

Chapter 1

Surface Modeling

Learning Objectives

After completing this chapter, you will be able to:

- *Create extruded, revolved, and swept surfaces*
- *Create lofted, planar, and boundary surfaces*
- *Create fill and radiated surfaces*
- *Extend, trim, and untrim surfaces*
- *Offset, fillet, and knit surfaces*
- *Create a Mid-surface*
- *Delete holes*
- *Replace and delete faces*
- *Move and copy surfaces*
- *Thicken a surface body*
- *Create a thickened surface cut*
- *Create a surface cut*

SURFACE MODELING

Surface modeling is a technique of creating planar or non-planar geometry of zero thickness. This zero thickness geometry is known as surface. Surfaces are generally used to create models of complex shapes. You can easily convert surface models into solid models. You can also extract a surface from a solid model using the surface modeling tools. This chapter deals with the surface modeling tools in SOLIDWORKS. Using these tools, you can create complex shapes as surfaces and then convert them into solid models, if required.

Most of the real world components are created using solid modeling. But sometimes, you may need to create some complex features that can only be created by surface manipulation. Surface manipulation is done by using surface modeling tools. After creating the required complex surface, you can convert it into a solid model. The reasons to convert a surface model into a solid model are that a surface is a zero-thickness geometry and it has no mass and mass properties. But, while designing real world models, you may need mass and mass properties. The other reason is that you can generate a section view only if the model is a solid.

In SOLIDWORKS, surface modeling is done in the **Part** mode and the tools used for surface modeling are available in the **Surfaces CommandManager**. The **Surfaces CommandManager** will not be available by default. Therefore, you need to right-click on any one of the **CommandManager** tabs and then choose the **Tabs > Surfaces** option from the shortcut menu. The surface modeling tools can also be invoked by choosing **Insert > Surface** from the SOLIDWORKS menus. You will notice that some of the tools available in the **Surfaces CommandManager**, such as extrude, revolve, sweep, and loft are similar to those discussed in the solid modeling.

The tools in the **Surfaces CommandManager** and the other advanced surface modeling tools are discussed next.

Creating an Extruded Surface

CommandManager: Surfaces > Extruded Surface
SOLIDWORKS menus: Insert > Surface > Extrude
Toolbar: Surfaces > Extruded Surface



In SOLIDWORKS, the **Extruded Surface** tool is used to extrude a closed or an open sketch for creating an extruded surface. To create an extruded surface, create a sketch in the sketching environment and then choose the **Extruded Surface** button from the **Surfaces CommandManager**; the **Surface-Extrude PropertyManager** will be displayed, as shown in Figure 1-1. Also, the preview of the extruded surface with the default values will be displayed in the drawing area. To define feature termination, select the required option from the **End Condition** drop-down list in the PropertyManager. The feature termination options are available in the **Direction 1** and **Direction 2** rollouts. The other options in this PropertyManager are the same as those discussed in part modeling. You can also define the extrusion depth dynamically by dragging the handle displayed in the drawing area. In

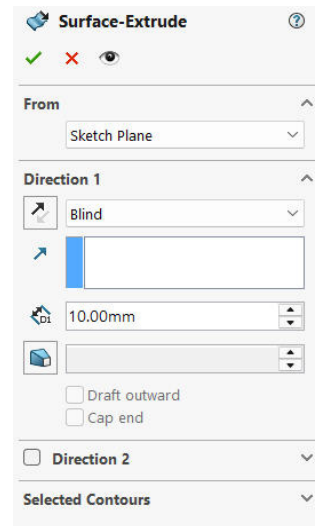


Figure 1-1 Partial view of the *Surface-Extrude PropertyManager*

SOLIDWORKS, you can extrude a 2D face such that all its edges are extruded and it results into an extruded surface. To extrude a 2D face, choose the **Extruded Surface** tool without drawing a sketch. Next, press the ALT key and select the surface; all edges of the selected 2D surface will be extruded. You can also select the surface of a solid model. If you do so, the edges of the model will be extruded as surfaces.

Figure 1-2 shows a closed sketch and Figure 1-3 shows the surface created by extruding the closed sketch. Figure 1-4 shows an open sketch and Figure 1-5 shows the surface created by extruding that open sketch.

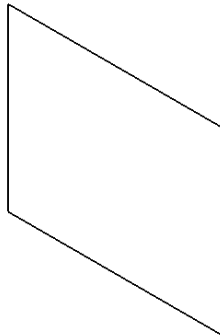


Figure 1-2 A closed sketch

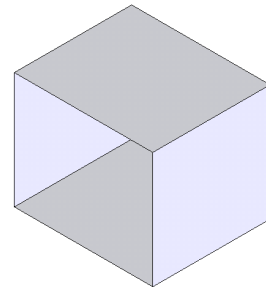


Figure 1-3 Surface created by extruding the closed sketch

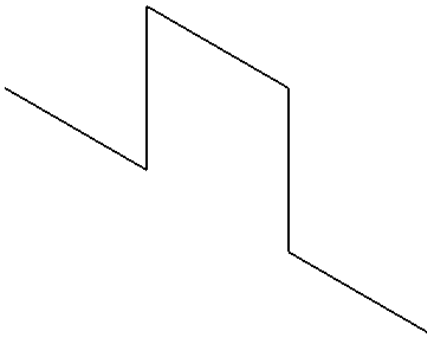


Figure 1-4 An open sketch

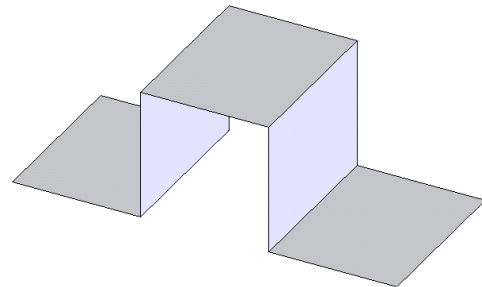


Figure 1-5 Surface created by extruding the open sketch

Creating a Revolved Surface

CommandManager: Surfaces > Revolved Surface
SOLIDWORKS menus: Insert > Surface > Revolve
Toolbar: Surfaces > Revolved Surface



You can also create a surface by revolving a closed or an open sketch along a centerline. Revolving a sketch along a centerline to create a revolved surface is similar to revolving a sketch along a centerline to create a solid feature. To create a revolved surface, first create a sketch and a centerline in the sketching environment. Next, choose the **Revolved Surface** button from the **Surfaces CommandManager**; the **Surface-Revolve PropertyManager** will be displayed, as shown in Figure 1-6. Also, preview of the revolved surface with the drag handle will be displayed in the drawing area. The feature termination options and other options in this PropertyManager are similar to those discussed while creating a solid revolved feature.

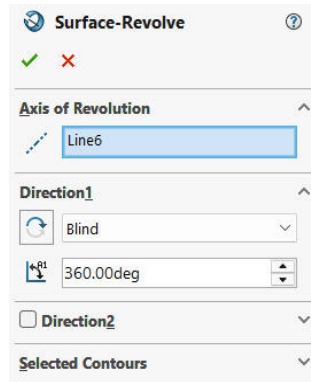


Figure 1-6 The *Surface-Revolve PropertyManager*

Figure 1-7 shows an open sketch for creating a revolved surface. Figure 1-8 shows the revolved surface created by revolving the sketch through an angle of 270 degrees.

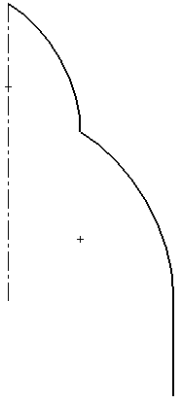


Figure 1-7 Sketch for creating a revolved surface

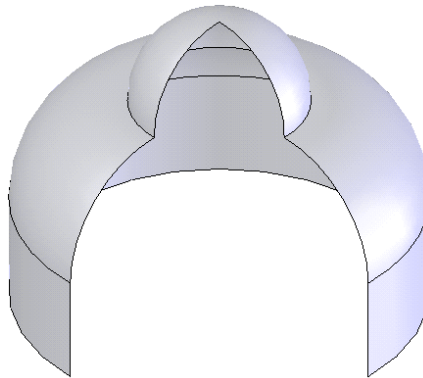


Figure 1-8 Surface created by revolving the sketch through an angle of 270 degrees

Creating a Swept Surface

CommandManager: Surfaces > Swept Surface
SOLIDWORKS menus: Insert > Surface > Sweep
Toolbar: Surfaces > Swept Surface



You can also create a swept surface by sweeping a closed or an open profile along a closed or an open path. To create a sketch profile, first draw a closed or an open sketch as a sweep profile and another sketch as a sweep path in the sketching environment. Next, choose the **Swept Surface** button from the **Surfaces CommandManager**; the **Surface-Sweep PropertyManager**

will be displayed, as shown in Figure 1-9. Also, you will be prompted to select a sweep profile. Select a closed sketch or an open sketch as the profile of the sweep feature; you will be prompted to select the sweep path. Select a closed or an open sketch as the sweep path. On doing so, the preview of the sweep feature will be displayed in the drawing area.

To create a circular swept surface using the **Circular Profile** option, you need to draw only a closed or an open sketch for sweep path in the sketching environment.

All other options used to create a swept surface are similar to those discussed while creating the solid sweep feature.

You can also select guide curves while creating the sweep surface.

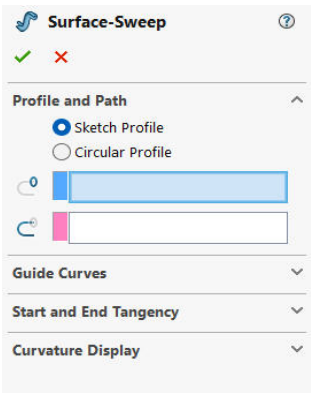


Figure 1-9 The Surface-Sweep PropertyManager

Figures 1-10 through 1-19 illustrate various ways to create a sweep feature. Figure 1-10 shows an open profile and an open path and Figure 1-11 shows the resultant sweep surface. Figure 1-12 shows a closed profile and an open path and Figure 1-13 shows the resultant sweep surface. Figure 1-14 shows an open profile and a closed path and Figure 1-15 shows the resultant sweep surface. Figure 1-16 shows a closed path for circular sweep and Figure 1-17 shows the resultant sweep surface. Figure 1-18 shows a profile, a path, and three guide curves and Figure 1-19 shows the resultant sweep surface.

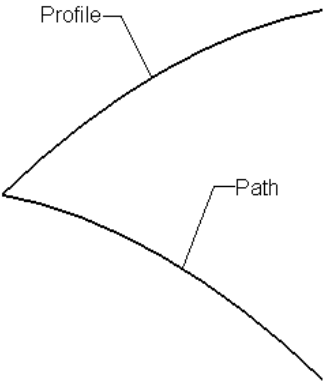


Figure 1-10 An open profile and an open path

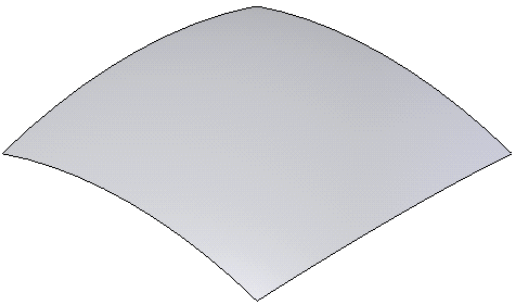


Figure 1-11 Resultant sweep surface

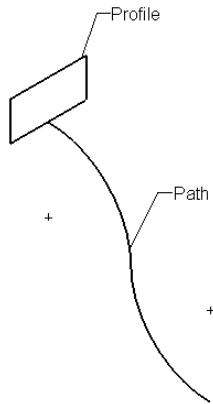


Figure 1-12 A closed profile and an open path

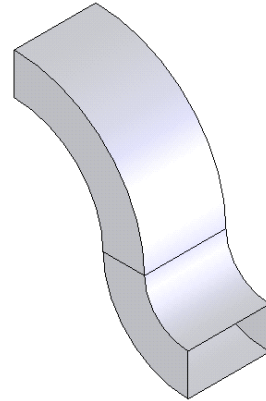


Figure 1-13 Resultant sweep surface

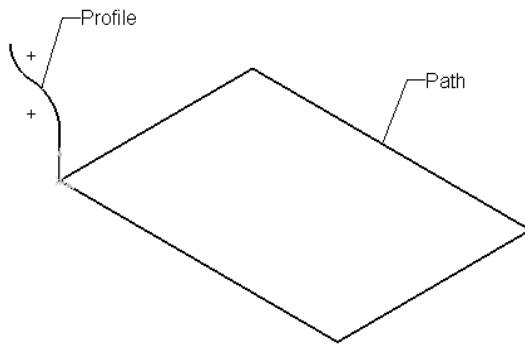


Figure 1-14 An open profile and a closed path

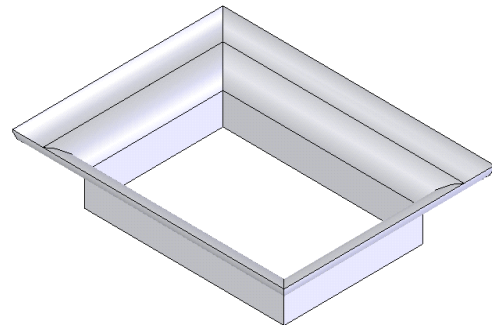


Figure 1-15 Resultant sweep surface

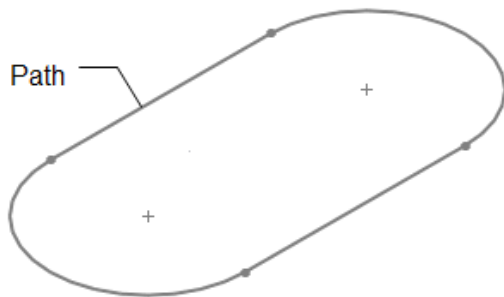


Figure 1-16 A closed path for circular sweep

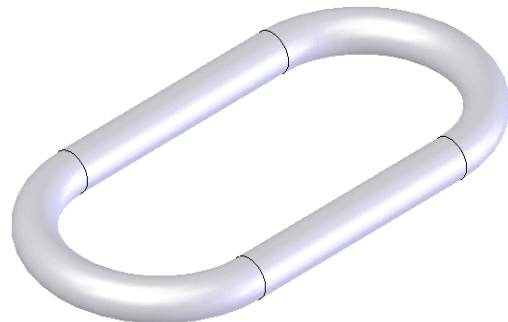


Figure 1-17 Resultant sweep surface

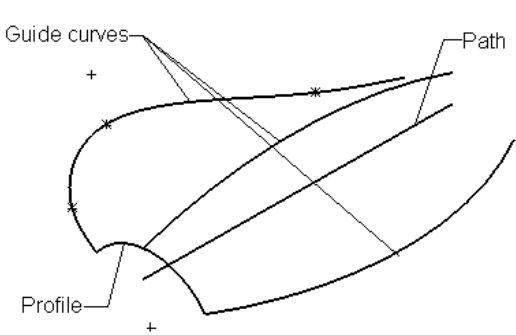


Figure 1-18 Profile, path, and guide curves

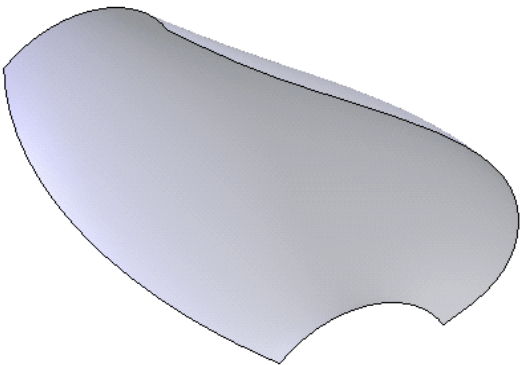



Figure 1-19 Resultant sweep feature

Creating a Lofted Surface

CommandManager:	Surfaces > Lofted Surface
SOLIDWORKS menus:	Insert > Surface > Loft
Toolbar:	Surfaces > Lofted Surface

 In SOLIDWORKS, you can also create a surface by lofting two or more sections. To create a lofted surface, choose the **Lofted Surface** button from the **Surfaces CommandManager**; the **Surface-Loft PropertyManager** will be displayed, as shown in Figure 1-20, and you will be prompted to select at least two profiles. Select the profiles to be lofted. All the options for creating a lofted surface are similar to those discussed while creating a solid lofted feature.

Note that if you want to create a lofted surface with open section, all the sections to be lofted must be opened. Similarly, if you want to create a closed lofted surface, all the sections must be closed. This means that in a lofted surface, the combination of closed and opened sections is not possible. Figure 1-21 shows the two open sections to be lofted and Figure 1-22 shows the resultant lofted surface.

Figure 1-23 shows two closed sections to be lofted and Figure 1-24 shows the resultant lofted surface. Figure 1-25 shows two sections and a centerline and Figure 1-26 shows the resultant lofted surface.

Figure 1-27 shows two sections and guide curves and Figure 1-28 shows the resultant lofted surface.

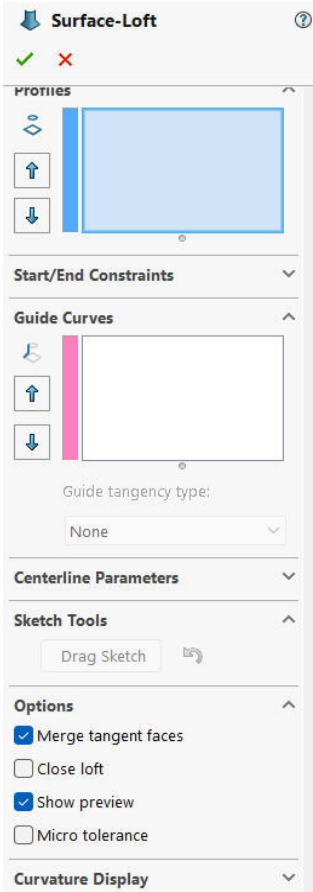


Figure 1-20 Partial view of the Surface-Loft PropertyManager

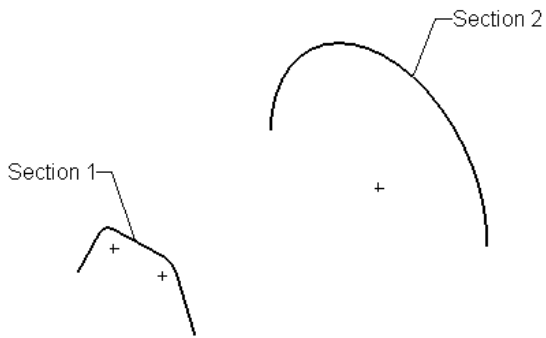


Figure 1-21 Open sections

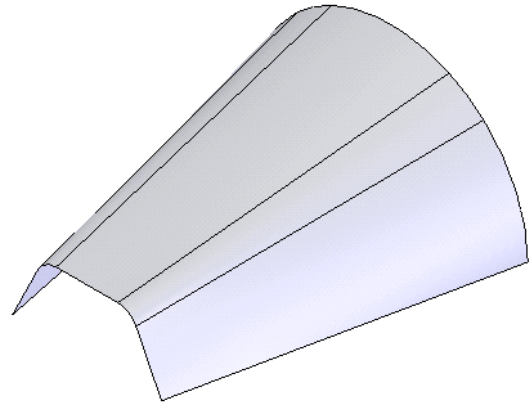


Figure 1-22 Resultant lofted surface

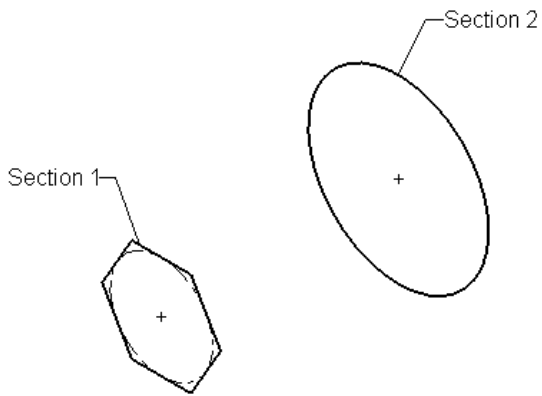


Figure 1-23 Closed sections

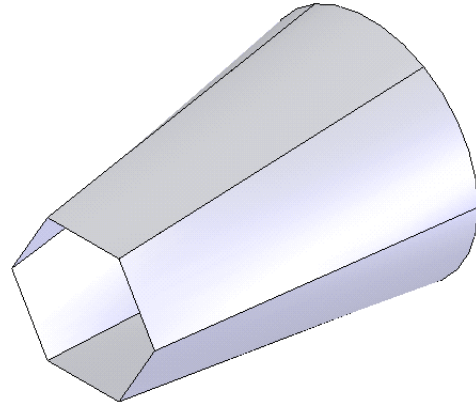


Figure 1-24 Resultant lofted surface

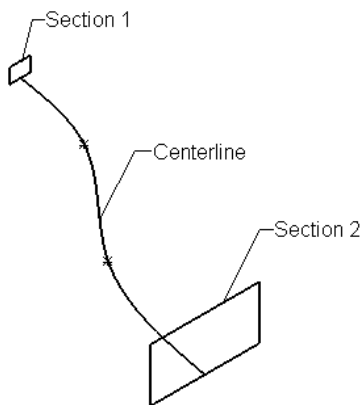


Figure 1-25 Sections and centerline

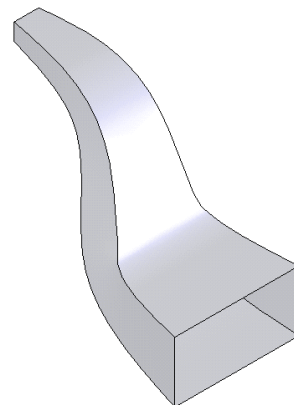


Figure 1-26 Resultant lofted surface

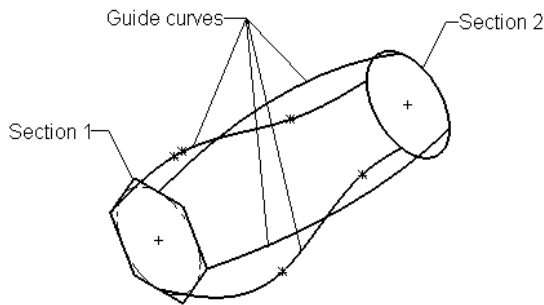


Figure 1-27 Sections and guide curves

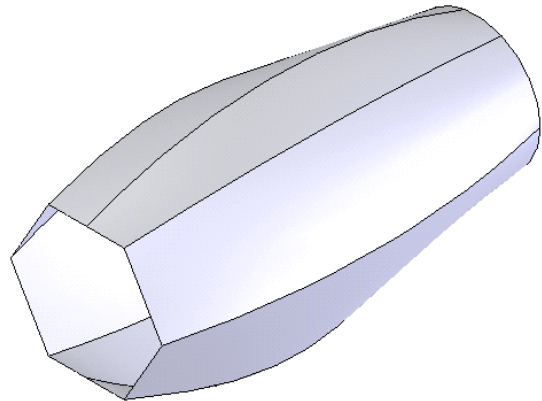


Figure 1-28 Resultant lofted surface

Creating a Boundary Surface

CommandManager: Surfaces > Boundary Surface
SOLIDWORKS menus: Insert > Surface > Boundary Surface
Toolbar: Surfaces > Boundary Surface



The **Boundary Surface** tool is used to create complex models with high accuracy as well as high surface quality, while maintaining the curvature continuity. To create a boundary surface, choose the **Boundary Surface** button from the **Surfaces CommandManager**; the **Boundary-Surface PropertyManager** will be displayed, as shown in Figure 1-29 and you will be prompted to select the profiles for the boundary surface. Select the curves from the drawing area; the selected curves will be displayed in the **Direction 1** rollout of the PropertyManager and the preview will be displayed in the drawing area. To select the curves for the **Direction 2** rollout, click in the **Curves** selection box of the **Direction 2** rollout and then select the curves from the drawing area; the selected curves will be displayed in the **Curves** selection box. The other options in this PropertyManager are discussed next.

