Chapter *15*

Surface Modeling

Learning Objectives

After completing this chapter you will be able to:
• *Create Extruded surface.*
• *Create Revolved surface.*
• *Create Swept surface.*
• *Create Lofted surface.*
• *Create Planar surface.*
• *Create Fill surface.*
• *Create Radiated surface.*
• *Offset the surfaces.*
• *Trim the surfaces.*
• *Untrim the surfaces.*
• *Extend the surfaces.*
• *Knit the surfaces.*
• *Fillet the surfaces.*
• *Create a Mid-surface.*
• *Delete Holes.*
• *Replace the faces.*
• *Delete the faces.*
• *Move and Copy the surfaces.*
• *Thicken the surface body.*
• *Create thicken surface cut.*
• *Create a surface cut.*

SURFACE MODELING

Surface modeling is a technique of creating a planar or non planar geometry of zero thickness. This zero thickness geometry is known as surface. The surfaces are generally used to create models of complex shapes. You can convert surface models in solid models. You can also extract a surface from a solid model using the tools available for surface modeling. This chapter explains the surface modeling tools available in SolidWorks. Using these tools, you can create complex shapes as surfaces and then convert them in 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. This manipulation of surfaces is done using surface modeling. After creating the required complex surface, you can convert it into a solid model. The reasons why you need to convert a surface model into solid model is because a surface is a zero thickness geometry, and so it has no mass and no mass properties. In real world design, many times you need mass and mass properties of a model. The other reason is that you can generate the section view only if the model is solid.

In SolidWorks, the surface modeling is done in the **Part** mode and the tools used for surface modeling are available in the **Surfaces** toolbar. This toolbar is not available by default in the **Part** mode. Therefore, you need to invoke it by choosing **View > Toolbars > Surfaces** from the menu bar. The tools used for surface modeling are also available in the menu bar. Choose **Insert > Surface** from the menu bar to invoke the surface options. You will notice that some of the tools available in the **Surfaces** toolbar are the same as those discussed in solid modeling. These tools are extrude, revolve, sweep, and loft.

The above mentioned tools and the other advanced surface modeling tools are discussed next.

Creating an Extruded Surface

Toolbar:	Surfaces > Extruded Surface
Menu:	Insert > Surface > Extrude

In SolidWorks, the **Extruded Surface** tool is provided to extrude a closed or an open sketch to create an extruded surface. An extruded surface is created using the **Surface-Extrude PropertyManager**. After creating the sketch in the sketcher environment, choose the **Extruded Surface** button from the **Surfaces** toolbar. The **Surface-Extrude PropertyManager** is displayed as shown in Figure 15-1. The preview of the extruded surface with the default value is also displayed in the drawing area. Using the drag handle provided in the drawing area, you can dynamically define the extrusion depth. Using the options available in the **End Condition** drop-down list, you can define the feature termination. The feature termination options available in the **Direction 1** and **Direction 2** rollouts and other options are the same as those discussed in part modeling.

Figure 15-2 shows a closed sketch and Figure 15-3 shows the surface created by extruding the closed sketch. Figure 15-4 shows an open sketch and Figure 15-5 shows the surface created by extruding that open sketch.
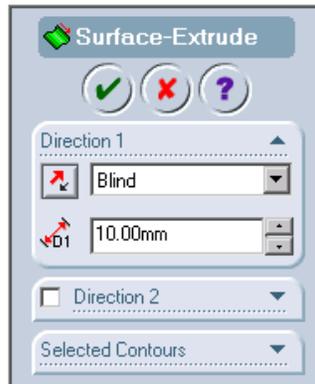
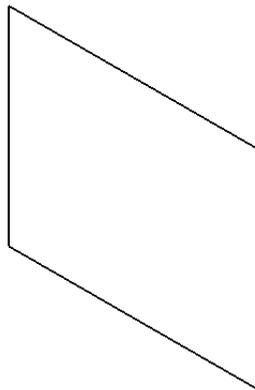*Figure 15-1 The **Surface-Extrude PropertyManager***
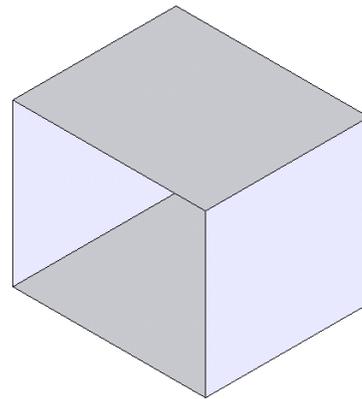

Figure 15-2 A closed sketch

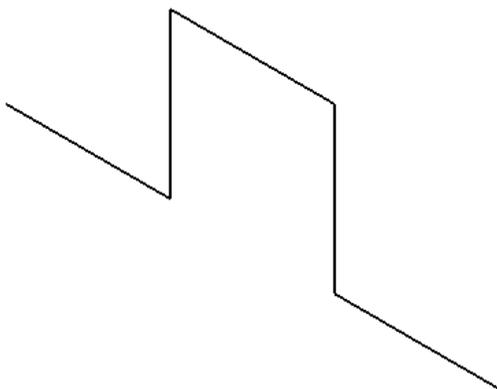*Figure 15-3 Surface created by extruding the closed sketch*

