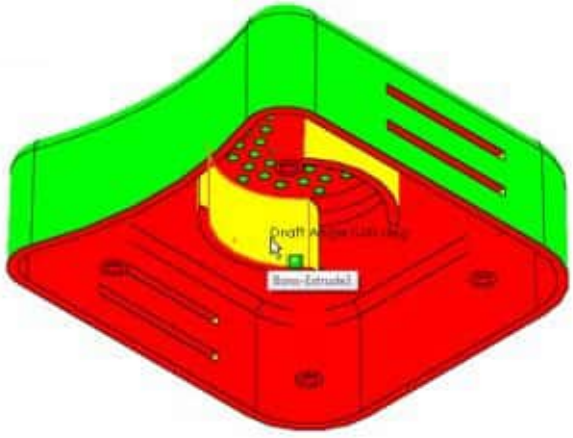


The faces displayed in green color are having a positive draft. Whereas, the faces highlighted in red color are having negative draft.

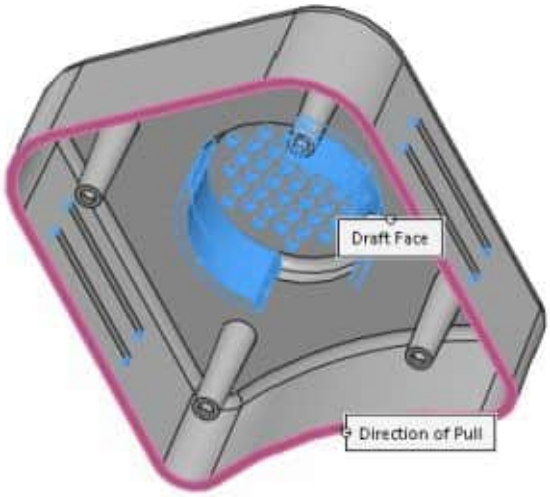
6. Place the pointer on the outer faces of the part and notice the positive draft angle.



7. Place the pointer on the faces highlighted in yellow color. Notice that the draft angle is zero. You need to apply draft to these faces.



3. Click **OK** on the PropertyManager.
9. Use the **Draft** tool and apply 1 degree draft to all the faces highlighted in yellow.



Notice that all the inner faces are red color and outer faces are green.

Applying Shrinkage allowance

Molded parts will shrink after they are removed from the mold and allowed to cool. To compensate for the shrinkage, you can scale the model to make it slightly larger.

1. To apply a scale feature, click the **Scale** button on the **Mold Tools** CommandManager.

In the **Scale about** drop-down, you can specify the scale factor about the centroid, origin, or coordinate system of the model.

2. For this tutorial, set **Scale about** to **Centroid**.
3. Check **Uniform scaling**.

Note

You can scale the model uniformly or non-uniformly. To scale the model uniformly, check **Uniform scaling**. To scale non-uniformly, uncheck **Uniform scalin**g and enter scale factors in the X, Y and Z directions. The non-uniform scaling will result in a deformed shape of the model.

4. Enter 1.02 as **Scale Factor**.
5. Click **OK** to scale the model.

Notice that the scale feature is listed in the FeatureManager Design Tree. The scale feature

will not affect the size of previously existing features or sketches.

Inserting Mold Folders

SOLIDWORKS adds mold folders to the FeatureManager Design Tree after creating the mold. However, it is recommended that you add the mold folders before creating the mold.

1. On the CommandManager, click **Mold Tools > Insert Mold Folders** .

Creating a Parting Line

A parting line acts as the boundary between the positively and negatively drafted faces of a part. It divides the model into a core and cavity. In addition, the two halves of the mold come into contact at the part line.

1. To create a parting line, click the **Parting Lines** button on the **Mold Tools** CommandManager.

The parting line uses the draft analysis tool to determine the positively and negatively drafted faces. In order to create a parting line, you need to specify the direction of pull.

2. Select the Top plane from the FeatureManager Design Tree to define the direction of pull.

3. Set **Draft Angle** to 1 degree.
4. Make sure that the **Use for Core/Cavity Split** and **Split faces** options are checked.

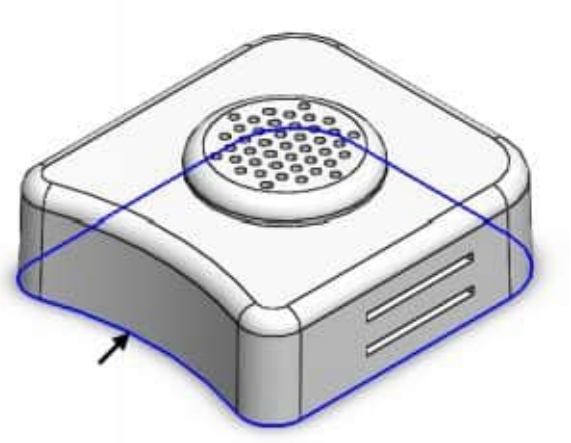
The **Use for Core/Cavity Split** option is checked to use the parting line to create the core and cavity. Note that if there are multiple parting lines created, the **Use for Core/Cavity Split** option identifies parting line to be used for splitting the mold.

5. Click the **Draft Analysis** button on the **Parting Line** PropertyManager. The parting

line appears on the model.

The parting line is created at the point where the positively and negatively drafted faces come together.

5. Click **OK** on the PropertyManager. The parting line is created and is displayed in the FeatureManager Design Tree.



Next, you need to create the shut-off surfaces

Creating Shut-off Surfaces

Shut-off surfaces are necessary when the part being molded has openings on it. To create the opening in the model, the two halves of mold called core and cavity will have to touch each other. A shut-off surface is needed to define the face where the core and cavity will touch.

1. To create shut-off surfaces, click the **Shut-off Surfaces** button on the **Mold Tools** CommandManager. The openings on the model are detected automatically.

The inner edges of the openings are automatically selected. These are used to create a shut-off surface.

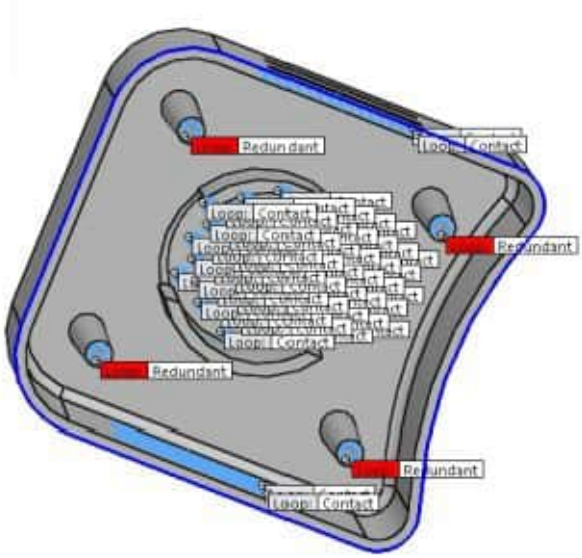
2. Select **All Tangent** from the **Reset All Patch Types** group.
3. Make sure that the **Knit** option is checked.

Note that the core and cavity surfaces are automatically created along with the shut-off

surfaces. The core surface acts as the boundary between the core and the part. The cavity surface acts as the boundary between the cavity and the part.

The **Knit** option combines or knits the shut-off surfaces to the core and cavity surfaces.

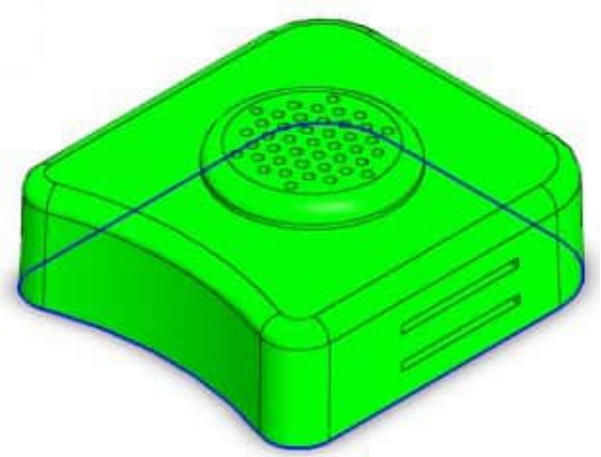
4. Rotate the model and notice the redundant loops on the mounting boss holes.



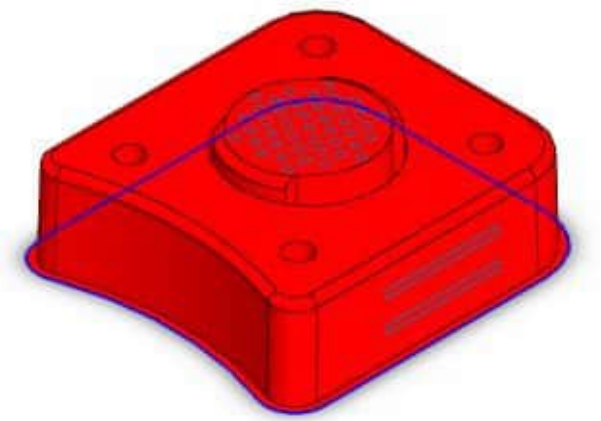
5. Press the Shift key and click on the loops highlighted as redundant. The redundant loops are deselected.
6. Click **OK** to create the shut-off surfaces.
7. To display the results, click the right mouse button on the **Solid Bodies** folder in the FeatureManager Design Tree.

7. Click **Hide** on the Pop-up menu displayed; the core and cavity surfaces are displayed.
8. Expand the **Surface Bodies** folder in the Feature Manager and hide any one of the surfaces to see the other surface.

Core Surface



Cavity Surface



Next, you need to create parting surfaces. In order to do so, you need to hide the core and cavity surfaces and show the solid body.

Creating Parting Surfaces

The parting surfaces extend away from the part and act as contact surface between the core and the cavity blocks.

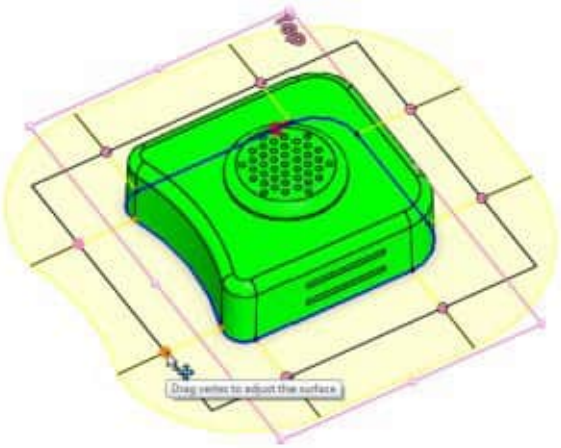
1. Click on the **Solid Bodies** folder and select **Show**.
2. To create parting surfaces, click the **Parting Surface** button the **Mold Tools** CommandManager.

Now, you need to define the method to create the parting surface.

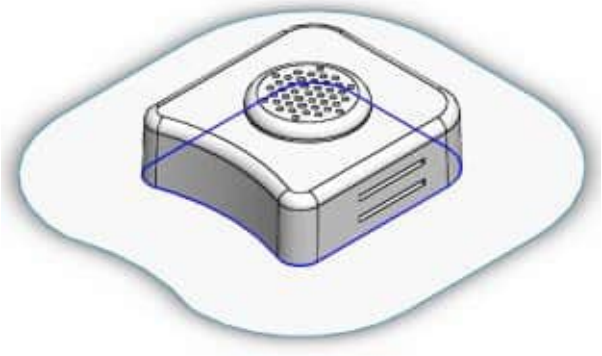
3. Select the **Perpendicular to pull** option. Notice that the parting line is selected, automatically.
4. Select the **Smooth** icon from the **Parting Surface** group.

5. Type-in 40 in the **Distance** box in the **Parting Surface** section.
6. In the **Options** section, check the **Manual mode** option. Notice the dotted handles on the parting surface.

You can drag these handles to adjust the parting surface.



7. Click **OK** to create the parting surface.

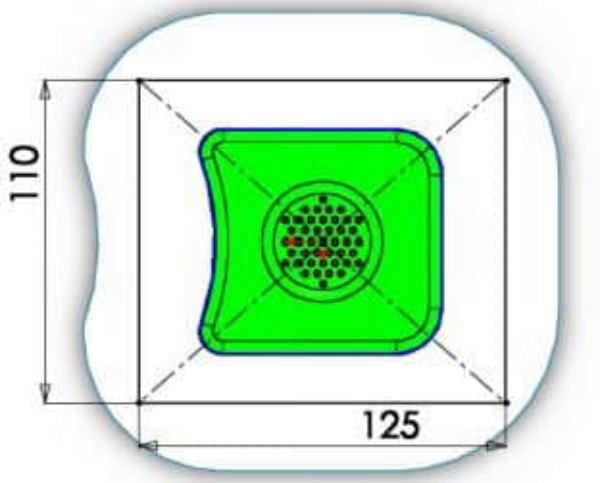


Next, you need to create the mold.

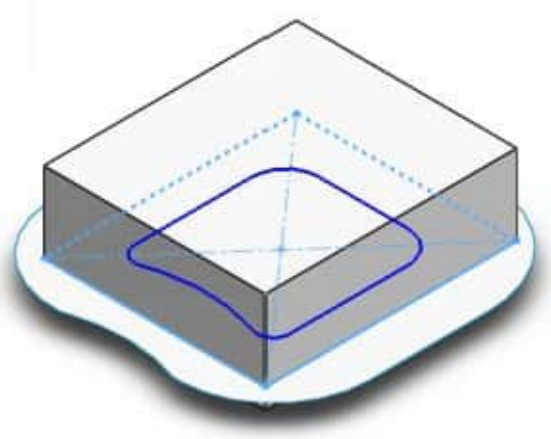
Creating the Tooling Split

First, you need to create a plane that will represent the interface between a core and cavity blocks.

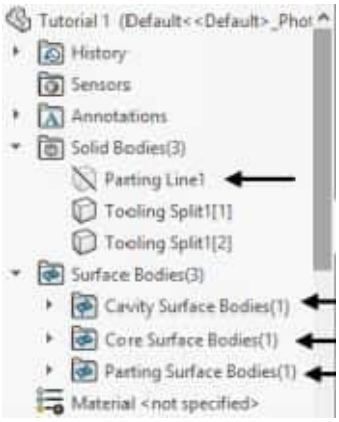
1. Click the **Tooling Split** button on the **Mold Tools** CommandManager and select the parting surface.
2. Create the sketch as shown in figure. You have to make sure that the sketch covers the part completely.



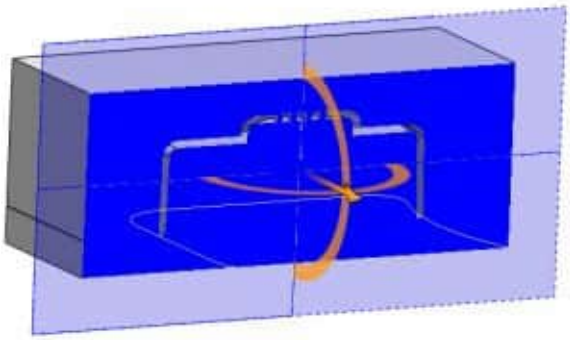
). Exit the sketch.
0. Specify the **Depth in Direction 1** as 40 in the **Block Size** group.
1. Specify the **Depth in Direction 2** as 10 in the **Block Size** group.
2. Click **OK** to create the mold.



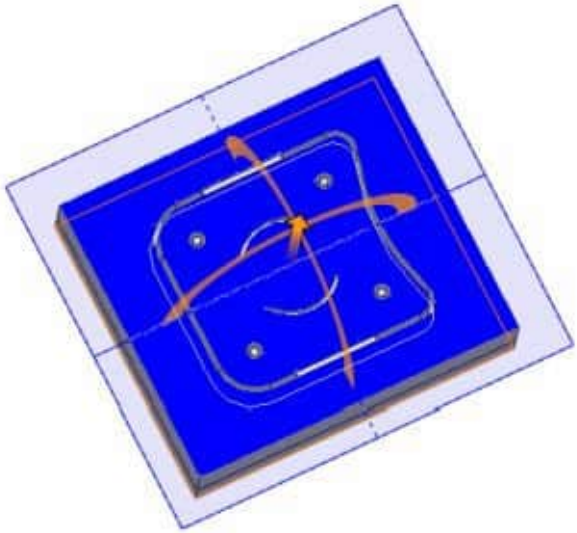
3. Hide the part body, parting surface, core surface, and cavity surface.