Direction 1 Rollout

The **Direction 1** rollout is used to control the tangency and curvature continuity of curves in direction 1. The boundary surface is created based on the order of curves selected from the drawing area. You can change the order of curves in the **Curves** selection box by choosing the **Move Up** and **Move Down** buttons. The options that affect curves in the direction 1 are discussed next.

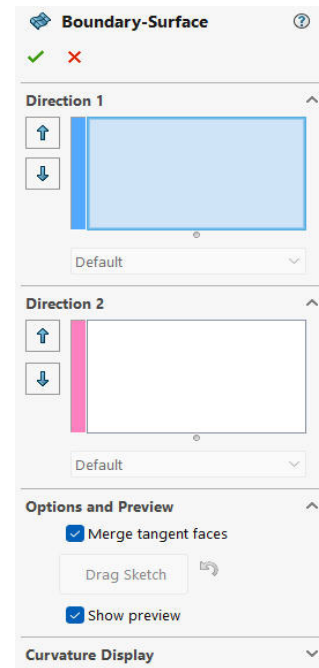


Figure 1-29 The Boundary-Surface PropertyManager

Tangent Type

The **Tangent Type** drop-down list is used to display options that control the tangency of the curvature. The options in this drop-down list are discussed next.

None: The **None** option is used to apply zero curvature or no tangency constraint to curves.

Direction Vector: The **Direction Vector** option is used to apply tangency constraint to curves. When you select the **Direction Vector** option from the **Tangent Type** drop-down list, the **Alignment** drop-down list and the **Direction Vector** selection box will be displayed below the **Tangent Type** drop-down list. Select the required alignment option from the **Alignment** drop-down list and then select the direction based on the selected curves. You can also specify the draft angle and tangent length for curves in the **Draft angle** and **Tangent Length** spinners, respectively.

Default: The **Default** option will be available in the **Tangent Type** drop-down list, only when at least three curves are selected in one direction.

Normal To Profile: The **Normal To Profile** option is used to apply the tangency constraint normal to the selected curves. You can also set the draft angle and tangent length for curves using this option.

Tangency To Face: This option will be available in the **Tangent Type** drop-down list only when you select the edges of existing surfaces as boundary curve. Select this option to make the surface tangent to the existing surface at the selected boundary curve. Select the required alignment option from the **Alignment** drop-down list available below the **Tangent Type** drop-down list. The options in the **Alignment** drop-down list control the flow of the boundary surface.

Curvature To Face: This option will be available in the **Tangent Type** drop-down list only when you select the edges of existing surfaces as boundary curve. This option makes the surface smoother and curvature continuous to the existing surface at the selected boundary curve. Select the required alignment option from the **Alignment** drop-down list available below the **Tangent Type** drop-down list. The options in the **Alignment** drop-down list control the flow of the boundary surface.

Direction 2 Rollout

The options in the **Direction 2** rollout are the same as those discussed in the **Direction 1** rollout.

Curve Influence Type

The **Curve Influence Type** drop-down list will be displayed in the **Direction 1** and **Direction 2** rollouts only when you select a curve for the second direction. The options available in this drop-down list are discussed next.

Global: The **Global** option is selected by default in this drop-down list. This option is used to extend the curve influence up to the entire boundary feature.

To Next Curve: The **To Next Curve** option is used to extend the curve influence up to the next curve only.

To Next Sharp: The **To Next Sharp** option is used to extend the curve influence up to the next sharp only. Sharp is a hard corner of the sketch entity. This option is applicable between two sketch entities that do not have a tangency and curvature relation with each other.

To Next Edge: The **To Next Edge** option is used to extend the curve influence up to the next edge only.

Linear: The **Linear** option is used to extend the curve influence linearly up to the entire boundary feature.



Note

*While selecting curves from the drawing area, select a point on the curve such that it follows the required path of the boundary feature. The selected points on the curves act as connectors of the boundary feature. You can also flip the boundary feature connectors. To do so, right-click in the drawing area; a shortcut menu will be displayed. Choose the **Flip Connectors** option from the shortcut menu to flip the direction of connectors.*

Options and Preview Rollout

You can merge the tangent faces of a boundary feature by selecting the **Merge tangent faces** check box available in this rollout. To separate the tangent faces of the boundary feature, you need to clear this check box. If you have selected the curves that lie in two different directions, then you can trim the surfaces up to the curve(s) by selecting the **Trim by direction 1** and **Trim by direction 2** check boxes. The **Close surface** check box is selected to create a closed surface. Note that the angle between the start and end sections should be more than 180 degrees to create the closed surface and also there should be at least three sections. To view the preview of the boundary feature, select the **Show preview** check box available in this rollout.

Curvature Display Rollout

The **Curvature Display** rollout is used to display the mesh preview, zebra stripes, and curvature combs of the boundary feature. The **Mesh preview** and **Curvature combs** check boxes are selected by default in the **Curvature Display** rollout of the **Boundary-Surface PropertyManager**. The **Mesh preview** check box allows you to toggle the mesh preview of the boundary surface. You can increase or decrease the number of lines of the mesh by using the **Mesh density** spinner available below the **Mesh preview** check box in this rollout. By selecting the **Zebra stripes** check box, you can visually determine the type of boundary existing between surfaces such as contact, tangency, and curvature continuous. Using the **Zebra stripes** check box, you can also identify wrinkles or defects in surfaces. On selecting the **Curvature combs** check box, you can visualize the continuity of the curve and also get a better idea of the quality of the surfaces that will be generated. It also helps you to magnify discontinuities in a curve. The **Direction 1** and **Direction 2** check boxes, available below the **Curvature combs** check box in this rollout, are used to toggle the display of curvature combs along the direction 1 and direction 2. You can also adjust the scale and density of curvature combs by using the **Curvature Comb Scale** and **Curvature Comb Density** spinners, respectively.

Figure 1-30 shows three curves for creating a boundary surface in direction 1 and Figure 1-31 shows the resultant boundary surface. Figure 1-32 shows six curves for creating a boundary surface in direction 1 and direction 2 and Figure 1-33 shows the resultant boundary surface.

Figure 1-34 shows the direction 2 curves extended beyond the direction 1 curves and Figure 1-35 shows the resultant preview of the boundary surface without trimming the direction 2 curves. Figure 1-36 shows the resultant preview of the boundary surface after trimming the direction 2 curves by using the direction 1 curves.

Figure 1-37 shows two sketches for creating a boundary surface in direction 1 and Figure 1-38 shows the resultant boundary surface with merge tangent faces. Figure 1-39 shows the resultant boundary surface without merge tangent faces.

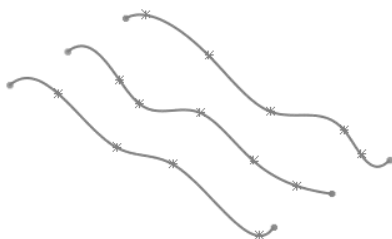


Figure 1-30 Three curves for creating a boundary surface in direction 1

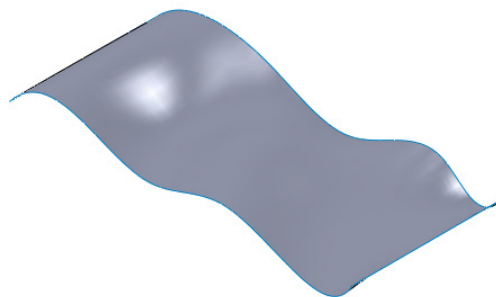


Figure 1-31 Resultant boundary surface

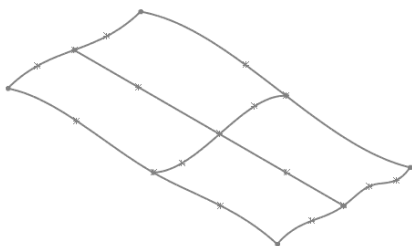


Figure 1-32 Curves for creating a boundary surface in direction 1 and direction 2

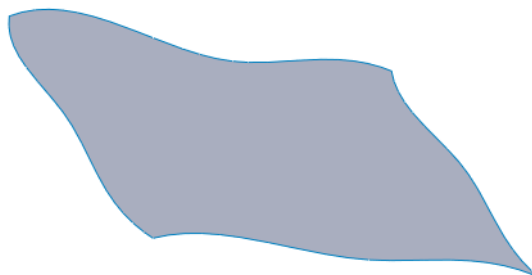


Figure 1-33 The resultant boundary surface

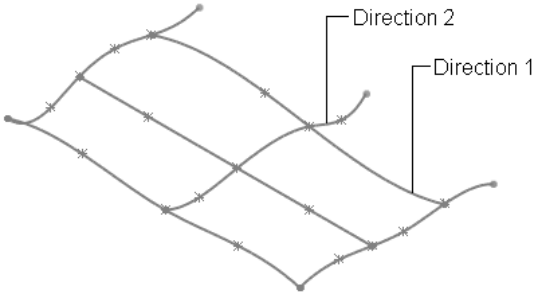


Figure 1-34 The direction 2 curves extended beyond the direction 1 curves

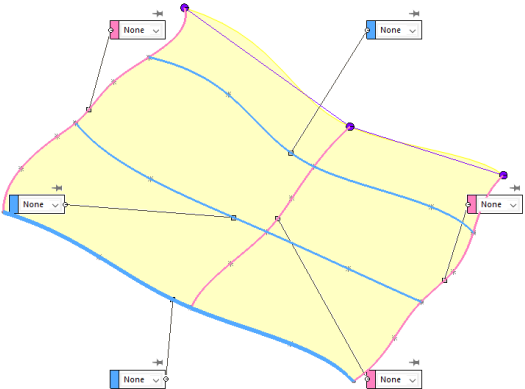


Figure 1-35 Preview of the boundary surface without trimming the direction 2 curves

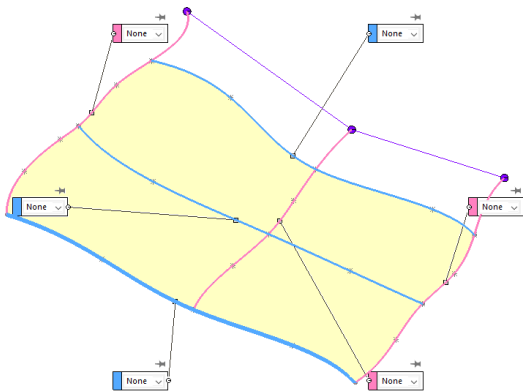


Figure 1-36 Preview of the boundary surface after trimming the direction 2 curves by using the direction 1 curves

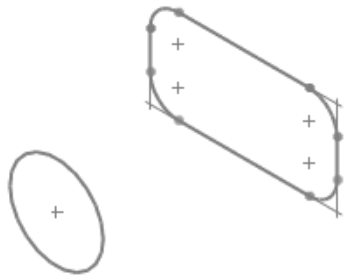


Figure 1-37 The sketches for creating a boundary surface

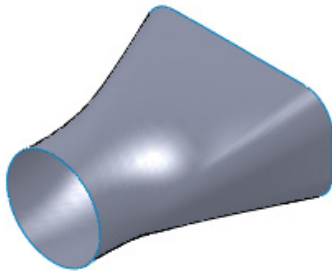


Figure 1-38 The boundary surface with merge tangent faces

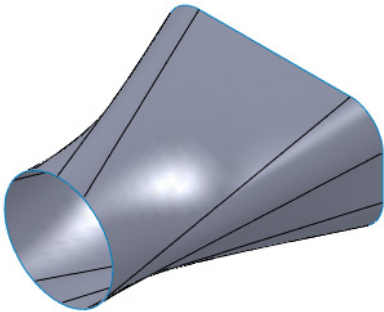



Figure 1-39 The boundary surface without merge tangent faces

Creating a Planar Surface

CommandManager: Surfaces > Planar Surface
SOLIDWORKS menus: Insert > Surface > Planar
Toolbar: Surfaces > Planar Surface

 A planar surface is generally used to fill gaps between surfaces using a planar patch. To create a planar surface, choose the **Planar Surface** button from the **Surfaces CommandManager** or choose **Insert > Surface > Planar** from the SOLIDWORKS menus; the **Planar Surface PropertyManager** will be displayed, as shown in Figure 1-40, and you will be prompted to select the bounding entities such as a sketch, an edge, or a curve.

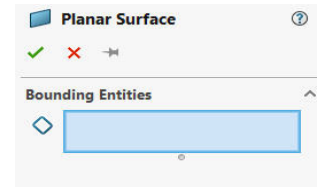


Figure 1-40 The Planar Surface PropertyManager

Select the bounding entities; the names of the bounding entities will be displayed in the **Bounding Entities** rollout. Next, choose the **OK** button from the **Planar Surface PropertyManager**; a planar surface will be created using the selected entities. Note that the bounding entities should be coplanar.

Figure 1-41 shows the bounding entities to be selected for creating a planar surface and Figure 1-42 shows the resultant planar surface.

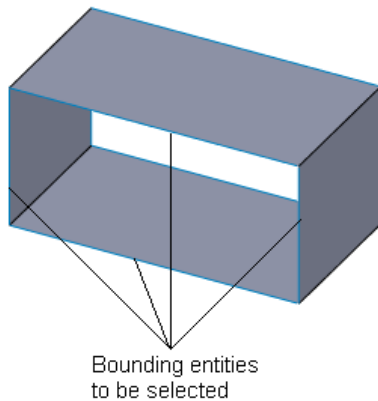


Figure 1-41 Bounding entities to be selected

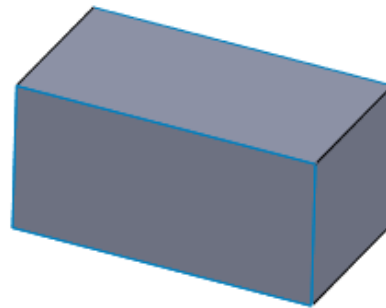



Figure 1-42 The resultant planar surface

Creating a Fill Surface

CommandManager: Surfaces > Filled Surface
SOLIDWORKS menus: Insert > Surface > Fill
Toolbar: Surfaces > Filled Surface

 The **Filled Surface** tool is used to create a surface patch along N number of sides. The sides to be selected for creating a filled surface can be the edges of the existing model, 2D or 3D sketch entities, or 2D or 3D curves. The difference between a planar surface and a fill surface is that you cannot create a planar surface using 3D curves or edges. For example, the 3D edge created in Figure 1-43 cannot be used to create a planar surface. But, you can fill this gap by selecting the 3D edge and creating a fill surface.

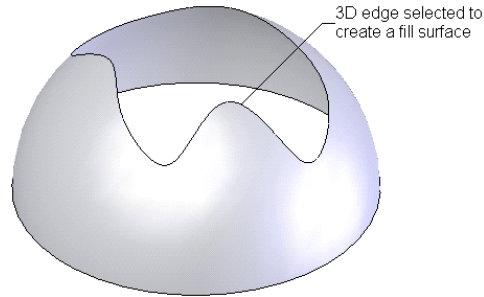


Figure 1-43 Entity selected for creating a fill surface

To create a fill surface, choose the **Filled Surface** button from the **Surfaces CommandManager**; the **Fill Surface PropertyManager** will be displayed, as shown in Figure 1-44, and you will be prompted to select bounding entities and set the required options. Select the entities that will define the boundary; the selected entities will be displayed in different color, and callouts will be attached to them. On selecting the last entity that will close the current selection chain, the preview of the fill surface along with the mesh will be displayed in the drawing area. Now, choose the **OK** the button from the **Fill Surface PropertyManager**. Figure 1-45 shows preview of the fill surface along with the mesh and Figure 1-46 shows the resultant fill surface.



Note

The surface model used in this example is created by trimming a surface. You will learn about trimming the surfaces later in this chapter.

The other options in the **Fill Surface PropertyManager** are discussed next.

Edge settings

The options in the **Edge settings** area are used to define various parameters to specify references with respect to the selected edges, type of curvature, and so on. These options are discussed next.

Alternate Face

The **Alternate Face** button in the **Edge settings** area is used to specify the face reference to be included while creating a fill surface for controlling the curvature of the fill surface. This option is only used when you are creating a fill surface on a solid body.

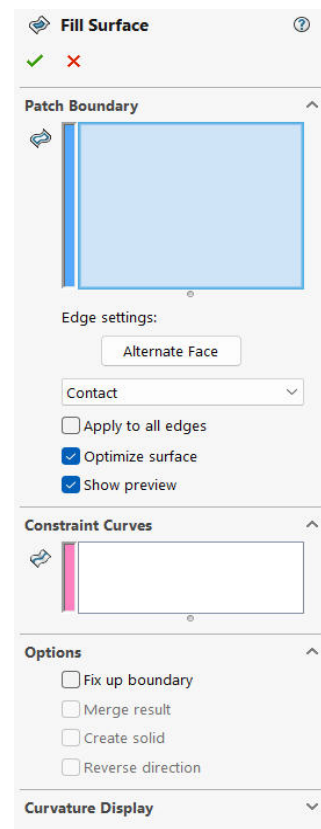


Figure 1-44 The **Fill Surface PropertyManager**

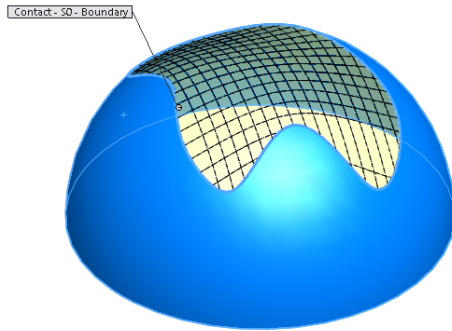


Figure 1-45 Preview of the fill surface along with the mesh

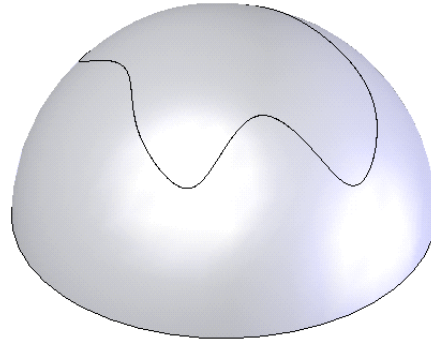


Figure 1-46 Resultant fill surface

Curvature Control

The **Curvature Control** drop-down list is used to define the type of curvature that you need to apply on the fill surface. There are different types of curvatures in this drop-down list that are discussed next.

Contact

The **Contact** option is selected by default and is used to create a patch using the fill surface option within the selected patch boundary.

Tangent

The **Tangent** option is selected to create a patch such that the resulting patch maintains tangency with the selected edges. On selecting this option for creating a patch, the **Reverse Surface** button is also displayed, if there is a possibility of creating a patch in the other direction. Choose this button to reverse the direction of the surface created.

Curvature

The **Curvature** option is selected to create a patch such that the resulting patch maintains curvature continuity with the selected edge.

Figure 1-47 shows the circular edge selected as the patch boundary. Figure 1-48 shows the fill surface created with the **Contact** option selected in the **Curvature Control** drop-down list. Figure 1-49 shows the fill surface created with the **Tangent** option selected in the **Curvature Control** drop-down list.