Figure 15-4 An open sketch
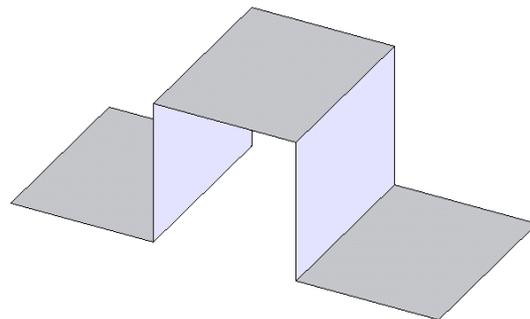
Figure 15-5 Surface created by extruding an open sketch

Creating a Revolved Surface

Toolbar:	Surfaces > Revolved Surface
Menu:	Insert > Surface > Revolve

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 surface is similar to revolving a sketch along a centerline to create a solid feature. To create a revolved surface, choose the **Revolved Surface** button from the **Surfaces** toolbar after creating the sketch and the centerline in the sketching environment. The **Surface-Revolve PropertyManager** will be displayed as shown in Figure 15-6. The preview of the revolved surface with the drag handle is displayed in the drawing area. The feature termination options and other options are similar to those discussed while revolving a solid feature.

*Figure 15-6 The **Surface-Revolve PropertyManager***

Figure 15-7 shows an open sketch for creating a revolve surface. Figure 15-8 shows the resultant revolve surface.
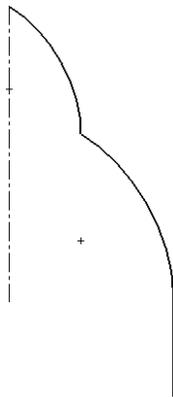

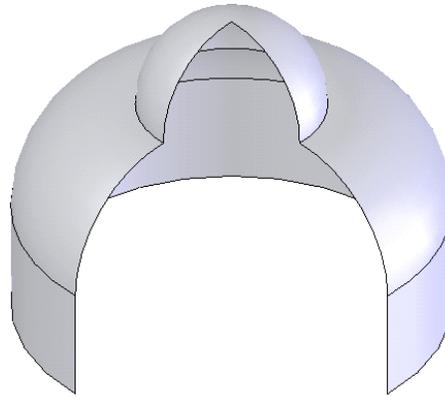*Figure 15-7 Sketch for creating a revolve surface*

Figure 15-8 Surface created by revolving a sketch through an angle of 270-degree

Creating a Swept Surface

|	
Toolbar:	Surfaces > Swept Surface
Menu:	Insert > Surface > Sweep

You can also create a sweep surface by sweeping a closed or an open profile along a closed or an open path. To create a swept feature, choose the **Swept Surface** button from the **Surfaces** toolbar. The **Surface-Sweep PropertyManager** is displayed as shown in Figure 15-9. You are prompted to select a sweep profile. Select a closed sketch or an open sketch as the profile of the sweep feature. Next, you are prompted to select the sweep path. Select a closed or an open sketch as the sweep path. The preview of the sweep feature will be displayed in the drawing area.
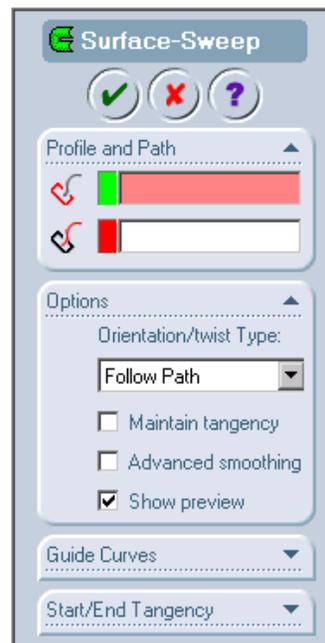