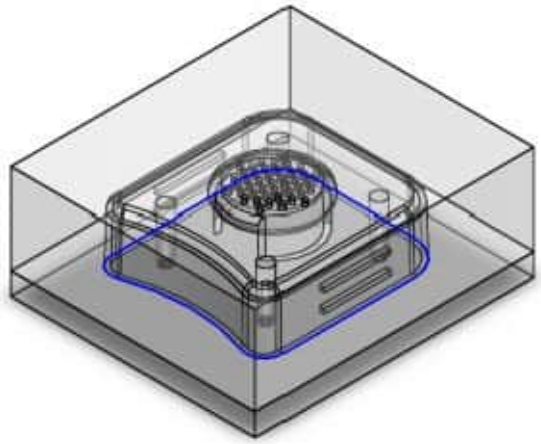
0. Click the **Section View** icon the View Heads Up toolbar.
1. Click the **Right** icon on the PropertyManager.
2. Drag the arrow and notice the gap in the mold. The gap will be filled with material to create the geometry.



3. Click the **Top** icon on the PropertyManager and drag the arrow. Notice the gap inside the mold.



4. Click **Cancel** on the PropertyManager.
5. In the FeatureManager Design Tree, expand the **Solid Bodies** folder, click the right mouse button on the tooling split and select **Change Transparency**.
6. Likewise, change the transparency of the other tooling split.



7. Click the right mouse button on the part body and select **Show**.

Performing the Undercut analysis

The undercut analysis is performed to detect the portions of the part that cannot be molded using the primary direction of pull.

1. Hide the tooling splits.
2. To perform undercut analysis, click the **Undercut Analysis** button on the **Mold Tools** CommandManager.

Note that the direction of pull is already specified and you do not need to specify it in this case.

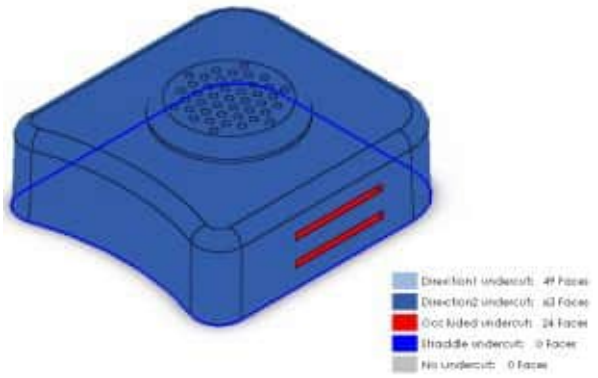
Notice the colors displayed on the geometry. Refer to the **Undercut Faces** section on the PropertyManager to get a good understanding of these colors.

The **Direction1 undercut** faces are one which are not visible from the first pull direction. A single arrow in the graphics window indicates the first pull direction.

The **Direction2 undercut** faces are one, which are not visible from the second pull direction. A double arrow in the graphics window indicates the second pull direction.

Notice the red faces of the model. These are defined as occluded undercuts, which are not visible from either pull directions.

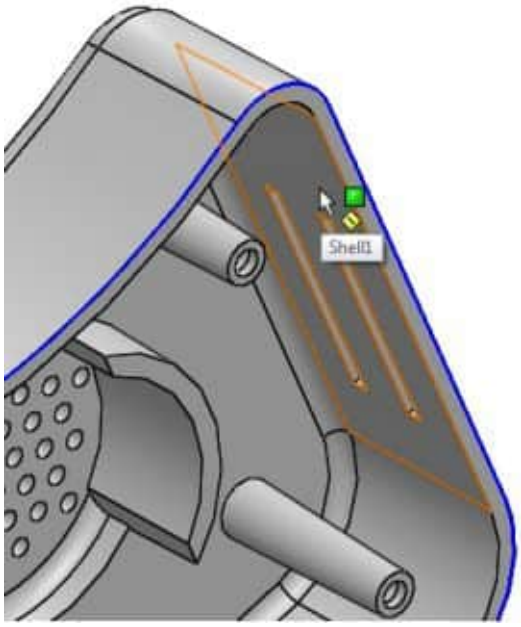
3. Click **Calculate**.
4. Click **OK** on the PropertyManager.



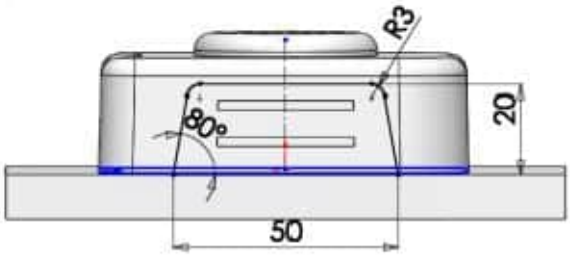
Next, you need to create side cores.

Creating side cores

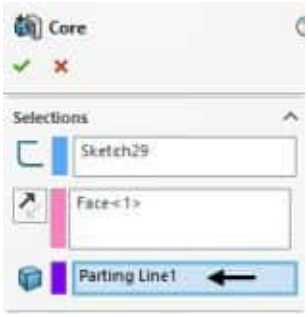
1. To create a side core, click the **Core** button on the **Mold Tools** CommandManager.
2. Select the inner face of the model, which is having openings.



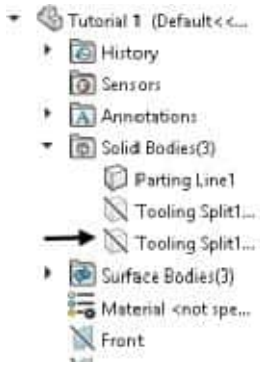
3. Show the lower tooling split.
4. Create the sketch as shown in figure.



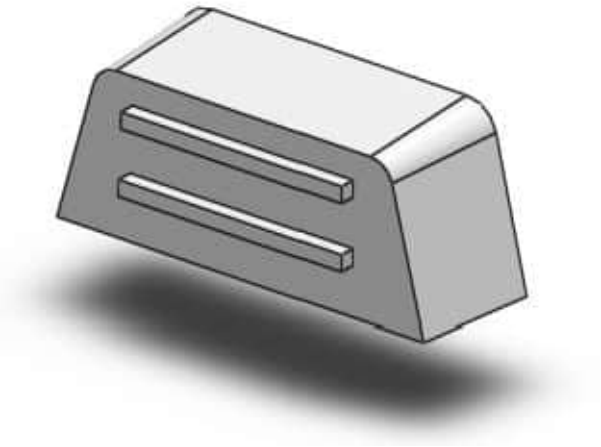
5. Exit the sketch.
6. Click in the **Core/Cavity body** field.



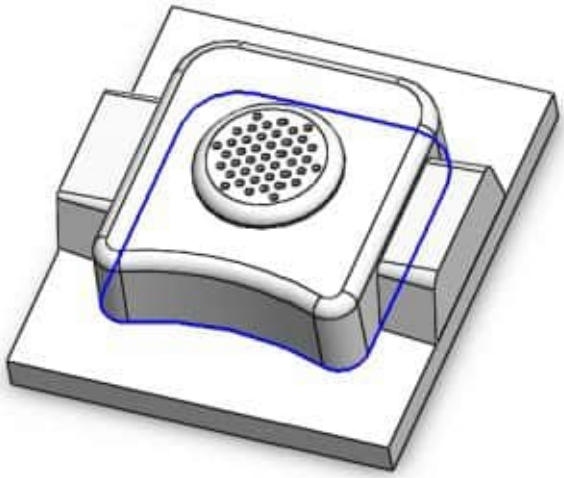
7. Select the upper tooling split from the **Solid Bodies** folder in the **Feature** tree.



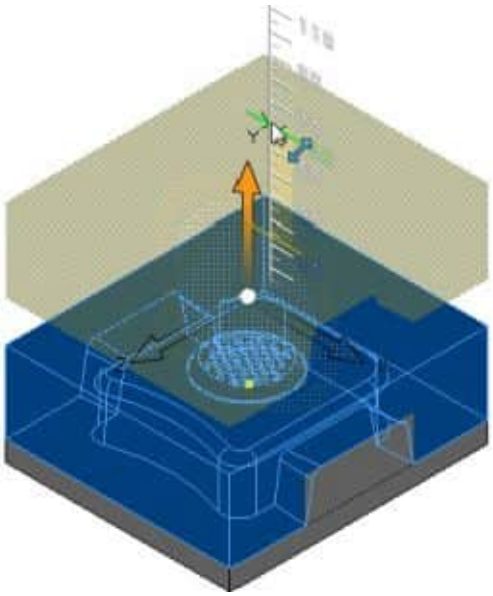
8. Type-in 4 in the **Draft Angle** box.
9. Select **Through All** from the second drop-down in the **Parameters** group.
10. Uncheck the **Cap ends** option.
11. Click **OK** to create the side core.
12. Hide the other bodies and display the side core body.



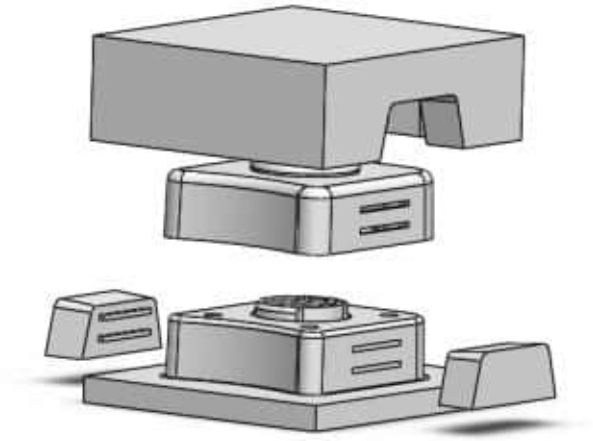
13. Likewise, create another side core for the other openings.



5. Show all the solid bodies.
6. On the CommandManager, click **Direct Editing > Move Copy Bodies** .
7. Select the upper tooling split and click the **Translate/Rotate** button on the PropertyManager.
8. Uncheck the **Copy** option.
9. Click and drag the Y-axis of the manipulator.



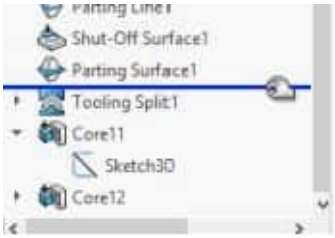
0. Click **OK**.
1. Likewise, move the bodies as shown.



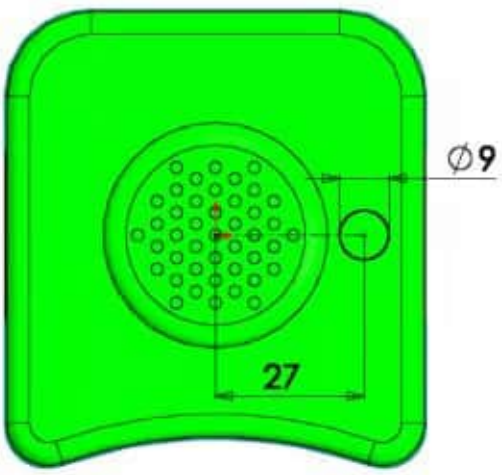
2. Save the part file.

Creating your own surfaces

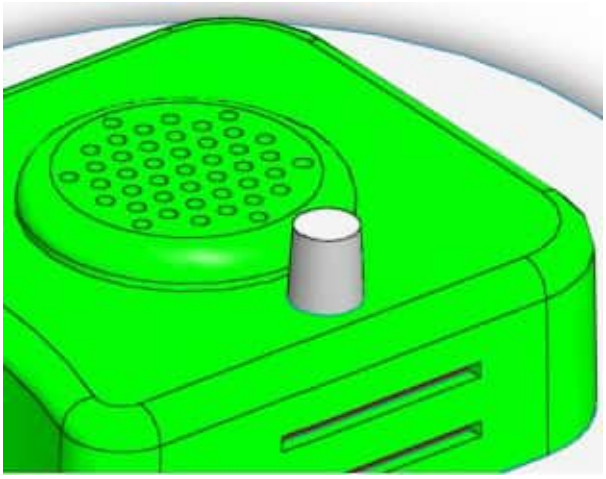
1. In the **FeatureManager Design Tree**, click and drag the bar located at the bottom.
2. Release it above the tooling split. The tooling split and cores are suppressed.



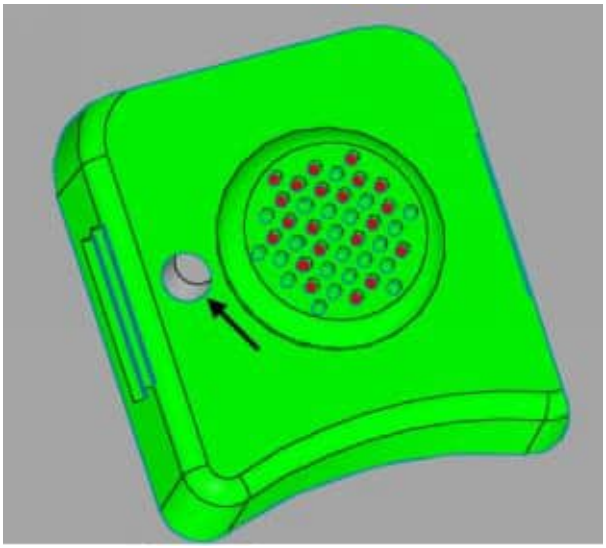
3. Hide the part body and show all the surface bodies.
4. Draw a circle on the top face of the cavity surface, as shown.



5. Create an extruded surface with the capped end and draft angle.

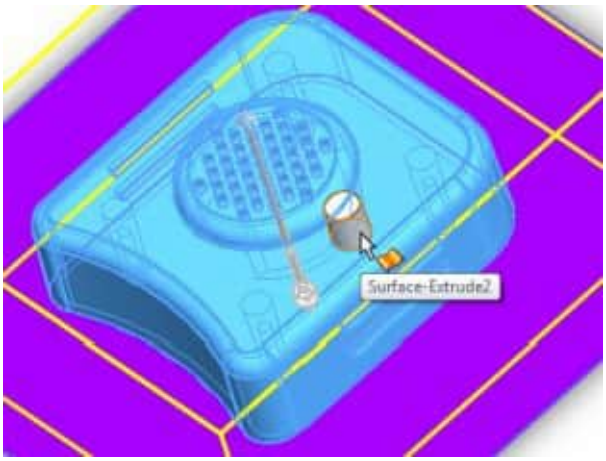


6. Hide the Core surface.
7. Trim the Cavity surface, as shown.

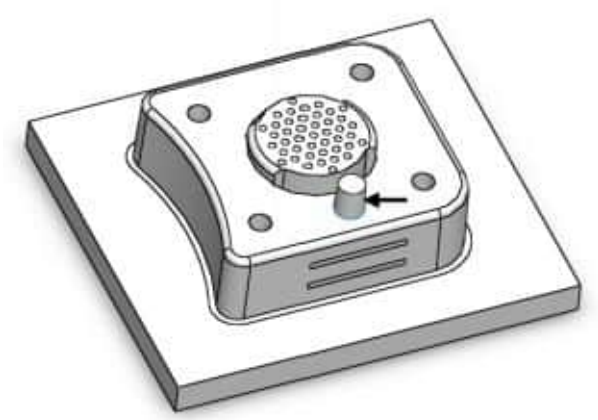


8. In the FeatureManager Design Tree, drag the bar and release it below the cores.
9. In the FeatureManager Design Tree, click on the Tooling Split and select the **Edit Feature**.