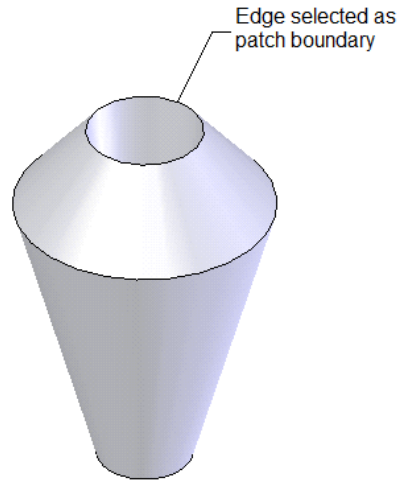


Figure 1-47 Edge selected as the patch boundary

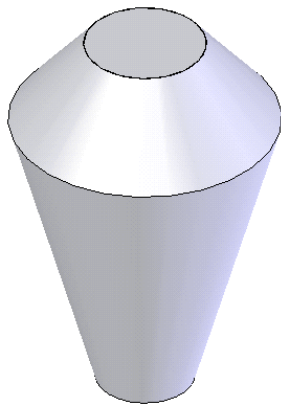


Figure 1-48 Fill surface created using the Contact option

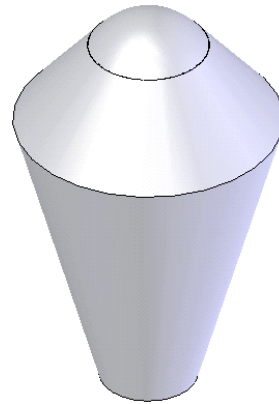


Figure 1-49 Fill surface created using the Tangent option

Apply to all edges

This option is used to apply curvature settings to all edges. If this check box is not selected, the current curvature setting will only be applied to the edge of the boundary that is selected in the **Patch Boundaries** selection box.

Optimize surface

The **Optimize surface** check box is selected to create a simple patch of a surface along the selected patch boundary. This check box is selected by default. Therefore, if you create a surface patch, the time taken to create a surface will be less and the model will be rebuilt faster. When you clear this check box, the **Resolution Control** rollout will be displayed, as shown in Figure 1-50. The slider available in this rollout is used to specify the resolution of the fill surface. If you specify higher resolution, the quality of the fill surface will be better

but it will take more time in rebuilding. In case of lower resolution, the quality of the surface will not be good. However, in such a case, the rebuilding of the model will take lesser time.

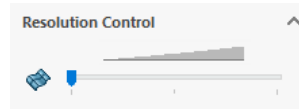


Figure 1-50 The *Resolution Control* rollout

Show preview

The **Show preview** check box is selected by default and is used to display the preview of the fill surface that is created using the selected patch boundary.

Reverse Surface

Choose the **Reverse Surface** button to change the direction of filling the surface. You can reverse the direction of the fill surface only when it has the tangency or curvature continuity with the existing surface to be patched. To change the tangency or curvature continuity of a fill surface, select the **Tangent** or **Curvature** option from the **Curvature Control** drop-down list in the **Edge settings** area of the **Patch Boundary** rollout.

Constraint Curves

The **Constraint Curves** rollout is used to define constraint curves while creating a fill surface. To create a fill surface using constraint curves, invoke the **Fill Surface PropertyManager** and then select the patch boundary. Now, click once in the **Constraint Curves** selection box of the **Constraint Curves** rollout to invoke the selection mode and then select the constraint curves. Note that the constraint curves to be selected can be sketched entities, an edge, or a curve. The selected constraint curve will be displayed in a different color and a callout will be attached to it. Also, the name of the selected entity will be displayed in the **Constraint Curves** selection box. When you select the constraint curves, the preview of the fill surface gets modified. After specifying all constraint curves, choose the **OK** button from the **Fill Surface PropertyManager**. Figure 1-51 shows the sketch and the constraint curves selected for patching the boundary. Figure 1-52 shows the resultant fill surface.

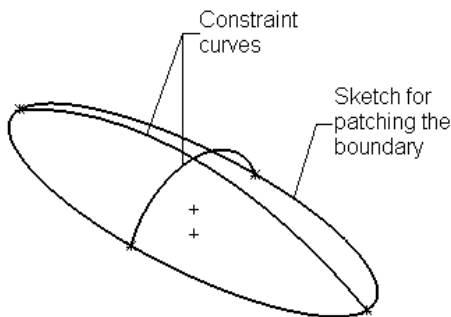


Figure 1-51 Sketch and constraint curves to be selected

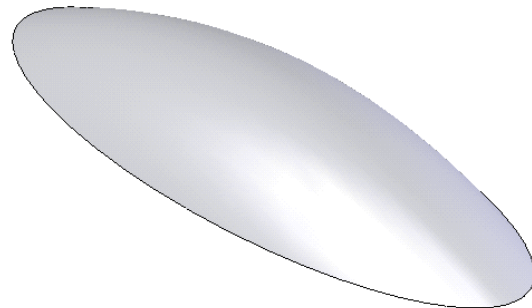


Figure 1-52 Resultant fill surface

Creating a Radiated Surface

CommandManager:	Surfaces > Radiate Surface (Customize to Add)
SOLIDWORKS menus:	Insert > Surface > Radiate
Toolbar:	Surfaces > Radiate Surface (Customize to Add)



In SOLIDWORKS, you can also create a surface by radiating a surface along an edge or a split line. The radiated surface is always created parallel to the plane or the face selected as the radiate direction reference. This type of surface is generally used in mold design as parting surface for extracting the core and cavity. To create a radiated surface, choose the **Radiate Surface** button from the **Surfaces CommandManager**; the **Radiate Surface PropertyManager** will be displayed, as shown in Figure 1-53.

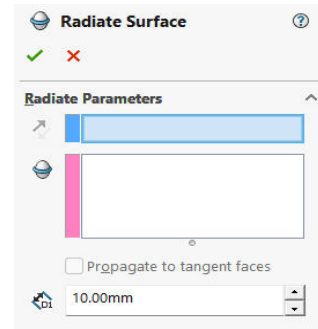


Figure 1-53 The **Radiate Surface PropertyManager**

You will observe that the **Radiate Direction Reference** selection box is activated by default in this PropertyManager. Therefore, first you need to select a plane or a planar face parallel to which the surface will be radiated. The selected reference will be highlighted in a different color and an arrow symbol normal to the selected face will be displayed. On selecting a face, the **Edges To Radiate** selection box will be activated. Now, select the edges along which the surface will be radiated; the names of the selected edges will be displayed in the **Edges To Radiate** selection box. Note that the arrows will be displayed in the drawing area, showing the direction in which the surface will be radiated. Now, set the value of the distance of the surface to be radiated in the **Radiate Distance** spinner. The **Propagate to tangent faces** check box is used to radiate surfaces along all the edges that are tangent to the selected edge. After setting all parameters, choose the **OK** button from the **Radiate Surface PropertyManager**.

Figure 1-54 shows the radiate direction reference and the edges to be selected. Figure 1-55 shows the resultant radiated surface with the **Propagate to tangent faces** check box selected.

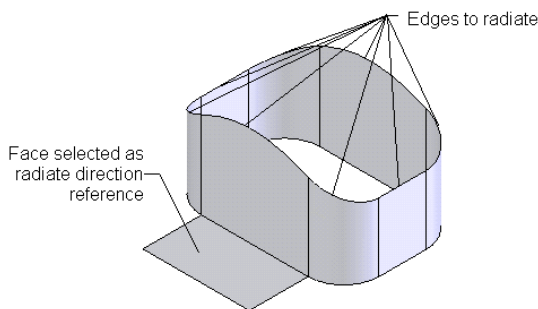


Figure 1-54 Reference and the edges to be selected

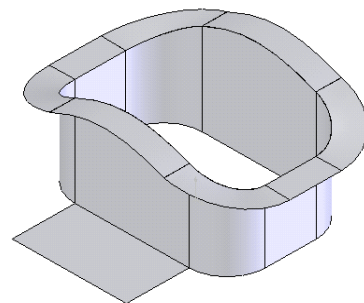


Figure 1-55 The radiated surface created with the **Propagate to tangent faces** check box selected

Offsetting Surfaces

CommandManager: Surfaces > Offset Surface
SOLIDWORKS menus: Insert > Surface > Offset
Toolbar: Surfaces > Offset Surface



The **Offset Surface** tool is used to offset a selected surface or surfaces to a given distance. To offset a surface, choose the **Offset Surface** tool from the **Surfaces CommandManager**; the **Offset Surface PropertyManager** will be displayed, as shown in Figure 1-56, and you will be prompted to select a face or a surface to offset.

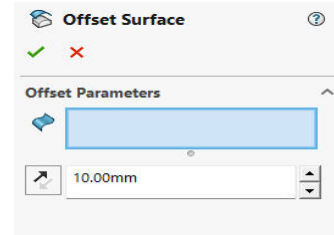


Figure 1-56 The *Offset Surface PropertyManager*

Now, select the face or the surface that you need to offset; the selected face or the surface will be highlighted in a different color and its name will be displayed in the **Surface or Faces to Offset** selection box. Also, preview of the offset surface with the default value will be displayed in the drawing area. Set the value of the offset distance using the **Offset Distance** spinner. You can flip the direction of the surface creation using the **Flip Offset Direction** button available on the left of the **Offset Distance** spinner. After setting all parameters, choose the **OK** button from the **Offset Surface PropertyManager**. Figure 1-57 shows the surface selected to offset and Figure 1-58 shows the resultant offset surface. If the **Offset Surface** tool fails to create offset, the **Offset Surface PropertyManager** lists and highlights the failing faces in the **Offset Parameters** area. Right-click on any of the failing faces then choose **Remove All Failing Faces** to remove all failing face from the short-cut menu.

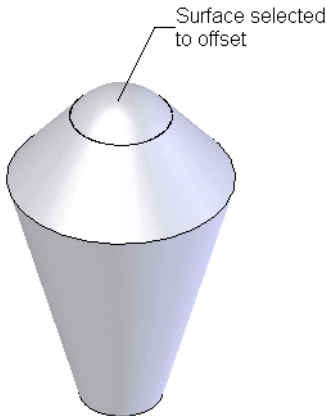


Figure 1-57 Surface selected to offset

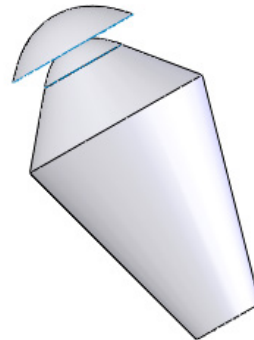


Figure 1-58 Resultant offset surface



Tip

If you want to extract a surface from a solid body or a surface body, invoke the **Offset Surface PropertyManager** and select the surfaces to be extracted. Next, enter **0** in the **Offset Distance** spinner; the **Offset Surface PropertyManager** turns to **Copy Surface PropertyManager**. Choose the **OK** button from the **Copy Surface PropertyManager**.

Trimming Surfaces

CommandManager: Surfaces > Trim Surface
SOLIDWORKS menus: Insert > Surface > Trim
Toolbar: Surfaces > Trim Surface



The **Trim Surface** tool is used to trim surfaces using an entity as the trim tool. A surface, a sketched entity, or an edge can be used as a trim tool. To trim a surface, choose the **Trim Surface** button from the **Surfaces CommandManager**; the **Trim Surface PropertyManager** will be displayed, as shown in Figure 1-59, and you will be prompted to select pieces to keep or remove.

There are two methods of trimming a surface, namely Standard trim and Mutual trim. The **Standard** radio button is selected by default in the **Trim Type** rollout of the PropertyManager. Therefore, while using the first method, if you select the trimming surface using the cursor, this surface will act as a trim tool. You can select a surface, a plane, a sketch, or an edge as a trimming surface. On doing so, the selected entity will be highlighted in a different color and the name of the trimming surface will be displayed in the **Trim tool** display area. By default, the **Keep selections** radio button is selected in the **Selections** rollout. Also, the selection mode in the **Pieces to Keep** selection box will be activated and you will be prompted to select pieces to keep. Also, the cursor will be replaced by the surface body cursor when you move it on the surface. Move the cursor on the surface being trimmed; the pieces of the surface on which you place the cursor will be displayed in a different color. Select the piece or pieces of the surface to keep; the selected pieces will be displayed in different color and their names will be displayed in the **Pieces to Keep** selection box. Next, choose the **OK** button from the **Trim Surface PropertyManager**. If you select the **Remove selections** radio button, the **Pieces to Keep** selection box will change into **Pieces to Remove** selection box. As a result, the selected surfaces will be removed.

Figure 1-60 shows the trimming surface and the piece to keep after trimming. Figure 1-61 shows the resultant trimmed surface. Figure 1-62 shows the sketch selected as a trimming entity and Figure 1-63 shows the resultant trimmed surface.

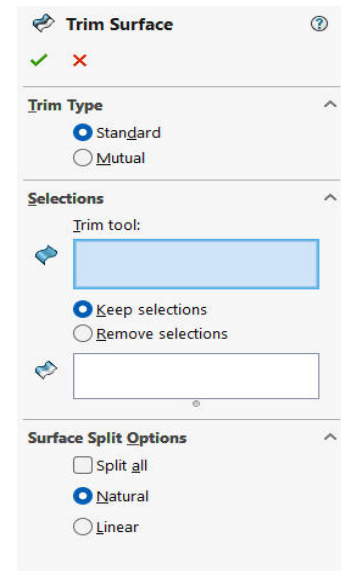


Figure 1-59 The Trim Surface PropertyManager

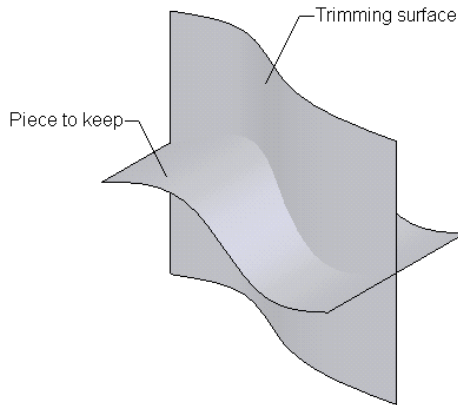


Figure 1-60 Trimming surface and the piece to keep after trimming the surface

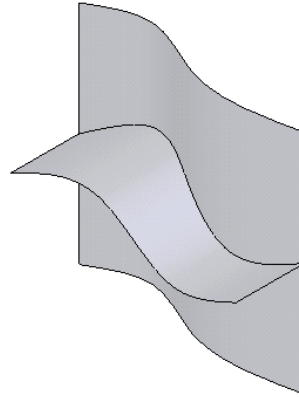


Figure 1-61 Resultant trimmed surface

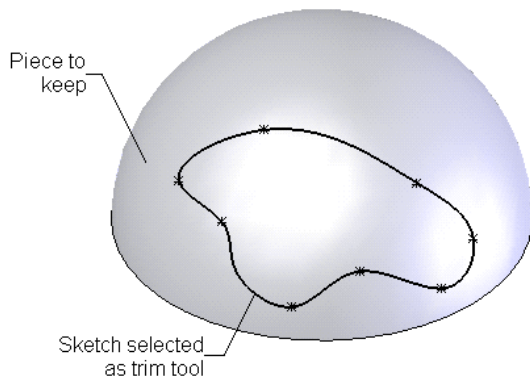


Figure 1-62 Sketch selected as a trimming entity

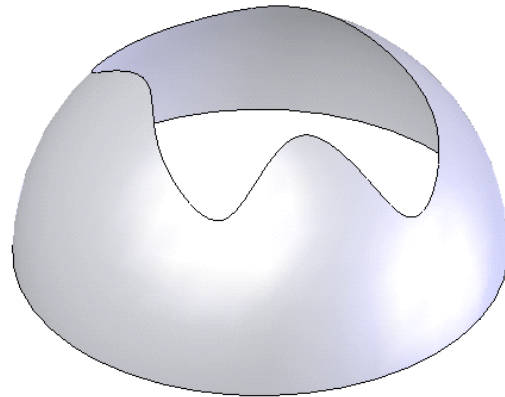


Figure 1-63 Resultant trimmed surface

The other method of trimming a surface is known as the Mutual trim method. In this method, you need to select two surfaces as the trimming surfaces. To trim these surfaces, invoke the **Trim Surface PropertyManager** and then choose the **Mutual** radio button from the **Trim Type** rollout; you will be prompted to select the surfaces to trim, followed by the pieces to keep. First, select the trimming surfaces and then select the pieces to keep, refer to Figure 1-64. After setting all parameters, choose the **OK** button from the **Trim Surface PropertyManager**. Figure 1-65 shows the resultant trimmed surface.

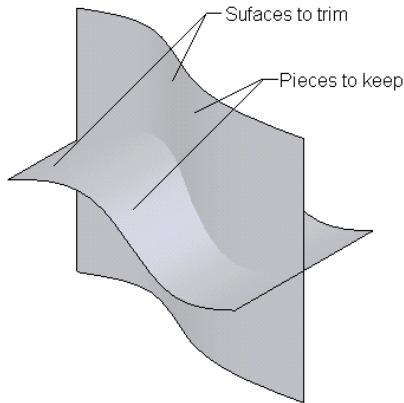


Figure 1-64 Surfaces to trim and pieces to keep

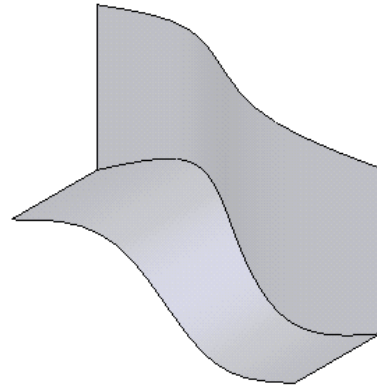


Figure 1-65 Resultant trimmed surface

If you select the **Split all** check box in the **Surface Split Options** rollout then all possible splits in the target surface are displayed. By default, the **Natural** radio button is selected in the **Surface Split Options** rollout. As a result, the endpoint of the split line will be extended tangentially to the boundary of target surface. If you select the **Linear** radio button from the **Surface Split Options** rollout then the endpoint of the split line will be extended linearly to the boundary of the target surface. Figure 1-66 shows the trimming of surface when the **Natural** radio button is selected and Figure 1-67 shows the trimming of surface when the **Linear** radio button is selected.

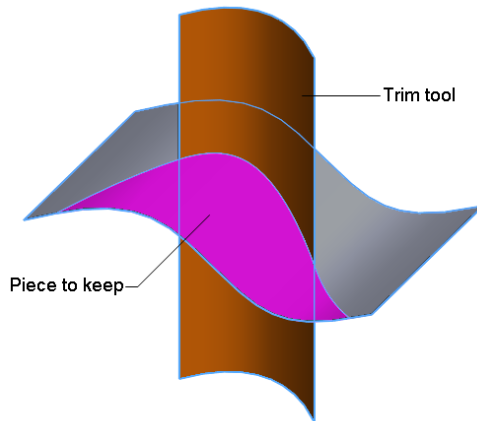


Figure 1-66 Piece to keep when the **Natural** radio button is selected

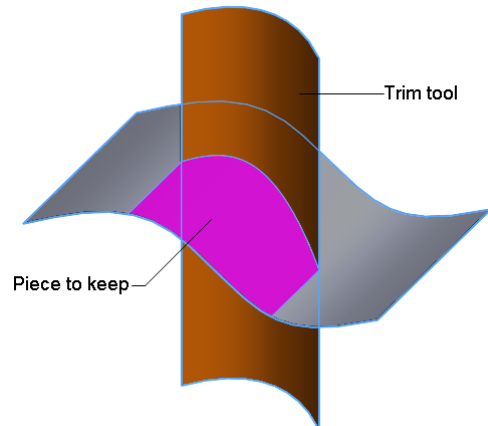


Figure 1-67 Piece to keep when the **Linear** radio button is selected

Untrimming Surfaces

CommandManager: Surfaces > Untrim Surface
SOLIDWORKS menus: Insert > Surface > Untrim
Toolbar: Surfaces > Untrim Surface



The **Untrim Surface** tool is used to create a surface patch by extending the existing surfaces. Using this tool, you can fill the trimmed portion of a surface with a surface patch. To untrim a surface, choose **Insert > Surface > Untrim** from the SOLIDWORKS menus. Alternatively, choose the **Untrim Surface** button from the **Surfaces CommandManager**; the **Untrim Surface PropertyManager** will be displayed, as shown in Figure 1-68 and you will be prompted to select the surface bodies or edges of a surface. Select the surface bodies to be untrimmed.

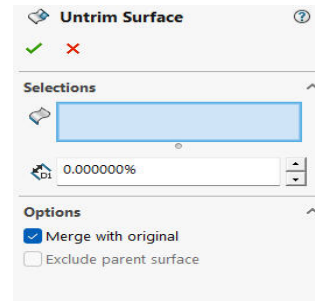


Figure 1-68 The Untrim Surface PropertyManager

In SOLIDWORKS, there are two methods to untrim surfaces. In the first method, you need to select the face that you want to untrim and in the second method, you need to select the edges of the trimmed portion of the surface. Both these methods are discussed next.

Untrimming Surfaces by Selecting Faces

In this method, you will untrim the surface by selecting the face or faces of the surface to be untrimmed. To do so, invoke the **Untrim Surface PropertyManager** and then select the face of the surface that needs to be untrimmed; the preview of the untrimmed surface with the default settings will be displayed in the drawing area. As soon as you select the face or faces of the surface to untrim, the **Options** rollout will be displayed with different options, as shown in Figure 1-69. These options are discussed next.