*Figure 15-9 The **Surface-Sweep PropertyManager***

You can also select the guide curves while creating the sweep surface. All the other options used to create a sweep surface are similar to those discussed while creating the solid sweep feature. Figure 15-10 shows an open profile and an open sketch and Figure 15-11 shows the resultant sweep surface. Figure 15-12 shows a closed profile and an open path and Figure 15-13 shows the resultant sweep surface Figure 15-14 shows an open profile and closed path and Figure 15-15 shows the resultant sweep surface.
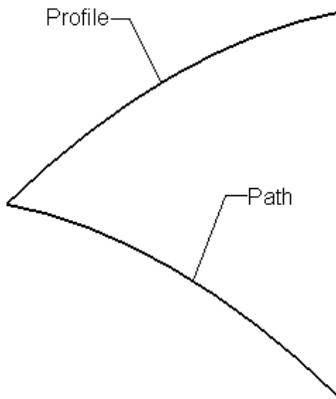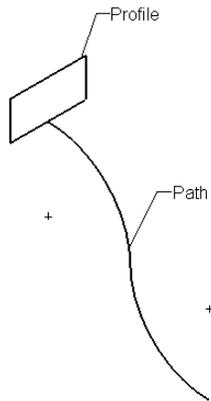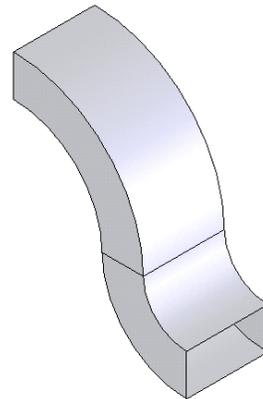
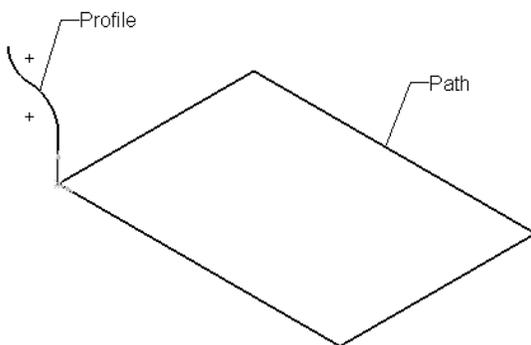***Figure 15-10*** *Open profile and open path*
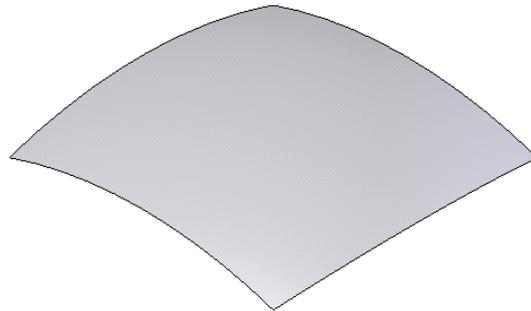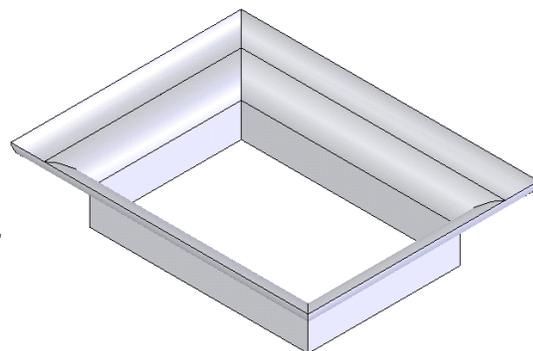

Figure 15-11 *Resultant sweep surface*

Figure 15-12 *Closed profile and open path*

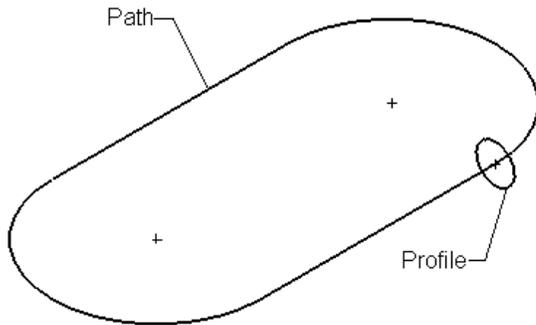***Figure 15-13*** *Resultant sweep surface*

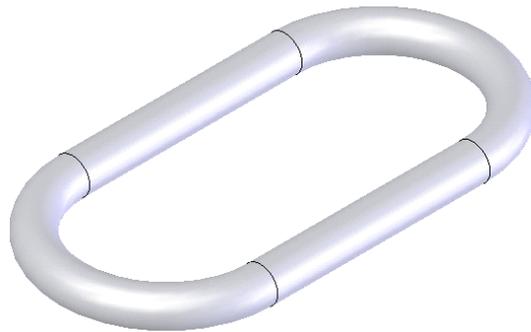***Figure 15-14*** *Open profile and closed path*

Figure 15-15 *Resultant sweep surface*

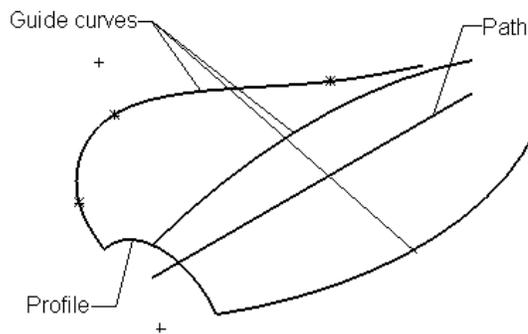Figure 15-16 shows a closed profile and a closed path and Figure 15-17 shows the resultant sweep surface.
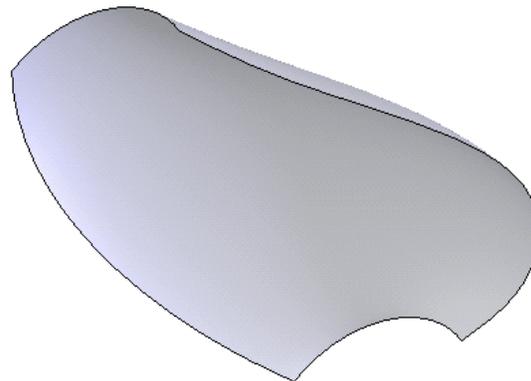
Figure 15-16 Closed profile and closed path *Figure 15-17 Resultant sweep surface*

Figure 15-18 shows the profile, path, and the guide curves and Figure 15-19 shows the resultant sweep surface.

*Figure 15-18 Profile, path, and guide curves* *Figure 15-19 Resultant sweep feature*

Creating a Lofted Surface

Toolbar:	Surfaces > Lofted Surface
Menu:	Insert > Surface > Loft

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** toolbar. The **Surface-Loft PropertyManager** is invoked as shown in Figure 15-20 and you are prompted to select at least 2 profiles. Select the profiles to be lofted. All the options of creating a lofted surface are similar to those are discussed while creating a solid lofted feature.