10. On the PropertyManager, click in the Cavity selection box and select the extruded surface.



1. Click **OK**.
2. Show the cavity solid body and notice the difference.



3. Save and close the file.

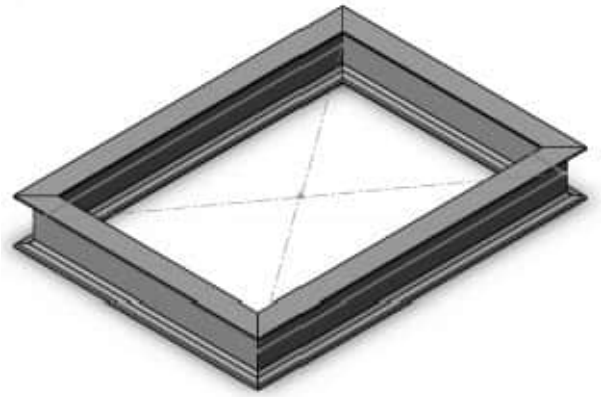
Chapter 3: Weldments

SOLIDWORKS provides a set of tools for creating welded frames and structures. In addition to creating welded frames and structures, you can use these tools for different applications. In this chapter, you will learn all of the tools used to create weldments. You will also learn to create custom weldment profiles and add them to your Library.

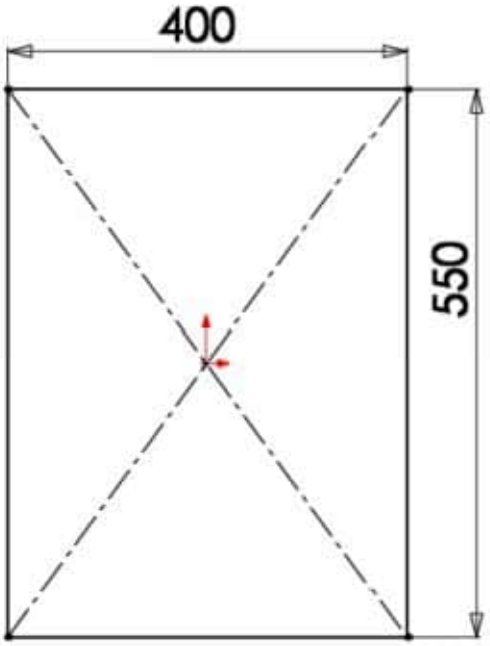
The weldment tools are located in the **Weldments** Command Manager. You need to customize the CommandManager to display this CommandManager. Click the right mouse button on anyone of the tabs on the CommandManager, and select **Weldments** from the shortcut menu.

TUTORIAL 1

In this tutorial, you create a simple weldment, as shown in figure below.



1. Start a new part file.
2. Click the **Options** button on the Quick access toolbar.
3. Click the **Document Properties** tab on the **Options** dialog.
4. Click the **Units** node.
5. Select the **MMGS (millimeter, gram, second)** option.
6. Click the **OK** button to close the dialog.
7. Create a layout sketch on the top plane, as shown.

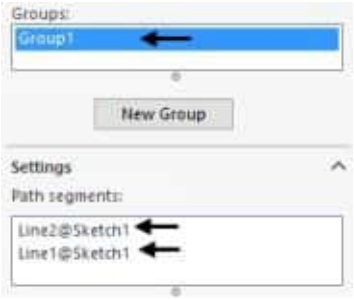


3. Exit the sketch.
9. Click the **Weldments** tab on the Command Manager.
10. Click the **Structural Member** button the **Weldments** tab.

In this **Structural Member** PropertyManager, you need to define the parameters of the structural member.

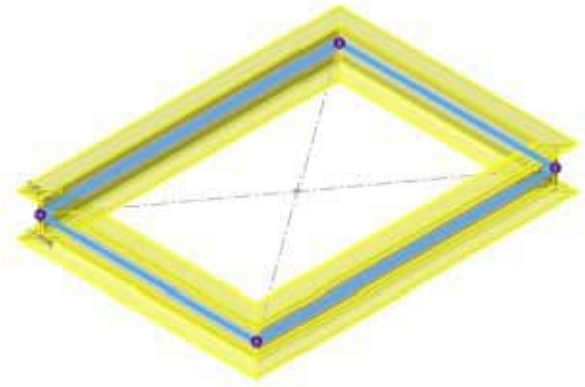
1. Select **iso** from the **Standard** drop-down.
2. Select **sb beam** from the **Type** drop-down.
3. Select **80X60** from the **Size** drop-down. Next, you need to define the position of the structural members.
4. Select the left vertical line and bottom horizontal line of the layout sketch. A structural member is created on the selected sketch segment.

You will notice that **Group1** is listed in the **Groups** selection box. In addition, the selected segments are listed in the **Path segments** selection box of the **Settings** section.

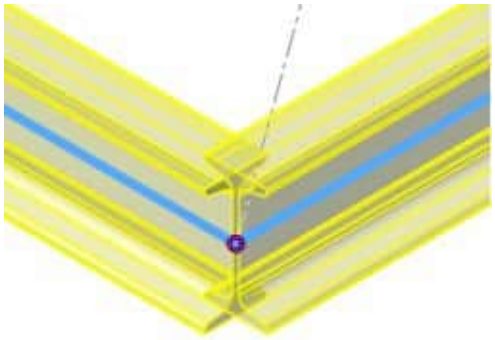


If you click the **New Group** button, a new group will be created. If you select the line segments from the layout sketch, they will be listed under the second group. Note that the size and parameter settings will be applied to all the groups. If you change the size of the structural member, it will be applied to members of all the groups.

5. Select the other two segments of the layout sketch.



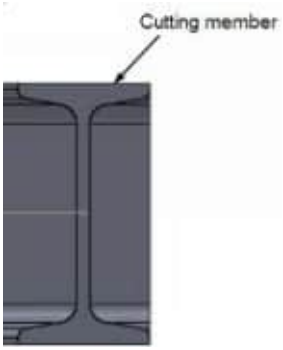
You can control the end condition of the structural members by using the **Apply corner treatment** option in the **Settings** section. By default, this option is checked. As result, the corner treatment is applied between the structural members. If you uncheck this option, the corner treatment will not be applied.



You can change the type of corner treatment by using the icons available below the **Apply corner treatment** option.

By using these icons, the corner treatment can be applied to all the structural members. If you select the **End miter** button, the **Merge miter trimmed bodies** option will be available. This option is used to merge the trimmed bodies into a single body.

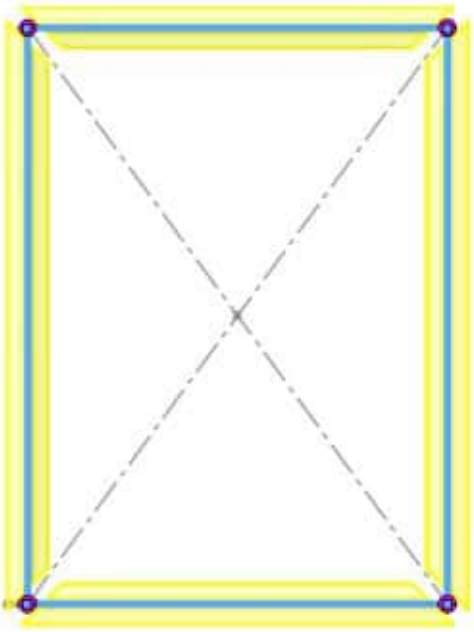
If you select the **End butt1** or **End butt2** button, the **Simple cut** and **Coped cut** buttons will be available. The **Simple cut** button is used to create a simple cut between the structural members. The **Coped cut** button creates a cut, which takes the shape of the cutting surface.



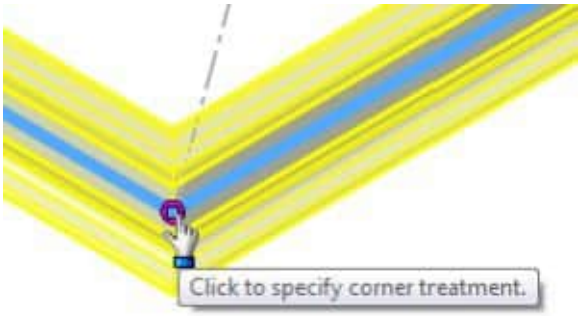
Simple Cut

Coped Cut

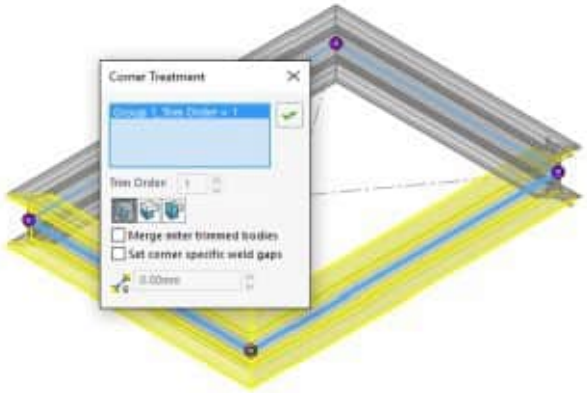
The **Gap distance** box is used to specify gaps at the corners of the structural members.



If you want to apply the corner treatment individually, click on the corner point that you want to change.



The **Corner Treatment** window appears.



You can select the required options from the **Corner Treatment** window and click the green check on the window; the corner treatment will be applied to the selected corner.

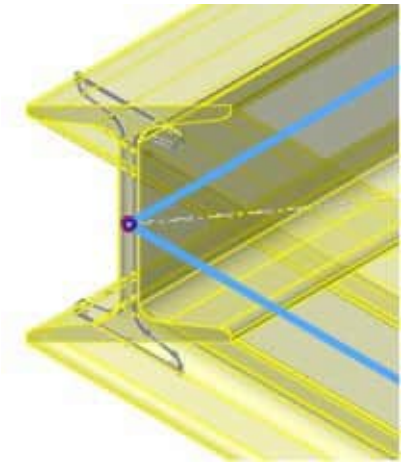
The **Alignment** option in the **Settings** section is used to specify the alignment of the structural member about the line segment. Click in the **Alignment** selection box and select the horizontal line; the **Align horizontal axis** and **Align vertical axis** options get active.

The **Align horizontal axis** option aligns the horizontal axis of the beam cross-section with the line segment.

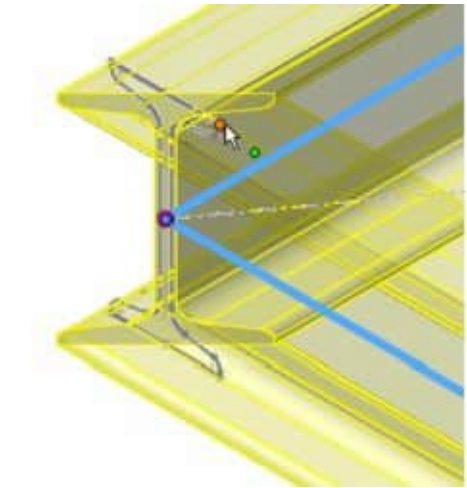
The **Align vertical axis** option aligns the vertical axis of the beam cross-section with the line segment.

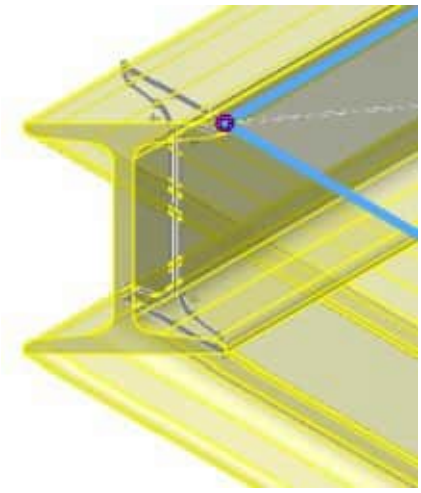
The **Rotation Angle** box allows you to align the structural member at an angle to the line segment.

The **Locate Profile** button allows you to locate the profile of the beam cross-section in the graphics window.



You can change the position of the profile by clicking any another point on the profile.

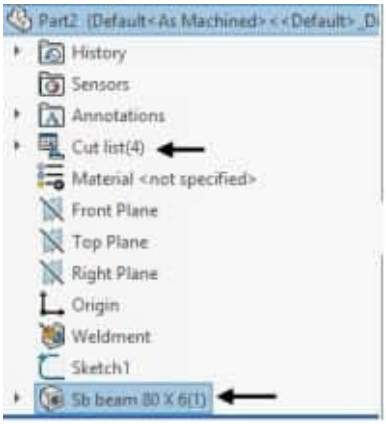




You can also use the **Mirror Profile** option and mirror the profile about the **Horizontal axis** or **Vertical axis**.

6. After specifying all the settings, click **OK** on the **PropertyManager**.

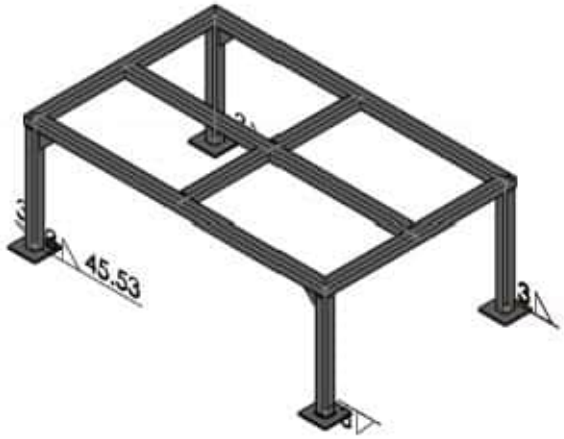
Notice the two features: **Cut list** and **Weldment** are listed in the **FeatureManager Design Tree**.



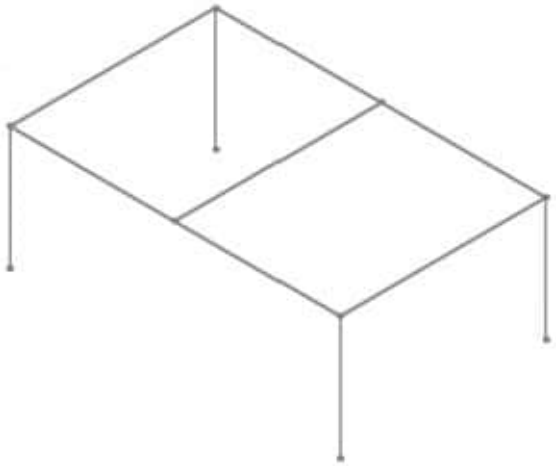
7. Save and close the document.

TUTORIAL 2

In this tutorial, you will create the weldment model as shown.



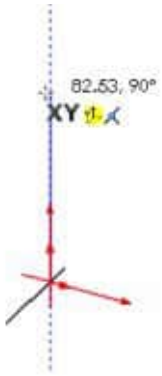
1. Start a new part document.
2. Create the layout sketch as shown.



This layout sketch can be created as a 3D sketch or a combination of 2D and 3D sketches.

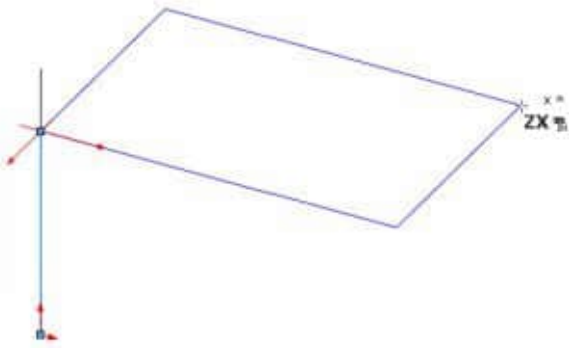
You can follow the steps given next to create the layout sketch.

3. On the CommandManager, click **Weldments > 3D Sketch** .
4. On the CommandManager, click **Sketch > Line** .
5. Select the origin point, move the pointer up along the dotted lines, and click.