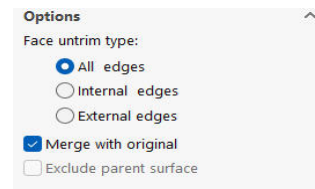


Figure 1-69 The Options rollout

Face untrim type

The **Face untrim type** area is used to specify the type of edges along which you want to untrim the surface. The options in this area are discussed next.

All edges

The **All edges** radio button is selected by default. As a result, all internal and external edges of the selected surface are extended to be untrimmed. Figure 1-70 shows the surface selected and Figure 1-71 shows the resultant untrimmed surface with the **All edges** radio button selected.

Internal edges

Select the **Internal edges** radio button to patch only the internal edges of a selected surface. Figure 1-70 shows the surface selected for untrimming and Figure 1-72 shows the resultant untrimmed surface with the **Internal edges** radio button selected.

External edges

Select the **External edges** radio button to patch only the external edges of a selected surface. Figure 1-70 shows the surface selected for untrimming and Figure 1-73 shows the resultant untrimmed surface with the **External edges** radio button selected.

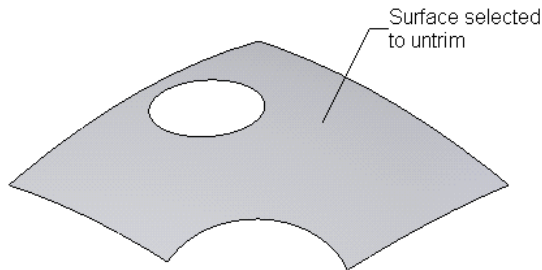


Figure 1-70 Surface selected to be untrimmed

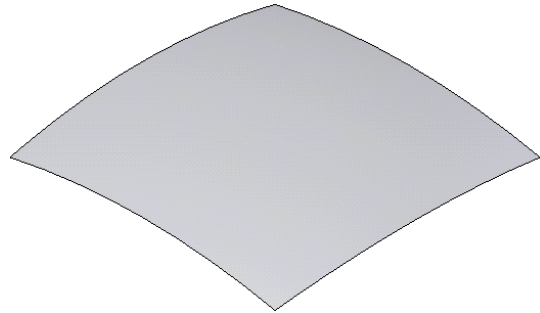


Figure 1-71 Resultant untrimmed surface with the **All edges** radio button selected

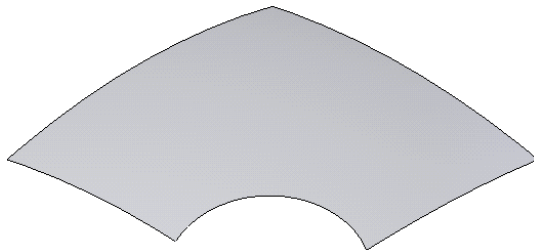


Figure 1-72 Resultant untrimmed surface with the **Internal edges** radio button selected

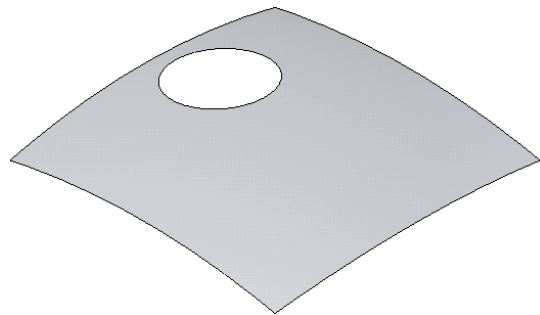


Figure 1-73 Resultant untrimmed surface with the **External edges** radio button selected

Merge with original

The **Merge with original** check box is used to merge the untrimmed surface created with the original surface. This check box is selected by default in the **Options** rollout. If you clear this check box, the resultant untrimmed surface will be a separate surface body.

You can also specify the percentage of distance up to which you need to extend the surface depending on the type of edges selected from the **Face untrim type** area of the **Options** rollout. The **Distance** spinner is used to define the percentage of distance for extending the surface. A preview of the surface extension is displayed in the drawing area.

Untrimming Surfaces by Selecting Edges

You can also patch a trimmed surface using the **Untrim Surface** tool by selecting the edges of the trimmed portion of the surface. To do so, invoke the **Untrim Surface PropertyManager** and then select the edge of the surface along which you want to patch the trimmed surface; the

preview of the patched surface will be displayed in the drawing area. As soon as you select the edge of the surface to patch the trimmed surface, the **Options** rollout will be displayed with different options, as shown in Figure 1-74. These options are discussed next.

Edge untrim type

The **Edge untrim type** area is used to specify the options for patching the trimmed surface by using the selected edges. The options in this area are discussed next.

Extend edges

The **Extend edges** radio button is selected by default and is used to extend the edge to create a corner for untrimming the trimmed surface.

Connect endpoints

The **Connect endpoints** radio button is selected to patch the trimmed surface by joining the endpoints of the selected edge.

Merge with original

The use of **Merge with original** check box is the same as discussed earlier.

Figure 1-75 shows the edge to be selected to untrim a surface. Figure 1-76 shows an untrimmed surface created with the **Extend edges** radio button selected. Figure 1-77 shows the untrimmed surface created with the **Connect endpoints** radio button selected.

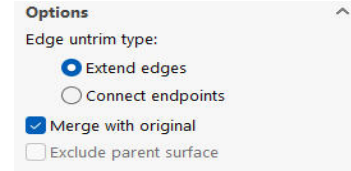


Figure 1-74 The **Options** rollout with different options

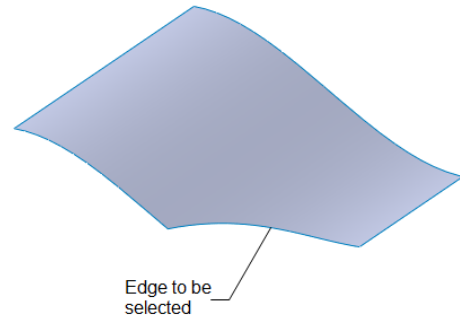


Figure 1-75 Edge to be selected to untrim the surface

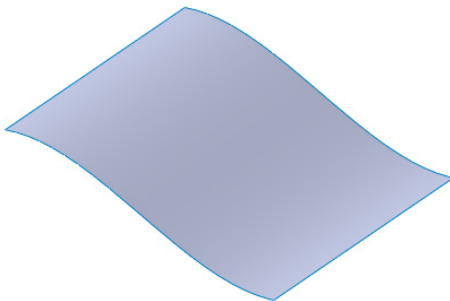


Figure 1-76 Untrimmed surface created by selecting the **Extend edges** radio button

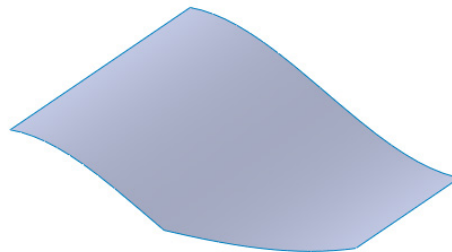



Figure 1-77 Untrimmed surface created by selecting the **Connect endpoints** radio button

Exclude Parent surface

Select the **Exclude parent surface** checkbox to exclude the parent surface from the resultant Untrim Surface features.

Extending Surfaces

CommandManager:	Surfaces > Extend Surface
SOLIDWORKS menus:	Insert > Surface > Extend
Toolbar:	Surfaces > Extend Surface

 The **Extend Surface** tool is used to extend a surface along a selected edge or a selected face. To extend a surface, choose the **Extend Surface** button from the **Surfaces CommandManager**; the **Extend Surface PropertyManager** will be displayed, refer to Figure 1-78. Also, you will be prompted to select a face or edge(s) and set the properties to extend. There are two methods to extend a surface and these are discussed next.

Extending a Surface Using the Same surface Option

You can extend a surface by using the **Same surface** radio button in the **Extension Type** rollout of the **Extend Surface PropertyManager**. Select this radio button to extend a surface by maintaining its curvature. After selecting this radio button, you need to select the edge or face that you need to extend. Note that when you select the face to extend the surface, the surface extends equally in all directions. You can extend a surface dynamically using the drag handle or set the extending distance in the **Distance** spinner available in the **End Condition** rollout. You can also use other feature termination options available in the **End Condition** rollout. By default, the **Distance** radio button is selected in the **End Condition** rollout. If you want to extend the surface up to a particular point or vertex, select the **Up to point** radio button from the **End Condition** rollout and then select the required point or vertex from the drawing area. If you want to extend the surface up to a particular surface, select the **Up to surface** radio button and then select the required surface from the drawing area. After setting all parameters, choose the **OK** button from the **Extend Surface PropertyManager**.

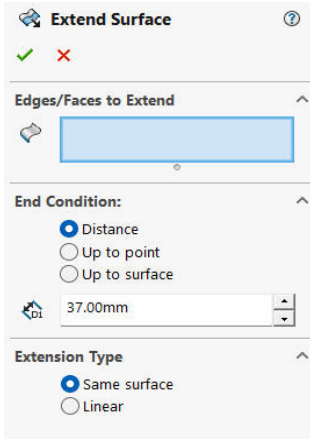


Figure 1-78 The Extend Surface PropertyManager

Figure 1-79 shows the edge selected to extend the surface. Figure 1-80 shows preview of the surface being extended by selecting the edge with the **Same surface** radio button selected. Figure 1-81 shows the face selected to extend the surface. Figure 1-82 shows preview of the surface being extended by selecting the face with the **Same surface** radio button selected.

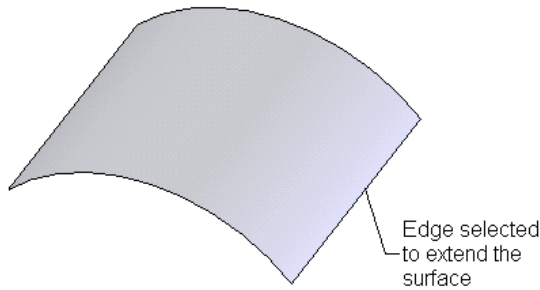


Figure 1-79 Edge selected to extend the surface

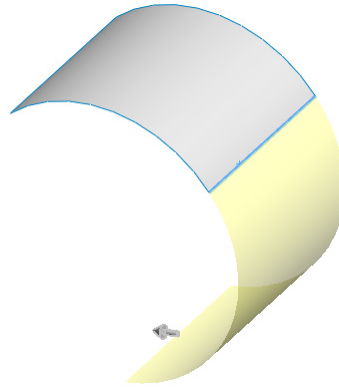


Figure 1-80 Preview of the extended surface with the **Same surface** radio button selected



Note

If any edge of the selected surface is merged with another surface, the surface will not extend along that edge.

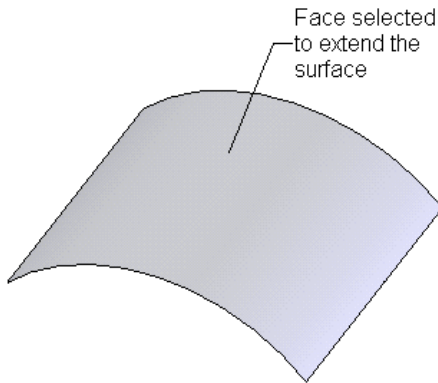


Figure 1-81 Face selected to extend the surface

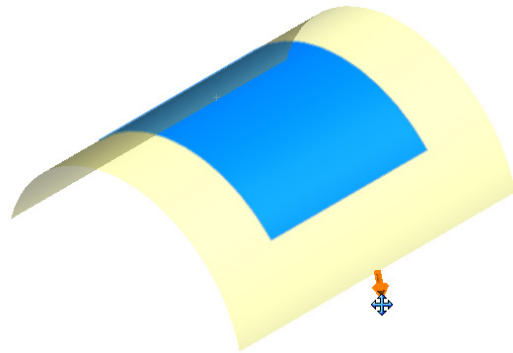


Figure 1-82 Preview of the extended surface with the **Same surface** radio button selected

Extending a Surface Using the Linear Option

You can extend a surface in the linear direction up to an existing surface by maintaining tangency. To do so, invoke the **Extend Surface PropertyManager** and then select the **Linear** radio button from the **Extension Type** rollout. Next, select the face or the edge along which you need to extend the surface and then specify the feature termination using the **End Condition** rollout. After setting all parameters, choose the **OK** button.

Figure 1-83 shows the edge selected to extend the surface and Figure 1-84 shows preview of the surface being extended with the **Linear** radio button selected. Figure 1-85 shows the face selected to extend the surface and Figure 1-86 shows the preview of surface being extended with the **Linear** radio button selected.

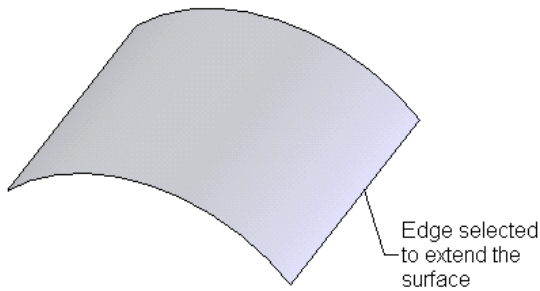


Figure 1-83 Edge selected to extend the surface

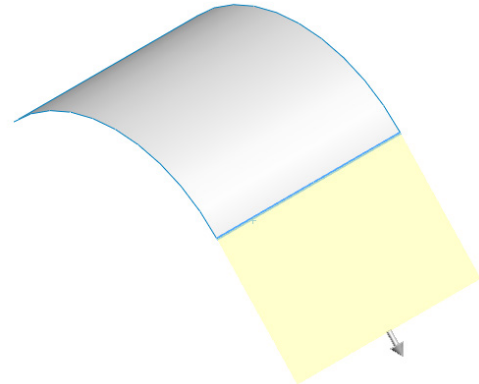


Figure 1-84 Preview of the extended surface with the **Linear** radio button selected

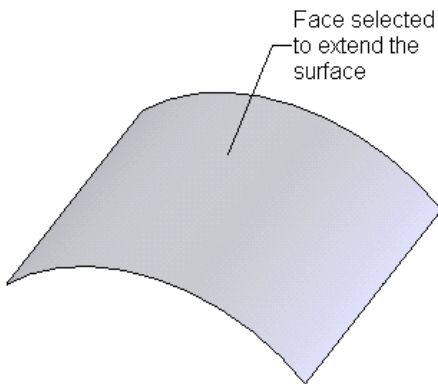


Figure 1-85 Face selected to extend the surface

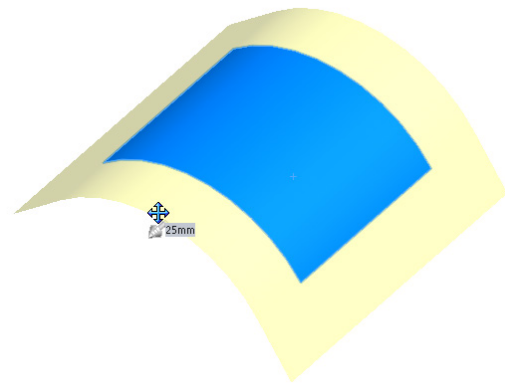


Figure 1-86 Preview of the extended surface with the **Linear** radio button selected

Knitting Surfaces

CommandManager:	Surfaces > Knit Surface
SOLIDWORKS menus:	Insert > Surface > Knit
Toolbar:	Surfaces > Knit Surface



The **Knit Surface** tool is used to knit multiple surfaces together to create a single surface. You can also knit a surface with the faces of a solid body. The surfaces to be knitted together must be in contact with each other. This means that you cannot knit disjointed surfaces or faces. The **Knit Surface** tool is widely used for extracting core and cavity while designing a mold.

To knit surfaces, choose the **Knit Surface** button from the **Surfaces CommandManager**; the **Knit Surface PropertyManager** will be displayed, refer to Figure 1-87. Select the surfaces to be knitted together; the names of surfaces will be displayed in the **Surfaces and Faces to Knit** selection box of the **Selections** rollout. Specify the knitting tolerance in the **Knitting tolerance** spinner. If the size of a gap is lower than the tolerance specified then the gap will be knitted

and closed. Specify a tolerance if you want to display the gaps, which are within that range. Depending on the knitting tolerance and range specified, the gaps will be listed in the list box, refer to Figure 1-87. Move the cursor near the check box in the list box; a tooltip with the message **Knit all gaps less than gap size** will be displayed. Select the check box to knit all gaps. Now, if you move the cursor near the selected check box, a tooltip with the message **Keep all gaps greater than gap size** will be displayed. Clear the check box to retain the gaps, if needed. Select the **Create solid** check box to convert the knitted entities to solid. Select the **Merge entities** check box to merge all knitted surfaces. After specifying all parameters, choose the **OK** button from the **Knit Surface PropertyManager**; a knitted surface will be created.

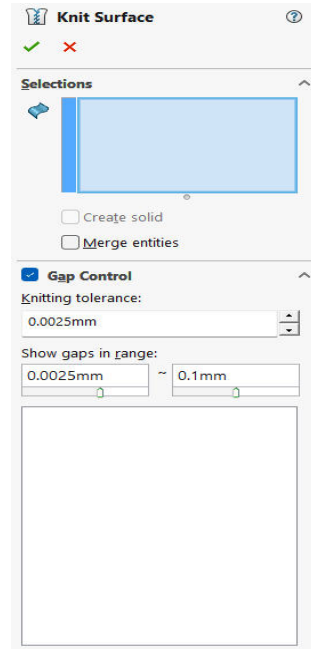


Figure 1-87 The **Knit Surface PropertyManager**

Filleting Surfaces

CommandManager: Surfaces > Fillet
SOLIDWORKS menus: Insert > Surface > Fillet/Round
Toolbar: Surfaces > Fillet (Customize to add)



You can add fillets on sharp edges of surfaces by using the **Fillet** tool. The procedure of filleting the surfaces is the same as filleting solid models discussed earlier. But there are some exceptions. These exceptions are discussed next.

1. While applying the face fillet to a surface, you need to define the direction in which the fillet needs to be added.
2. You cannot use the **Keep features** option while filleting a surface.
3. You cannot select a surface using the **FeatureManager Design Tree** while filleting all the edges in a surface.

While applying face fillet, by default the **Trim and attach** radio button is selected in the **Trim surfaces** area of the **Fillet Options** rollout. As a result, the filleted faces will be trimmed and knit into a single surface. Select the **Don't trim or attach** radio button if you want to create the fillet as a new surface and not as a single trimmed and knitted surface.



Note

You can only fillet the edge of surface that is created at the intersection of two surfaces. Make sure that if an edge is created using two surfaces, then you need to knit them before filleting.

Creating a Mid-Surface

CommandManager:	Surfaces > Mid-Surface (Customize to add)
SOLIDWORKS menus:	Insert > Surface > Mid Surface
Toolbar:	Surfaces > Mid-Surface (Customize to add)

The **Mid-Surface** tool is used to create a surface between the two parallel faces of a solid model. You can define the placement of a surface in terms of percentage value with respect to the face selected first. Note that the faces to be selected to create a mid-surface should be two parallel faces or two concentric curved faces. To create a mid surface, choose the **Mid-Surface** button from the **Surfaces CommandManager**; the **MidSurface1 PropertyManager** will be displayed, as shown in Figure 1-88. Also, you will be prompted to either select the face pairs manually or use the **Find Face Pairs** button to automatically recognize the face pairs. Next, select the faces between which you need to create the mid-surface. On doing so, both the selected faces will be highlighted in different colors. Also, the names of the selected faces will be displayed in the **Face pairs** display area. By default, the mid-surface is placed in the middle of the selected faces. You can also define the percentage distance for the placement of the mid-surface using the **Position** spinner that is available in the **Selections** rollout of the PropertyManager. The position of the mid-surface is defined from the first selected surface. After setting all the parameters, choose the **OK** button from the **MidSurface1 PropertyManager**.

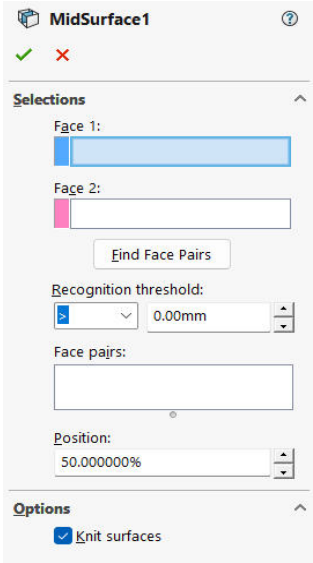


Figure 1-88 The MidSurface1 PropertyManager

Figure 1-89 shows the offset faces to be selected and Figure 1-90 shows the mid-surface created in the middle of the selected faces.

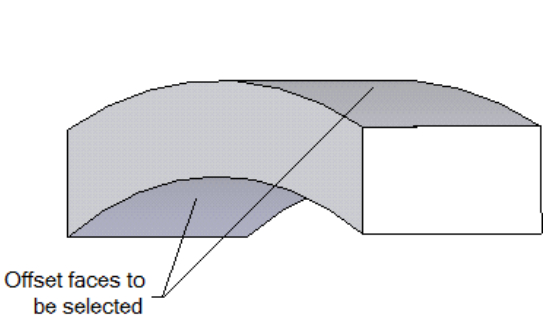


Figure 1-89 Offset faces to be selected

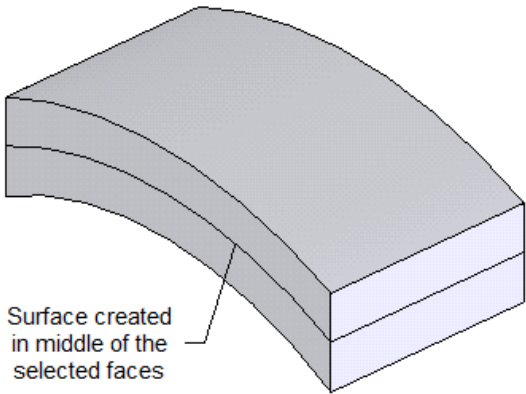


Figure 1-90 Resultant mid-surface

**Note**

To access the mid-surface, right-click on the solid bodies in the Feature Manager Design Tree and click the eye icon to hide the associated feature.