Remember that if you need to create the loft surface with open section, then all the sections to
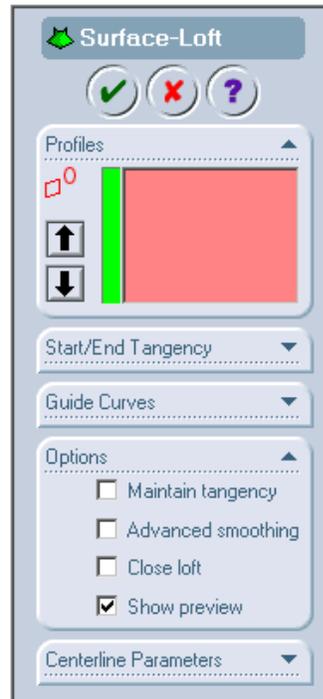
*Figure 15-20 The **Surface-Loft PropertyManager***

be lofted must be open. Similarly, if you need to create a closed lofted surface, then all the sections must be closed. The combination of closed and open sections in a loft surface is not possible. Figure 15-21 shows two open sections to be lofted and Figure 15-22 shows the resultant lofted surface.
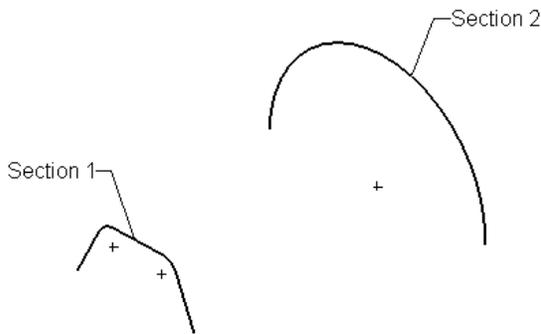
Figure 15-21 Open sections   *Figure 15-22 Resultant lofted surface*
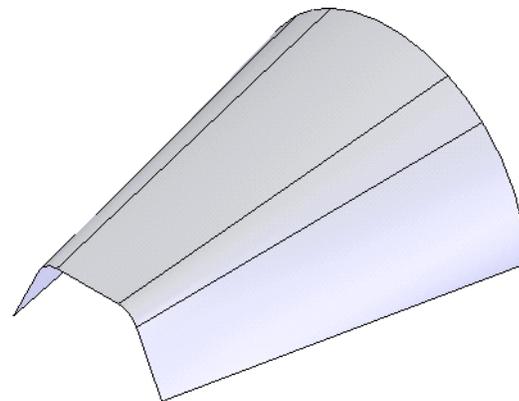
Figure 15-23 shows two closed sections to be lofted and Figure 15-24 shows the resultant lofted surface. Figure 15-25 shows two sections and the centerline and Figure 15-26 shows the resultant lofted surface.

Figure 15-23 *Closed sections*
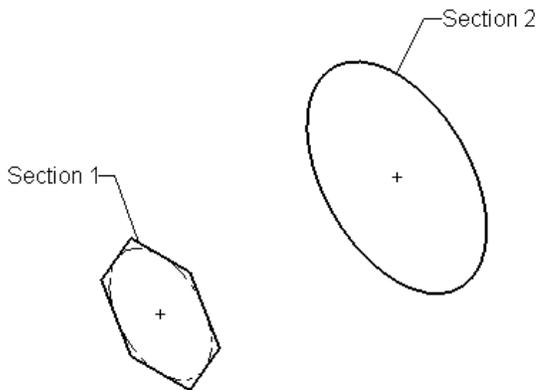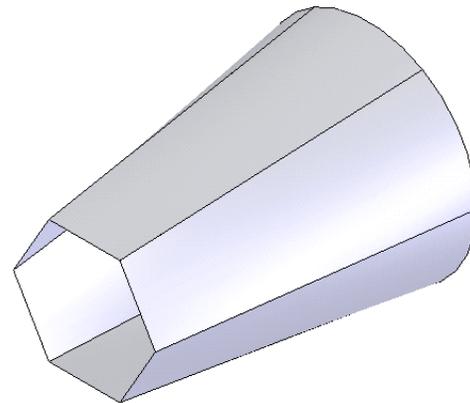

Figure 15-24 *Resultant lofted surface*

Tip. *You do not need to add **Pierce** constraint if you are lofting the sections using the centerline.*
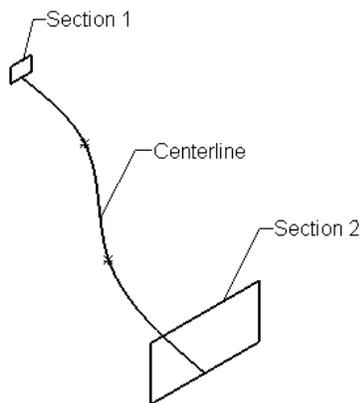
Figure 15-25 *Sections and centerline*
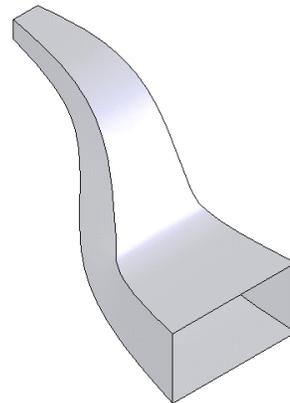
Figure 15-26 *Resultant lofted surface*

Figure 15-27 shows two sections and the guide curves and Figure 15-28 shows the resultant lofted surface.