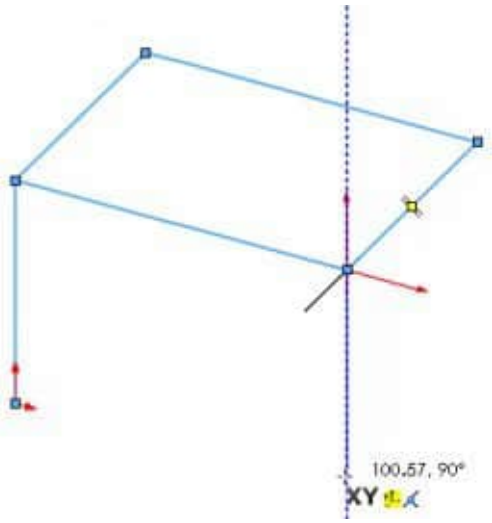


6. On the CommandManager, click **Sketch > Rectangle drop-down > Corner Rectangle** .

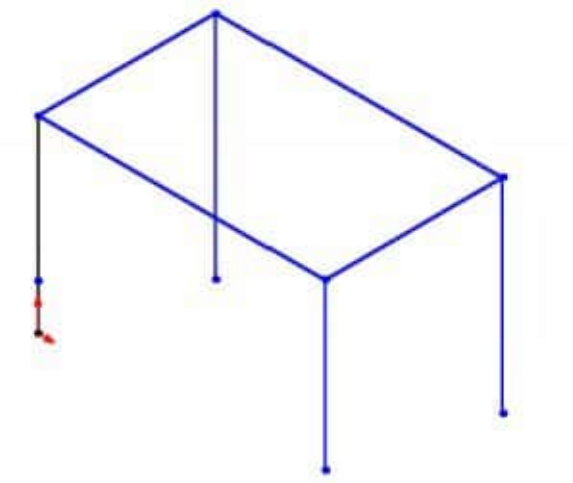
7. Select the end point of the line to define the first corner of the rectangle.
8. Press the Tab key until the rectangle is oriented, as shown.



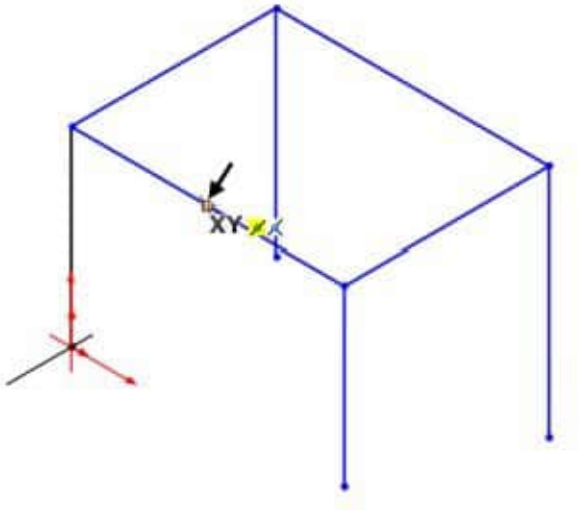
9. Move the pointer and click to create the rectangle.
10. On the CommandManager, click **Sketch > Line** .
11. Select the corner point of the rectangle, as shown.
12. Press the Tab key to switch to the XY plane.
13. Move the pointer the downward and click.
14. Press Esc to deactivate the tool.



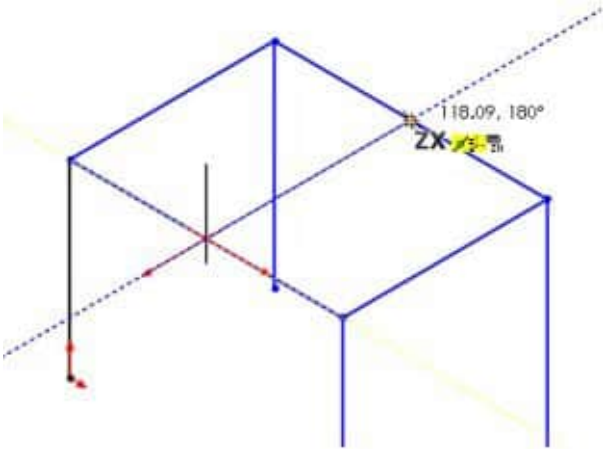
15. Likewise, create other lines as shown.



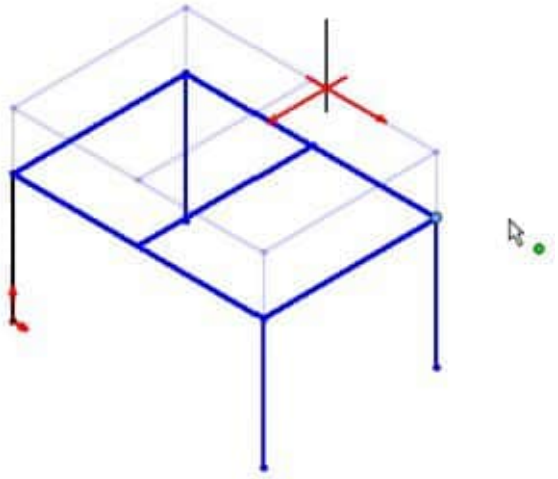
16. Activate the **Line** tool and select the midpoint of the horizontal line, as shown.



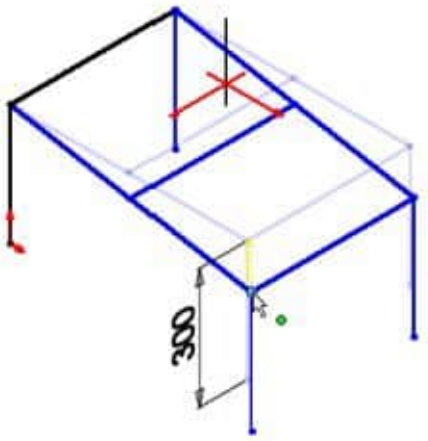
7. Press the Tab key to switch to the ZX plane.
8. Select the midpoint of the other horizontal line and press Esc.



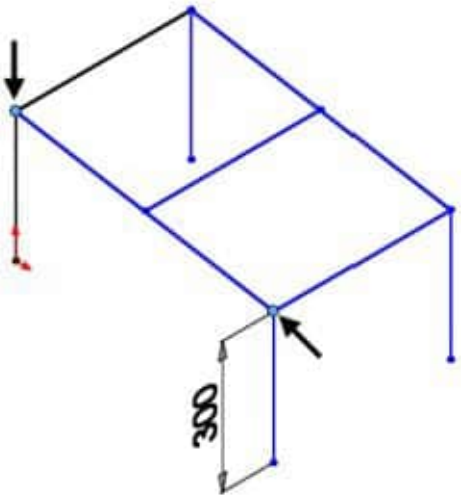
9. Press the Ctrl key and select all the vertical lines.
0. Click the **Equal** = icon on the PropertyManager.
1. Click and drag the corner point of the rectangle and notice that the vertical line length changes.



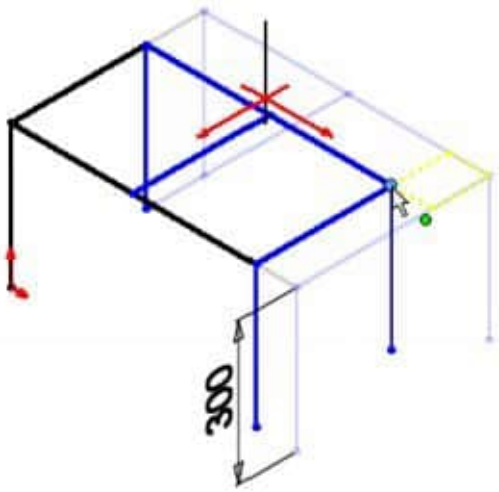
2. Add 300 mm dimension to anyone of the vertical lines.
3. Click and drag the corner of the rectangle as shown. Notice that one side of the rectangle is fixed but the other one is moveable.



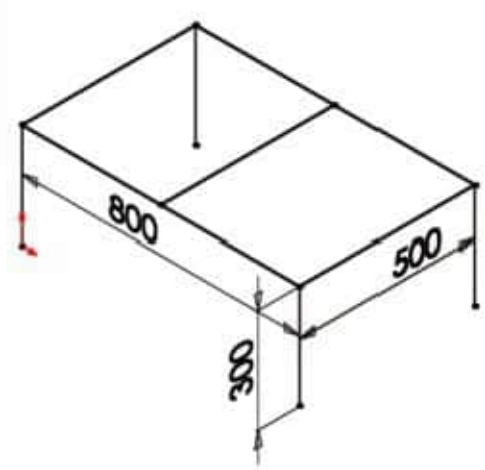
4. Press the Ctrl key and click on the corner points of the rectangle, as shown.



5. Click **Along X** icon on the PropertyManager.
6. Click and drag the corner point of the rectangle, as shown. Notice that the length and width of the rectangle changes.



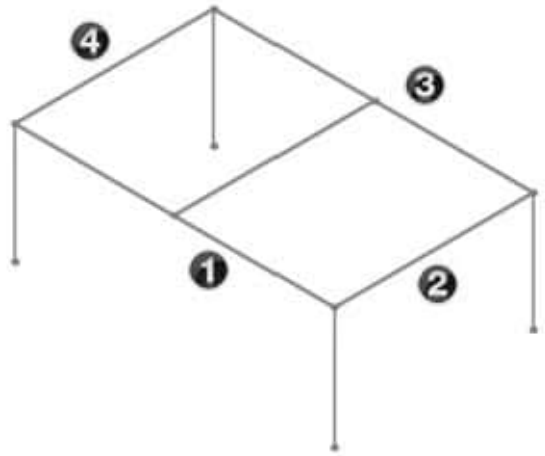
7. Add dimensions to the length and width of the rectangle. The sketch is fully defined and turned into black.
8. Exit the sketch.



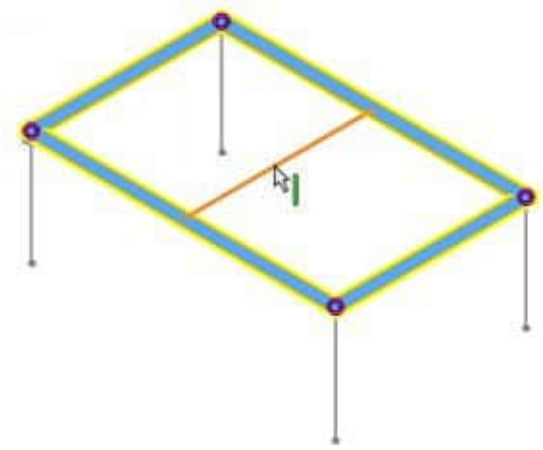
After creating the layout sketch, you need to create the structural members.

Adding Structural members

1. Click the **Structural Member** button on the **Weldment** CommandManager.
2. Select **iso** from the **Standard** drop-down.
3. Select **square tube** from the **Type** drop-pdown.
4. Select **30X30X2.6** from the **Size** drop-down.
5. Start selecting the line segments of the sketch layout.



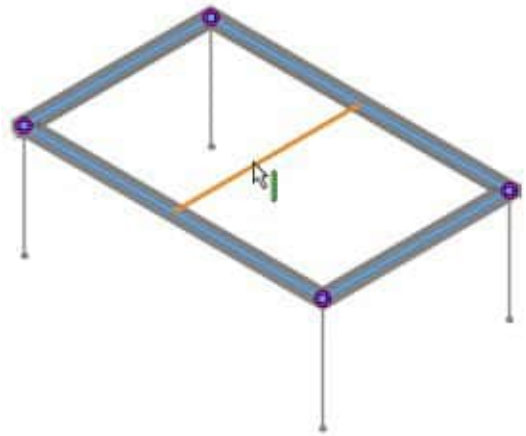
Note that you can select either the continuous or parallel line segments. If you try to select the non-continuous or parallel segment, you will be not allowed to do so.



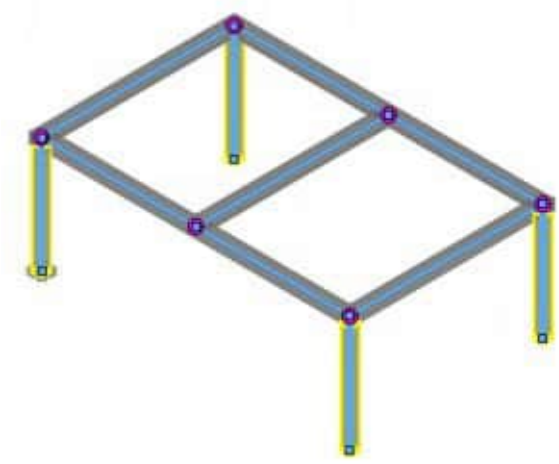
To add structural members to the non-continuous segments, you need to create a new group.

5. Click the **New Group** button below the **Groups** selection box and select the line segments.

Group 2



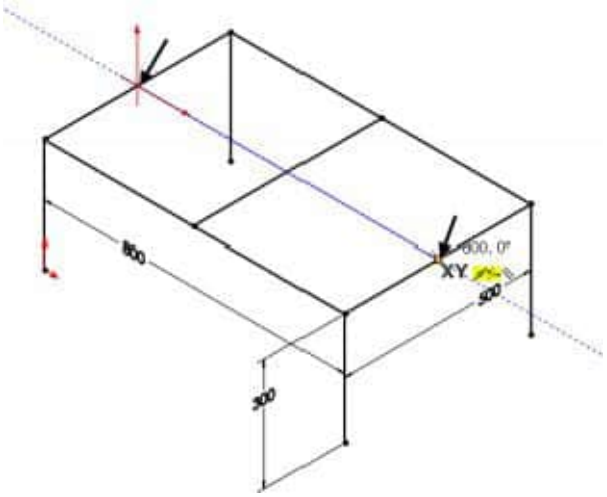
Group 3



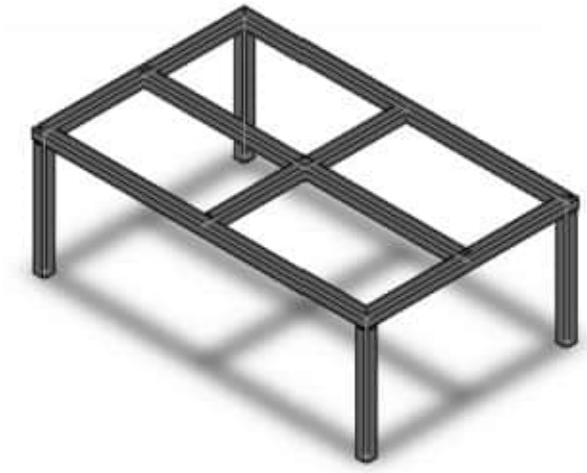
Note that the order in which the structural members are created defines the members that will be trimmed. The structural members that are crossing already created members will

be trimmed.

7. Click the **OK** button on the **PropertyManager** to accept the selection.
8. In the FeatureManager Design Tree, click on the **3DSketch** and select **Edit Sketch**.
9. Draw a line connecting the midpoints of the horizontal lines, as shown.



10. Exit the sketch.
11. Activate the **Structural Member** tool.
12. Select **iso** from the **Standard** drop-down.
13. Select **square tube** from the **Type** drop-down.
14. Select **30X30X2.6** from the **Size** drop-down.
15. Select the line segment and create a structural member, as shown.



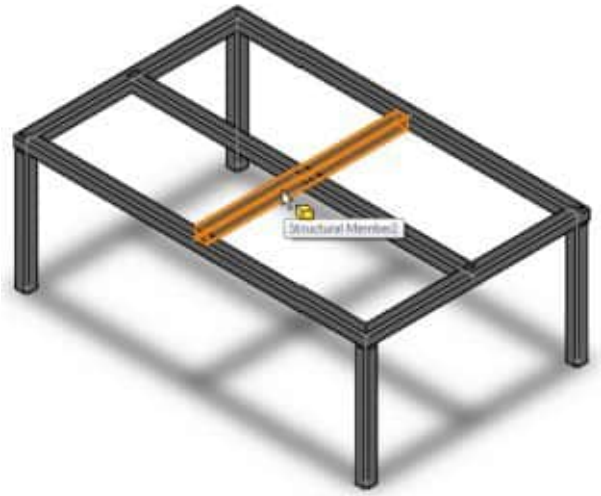
Trimming the Structural Members

The structural members will be trimmed automatically if you have created them as a single instance. However, if you have created the structural members by activating the **Structural Member** tool several times, they will not be trimmed automatically. You need to trim those members by using the **Trim/Extend** tool.