The **Find Face Pairs** button is used to find the faces that are adjacent to the selected face. The options in the **Recognition threshold** area are used to filter the faces depending on the wall thickness of the face searched using the **Find Face Pairs** option. Using the **Threshold Operator** drop-down list in the **Recognition threshold** area, you can set the mathematical operators such as $>$, $<$, $=$, and so on. Using the **Threshold Thickness** spinner, you can specify the threshold thickness.

Deleting Holes from Surfaces

CommandManager: Surfaces > Delete Hole

SOLIDWORKS menus: Insert > Surface > Delete Hole



The **Delete Hole** tool is used to delete holes from a surface or any closed contours that cut surface. To do so, select an edge from the drawing area and choose the **Delete Hole** tool from the **CommandManager**; the **Delete Hole PropertyManager** will be displayed, as shown in Figure 1-91; the empty area of the selected edge will be patched and the tangency and curvature with the surrounding surfaces will be maintained.

You can also select more edges to delete the hole; the name of selected edge/s will be displayed in the **Selected Edges to remove** selection box in the **Selections** rollout. Choose the **OK** button from this PropertyManager; you will notice that the **DeleteHole** node is added in the **FeatureManager Design Tree**.

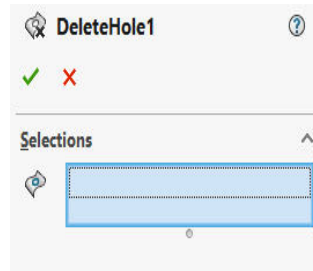


Figure 1-91 The *Delete Hole PropertyManager*

Figure 1-92 shows the edge of the closed contour to be selected. Figure 1-93 shows the hole deleted using the **Delete Hole** tool.

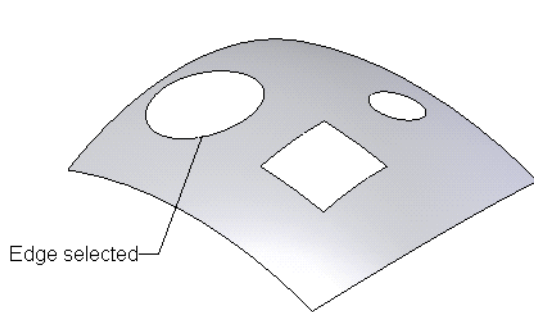


Figure 1-92 Edge selected

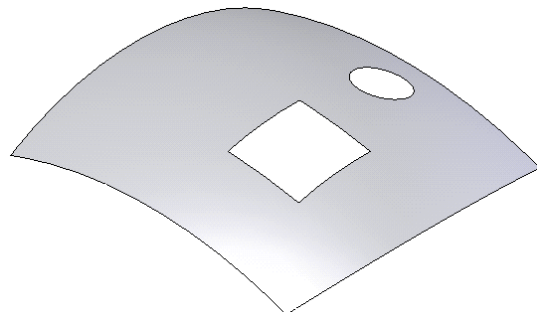


Figure 1-93 Resultant surface

Replacing Faces

CommandManager:	Surfaces > Replace Face
SOLIDWORKS menus:	Insert > Face > Replace
Toolbar:	Surfaces > Replace Face



In SOLIDWORKS, you can replace the selected faces of a solid body with one or more surfaces. When you replace the selected faces with another surface or surfaces, the resultant solid body retains the shape of the replaced surface by adding or subtracting material from the solid body. To replace a surface, choose the **Replace Face** button from the **Surfaces CommandManager**; the **Replace Face1 PropertyManager** will be displayed, as shown in Figure 1-94.

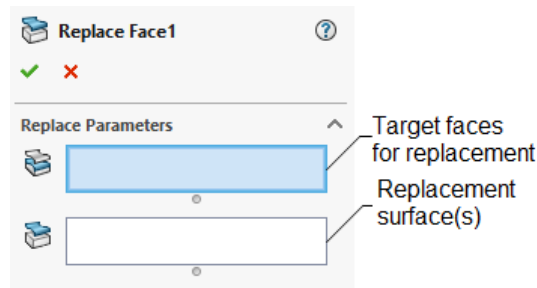


Figure 1-94 The *Replace Face1 PropertyManager*

Select the face to be replaced; the name of the selected face will be displayed in the **Target faces for replacement** selection box. Next, click once in the **Replacement surface(s)** selection box to invoke the selection mode. Now, select the replacement surface; the name of the replacement surface will be displayed in the **Replacement surface(s)** selection box. Choose the **OK** button from the **Replace Face1 PropertyManager**; the selected face of the solid body will be replaced.

Figure 1-95 shows the target face to be replaced and the replacement surface. Figure 1-96 shows the resultant replaced face. Figure 1-97 shows the solid body after hiding the surface body. To hide the surface body, select the surface body and then invoke the shortcut menu. Then, choose the **Hide** option from the shortcut menu.

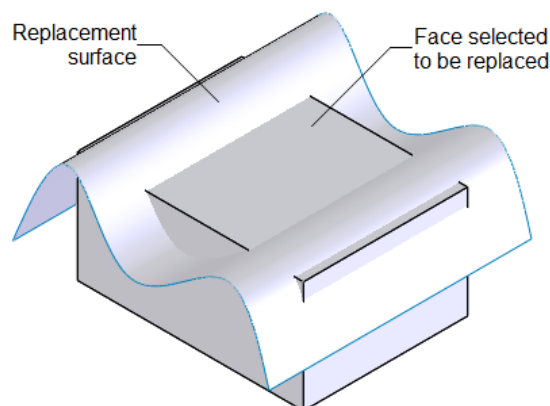


Figure 1-95 The target face and the replacement surface

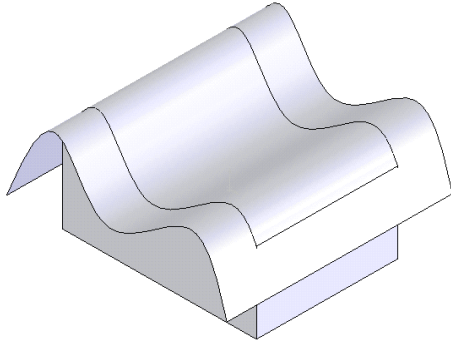


Figure 1-96 Resultant replaced face

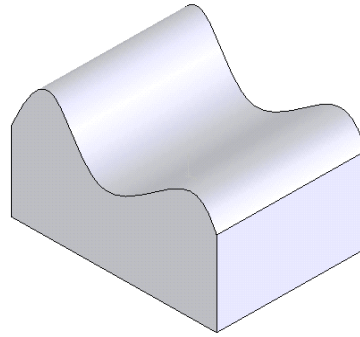


Figure 1-97 Model after hiding the surface

Deleting Faces

CommandManager: Surfaces > Delete Face
SOLIDWORKS menus: Insert > Face > Delete
Toolbar: Surfaces > Delete Face



The **Delete Face** tool is used to delete the faces of the selected surface or the solid body. When you delete a face of a solid body, the solid body is converted into a surface body. If you patch the deleted face, the solid body will not be converted into a surface body. To delete a face, choose the **Delete Face** button from the **Surfaces CommandManager**; the **Delete Face PropertyManager** will be displayed, as shown in Figure 1-98.

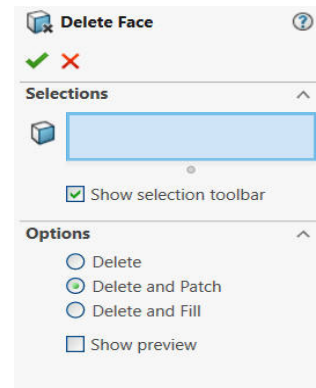


Figure 1-98 The **Delete Face PropertyManager**

The **Delete** radio button in the **Options** rollout of the **Delete Face PropertyManager** is used to delete the faces of a solid body without patching and trimming it. In this case, the solid body is converted into a surface body. The **Delete and Patch** radio button is selected by default. It is used to delete the faces of a solid or a surface body by patching and trimming it. The **Show preview** check box will be invoked only when the **Delete and Patch** radio button is selected. And it is used to display the preview of the model in the drawing area. The **Delete and Fill** radio button is used to delete multiple faces and generate a single face.

Select the face or faces to be deleted; the name of the selected face will be displayed in the **Faces to delete** selection box of the **Selections** rollout. Next, select the **Delete and Patch** radio button, if it is not selected by default; the preview of the model will be displayed in the drawing area. If the preview of the patch is not displayed, it confirms that the selected face cannot be deleted and patched. In this case, you need to select the **Delete** radio button from the **Options** rollout so that the face gets only deleted, but not patched. After setting all parameters, choose the **OK** button from the **Delete Face PropertyManager**; the resulting model will be displayed.

Figure 1-99 shows the face selected to be deleted. Figure 1-100 shows the face deleted with the **Delete and Patch** radio button selected. Figure 1-101 shows the face deleted with the **Delete** radio button selected. Figure 1-102 shows multiple faces to be selected to generate a single face. Figure 1-103 shows a single face generated using the **Delete and Fill** radio button.

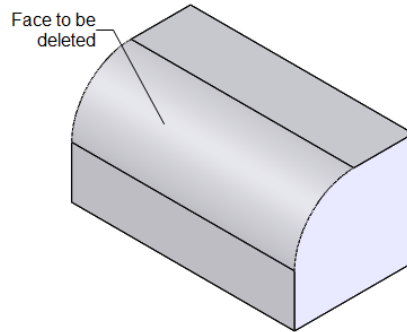


Figure 1-99 Face to be deleted

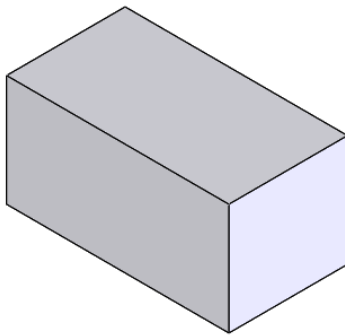


Figure 1-100 Face deleted with the **Delete and Patch** radio button selected

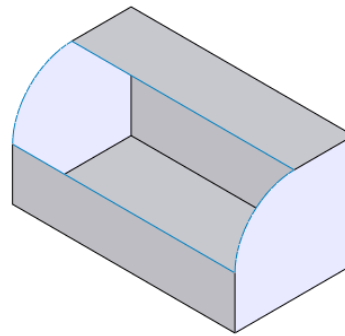


Figure 1-101 Face deleted with the **Delete** radio button selected

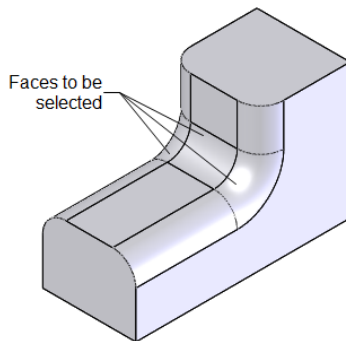


Figure 1-102 Faces to be selected to generate a single face

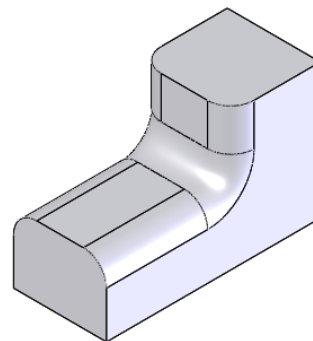


Figure 1-103 Face deleted with the **Delete and Fill** radio button selected



Note

You can also delete surface bodies using the **Delete/Keep Body PropertyManager**. The procedure of deleting surface bodies is the same as deleting solid bodies discussed earlier.

Moving and Copying Surfaces

You can move and copy the surfaces using the **Move/Copy Bodies** tool. The methods of moving and copying surface bodies using the **Move/Copy Bodies** tool are the same as those discussed for moving and copying solid bodies.

Mirroring Surface Bodies

You can mirror the surface bodies using the **Mirror** tool. The procedure of mirroring surface bodies is the same as that discussed in the solid bodies.

Adding Thickness to Surface Bodies

CommandManager: Surfaces > Thicken
SOLIDWORKS menus: Insert > Boss/Base > Thicken



In SOLIDWORKS, you can also add thickness to the surface bodies. There are two methods of adding thickness to surface bodies. In the first method, you need to add wall thickness to the surface body. In the second method, you need to solidify the closed, stitched surface body to create a solid body. These two methods are discussed next.

Adding Thickness to a Surface Body

To add thickness to a surface body, choose the **Thicken** tool from the **Surfaces CommandManager** or choose **Insert > Boss/Base > Thicken** from the SOLIDWORKS menus; the **Thicken PropertyManager** will be displayed, as shown in Figure 1-104.

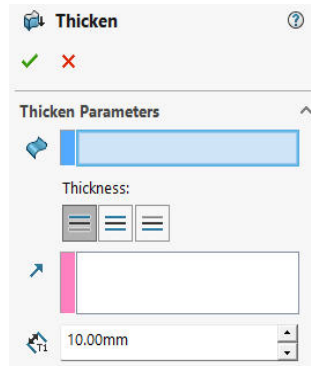


Figure 1-104 The Thicken PropertyManager

Also, you will be prompted to select the surface to thicken. Select the surface; preview of the thickened body with default values will be displayed in the drawing area. Using the buttons in the **Thickness** area of this PropertyManager, you can specify the side on which you want to thicken the surface. You can specify the direction of thickness by selecting any linear sketch entity, sketch point, reference plane, reference axis, or linear edge in the drawing area. The selected entity will get displayed in the **Direction of Thicken** box. You can also specify the wall thickness by using the **Thickness** spinner available in the **Thicken Parameters** rollout. Select the **Merge result** check box if you want to merge existing solid body from the drawing area with the thickened surface. This check box will only be available if a surface and a solid is present in the drawing area. After setting all the parameters, choose the **OK** button from the **Thicken**

PropertyManager; thickness will be added to the surface body. Figure 1-105 shows surface body to be thickened and Figure 1-106 shows the model after adding thickness to surface body.

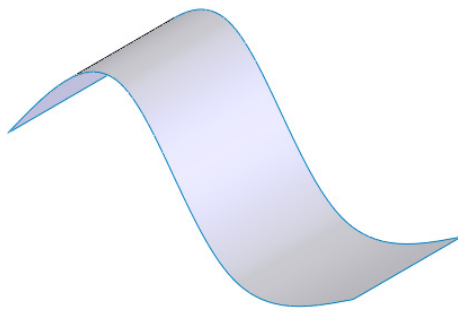


Figure 1-105 Surface to be thickened

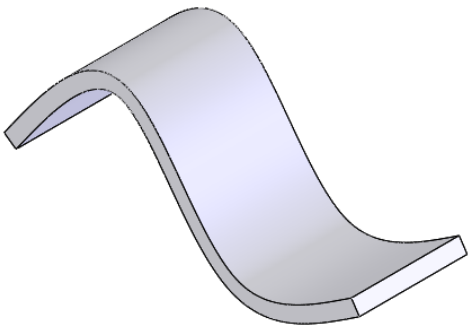


Figure 1-106 Model after thickening the surface

Solidifying a Closed Surface Body

To solidify a closed surface body, the surface body needs to be free from any type of gap and all surfaces should be stitched together using the **Knit** tool. To solidify the surface body, invoke the **Thicken PropertyManager** and then select a closed surface body; the **Create solid from enclosed volume** check box will be displayed. Select this check box; the **Thickness** area and the **Thickness** spinner will be disabled. Next, choose the **OK** button from the **Thicken PropertyManager**; a solid volume will be created from the closed surface.



Tip

If the surface model to be thickened consists of multiple joined surface bodies, you first need to knit the surfaces together and then add thickness to them.

Creating a Thicken Surface Cut

CommandManager: Surfaces > Thickened Cut
SOLIDWORKS menus: Insert > Cut > Thicken

In SOLIDWORKS, you can cut a solid body by thickening a surface. To do so, create a solid body and a surface intersecting each other and then choose the **Thickened Cut** button from the **Surfaces CommandManager**; the **Cut-Thicken PropertyManager** will be displayed, as shown in Figure 1-107. Also, you will be prompted to select the surface to be thickened. Select the surface that you want to use as the cutting tool and then specify the parameters for defining the side in which you want to add thickness and thickness of cut. On doing so, a preview will be displayed in the drawing area. Now, choose the **OK** button from the **Cut-Thicken PropertyManager**. If the thicken cut results in the creation of multiple bodies, the **Bodies to Keep** dialog box will be displayed. Using this dialog box, you can define the bodies that you need to keep.

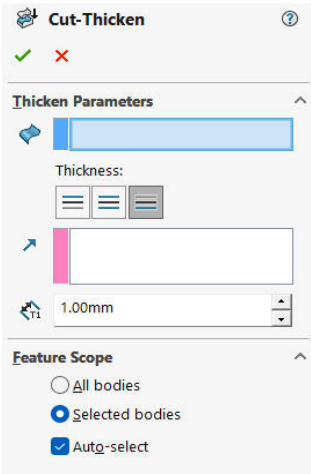


Figure 1-107 The Cut-Thicken PropertyManager

Figure 1-108 shows the surface selected for creating the thicken cut. Figure 1-109 shows the resultant thicken surface cut. You can specify direction of thicken same as **Thicken** tool.



Tip

If you create a thicken surface cut or a surface cut on multiple solid bodies, the **Feature Scope** rollout will be displayed with different options and you can specify the bodies on which you need to add this feature.

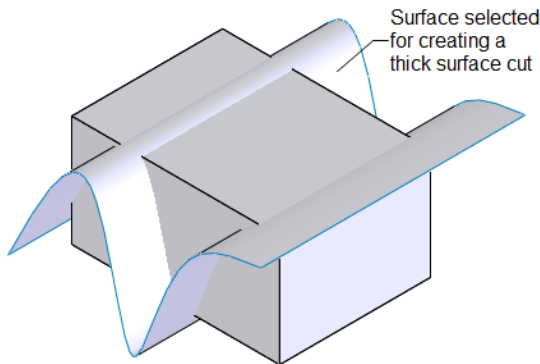


Figure 1-108 Surface to be selected

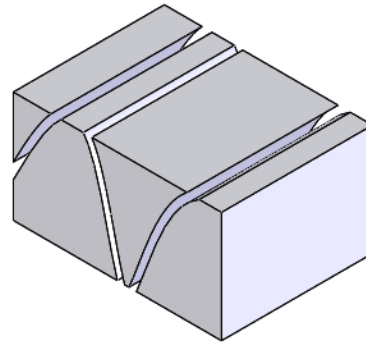


Figure 1-109 Resultant thicken surface cut

Creating a Surface Cut

CommandManager: Surfaces > Cut With Surface
SOLIDWORKS menus: Insert > Cut > With Surface



In SOLIDWORKS, you can also cut a solid body by using a surface. To create this type of surface cut, choose the **Cut With Surface** button from the **Surfaces CommandManager**; the **SurfaceCut PropertyManager** will be displayed with the **Surface Cut Parameters** rollout. But, if there are multiple bodies, then on choosing the **Cut With Surface** button, the modified **SurfaceCut PropertyManager** will be displayed, as shown in Figure 1-110.

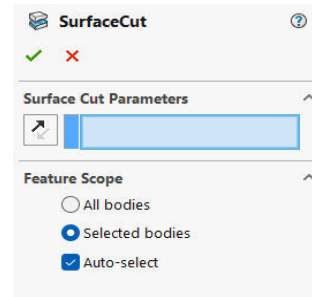


Figure 1-110 The modified **SurfaceCut PropertyManager**

On invoking this PropertyManager, you will also be prompted to select the cutting surface. Select the cutting surface; the name of the selected surface will be displayed in the **Selected surface for cut** selection box in the **Surface Cut Parameters** rollout. Also, an arrow will be displayed in the drawing area, indicating the direction of removal of the material. Using the **Flip Cut** button, available on the left of the **Selected surface for cut** selection box or the arrow displayed in the drawing area, you can flip the direction of material removal. Choose the **OK** button from the **SurfaceCut PropertyManager**; the surface cut will be created. Figure 1-111

shows the surface to be selected to create a surface cut. Figure 1-112 shows the resultant surface cut after hiding the surface body.