Creating a Planar Surface

Toolbar:	Surfaces > Planar Surface
Menu:	Insert > Surface > Planar

A planar surface is generally used to fill the gaps between the surfaces using a planar patch. To create a planar surface, you need to choose the **Planar Surface** button from the **Surfaces** toolbar or choose **Insert > Surface > Planar** from the menu bar. The **Planar Surface PropertyManager** is displayed as shown in Figure 15-29 and you are prompted to select a sketch, edge, or curve.

Figure 15-27 *Sections and guide curves*

Figure 15-28 *Resultant lofted surface*

Figure 15-29 *The **Planar Surface PropertyManager***

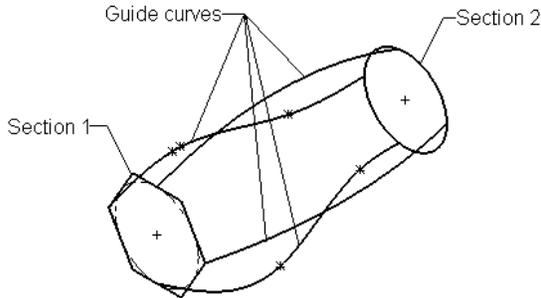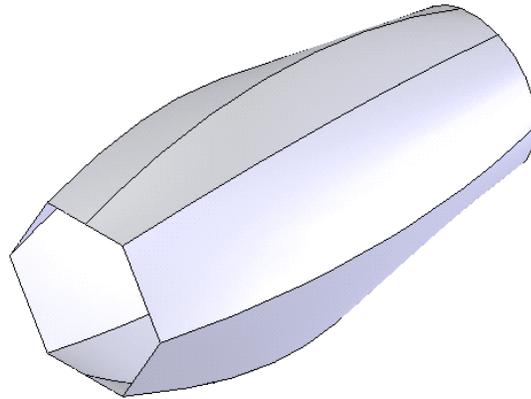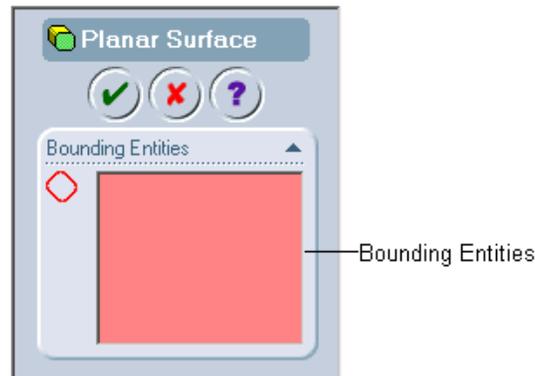After invoking the **Planar Surface PropertyManager**, you need to select the bounding entities. The bounding entities can be sketch, edges, or curves. Select the bounding entities, the name of the bounding entity is displayed in the **Bounding Entities** rollout. After selecting the bounding entities, choose the **OK** button from the **Planar Surface PropertyManager**. A planar surface will be created using the selected entities.

Creating a Fill Surface

Toolbar:	Surfaces > Filled Surface
Menu:	Insert > Surface > Fill

The **Filled Surface** tool is used to create a surface patch along N number of sides. The sides to be selected for creating a fill surface can be the edges of the existing model, 2D or 3D sketch entities, or 2D or 3D curves. The difference between planar surface and fill surface is that you cannot create planar surface using 3D curves or edges. For example, the 3D edge created in Figure 15-30 cannot be used to create planar surface. But you can fill this gap by creating a fill surface on the 3D edge.

To create a fill surface, choose the **Filled Surface** button from the **Surfaces** toolbar. The **Fill**