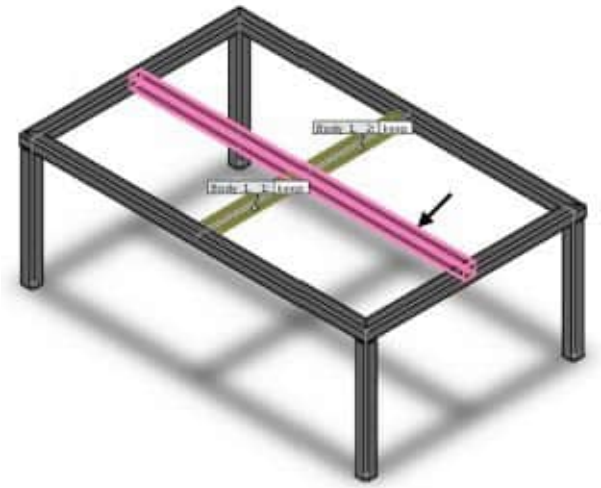
16. To trim the structural members, click the **Trim/Extend** button on the **Weldments**

CommandManager.

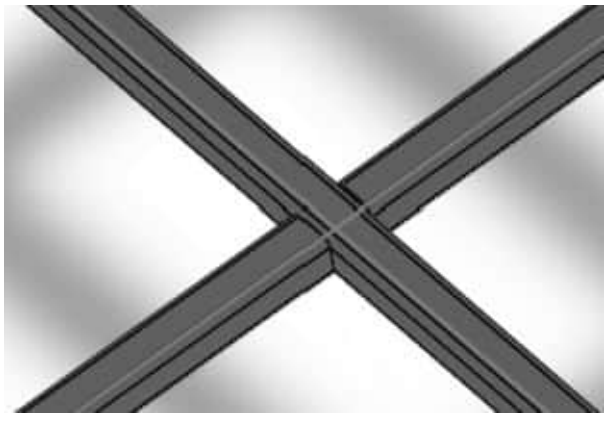
2. Select the structural member, as shown.



3. Select the **Bodies** option from the **Trimming Boundary** section.
4. Click in the **Trimming Boundary** selection box and select the horizontal member, as shown.

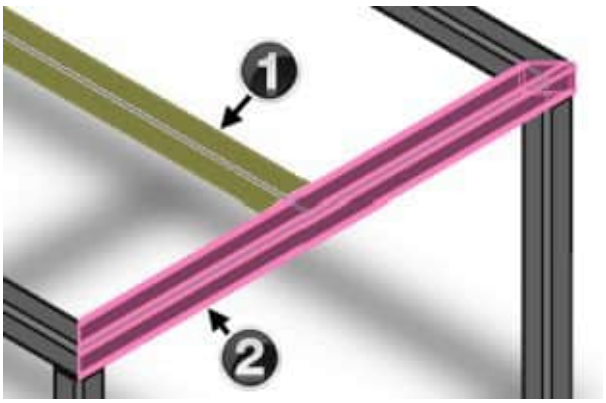


5. Click the **Simple cut between bodies** button.
6. Click **OK**. The structural member will be trimmed.

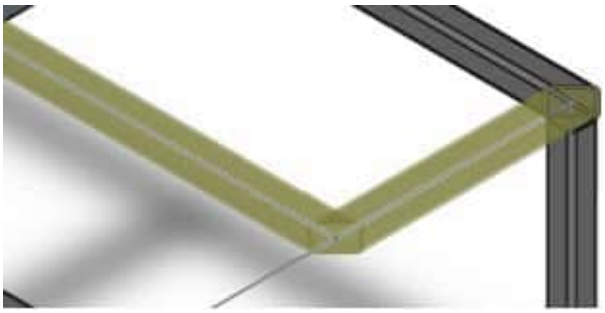


7. Again, activate the **Trim/Extend** tool.
8. Select the body to be trimmed, as shown.

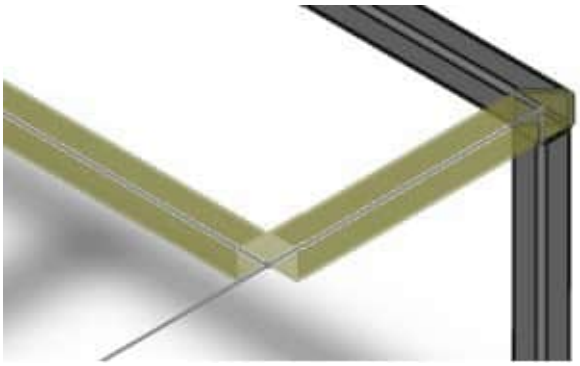
9. Click in the selection box in the **Trimming Boundary** section.
10. Select the structural member intersecting first body.



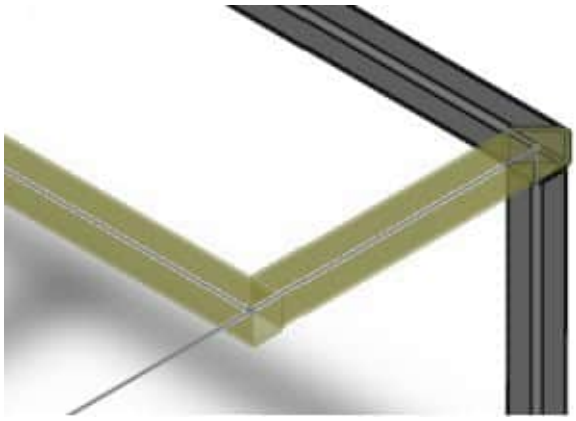
1. Click the **End Miter** icon on the PropertyManager. The two structural members will be trimmed to form a mitered corner.



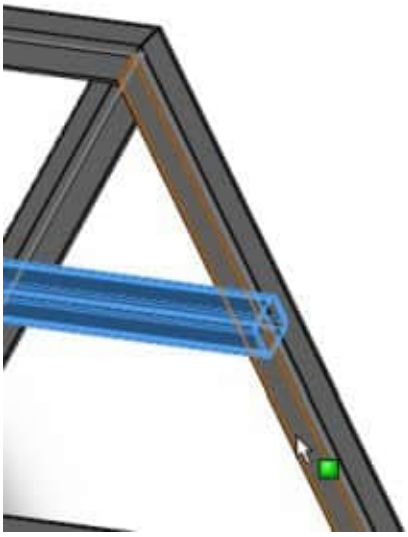
2. Click the **End Butt1** icon.

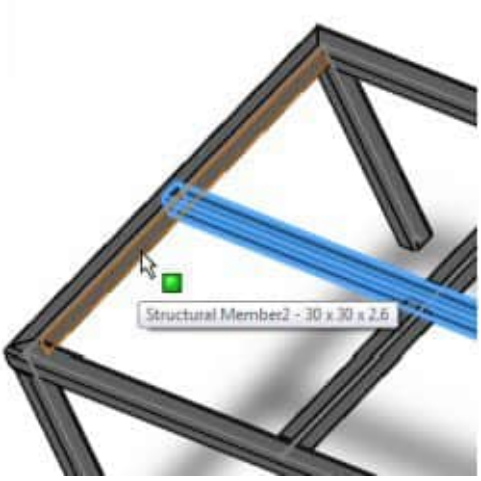


3. Click the **End Butt2** icon.



4. Click the **End Trim** icon on the PropertyManager and select the **Face/plane** option.
5. Select the side faces of the intersecting structural members.





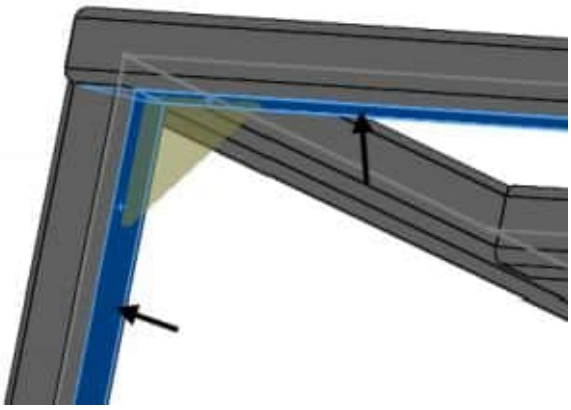
6. Click **OK** to trim the structural member.

Creating Gussets

Gussets are plates used to reinforce corners. Gussets can be created between any two faces

that intersect at an angle between 0 and 180 degrees.

1. To create a gusset, click the **Gusset** button on the **Weldments** CommandManager.
2. Select the two faces to join with a gusset, as shown.

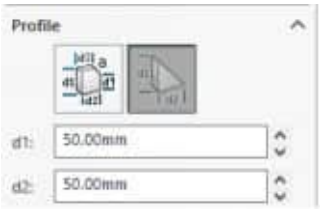


The gusset can be created in a triangular or polygonal shape.

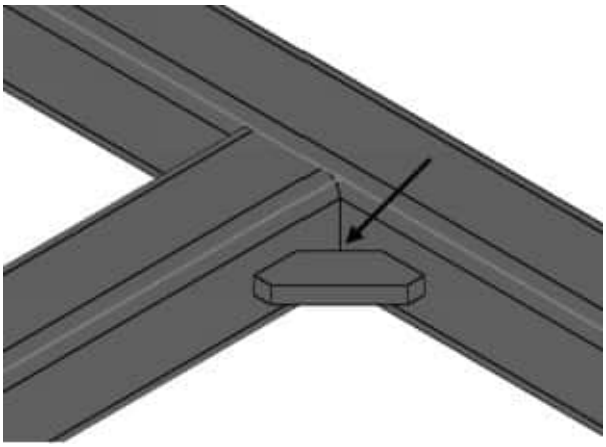
3. Select the **Triangular Profile** icon from the **Profile** section.

The size of the gusset can be controlled by the options available in the **Profile** section. Each option corresponds to the dimension shown on the profile button.

4. Enter the values as shown.



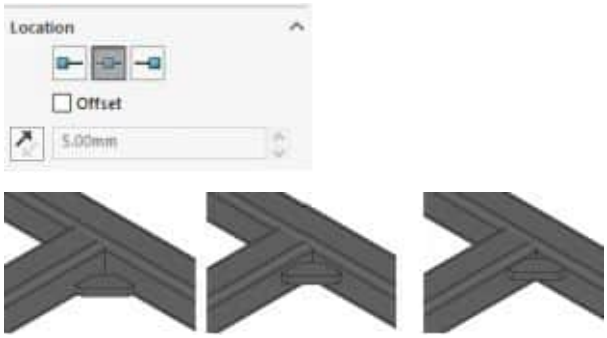
The **Chamfer** button in the **Profile** section allows you to add a chamfer to the inside edge of the gusset.



The thickness of the gusset can be specified using the thickness buttons and the **Gusset Thickness** edit box.

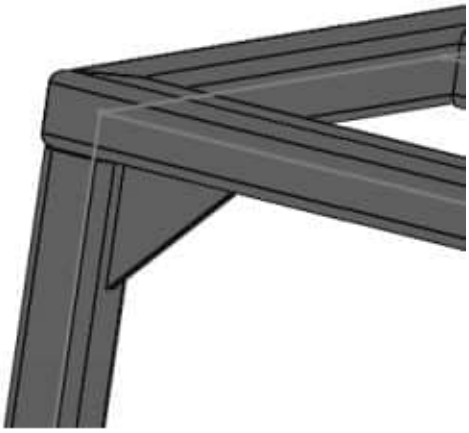
5. Click the **Both Sides** icon and enter 6 in the **Gusset Thickness** box.

The location of the gussets can be defined by using the options in the **Location** section.



If you want to define the gusset at an offset from the location, check the **Offset** option and enter the offset value in the **Offset Value** box.

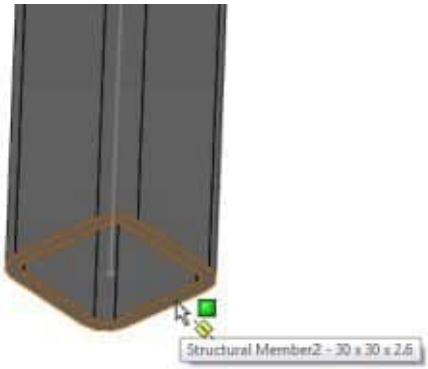
6. Click the **Mid Point** button in the **Location** section.
7. Click **OK** to create the gusset.



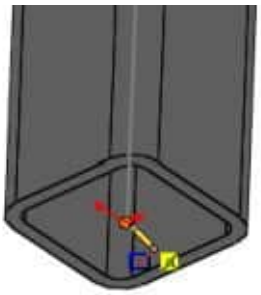
You can also use the **Extruded Boos/Base** tool to create gussets with custom shapes and sizes.

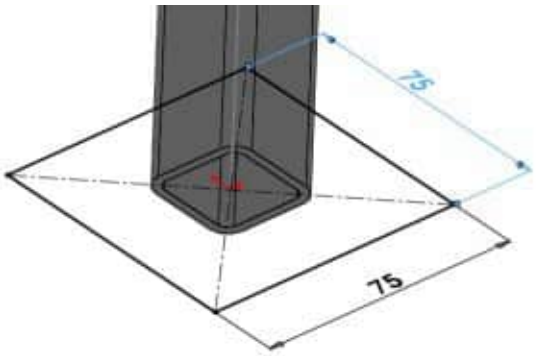
Creating Base Plates

1. On the CommandManager, click **Weldments > Extruded Boss/Base** .
2. Zoom and rotate the model to display the bottom face of the front left column.
3. Select the bottom face of the column.

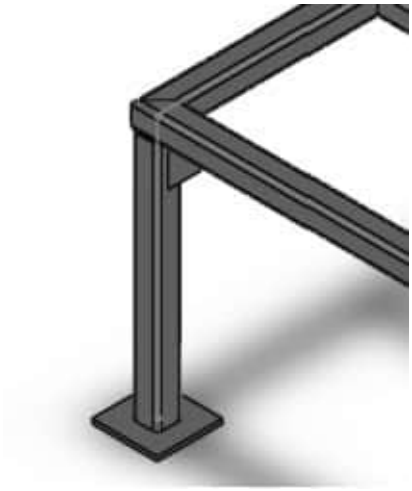


4. On the **Sketch** tab, click **Rectangle drop-down > Center Rectangle** .
5. Select the origin point and create a centered rectangle.



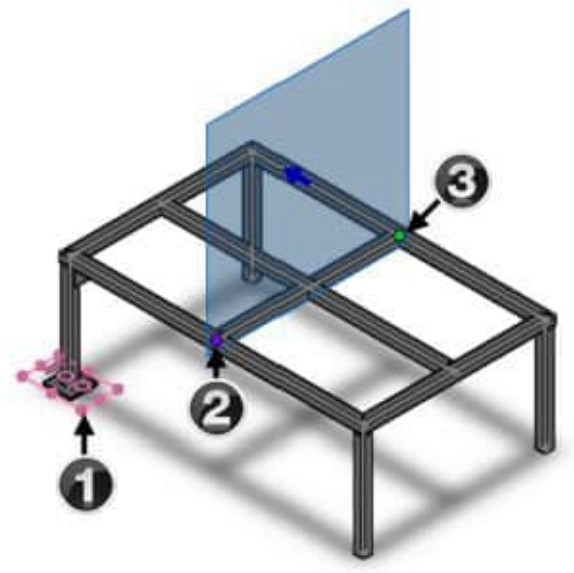


5. Exit the sketch.
7. Leave the **Merge result** option unchecked.
8. Type-in 5 in the **Depth** box and extrude the sketch downwards.



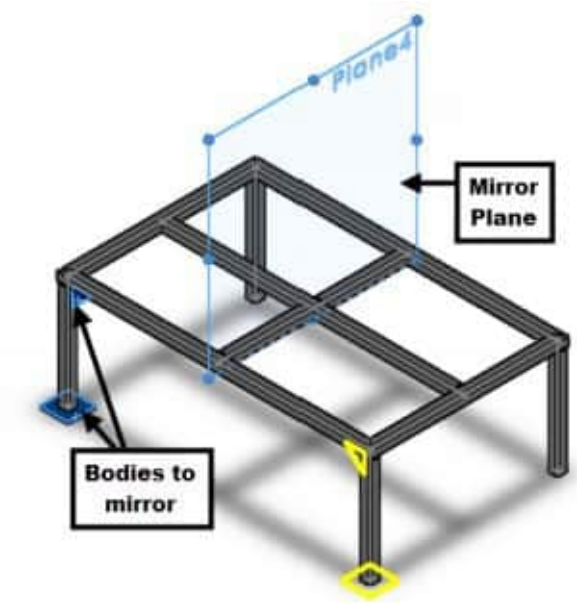
Mirroring Gussets and Base plates

1. On the CommandManager, click **Weldments > Reference Geometry > Plane** .
2. Select the Top plane from the FeatureManager Design Tree.
3. Select the two points from the layout sketch, as shown.