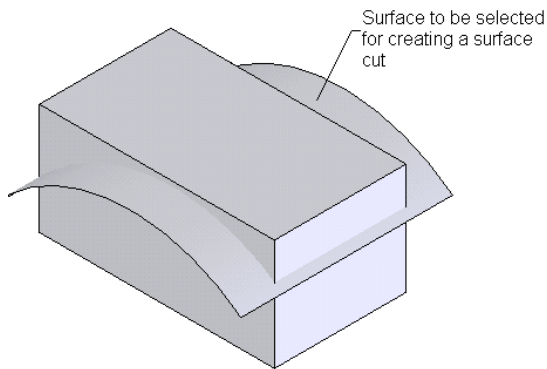


Figure 1-111 Surface to be selected to create a surface cut

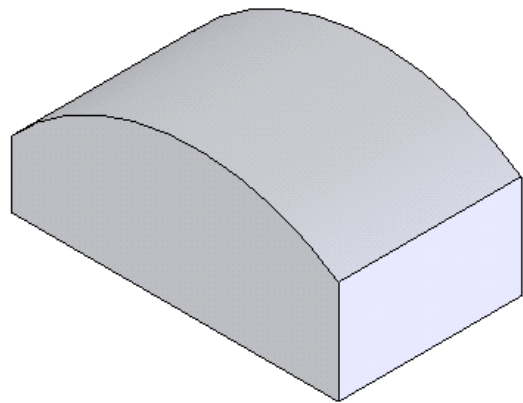


Figure 1-112 Resultant surface cut after hiding the surface body

In Figure 1-113, an example is shown in which a surface cuts a solid in several parts. In such cases, once you select a cut surface and choose **OK** from the **SurfaceCut** Property Manager; the **Bodies to Keep** dialog box will be displayed refer to Figure 1-114. The options in the dialog box are discussed next. By default, the **All bodies** radio button under the **Bodies** area is selected and used to create a surface cut by keeping all bodies in selected direction. The **Selected bodies** radio button is used to retain the selected bodies, refer to Figure 1-115.

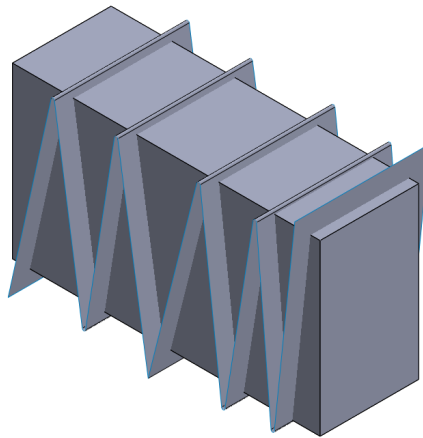


Figure 1-113 Surface cutting solid in several parts

Figure 1-116 shows the resultant surface cut after selecting **Body 1**, **Body 3** and **Body 5** bodies from the **Bodies** area.

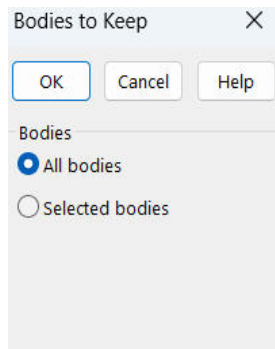


Figure 1-114 The **Bodies to Keep** dialog box with the **All bodies** radio button selected

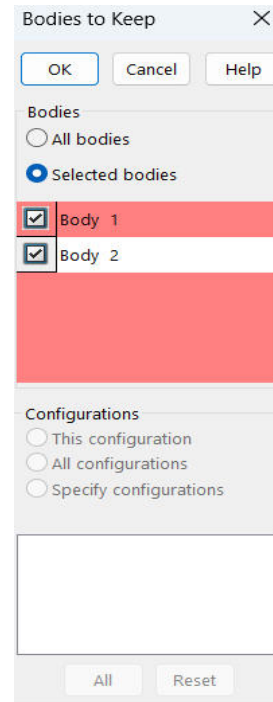


Figure 1-115 The **Bodies to Keep** dialog box with the **Selected bodies** radio button selected

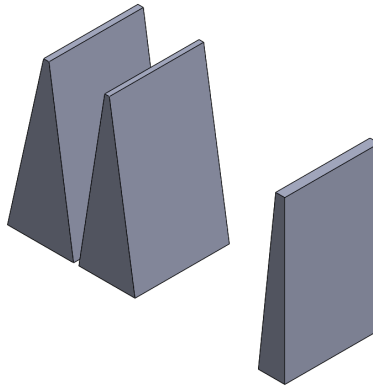


Figure 1-116 The resultant surface

TUTORIALS

The tutorials given next are available in video format. Scan the QR code or visit the following link to get access to the video tutorials.
<https://www.cadcim.com/adv-solidworks-2026-tutorial-videos>



Tutorial 1

In this tutorial, you will create the model shown in Figure 1-117. You will create the model using the surface modeling tools available in the **Surfaces CommandManager** and then add wall thickness to the surface model. The views and dimensions of the model are shown in Figure 1-118.

(Expected time: 1hr)

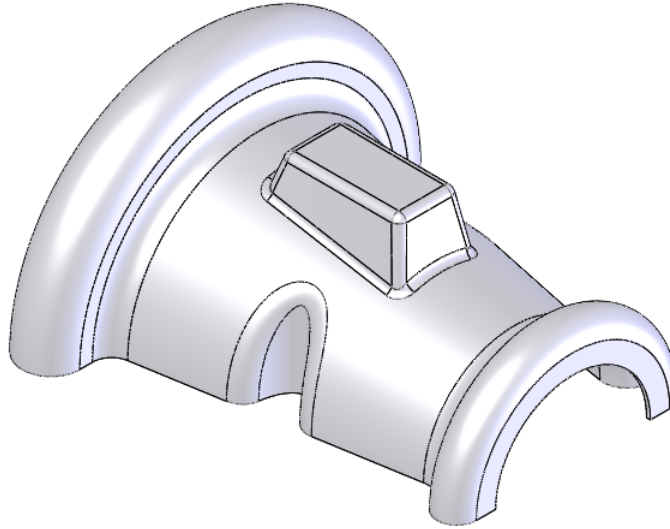


Figure 1-117 Model for Tutorial 1

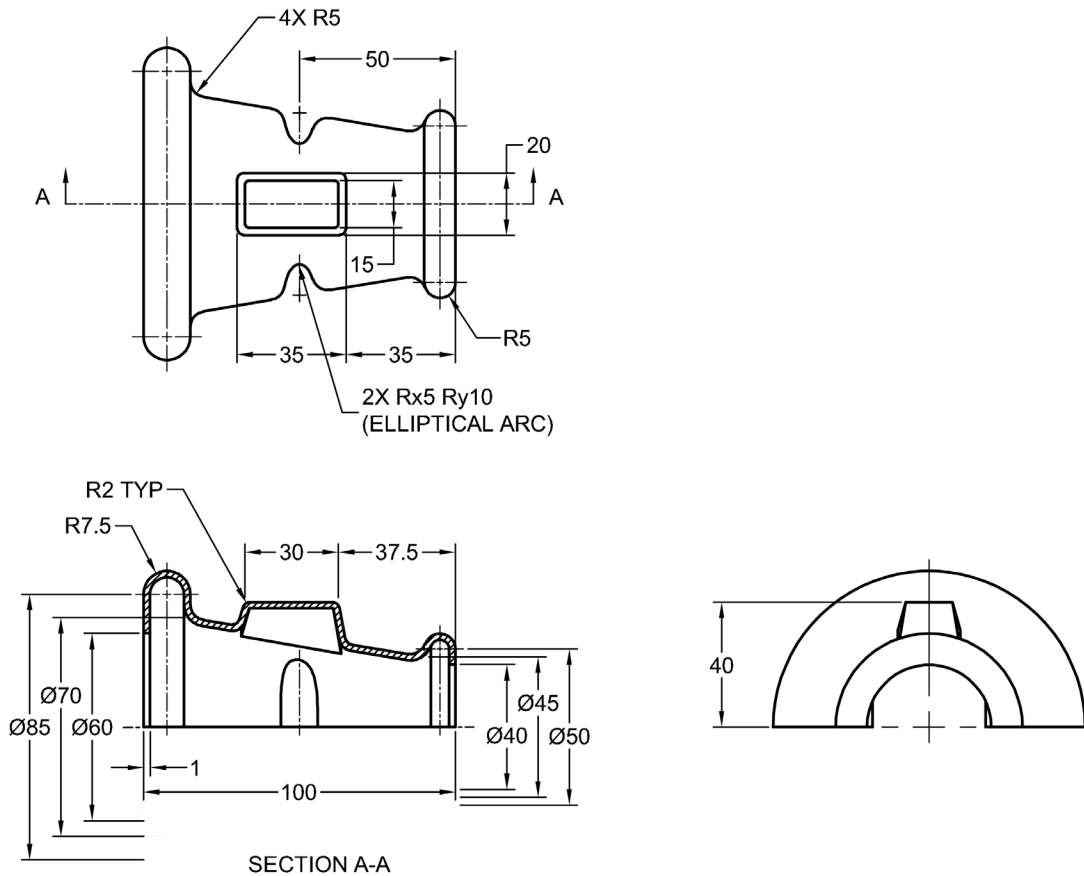


Figure 1-118 Views and dimensions of the model for Tutorial 1

The following steps are required to complete this tutorial:

- Create the base surface of the model by revolving the sketch using the **Mid Plane** option through 180 degrees.
- Create the second surface feature.
- Trim the extruded surface using the trim tool.
- Add fillet to the trimmed base surface.
- Create a plane at an offset distance of 40 mm from the Top plane.
- Create a lofted surface.
- Create a planar surface on the top of the lofted feature and trim the base feature using the lofted feature.
- Trim and knit all surfaces together.
- Add fillets to the surface model.
- Add thickness to the knitted surface.

Creating the Base Surface

To create this model, first you need to create the base surface. The base surface will be created by revolving the sketch created on the Front plane.

1. Start SOLIDWORKS part document using the **New SOLIDWORKS Document** dialog box.
2. Invoke the sketching environment using the Front plane as the sketching plane and create the sketch of the base surface, as shown in Figure 1-119.
3. Choose the **Revolved Surface** button from the **Surfaces CommandManager**; the **Surface-Revolve PropertyManager** is displayed.
4. In the **Surface-Revolve PropertyManager**, select the **Mid Plane** option from the **Revolve Type** drop-down list and set the value of the **Angle** spinner to **180**.
5. Choose the **OK** button from the **Surface-Revolve PropertyManager**. Figure 1-120 shows the resulting revolved base feature.

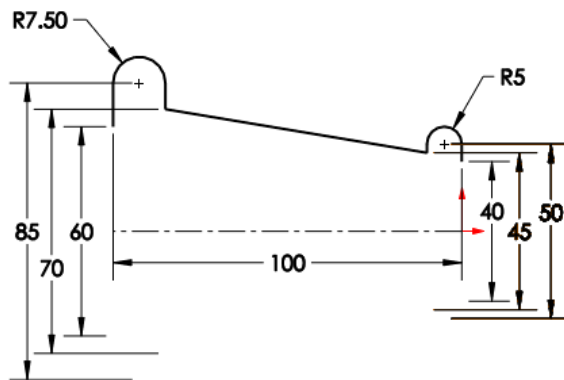


Figure 1-119 Sketch of the base surface

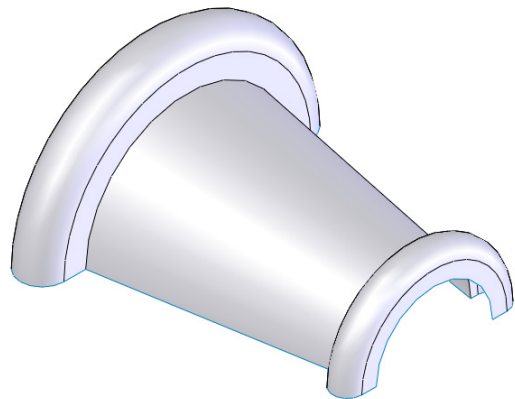


Figure 1-120 Revolved base surface

Creating the Second Surface Feature

The second surface feature is an extruded surface. This surface will be created by extruding a sketch created on the Top plane.

1. Select the Top plane as the sketching plane and invoke the sketching environment.
2. Create the sketch of the second surface feature, as shown in Figure 1-121. Note that you may have to apply the Horizontal or Vertical relation between the points of the ellipse to constrain them. Also, the centre point of the ellipse should coincide with the edge of the surface.
3. Next, choose the **Extruded Surface** button from the **Surfaces CommandManager**; the **Surface-Extrude PropertyManager** is displayed.

- Set **40** in the **Depth** spinner and choose the **OK** button from the **Surface-Extrude PropertyManager**.

The surface created after extruding the sketch is shown in Figure 1-122.

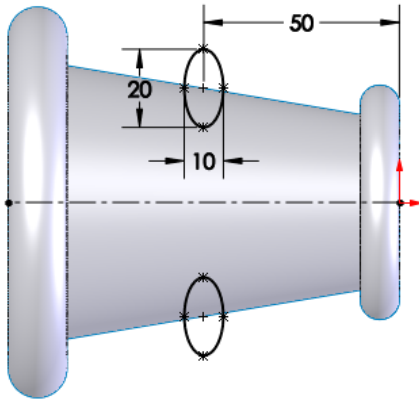


Figure 1-121 Sketch of the second surface feature

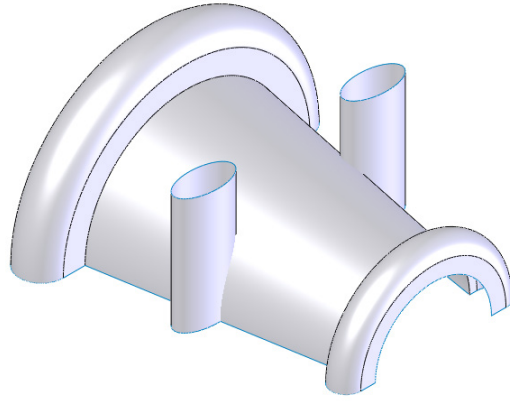


Figure 1-122 Extruded surface

Trimming the Base Surface Using the Extruded Surface

Next, you need to trim the unwanted portions of the base surface by using the extruded surface.

- Choose the **Trim Surface** button from the **Surfaces CommandManager**; the **Trim Surface PropertyManager** is displayed.
- Select the **Mutual** radio button from the **Trim Type** rollout and then select the surfaces, refer to Figure 1-123.
- Next, click in the **Keep selections** box to invoke the selection mode. Next, select the pieces to be kept, refer to Figure 1-123.
- Choose the **OK** button from the **Trim Surface PropertyManager**. The model after trimming the base surface is displayed in Figure 1-124.

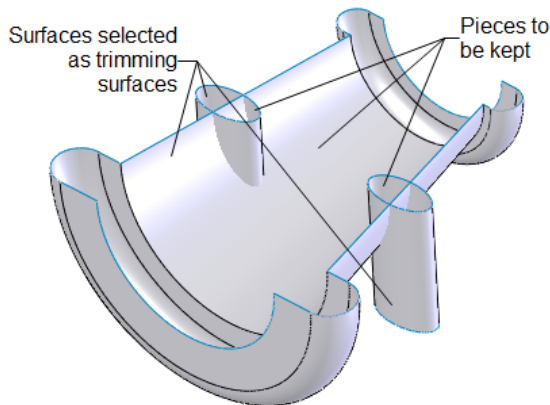


Figure 1-123 Surfaces selected for trimming

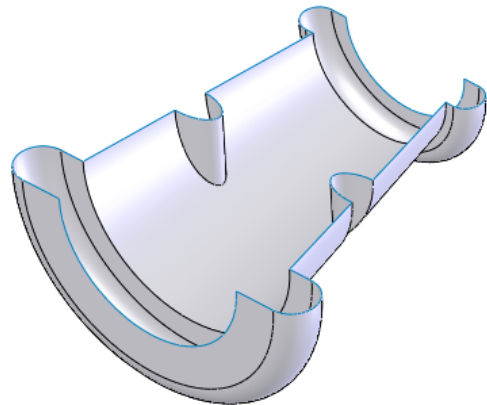


Figure 1-124 Model after trimming the base surface

Filletting the Edges of the Trimmed Surface

Next, you need to fillet the edges created at the intersection of the base surface with the extruded surfaces.

1. Choose the **Fillet** button from the **Surfaces CommandManager**; the **Fillet PropertyManager** is displayed.
2. Select the edges to fillet, as shown in Figure 1-125.
3. Set **5** in the **Radius** spinner and then choose the **OK** button from the **Fillet PropertyManager**. Figure 1-126 shows the model after adding fillet.

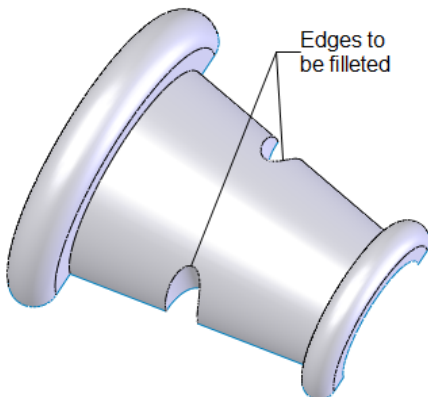


Figure 1-125 Edges to be filleted

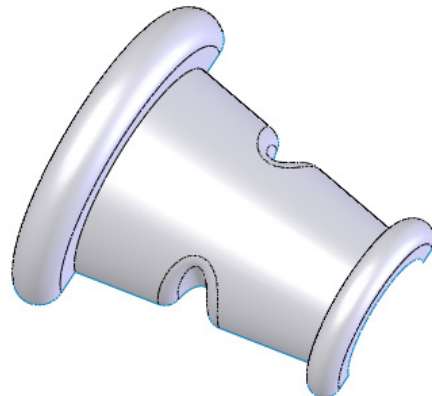


Figure 1-126 Resultant fillet feature

Creating the Lofted Surface

Next, you need to create a lofted surface. The lofted surface will be created between two curves. The first curve is to be created on the Top plane and projected on the surface. The next curve is to be created on an offset plane.

1. Create a plane at an offset distance of 40 mm from the Top plane and then invoke the sketching environment using the top plane as the sketching plane.
2. Create the sketch, as shown in Figure 1-127, and exit the sketching environment.

Next, you need to project this sketch on the base surface.

3. Choose **Curves > Project Curve** from the **Surfaces CommandManager**; the **Projected Curve PropertyManager** is invoked.
4. Select the **Sketch on faces** radio button from the **Projection type** area.
5. Next, select the sketch. When you select the sketch, the **Projection Faces** selection box gets activated.
6. Select the middle portion of the base surface. Choose the **OK** button from the **Projected Curve PropertyManager**.

Next, you need to create second curve for creating the loft surface.

7. Invoke the sketching environment with the newly created plane as the sketching plane and then create the sketch, as shown in Figure 1-128. Then, exit the sketching environment. Figure 1-129 shows the model after creating the sketch and the projected curve.

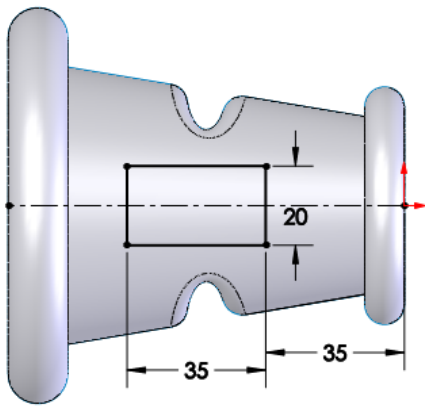


Figure 1-127 Sketch to create the projected curve

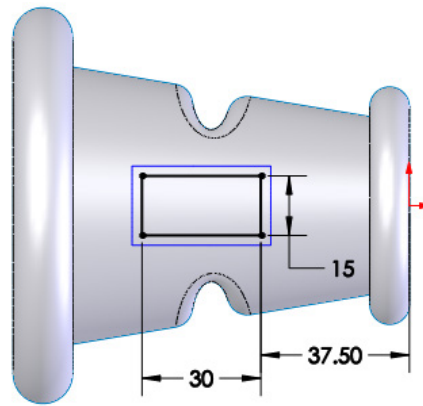


Figure 1-128 Second sketch for the loft surface

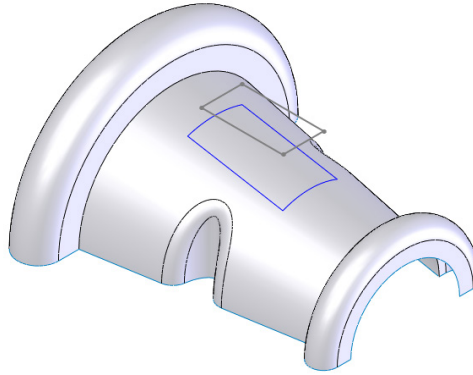


Figure 1-129 Model after creating the sketch and the projected curve

8. Choose the **Lofted Surface** button from the **Surfaces CommandManager**; the **Surface-Loft PropertyManager** is displayed.
9. Select the loft section, as shown in Figure 1-130, and choose the **OK** button from the **Surface-Loft PropertyManager**. Figure 1-131 shows the model after creating the lofted surface.

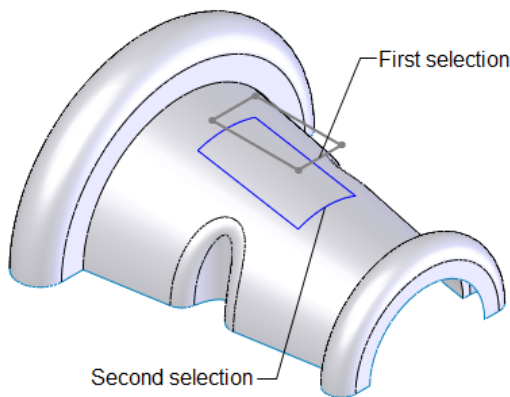


Figure 1-130 Section selected for the lofted surface

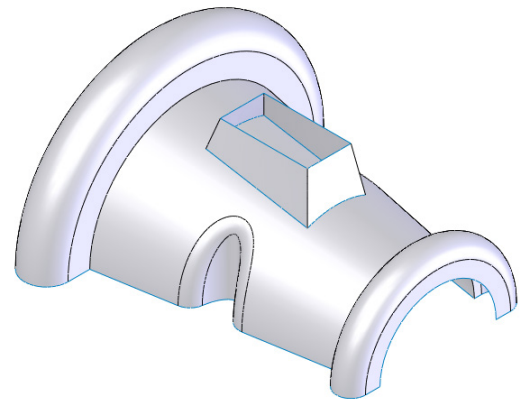


Figure 1-131 Resultant lofted surface

Creating the Planar Surface

Next, you need to create the planar surface by using the top edges of the lofted surface.