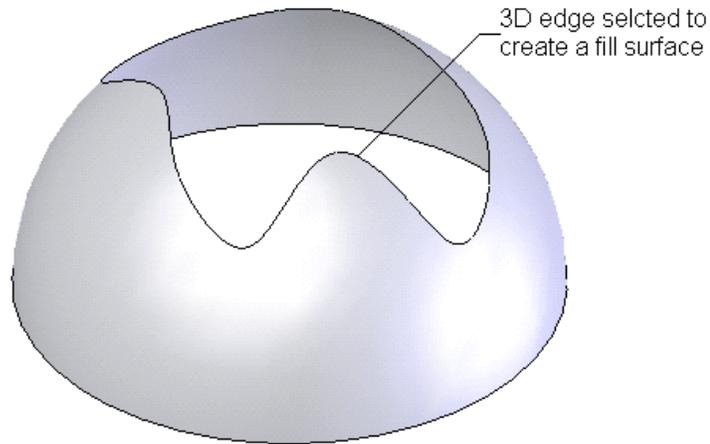3D edge selcted to create a fill surface

Figure 15-30 Entities selected for creating a fill surface

Surface PropertyManager is displayed as shown in Figure 15-31 and you are prompted to select the bounding entities and set the options. Select entities which will define the boundary.
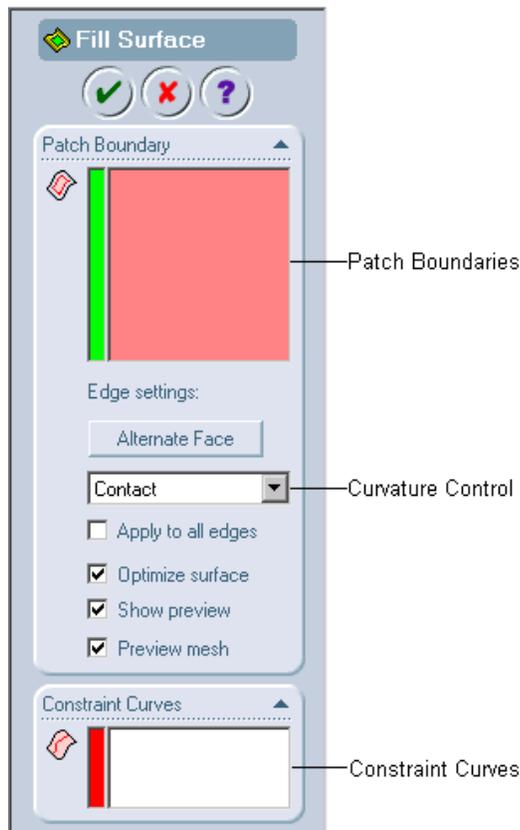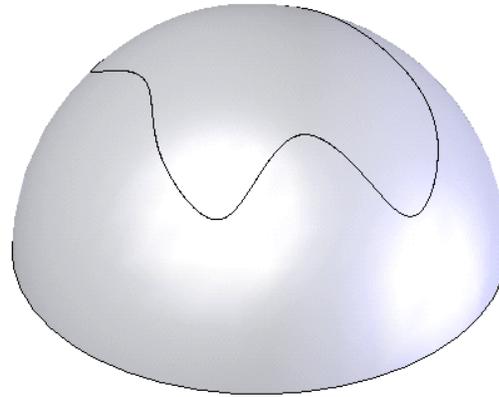

Figure 15-31 The Fill Surface PropertyManager

The selected entities are displayed in green and the callouts attached to them are also displayed. After selecting the last entity that will close the current selection chain, the preview of the surface is displayed in the drawing area and the mesh is also displayed. Now, choose the **OK** button from the **Fill Surface PropertyManager**. Figure 15-32 shows the preview of the fill surface along with the mesh and Figure 15-33 shows the resultant fill surface.
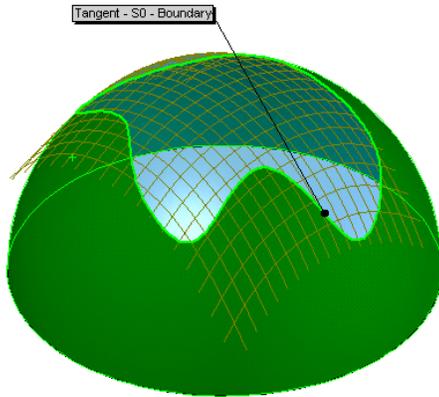
Figure 15-32 Preview of the fill surface along with mesh

Figure 15-33 Resultant fill surface

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

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

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

Alternate Face
The **Alternate Face** button available 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.

Curvature Control
The **Curvature Control** drop-down list is used to define the type of curvature you need to apply on the fill surface that you are creating. There are two types of curvatures that you can apply, these two types of curvatures are discussed next.
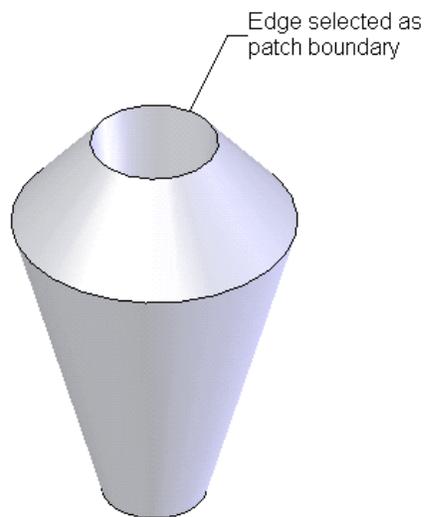
Contact
The **Contact** option is selected by default and is used to create a patch using the fill surface option within the selected patch boundary.

Tangent

The **Tangent** option is used to create a patch using the fill surface option within the selected patch boundary but it also maintains the tangency of the patch created with the selected edges. When you use this option for creating a patch, the **Reverse Surface** button is also displayed. Using this button, you can reverse the direction of the surface created.

Figure 15-34 shows the circular edge selected as patch boundary. Figure 15-35 shows the fill surface created using the **Contact** option selected from the **Curvature Control** drop-down list. Figure 15-36 shows the fill surface created using the **Tangent** option selected from the **Curvature Control** drop-down list.

Figure 15-34 Edge selected as patch boundary
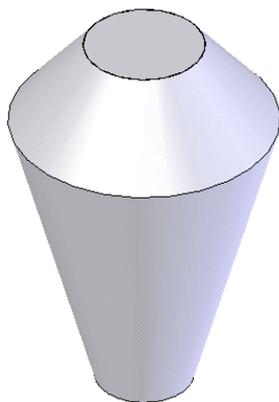
Figure 15-35   Fill surface created using the *Contact* *option*
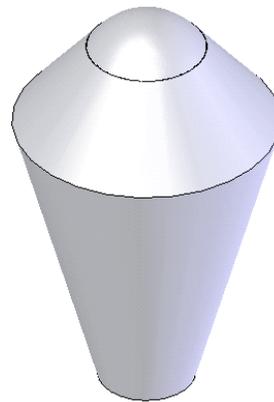
Figure 15-36   Fill surface created using the *Tangent* *option*

Apply to all edges

This option is used to apply the curvature settings discussed earlier to all the edges. If this check box is not selected, the current curvature setting is only applied to the edge of the boundary selected in the **Patch boundary** display area.