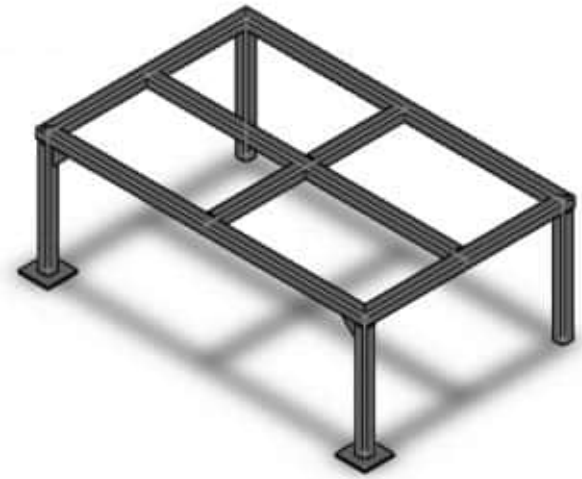


4. Click **OK** on the PropertyManager.
5. On the CommandManager, click **Features > Mirror** .
6. Select the new plane to define the mirror plane.

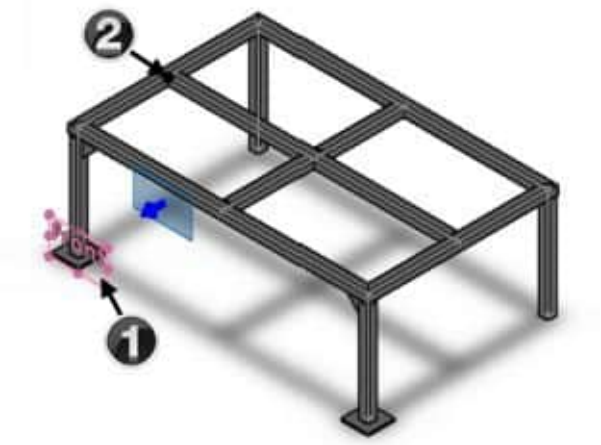
7. On the PropertyManager, expand the **Bodies to Mirror** section.
8. Select the gusset and base plate, and then click **OK**.



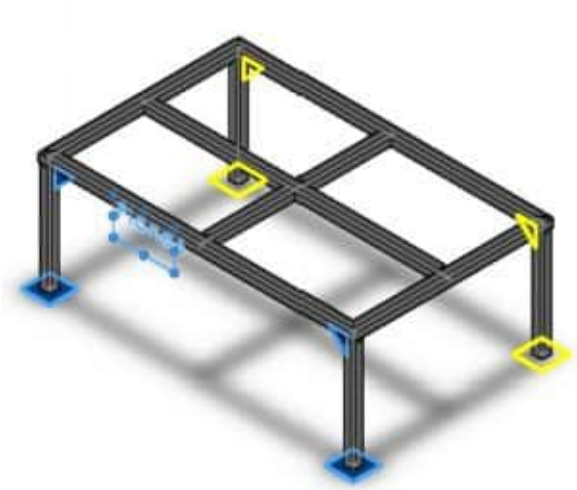
9. Hide the plane.



10. Activate the **Plane** tool, and select the Front plane and the point, as shown.



11. Mirror the base plates and gusset bodies about the new plane.



Creating Fillet Beads

You can create fillet beads to add weld information to the model. The **Fillet Bead** is the older SOLIDWORKS tool. The **Weld Bead** tool can be used instead of this tool for performance and new enhancements. However, this tool is explained to know its basic functionality.

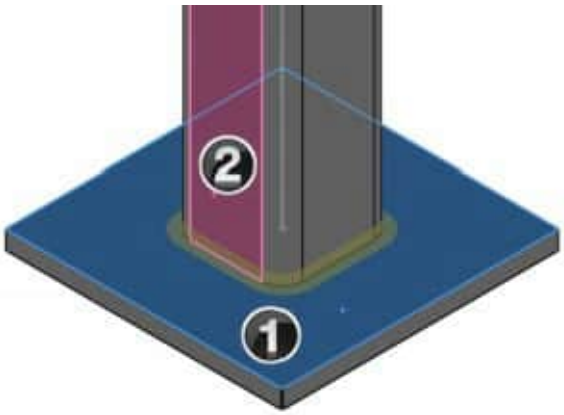
1. To create a fillet bead, click **Insert > Weldments > Fillet Bead** from the Menu bar .

You can create three types of fillet beads: **Full Length**, **Intermittent** and **Staggered**.

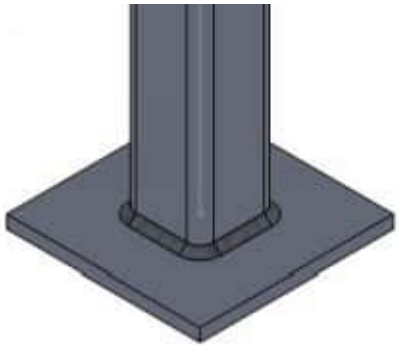
2. Select **Full Length** from the drop-down available in the **Arrow Side** section.
3. Enter **3** in the **Fillet size** box.

Next, you need to select intersecting faces to create a fillet weld between then.

4. Zoom to base plate of the front right column.
5. Select the top face of the base plate.
6. Click in the **Face set 2** selection box and select a face of the column, as shown.



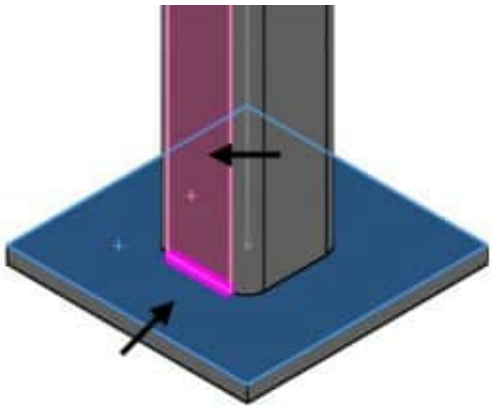
7. Make sure that the **Tangent Propagation** option is checked. You will notice that the bead is created around the structural member.
8. Leave the **Add weld symbol** option checked.
9. Click **OK** to create the fillet bead.



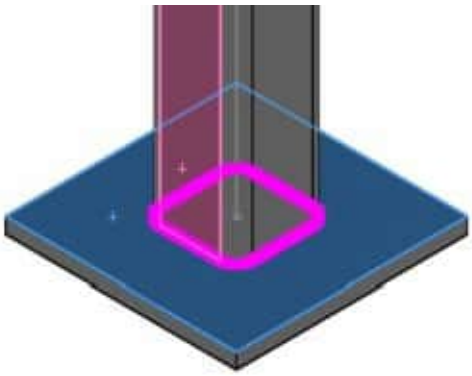
Creating Weld Beads

You can add weld beads to parts, assemblies or multi-body parts.

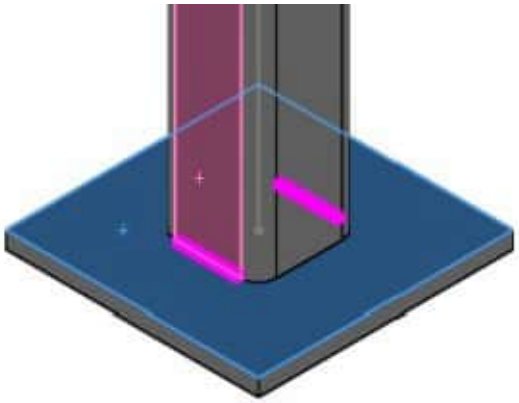
1. To add a weld bead, click the **Weld Bead** button on the **Weldments** CommandManager.
2. Zoom to the base plate of the front left column.
3. Select the top face of the base plate.
4. Select the faces of the column, as shown.



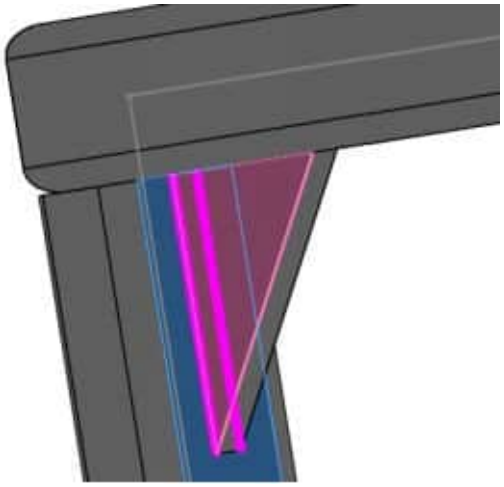
5. Check the **Tangent Propagation** option and notice that all the tangent edges are selected. The **All round** option also gives the same result.



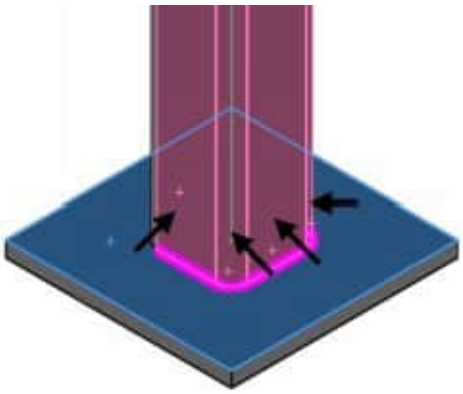
6. Uncheck the **Tangent Propagation** option and select the **Both sides** option. The opposite edge also is selected.



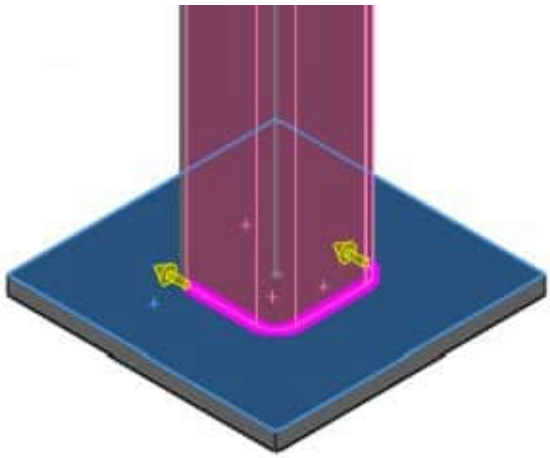
The **Both sides** option will be more useful while adding a weld to the gussets or plates.



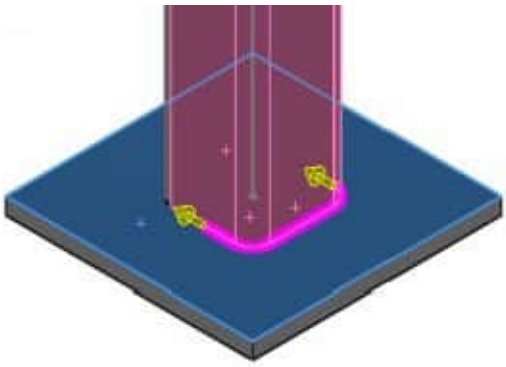
7. Select the **Selection** option and select faces, as shown.



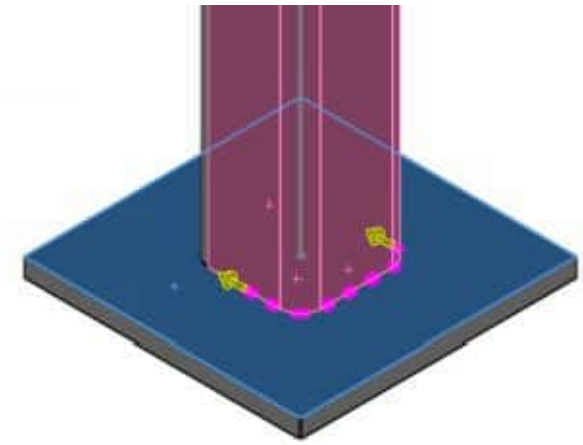
3. Check the **Form/To Length** option on the PropertyManager. Notice two arrows at the start and ends of the weld path.



9. Type-in 10 in the **Start point** box and notice that the start point is offset by 10 mm. In addition, the weld length is updated, automatically.
10. Click the **Reverse Direction** icon next to the **Start point** box. The start and ends of the weld are reversed.



You can also check the **Intermittent Weld** option to create gaps along the weld.

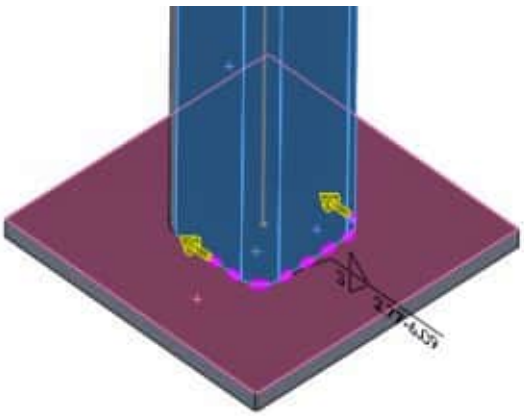


You can create an intermittent weld by specifying the **Gap and weld length** or **Pitch and weld length**.

1. Enter 3 in the **Bead Size** box
2. Click the **Define Weld Symbol** button in the **Settings** section.
3. Click the **Weld Symbol** button on the top of the leader.
4. Select the **Fillet** symbol from the flyout.

You can also click **More Symbols** on the flyout to open the **Symbol library** dialog. You can select symbols from a wide range of categories. Click **OK** after selecting the desired symbol.

5. Click **OK** on the dialog. Notice the weld symbol.

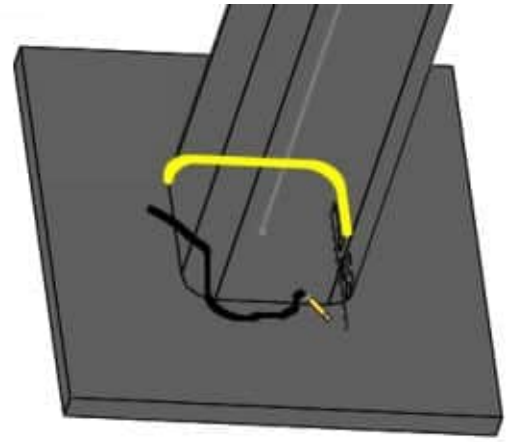


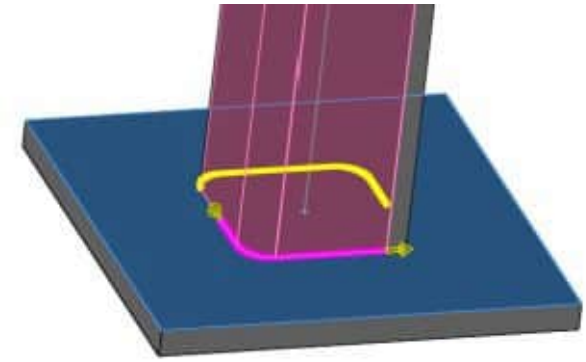
You can create multiple weld paths in a single instance.

6. Click the **New Weld Path** button in the **Weld Path** section.

Now, you need to define the weld path. You can define the weld path by selecting faces from the model or using the **Smart Weld Selection Tool**.

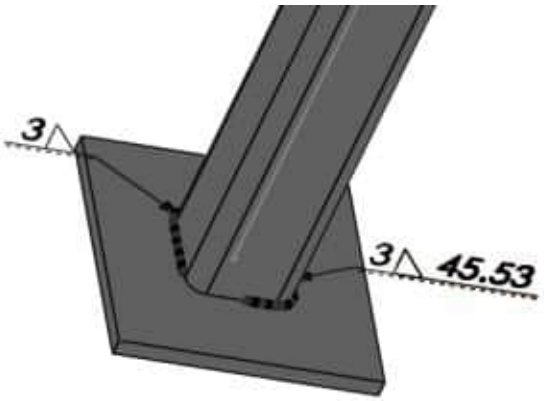
7. Click **Smart Weld Selection Tool** in the **Weld Path** section.
8. Click and drag the pointer crossing the edges, as shown. The edges are selected as soon as you release the pointer.