1. Choose the **Planar Surface** button from the **Surfaces CommandManager**; the **Planar Surface PropertyManager** is displayed.
2. Select the edges, as shown in Figure 1-132, to create the planar surface; the names of the selected edges are displayed in the **Bounding Entities** selection box of the **Planar Surface PropertyManager**.

- Choose the **OK** button from the **Planar Surface PropertyManager**; the planar surface is created. Figure 1-133 shows the resultant planar surface.

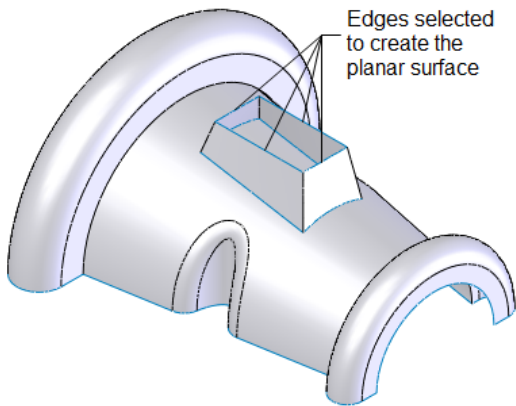


Figure 1-132 Edges selected to create the planar surface

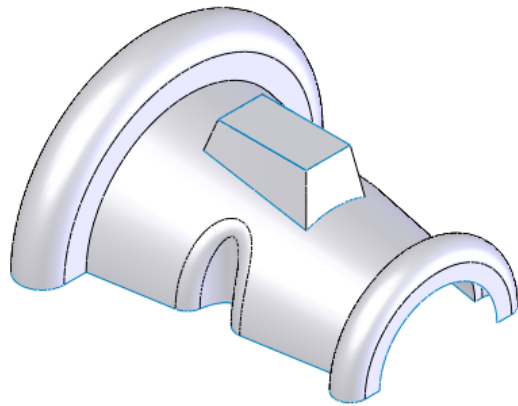


Figure 1-133 Resultant planar surface

Trimming the Base Surface Using the Lofted Surface

Next, you need to trim the base surface by using the lofted surface.

- Choose the **Trim Surface** button from the **Surfaces CommandManager** to invoke the **Trim Surface PropertyManager**.
- Select the **Standard** radio button from the **Trim Type** rollout. Next, select the lofted surface as the trimming tool and then select the base surface as the piece to be retained.
- Choose the **OK** button from the **Trim Surface PropertyManager**.

Figure 1-134 shows the model after the base feature has been trimmed by using the lofted surface.

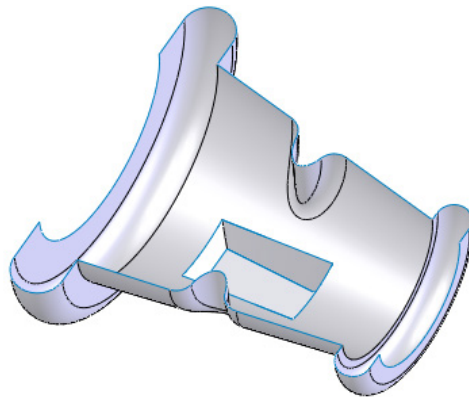


Figure 1-134 Model after trimming the base surface

Knitting All Surfaces

Next, you need to knit the base surface, the lofted surface, and the planar surface together by using the **Knit Surface** tool to create fillets on the edges. Note that, if you want to add wall thickness to the surface model created using multiple surfaces, you need to knit all the surfaces first.

1. Choose the **Knit Surface** button from the **Surfaces CommandManager**; the **Knit Surface PropertyManager** is invoked.
2. Select the base surface, the lofted surface, and the planar surface from the drawing area; the names of the selected surfaces are displayed in the **Surfaces and Faces to Knit** selection box.
3. Next, choose the **OK** button from the **Knit Surface PropertyManager**; the selected surfaces are knitted.
4. Add required fillets to the model. For dimensions of the fillets, refer to Figure 1-118. Final surface model after adding the fillets is shown in Figure 1-135.

Adding Thickness to the Surface Model

Next, you need to add thickness to the surface model.

1. Choose the **Thicken** button from the **Surfaces CommandManager** to invoke the **Thicken PropertyManager**.
2. Select the surface model and then set **1** in the **Thickness** spinner. Next, choose the **Thicken Side 1** button to add thickness in the other direction.
3. Choose the **OK** button from the **Thicken PropertyManager**.

Figure 1-136 shows the rotated view of the final model after thickening the surface.

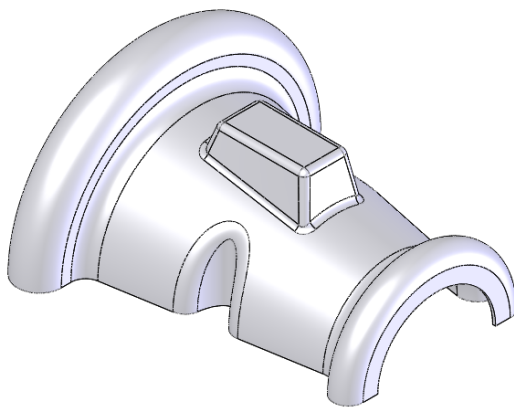


Figure 1-135 Final surface model

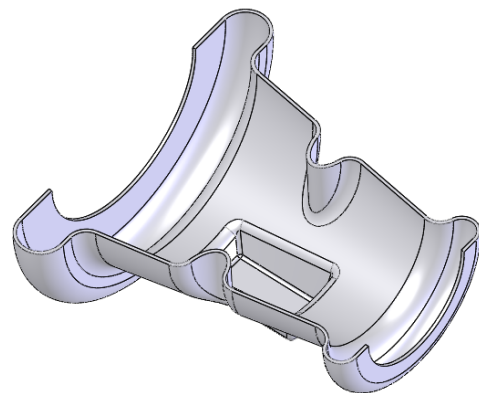


Figure 1-136 Rotated view of the final model after thickening the surface

Saving the Model

1. Choose the **Save** button from the Menu Bar and save the drawing with the name *c01_tut01* at the following location and close the file.

\Documents\SOLIDWORKS\c01

Tutorial 2

In this tutorial, you will create the cover of a hair dryer, as shown in Figure 1-137. First, you will create the model by using surfaces and then thicken it. The views and dimensions of the model are shown in Figure 1-138. **(Expected time: 1.5 hr)**

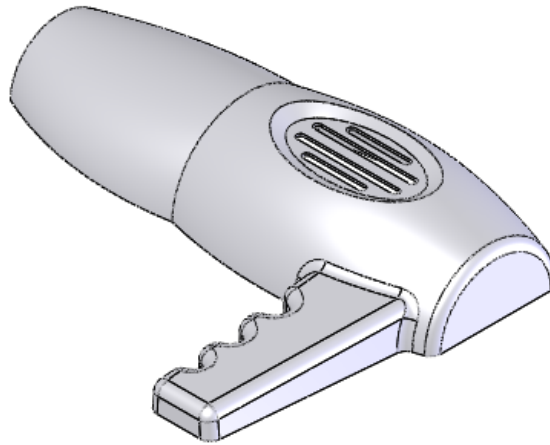


Figure 1-137 Cover of hair dryer

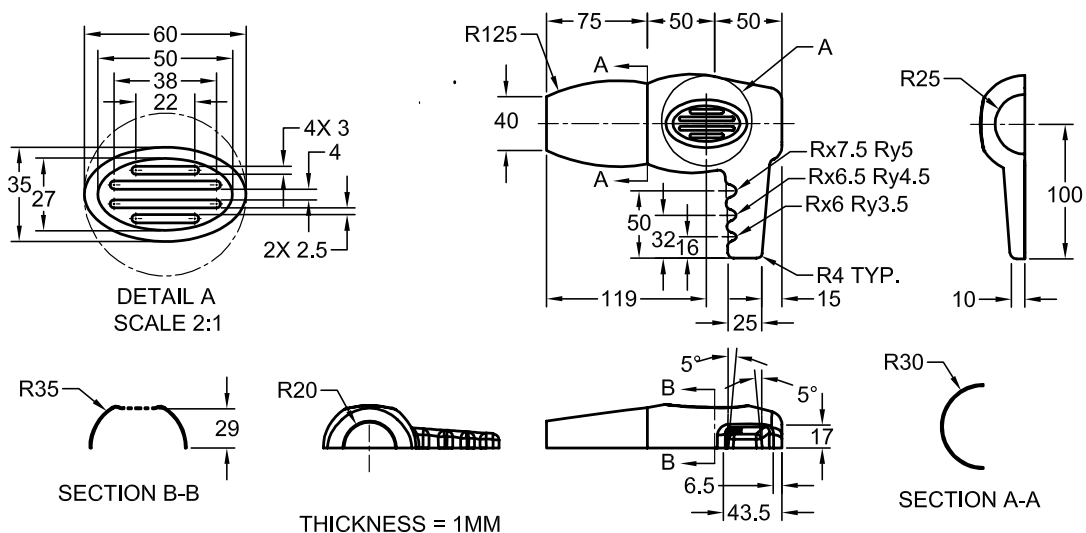


Figure 1-138 Views and dimensions of the model for Tutorial 2

The following steps are required to complete this tutorial:

- a. First, create the base surface. The base surface is created by lofting the open sections along the guide curves.
- b. Create a planar surface to close the right face of the base surface.
- c. Create the basic structure of the handle of the hair dryer cover by creating a lofted surface between two open sections.
- d. Trim the unwanted portion of the lofted surface that is used to create the handle.
- e. Create a planar surface to close the front face of the handle.
- f. Extrude the elliptical sketches to create the grips of the handle and then trim the unwanted surfaces.
- g. Create a dip on the top surface of the hair dryer.
- h. Trim the surface to create air vents.
- i. Knit all surfaces together and add fillets to the model.
- j. Thicken the surface.

Creating the Base Surface

To create the hair dryer cover, you first need to create the base surface of the model. The base surface will be created by lofting semicircular sections along the guide curves. These sections will be created on different planes. Therefore, you first need to create three planes at an offset distance from the Right plane.

1. Start SOLIDWORKS part document using the **New SOLIDWORKS Document** dialog box.
2. Create three planes at an offset distance from the Right plane, as shown in Figure 1-139. For the offset distance of planes, refer to Figure 1-138.
3. Create sections and guide curves to create a lofted surface, as shown in Figure 1-139. For dimensions, refer to Figure 1-138.
4. Invoke the **Lofted Surface** tool and create a lofted surface, as shown in Figure 1-140.
5. Invoke the sketching environment by selecting **Plane3** as the sketching plane.

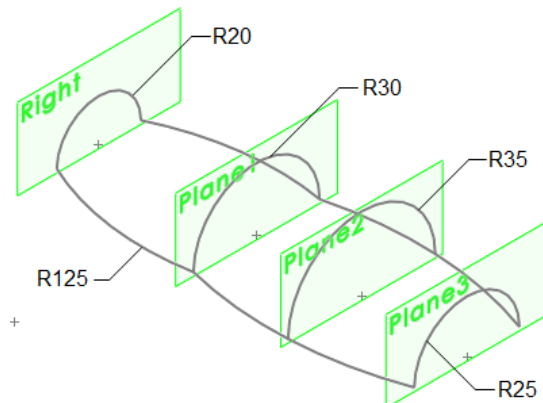


Figure 1-139 Sections and guide curves to create a lofted surface

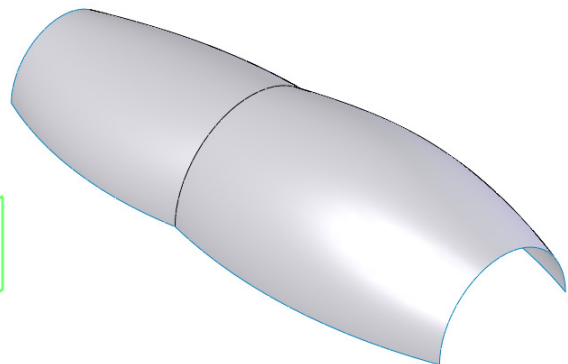


Figure 1-140 Resultant lofted surface

6. Create a closed sketch to create a planar surface, as shown in Figure 1-141.
7. Invoke the **Planar Surface PropertyManager** and then select the closed sketch from the drawing area. Next, choose the **OK** button from the **Planar Surface PropertyManager**; the planar surface is created, as shown in Figure 1-142.

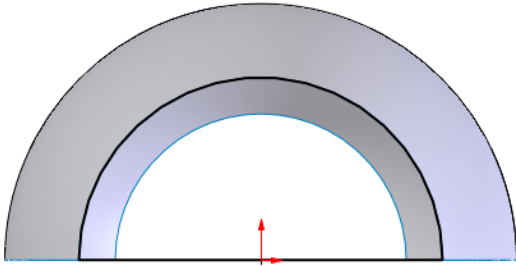


Figure 1-141 Sketch for creating the planar surface

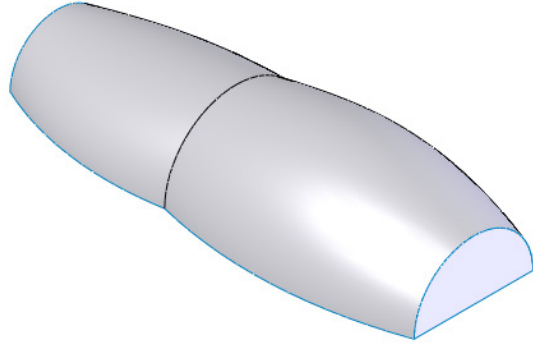


Figure 1-142 Resultant planar surface

Creating the Base Surface for the Handle

Next, you need to create the base surface for the handle. The base surface for the handle will be created by lofting two open sections. The first section for the lofted surface will be created on a plane at an offset distance from the Front plane and the second section will be created on the Front plane. Therefore, you first need to create a plane at an offset distance from the Front plane.

1. Create a plane at an offset distance of **100** mm from the Front plane.
2. Invoke the sketching environment using the newly created plane as the sketching plane.
3. Create an open sketch, as shown in Figure 1-143, and exit the sketching environment.
4. Next, invoke the sketching environment by using the Front plane as the sketching plane.
5. Create an open sketch, as shown in Figure 1-144, and exit the sketching environment.
6. Create the lofted surface by using the **Lofted Surface** tool, as shown in Figure 1-145.

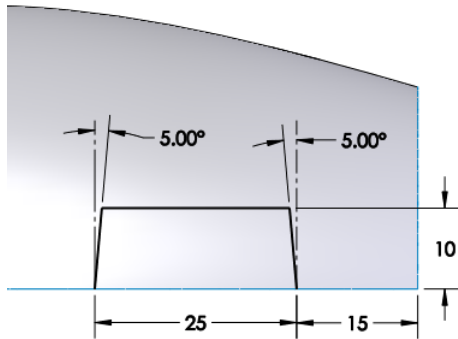


Figure 1-143 Sketch of the first section for creating the lofted surface of the handle

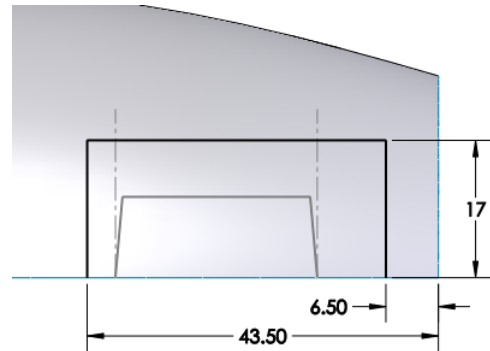


Figure 1-144 Sketch of the second section for creating the lofted surface of the handle

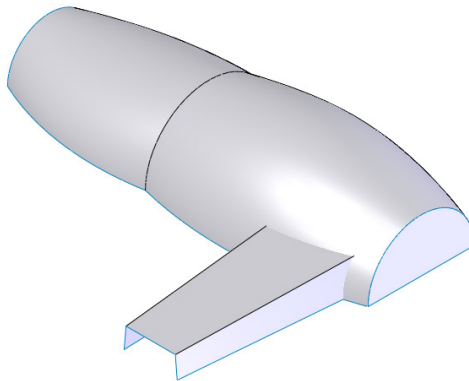


Figure 1-145 Resultant lofted surface

Trimming the Unwanted Portion from the Lofted Surface of the Handle

If you rotate the model after creating the lofted surface for the handle, you will observe that a portion of the lofted surface needs to be trimmed. The method for trimming the unwanted portion of the lofted surface is discussed next.

1. Invoke the **Trim Surface PropertyManager** and then select the **Mutual** radio button from the **Trim Type** rollout.
2. Select the trimming surfaces and the pieces to be kept, as shown in Figure 1-146. Next, choose the **OK** button from the **Trim Surface PropertyManager**. Figure 1-147 shows the resultant trimmed surface.

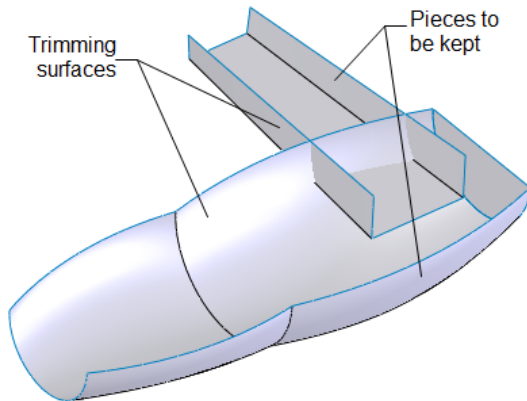


Figure 1-146 Trimming surfaces and pieces to be kept

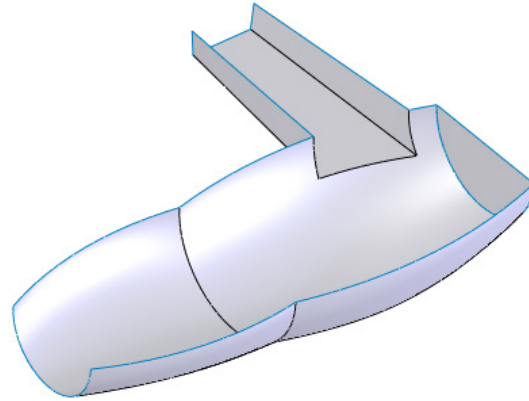


Figure 1-147 Resultant trimmed surface

Creating the Planar Surface

Next, you need to create a planar surface to close the front face of the handle.

1. Select the plane created at an offset distance from the Front plane and invoke the sketching environment.
2. Create a closed sketch to create the planar surface and then exit the sketching environment.
3. Choose the **Planar Surface** button from the **Surfaces CommandManager**; the **Planar Surface PropertyManager** is displayed. Next, select the closed sketch from the drawing area; the name of the selected sketch is displayed in the **Bounding Entities** selection box of the **Planar Surface PropertyManager**. Also, preview of the planar surface is displayed in the drawing area. Next, choose the **OK** button from the PropertyManager.

The model after creating the planar surface is shown in Figure 1-148.

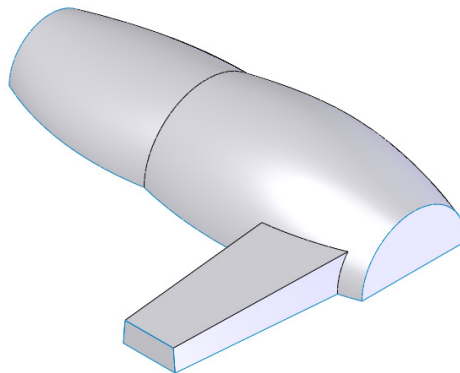


Figure 1-148 Model after creating the planar surface

Creating Grips on the Handle

Next, you need to create grips on the handle of the hair dryer. The grip will be created by extruding the elliptical surface and then trimming unwanted portion of the surfaces.

1. Invoke the sketching environment using the Top plane as the sketching plane.
2. Create a sketch for extruding the surface, as shown in Figure 1-149.
3. Invoke the **Surface-Extrude PropertyManager** and extrude the sketch up to a depth of 25 mm. The extruded surface is displayed, as shown in Figure 1-150.

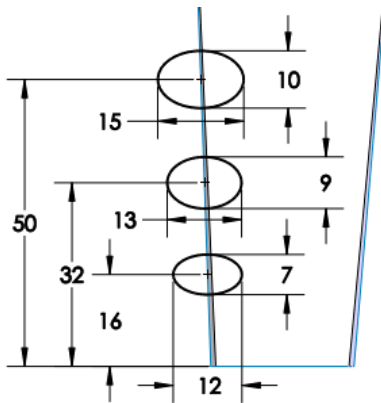


Figure 1-149 Sketch created for extruding the surface

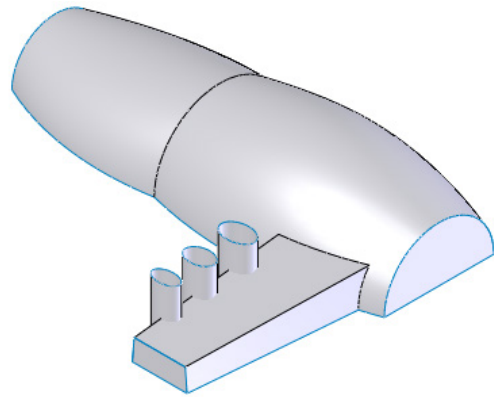


Figure 1-150 Resultant extruded surface

Next, you need to trim the handle and unwanted portions of the extruded surface to achieve desired shape of the grips.