Optimize surface

The **Optimize surface** option is used to create the simplest patch of surface along the selected patch boundary. The **Optimize surface** check box is selected by default. When you create a surface patch using the **Optimize surface** check box selected, the time taken to create the surface is less and the model rebuilds faster. When you clear this check box, the **Resolution Control** rollout is displayed as shown in Figure 15-37. The slider available in this rollout is used to specify the resolution of the fill surface. The higher the resolution, better will be the quality of the surface and the more time it will take to rebuild the model. Lower the resolution, the quality of the surface will not be good. However in this case, the rebuilding of the model will take lesser time.

*Figure 15-37 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 created using the selected patch boundary.
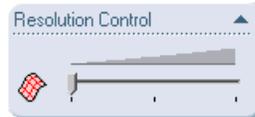
Preview Mesh

The **Preview Mesh** check box is selected by default and is used to display the mesh surface in the preview of the fill surface. This option is only available if the **Show Preview** check box is selected earlier.

Constraint Curves

The **Constraint Curves** rollout is used to define the constraint curves while creating the fill surface. To create a fill surface using the constraining curves, invoke the **Fill Surface PropertyManager** and then select the patch boundary. Now, click once in the **Constraint Curves** area of the **Constraint Curves** rollout to invoke the selection mode and then select the constraint curves. The constraint curve to be selected can be a sketched entity, edge, or a curve. The select constraint curve is displayed in red and the callout is displayed attached to it. The name of the selected entity is displayed in the **Constraint Curve** display area. The preview of the fill surface modifies as you select the constraint curves. After specifying all the constraint curves, choose the **OK** button from the **Fill Surface PropertyManager**. Figure 15-38 shows the sketches selected for patching the boundary and the constraint curves. Figure 14-39 shows the resultant fill surface.
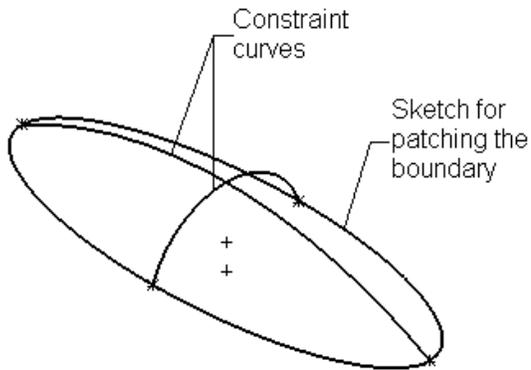
Figure 15-38 *Sketches to be selected for patching the boundary and the constraint curves*
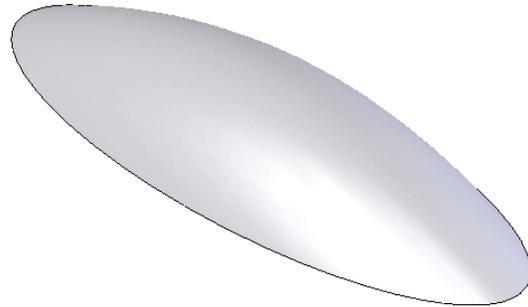

Figure 15-39 *Resultant fill surface*

Creating a Radiated Surface

Toolbar:	Surfaces > Radiate Surface
Menu:	Insert > Surface > Radiate

In SolidWorks, you can also create a surface by radiating a surface along an edge or edges or a split line. The radiated surface is always created parallel to the plane or face selected as radiate direction reference. This type of surface is generally used in mold design as the parting surface for extracting the core and cavity. For creating a radiated surface, choose the **Radiate Surface** button from the **Surfaces** toolbar. The **Radiate Surface PropertyManager** is displayed as shown in Figure 15-40.
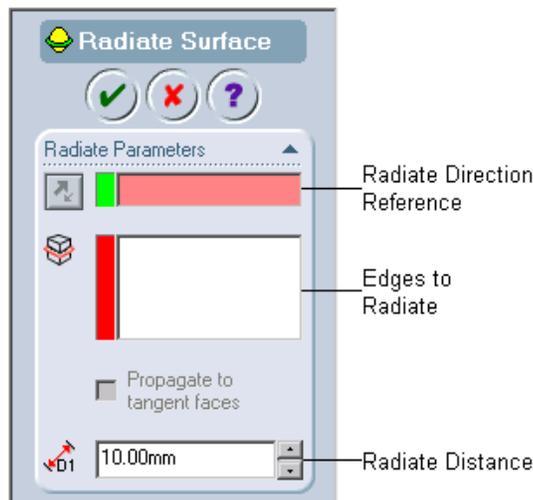

**Figure 15-40** *The **Radiate Surface PropertyManager***

You will observe that the selection is active in the **Radiate Direction Reference** display area. 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 green and an arrow symbol normal to the selected face will be displayed. Now, the selection mode in the **Edges to Radiate** display area is invoked and you need to select the edges along which the surface will be radiated. Select the edges. The name of the selected edges is displayed in the **Edges to Radiate** display area and the arrows showing the direction in which the surface will be radiated is also displayed attached to the selected edges. Now, set the value of the distance of the surface to radiate in the **Radiate Distance** spinner. After setting the radiate distance, choose the **OK** button from the **Radiate Surface PropertyManager**.