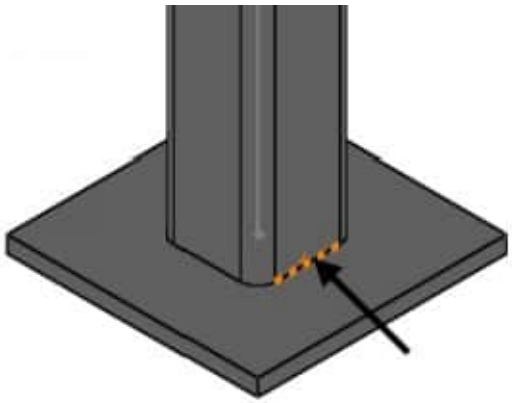


Notice that the offset is applied to the new weld path as well. You can uncheck the **From/To Length** option to remove the offset for the selected weld path.

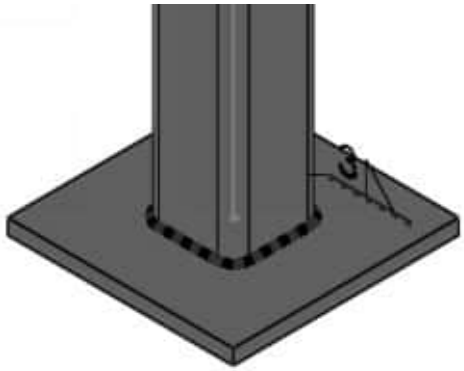
9. Click **OK** on the Property Manager. The weld is added to the model.



0. Activate the **Weld Bead** tool and **Weld Path** under the **Settings** section on the PropertyManager.
1. Zoom to the base plate of the back right column.
2. On the PropertyManager, select the **Weld Path** option.
3. Click the right mouse button on the horizontal edge of the column and select **Select Tangency**. All the tangentially connected edges are selected.

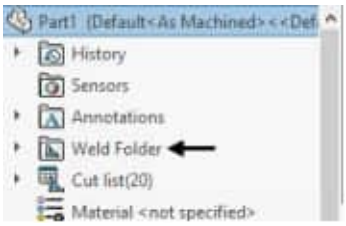


4. Leave the settings on the PropertyManager and click **OK**.

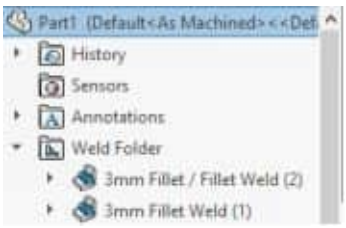


5. Likewise, add the weld bead to the other column.

Notice that there is a **Weld Folder** in the FeatureManager Design Tree. It is added to the FeatureManager Design Tree as you create weld beads.



On expanding this folder, you can see the welds that are created.



You can edit the properties of the weld by right clicking on it and selecting **Properties**.

The **Weld Bead Properties** dialog appears.

You can change the properties of the weld bead in this dialog. Note that weld beads are for representation same as cosmetic thread. They do not have any mass properties. When you have calculated the total mass of the model, the mass of the weld bead is excluded from model. You can add the **Total Weld Mass** value to the total mass of the model.

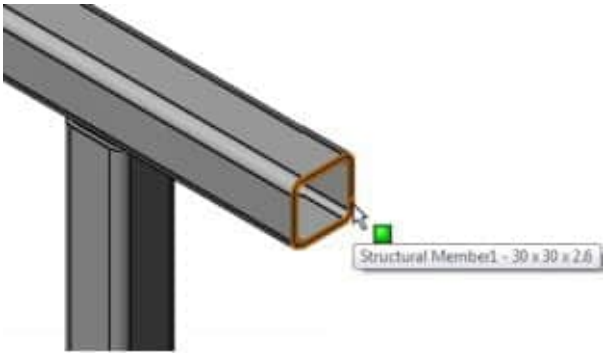
To edit a weld bead, right click on it and select **Edit Feature**.

TUTORIAL 3 (Creating End Caps)

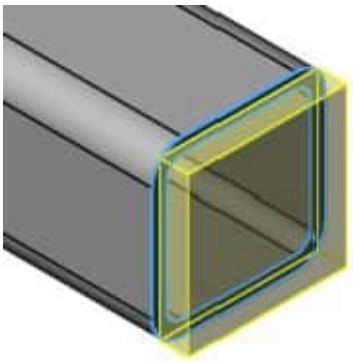
End caps are used to close the open ends of the structural members. Note that you can only apply end caps to structural members having linear edges. You cannot apply them to round tubes.

1. Start a new part document and create the model, as shown.

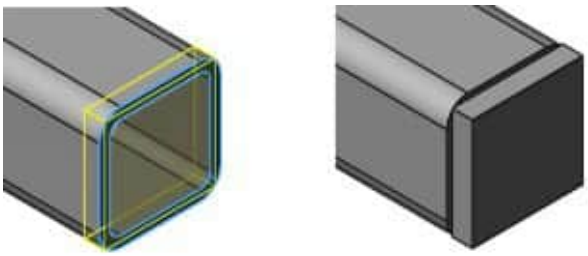
2. To create an end cap, click the **End Cap** button on the **Weldments** Command Manager.
3. Select the end face of the structural member, as shown.



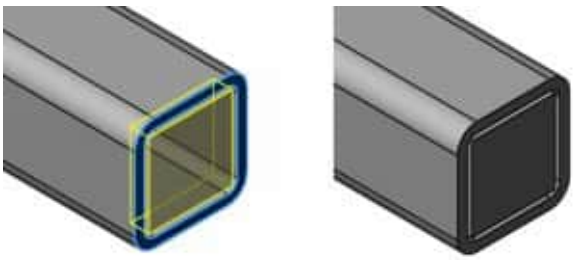
4. Click the **Outward** icon under the **Thickness direction** section. The preview of the end cap appears on the outer face.



5. Click the **Inward** icon under the **Thickness direction** section. The preview of the end cap appears inside the structural member. However, it will be created on the end face of the member. The inward end cap means that it will be created inside the original length of the structural member.



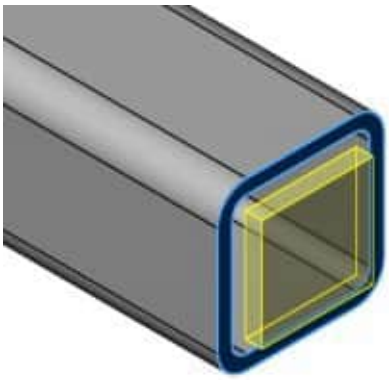
6. Click the **Internal** icon under the **Thickness direction** section. The preview of the end cap appears inside the structural member. You can further inset the end cap by entering a value in the **Inset Distance** box.



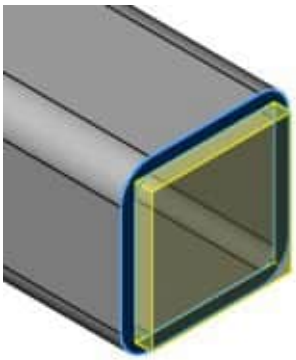
7. Click the **Outward** icon and specify the **Thickness** as **3**.

The size of the end cap can be defined using the options in the **Offset** section.

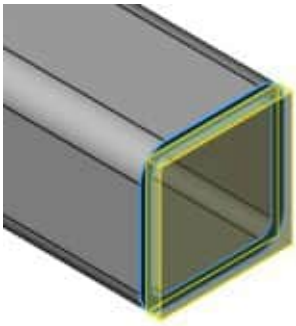
8. Select **Offset value** and enter 5 in the **Offset value** box. The size of the end cap is reduced by 5 mm. You can click the **Reverse Direction** icon to increase the end cap size.



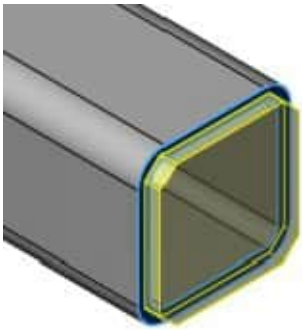
9. Select **Thickness Ratio** and enter 1 in the **Thickness Ratio** box.
10. Deselect the Reverse Direction icon. The size of the end cap is reduced by the weldment thickness value.



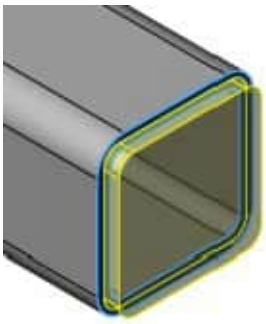
11. Change the **Thickness Ratio** to 0.5. The size of the weldment is reduced by the half of the weldment thickness value.



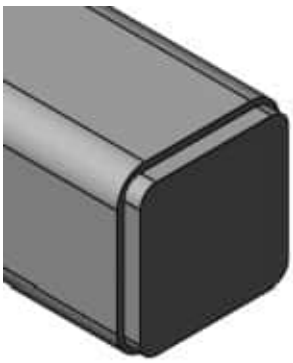
12. Check the **Corner Treatment** option to apply chamfer or fillet the end cap corners.
13. Select **Chamfer** and type-in 3 in the **Chamfer Distance** box.



4. Select **Fillet** and type-in 3 in the **Fillet Radius** box.



5. Click **OK** to create the end cap.



TUTORIAL 4 (Working with Cut lists)

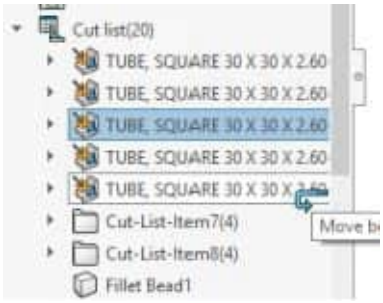
1. Open the Tutorial 2 file.

Cut lists are used to document weldment designs. They contain the information about the type of structural members, the lengths of the individual members and any other information necessary to manufacture a weldment. SOLIDWORKS automatically creates a cut list as you design a weldment. As soon as a weldment is added, the **Cut list** folder is added to the FeatureManager Design Tree.

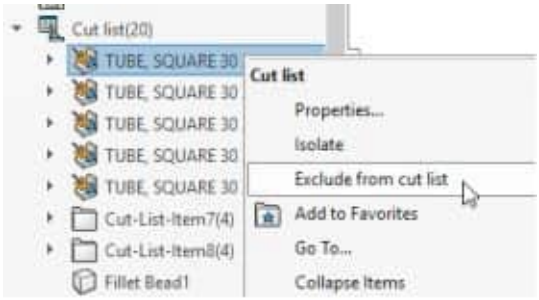
You will notice that the **Solid Bodies** folder is missing. When a structural member is added to the part making it a weldment, the **Solid Bodies** folder is replaced with the **Cut list** folder. As you continue to add weldment features, these features are listed individually within the **Cut list** folder.

The Cut list is generated and updated automatically as the **Update automatically** option is selected. You can click the right mouse button on the Cut list and deselect this option.

You can see that several subfolders are created within the **Cut list** folder. All the like items such as plates, end caps, and gussets are grouped into individual subfolders. Structural Members and weldment components are represented by the weld symbol . Other items are listed in normal folders. You can also reorder these folders by clicking and dragging them.



You can also exclude items from the Cut list by right-clicking on them and selecting **Exclude from cut list**.

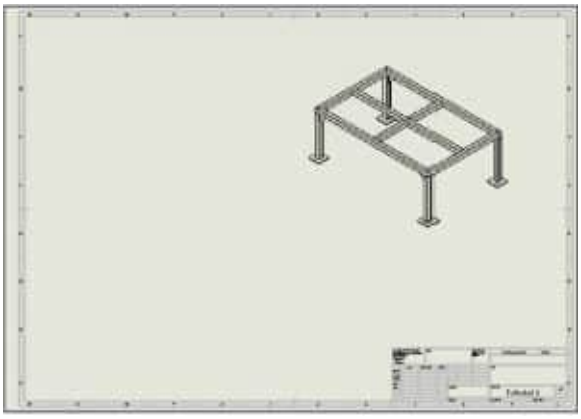


You can also include the excluded items by right-clicking and selecting **Include in cut list**. It is recommended that you make changes to the **Cut list** only after completing the weldment design. If you add any additional members to the model, you will lose all of the changes made to the Cut list order.

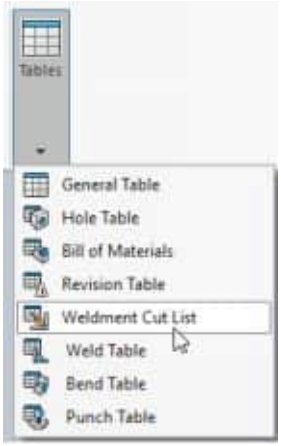
Adding Cut list to the Weldment Drawing

1. Create a drawing by selecting **New > Make Drawing from Part/Assembly**.

2. Select **A2 (ISO)** from the **Sheet Format/Size** section and click **OK**.
3. On the **View Layout** CommandManager, click the **Model View** tool.
4. Double-click on the Tutorial 2.
5. Place the Isometric view on the drawing sheet.



6. Click **Annotation > Tables > Weldment Cut List** on the CommandManager**.**

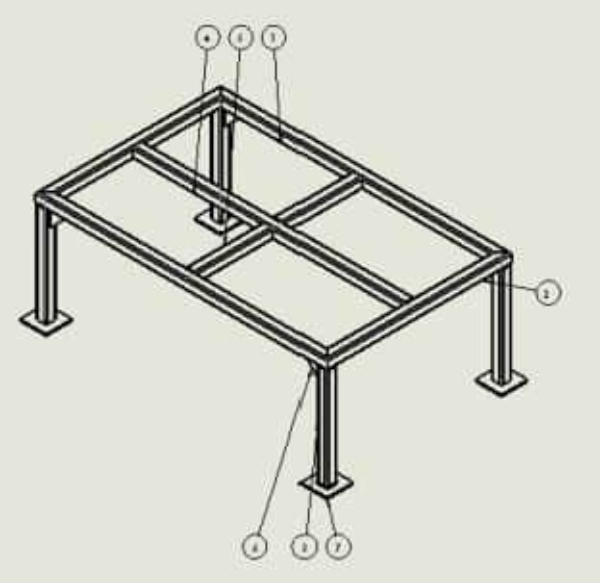


7. Select the view from the drawing sheet. The **Weldment Cut List** PropertyManager appears. The options in this PropertyManager are similar to the **Bill of Materials** one.
8. Click **OK** on the Property Manager and place the cut list table on the drawing sheet.

ITEM NO.	QTY	DESCRIPTION	LENGTH
1	2	TUBE, SQUARE 30 X 30 X 2.60	830
2	2	TUBE, SQUARE 30 X 30 X 2.60	530
3	4	TUBE, SQUARE 30 X 30 X 2.60	285
4	1	TUBE, SQUARE 30 X 30 X 2.60	770
5	2	TUBE, SQUARE 30 X 30 X 2.60	220
6	4		
7	4		

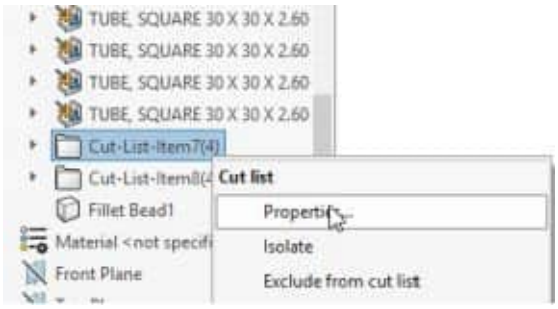
The **Description** and **Length** may or may not be populated in the cut list. It depends on the type of weldment. The **Description** and **Length** of the structural members are automatically populated. Notice that the **Description** and **Length** of the members such as gussets, base plates, and end caps are not populated automatically. To know about the members that are not populated, you need to add balloons to the model view.

9. On the CommandManager, click **Annotation > Auto Balloon**.
10. Click **OK**. Notice that the gussets and base plate are annotated with the 6 and 7 number balloons.