4. Invoke the **Trim Surface PropertyManager** and then select the **Mutual** radio button from the **Trim Type** rollout.
5. Select the trimming surfaces and the pieces to be kept, as shown in Figure 1-151.
6. Choose the **OK** button from the **Trim Surface PropertyManager**; the surface is trimmed. Figure 1-152 shows the resultant trimmed surface.

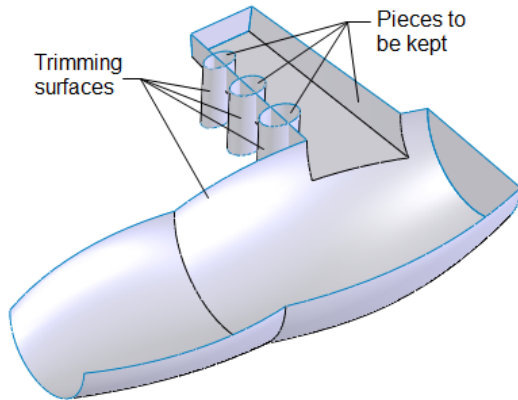


Figure 1-151 Trimming surfaces and the pieces to be kept

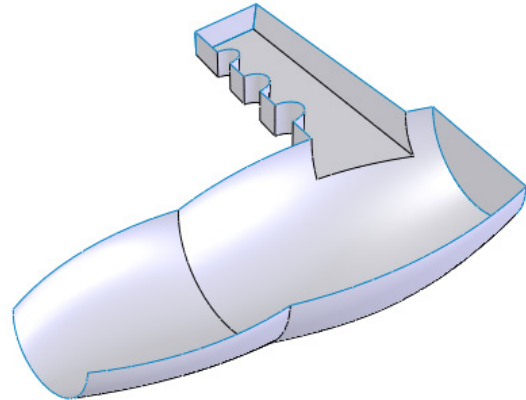


Figure 1-152 Resultant trimmed surface

Creating a Dip on the Upper Surface of the Base Surface

Next, you need to create a dip on the base surface. To do so, you need to use various tools for offsetting planes, creating lofted surface, trimming unwanted surfaces, and creating a planar surface.

1. Invoke the sketching environment using the Top plane as the sketching plane.
2. Create a sketch, as shown in Figure 1-153, and then exit the sketching environment.
3. Next, choose **Curves > Project Curve** from the **Surfaces CommandManager** and project the newly created sketch on the base surface. The model after projecting the sketch is displayed, as shown in Figure 1-154.

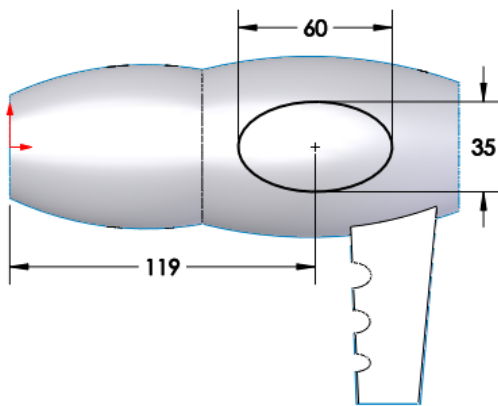


Figure 1-153 Sketch created

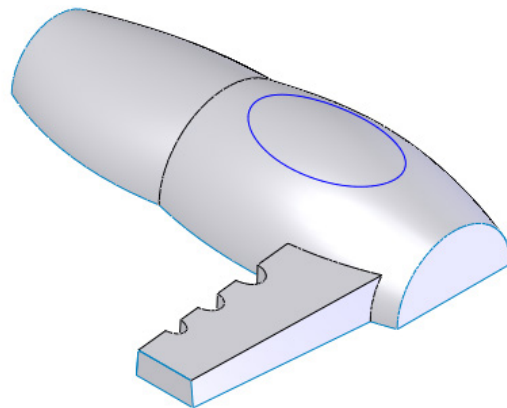


Figure 1-154 Resultant projected curve

4. Create a plane at an offset distance of **29** mm from the Top plane in an upward direction.

5. Next, invoke the sketching environment by using the newly created plane as the sketching plane and create a sketch, as shown in Figure 1-155. Next, exit the sketching environment.
6. Invoke the **Surface-Loft PropertyManager** by choosing the **Lofted Surface** button. Next, create a lofted surface by using the sketch and the projected curve created earlier. The lofted surface after hiding the base surface is shown in Figure 1-156.

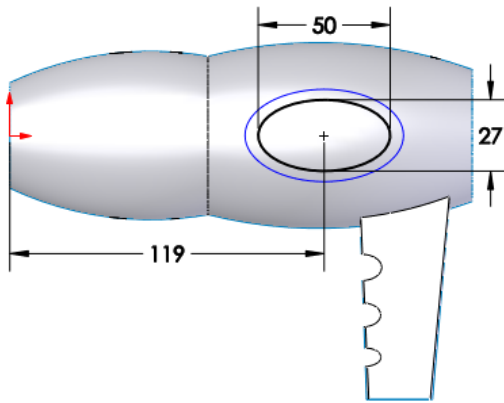


Figure 1-155 Sketch to be created

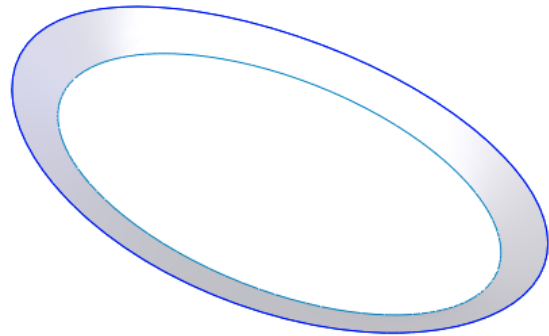


Figure 1-156 Lofted surface

7. Expand the **Surface Bodies** folder in the **FeatureManager Design Tree**. Next, select the surface and right-click on it to invoke the shortcut menu. Next, choose **Hide/Show** from the shortcut menu; the view of the selected surface gets modified.
8. Invoke the **Trim Surface** tool and trim the base surface using the newly created lofted surface. The model after trimming the surface is shown in Figure 1-157. Next, create the planar surface by using the **Planar Surface** tool, as shown in Figure 1-158.

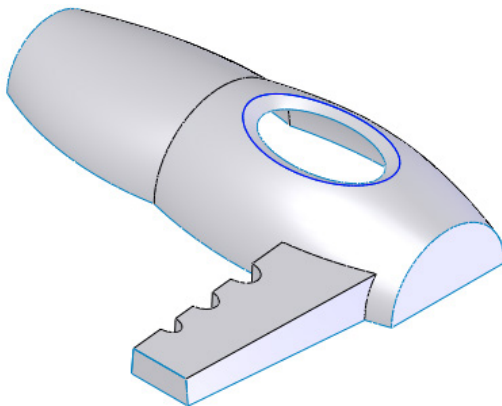


Figure 1-157 Model after trimming the surface

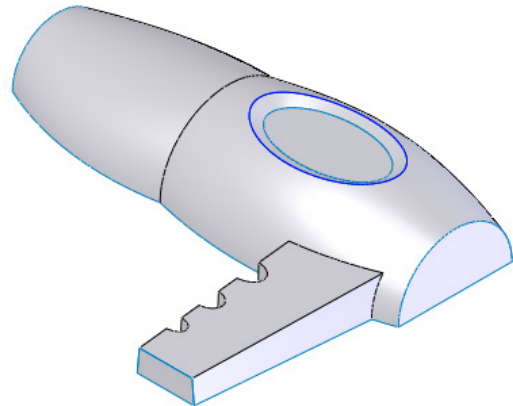


Figure 1-158 Planar surface

Creating Air Vents

Next, you need to create air vents on the newly created planar surface. Air vents are created by drawing the sketch of air vents on the planar surface and then trimming the planar surface.

1. Select the newly created planar surface as the sketching plane and then invoke the sketching environment.
2. Create the sketch of air vents. For dimensions, refer to Figure 1-138.
3. Invoke the **Trim Surface** tool by choosing the **Trim Surface** button from the **Surfaces CommandManager**. Then, select the pieces to be kept or removed from the planar surface.

The surface model after creating air vents is shown in Figure 1-159.

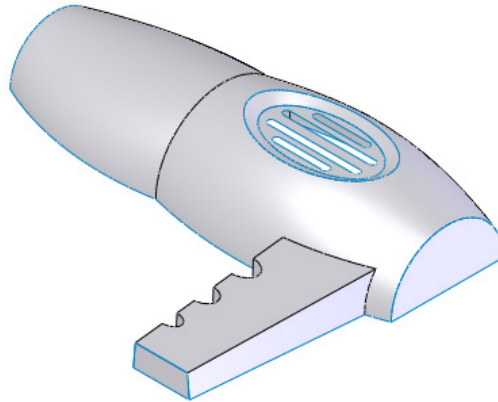


Figure 1-159 Surface model after trimming the planar surface

Knitting all Surfaces

After creating all the surfaces, you need to knit all the surfaces together and then add fillets to surfaces and thicken the model.

1. Choose the **Knit Surface** button from the **Surfaces CommandManager**; the **Knit Surface PropertyManager** is displayed and you are prompted to select the surfaces to be knit.
2. Expand the **Surface Bodies** folder in the design area of the **FeatureManager Design Tree** and then select all surface bodies from it.
3. Next, choose the **OK** button from the **Knit Surface PropertyManager**. If the surfaces do not knit properly then adjust the gaps using the list box provided at the bottom of the **Knit Surface PropertyManager**.
4. Add required fillets to the surface model. The model after adding fillets is displayed in Figure 1-160.

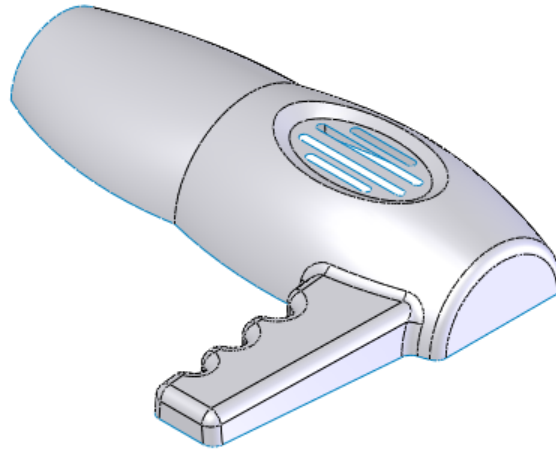


Figure 1-160 Surface model after adding fillets

Adding Thickness to the Surface Model

After creating the entire model, you need to add thickness to the surface model.

1. Choose the **Thicken** button from the **Surfaces CommandManager**; the **Thicken PropertyManager** is invoked and you are prompted to select the surface to thicken.
2. Set the value as **1** in the **Thickness** spinner then choose the **Thicken Side 2** button. Next, select the surface model from the drawing area; preview of the thickened model is displayed in the drawing area.
3. Next, choose the **OK** button from the **Thicken PropertyManager**; the final model is displayed, as shown in Figure 1-161.

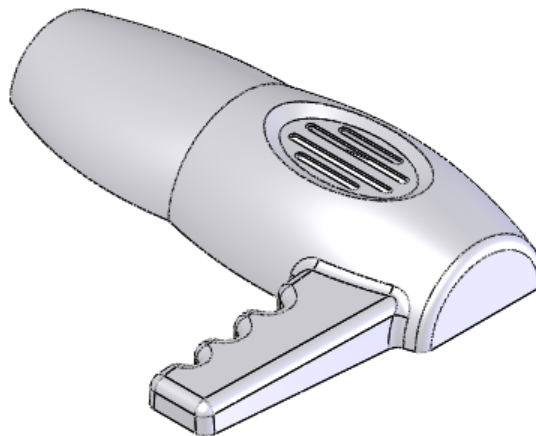


Figure 1-161 Final model

Saving the Model

1. Choose the **Save** button from the Menu Bar and save the drawing with the name *c01_tut02* at the following location and close the file.

\Documents\SOLIDWORKS\c01

Self-Evaluation Test

Answer the following questions and then compare them to those given at the end of this chapter:

1. In SOLIDWORKS, the _____ tool is used to extrude a closed or an open sketch.
2. The _____ **PropertyManager** is used to create a revolved surface.
3. The _____ tool is used to create a surface patch by extending existing surfaces.
4. The _____ tool is used to offset a surface or surfaces up to a given distance.
5. The _____ **PropertyManager** is used to create a lofted surface.
6. You cannot patch the deleted faces of a solid model using the **Delete Face** tool. (T/F)
7. Using the **Knit Surface** tool, you can knit multiple surfaces together to create a single surface. (T/F)
8. You cannot create a filled surface by selecting a 3D sketch as a patch boundary. (T/F)
9. The **Curvature Control** drop-down list is used to define curvature type for creating a fill surface. (T/F)
10. The **SurfaceCut PropertyManager** is used to create a surface cut. (T/F)

Review Questions

Answer the following questions:

1. Which of the following PropertyManagers is used to create a fillet surface?

(a) Fill Surface	(b) Surface Fill
(c) Fillet	(d) None of these
2. Which of the following buttons in the **Surfaces CommandManager** is used to invoke the **Replace Face1 PropertyManager**?

(a) Face Replace	(b) Replace Face
(c) Offset Surface	(d) Fillet Surface

3. Which of the following rollouts is used to define constraint curves while creating a fill surface?
- (a) **Define Constraint Curves** (b) **Constraint Curves**
(c) **Patch Boundaries** (d) None of these
4. Which of the following PropertyManagers is used to create a surface by radiating a surface along an edge or a split line?
- (a) **Surface Fill** (b) **Surface-Sweep**
(c) **Radiate Surface** (d) **Thicken**
5. The _____ **PropertyManager** is used to add thickness to a surface body.
6. To extend a surface linearly, invoke the **Extend Surface PropertyManager** and select the _____ radio button from the **Extension Type** rollout.
7. The _____ check box is used to solidify a closed surface model.
8. The _____ tool is used to delete the faces of a surface or a solid body.
9. The **Mid-Surface** tool is used to create a surface between the two parallel faces of a solid model. (T/F)
10. At least two sections are required to create a closed boundary surface. (T/F)

EXERCISES

Exercise 1

In this exercise, create the model shown in Figure 1-162 by using surfaces. After creating and knitting all surfaces, add thickness to the model. The views and dimensions of the model are shown in the same figure. For better visualization, some fillets are hidden. **(Expected time: 1hr)**

Hint:

In this model, first you need to knit the surfaces and then fillet the edges of the resultant knit surface.

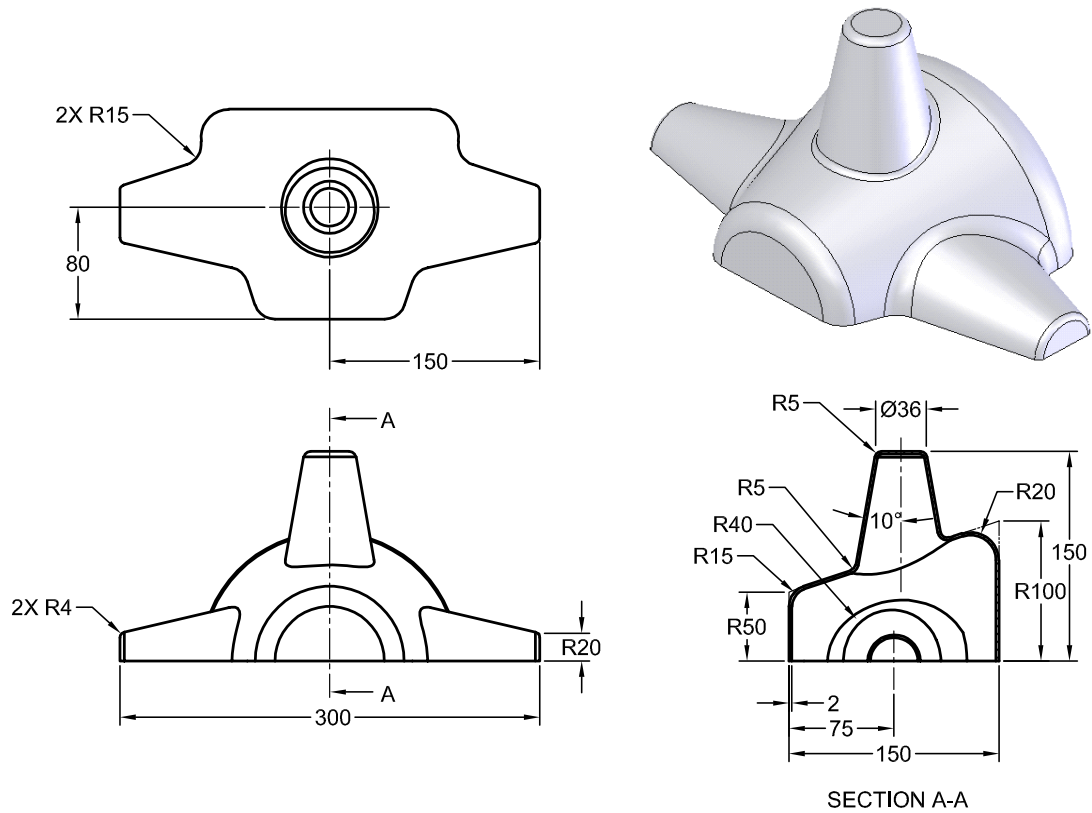


Figure 1-162 Views and dimensions of the model for Exercise 1

Exercise 2

In this exercise, create the model of binoculars shown in Figure 1-163 by using surfaces. First, create a closed surface model, knit all the surfaces together, and then solidify it. The views and dimensions of the model are shown in Figure 1-164.

(Expected time: 1hr)

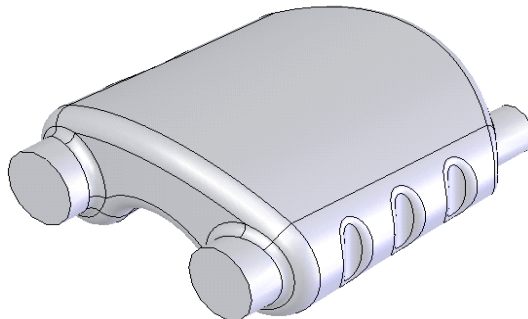


Figure 1-163 Model for Exercise 2

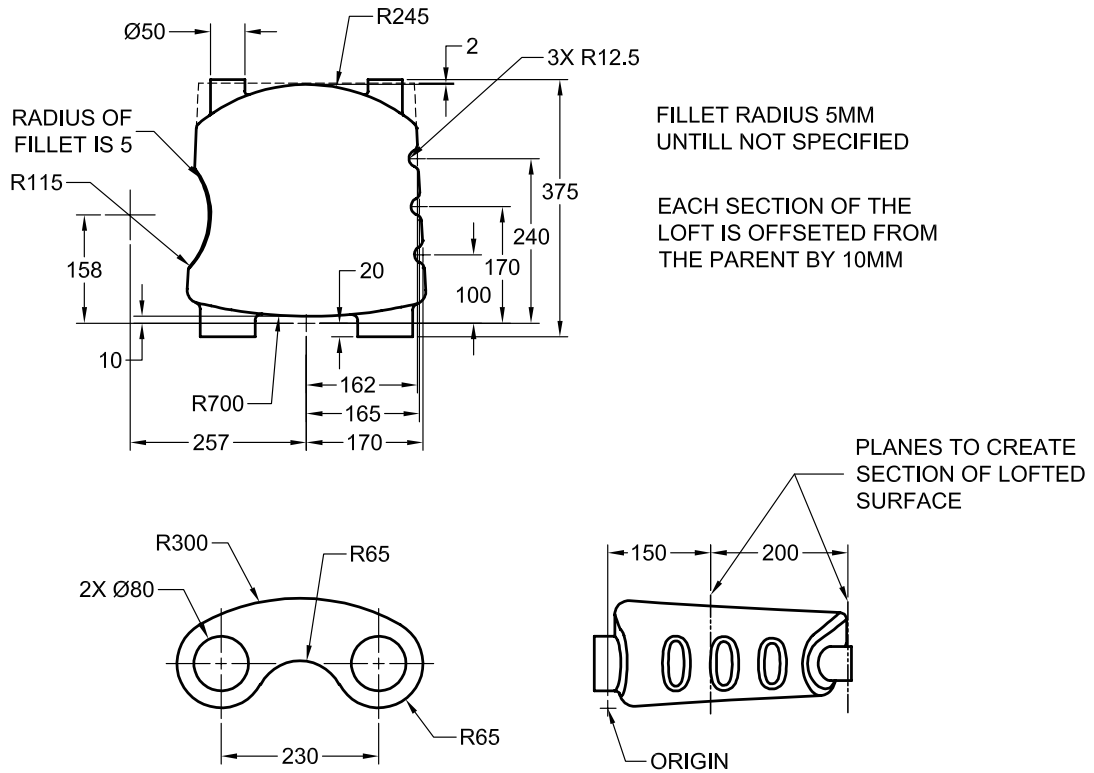


Figure 1-164 Views and dimensions of the model for Exercise 2

Answer to Self-Evaluation Test

1. Extruded Surface, 2. Surface-Revolve, 3. Untrim Surface, 4. Offset Surface, 5. Surface-Loft, 6. F, 7. T, 8. F, 9. T, 10. T