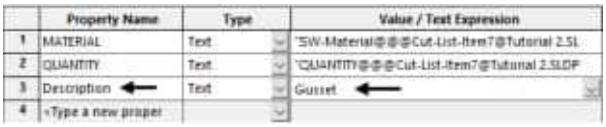


You need to populate these properties manually. To do so, you need to switch to the part document and add properties to these members.

1. Open the part document of the weldment.
2. Expand the **Cut list** folder in the **FeatureManager Design Tree**.
3. Right-click on the **Cut-list-Item7** subfolder under the Cut list folder and select **Properties**; the **Cut List Properties** dialog appears.



4. In the **Cut List Properties** dialog, enter **DESCRIPTION** in the **Property Value** field and **Gusset** in the **Value/Text Expression** field.



5. Click the **Cut-list-Item8** folder and enter **DESCRIPTION** in the **Property Value** field and **Base Plate** in the **Value/Text Expression** field.

6. Click **OK**.
7. Switch to the Drawing document of the weldment. Notice that the **Description** column is populated in the Cut list table.

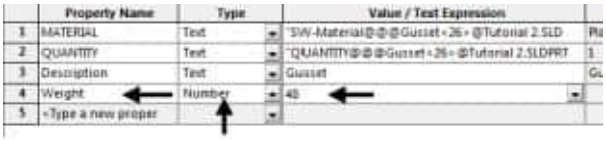
ITEM NO.	QTY.	DESCRIPTION	LENGTH
1	2	TUBE, SQUARE 30 X 30 X 2.60	830
2	2	TUBE, SQUARE 30 X 30 X 2.60	530
3	4	TUBE, SQUARE 30 X 30 X 2.60	285
4	1	TUBE, SQUARE 30 X 30 X 2.60	770
5	2	TUBE, SQUARE 30 X 30 X 2.60	220
6	4	Gusset	
7	4	Base Plate	

Adding Columns to the Cut list table

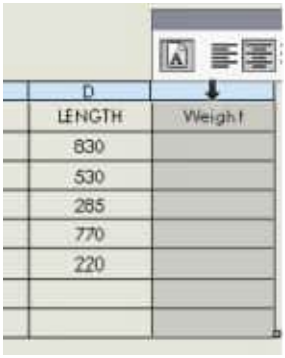
You can add more columns to the cut list table and save it as a template for the later use.

1. Place the pointer on the table and notice that row and column headers.
2. Click the right mouse button on the right most column and click **Insert > Column Right**.
3. On the PropertyManager, enter **Weight** in the **Title** box and click **OK**.
4. Open the part document of the weldment.

5. In the FeatureManager Design Tree, click the right mouse button on the Gusset folder and select **Properties**.
6. On the **Cut List Properties** dialog. enter **Weight** in the **Property Value** field and **48** in the **Value/Text Expression** field.



7. Click **OK**.
8. Switch to the drawing and notice that the value is not updated in the cut list table.
9. Click in the **Weight** column header.



10. On the PropertyManager, select **Cut list item property**.
11. Select **Weight** from the drop-down menu.
12. Click the green check to update the table.

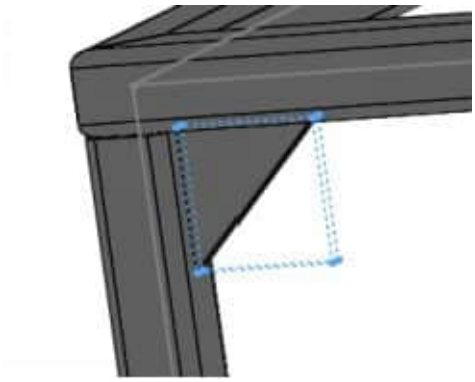
13. To save the cut list table as a template, right-click on the table and select **Save As**.
14. Type **Custom_cutlist** and click **Save**.

Creating Bounding box

Bounding box is the smallest box around a weldment. It gives you an idea about the weldment dimensions. You can use the bounding box to add some properties to the Cut list, automatically.

1. Open the part document of the weldment model.
2. In the FeatureManager Design Tree, expand the Cut list folder.
3. Click the right mouse button on the **Gusset** folder and select **Create Bounding Box**.
4. Expand the Gusset folder and notice a 3D sketch inside it.

5. Click on the 3D sketch and notice a dotted box around the gusset.



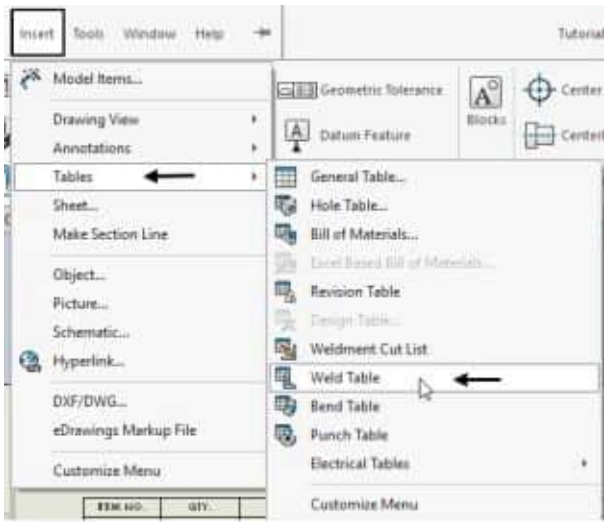
6. Click the right mouse button on the Gusset folder and select **Properties**. Notice the thickness, width, length, and volume properties on the **Cut list Properties** dialog.
7. Click **OK** to close the dialog.

You can edit or delete the bounding box by simply clicking the right mouse button on the folder and selecting **Edit Bounding Box** or **Delete Bounding Box**.

Adding a Weld table to the Weldment Drawing

You can show the weld beads or fillet beads of a weldment in a drawing document. You can show them in the form of a Weld table in the drawing.

1. To add a weld table, click **Insert > Tables > Weld Table** on the Menu bar.



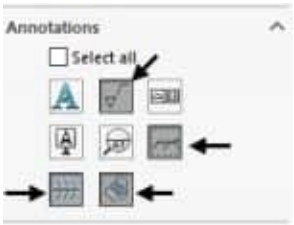
2. Select the Isometric view from the drawing.
3. Click **OK** on the Property Manager.
4. Place the weld table on the drawing sheet.

ITEM NO.	WELD SIZE	SYMBOL	WELD LENGTH	WELD MATERIAL	QTY.
1			23.36		1
2	3		2.77-6.54[23.36]		1
3			19.39		1
4	3		2.77-6.54[19.39]		1
5	3		111.07		2

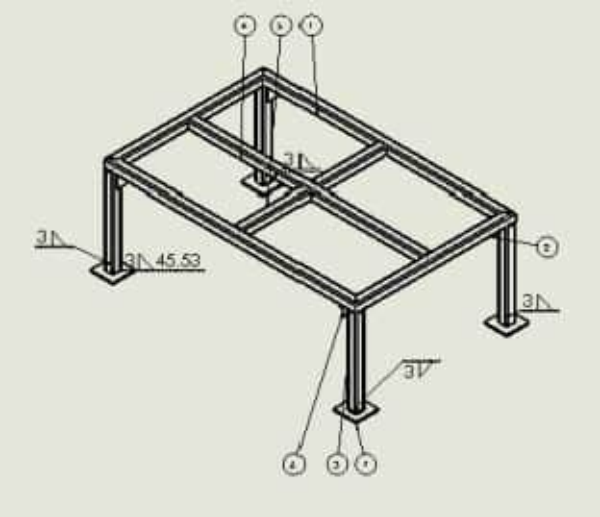
You can also add weld symbols and other annotations to the model view on the drawing.

Adding a Weld Symbols

1. To add weld symbols or other annotations, click **Annotation > Model Items** on the Command Manager.
2. On the **Model Items** Property Manager, set the **Source** to **Entire model**.
3. Deselect the **Marked for drawing** icon.
4. Select the icons from the **Annotations** section, as shown.



5. Click **OK**. The annotations are displayed on the model view.

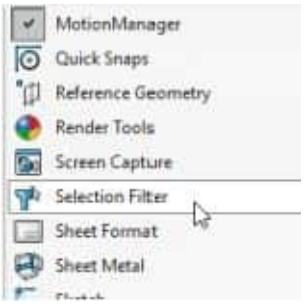


5. Save the Part document and the drawing document.

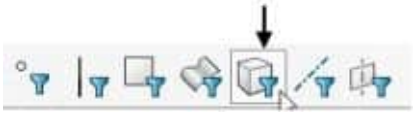
Creating Sub Weldments

You can break down a weldment into sub weldments to make it easy to manage them.

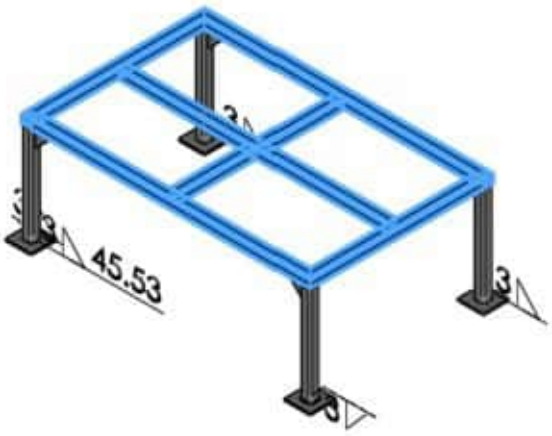
1. Open a Part document created in Tutorial 2.
2. Right-click on the Command Manager and select **Selection Filter**. The **Selection Filter** toolbar appears at the bottom.



3. Click the **Filter Solid bodies** button on the **Selection Filter** toolbar.

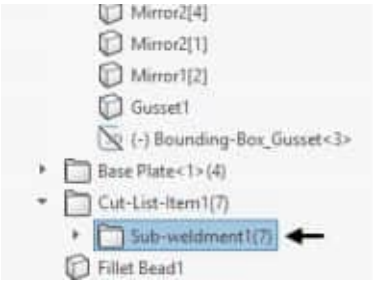


4. Press the Ctrl key and select the structural members as shown.



5. Right-click and select **Create Sub-Weldment**.
6. Expand the **Cut list** folder in the FeatureManager Design Tree.

You will notice a new folder under the cut list folder.

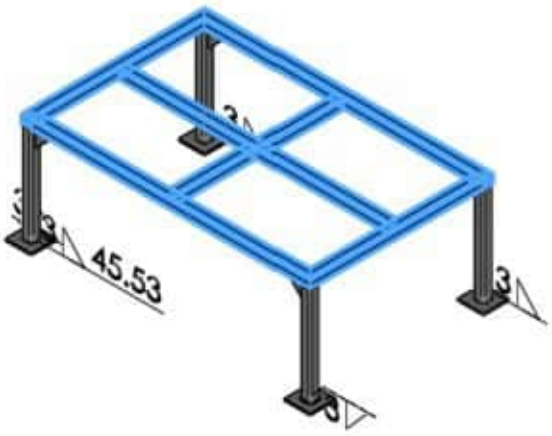


7. Open the drawing document of the weldments model. Place the cut list table on the drawing sheet.

ITEM NO.	QTY.	DESCRIPTION	LENGTH	Weight
1	1			
1	4	TUBE SQUARE 1.1 8 X 1.16 X 0.10	11.22	
3	4			46
4	4	Base Plate		

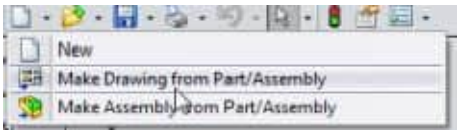
You will notice that the **Description** and **Length** values are empty in the cut list. This is because sub weldments are not recognized as weldment parts. You need to save the sub weldments as a separate weldment part.

8. Switch to the part document.
9. Press the Ctrl key and select the structural members as shown.

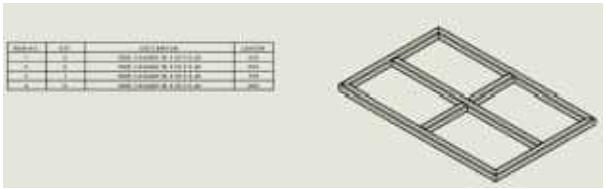


0. Click the right mouse button and select **Insert into New Part**.
1. On the PropertyManager, check the **Cut list properties to** option and select **Cut list properties**. This option allows you to insert the cut list table into the drawing.
2. Click **OK** on the PropertyManager.
3. Save the file as Tutorial 2-Sub-weldment1.

8. Create a drawing by selecting **New > Make Drawing from Part/Assembly**.



4. Insert the isometric view and cut list table into the drawing.

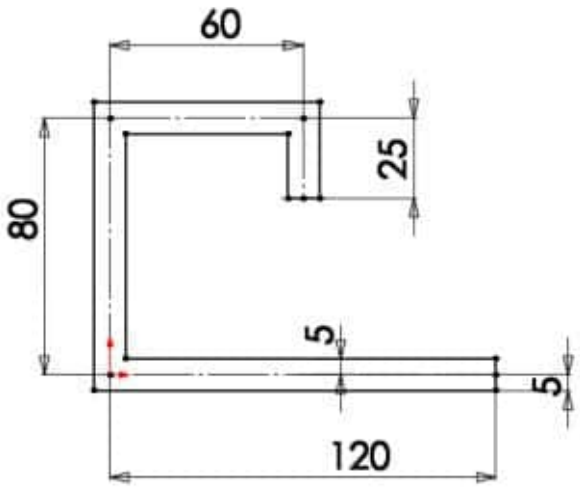


5. Save it as a separate part file.

TUTORIAL 5 (Creating Custom Profiles for structural members)

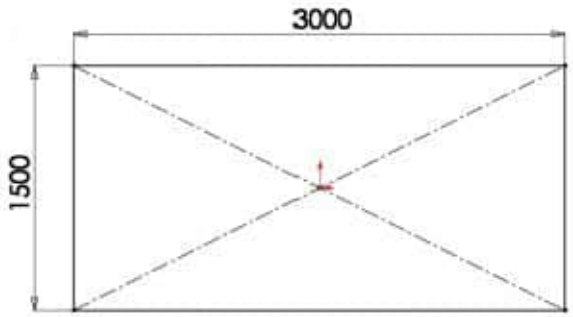
If you want to create a structural member using a profile that is not available in SOLIDWORKS, you can create a custom profile.

1. Close the SOLIDWORKS application, if opened.
2. Run SOLIDWORKS as administrator.
3. Open a new Part document.
4. Create a sketch as shown in figure.



5. Exit the sketch.
6. On the Quick Access Toolbar, click the **Options** icon.
7. On the **System Options** dialog, click **File Locations** in the tree.
8. Select **Weldment Profiles** from the **Show folders for** drop-down.
9. See the location of the weldment profiles and click **OK**.
10. Select the sketch from the FeatureManager Design Tree.
11. Click **File > Save As** on the Menu bar; the **Save As** dialog appears.
12. On this dialog, set the **Save as type** as **Lib Feat Part (*.sldlfp).**
13. Browse to the location of the weldment profiles.
14. Create the **Custom Profiles** folder.

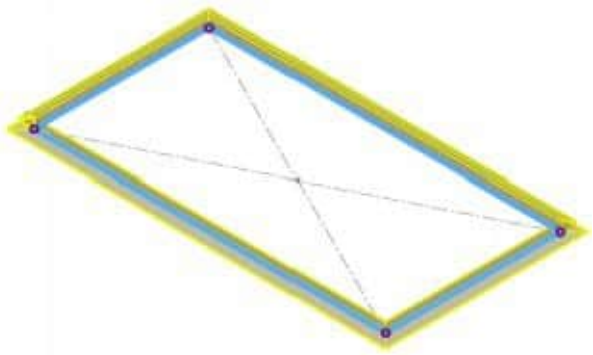
15. Create a sub folder with the name **C-Shape** in the **Custom Profiles** folder.
16. Save the file in the **C-Shape** folder as **C120.**
17. Open a new Part document and create a sketch as shown in figure.



18. Exit the sketch.

9. Click **Weldments > Structural Member** on the Command Manager.
0. On the Property Manager, make the selections, as shown in figure.

1. Select the line segments from the sketch. The structural members are created.



You can also click the **Locate Profile** button and change the position of the profile.

2. Click **OK**.

3. Save the file.