The **Propagate to tangent faces** check box is used to radiate the surfaces along all the edges that are tangent to the selected edge.

Figure 15-41 shows the radiate direction reference and the edge to be selected. Figure 15-42 shows the resultant radiated surface with the **Propagate to tangent faces** check box selected.
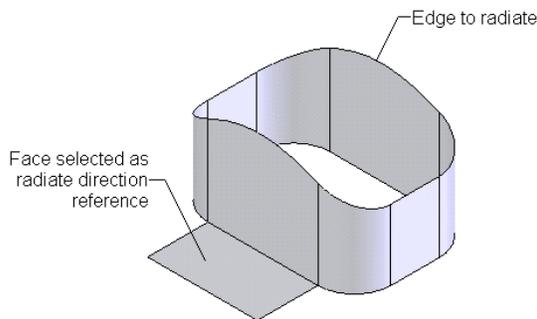
Figure 15-41 Reference and edge to be selected
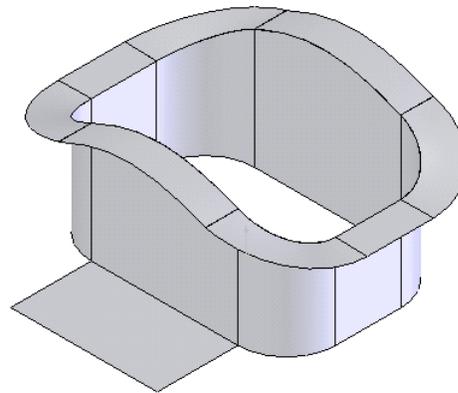
Figure 15-42 Resultant radiated surface with the **Propagate to tangent faces** *check box selected*

Offsetting the Surfaces

Toolbar:	Surfaces > Offset Surface
Menu:	Insert > Surface > Offset

The **Offset Surface** tool is used to offset the selected surface or surfaces to a given distance. For offsetting a surface, choose the **Offset Surface** button from the **Surfaces** toolbar. The **Offset Surface PropertyManager** is displayed as shown in Figure 15-43 and you are prompted to select a face or a surface to offset.

Now, select the face or surface that you need to offset. The selected face or surface is highlighted in green and its name is displayed in the **Surface or Faces to Offset** display area. The preview of the offset surface with the default value is also displayed in the drawing area. Set the value of the offset distance using the **Offset Distance** spinner. You can also flip the direction of the surface creation using the **Flip Offset Direction** button available on the left of the **Offset**
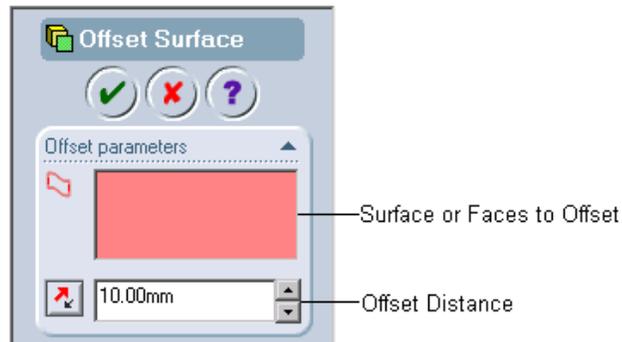
Figure 15-43 The Offset Surface PropertyManager

Direction spinner. After setting all the parameters, choose the **OK** button from the **Offset Distance PropertyManager**. Figure 15-44 shows the surface selected to offset and Figure 15-45 shows the resultant offset surface.
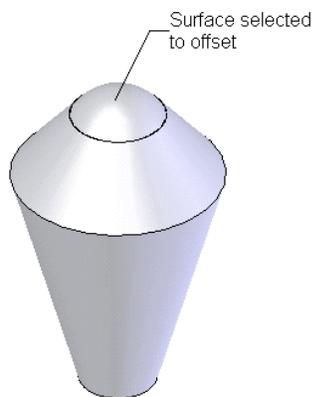

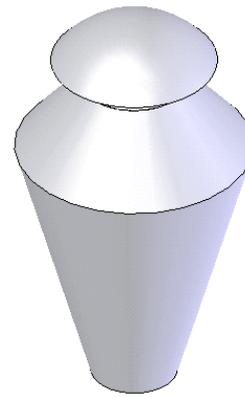*Figure 15-44 Surface selected to offset*


Figure 15-45 Resultant offset surface

Tip. *If you need to extract a surface from the solid or a surface body without specifying the offset distance, then invoke the **Offset Surface PropertyManager** and select the surfaces that you need to extract. Set the value of the **Offset Distance** spinner to **0** and choose the **OK** button from the **Offset Surface PropertyManager**.*

Trimming the Surfaces

Toolbar:	Surfaces > Trimmed Surface
Menu:	Insert > Surface > Trim

The **Trimmed Surface** tool is used to trim the surface using a trim tool. A trim tool can be a surface, sketched entity, or an edge. To trim a surface, choose the **Trimmed Surface** button from the **Surfaces** toolbar. The **Trim Surface PropertyManager** is displayed as shown in Figure 15-46 and you are prompted for selecting a trimming surface or sketch followed by the pieces to keep.
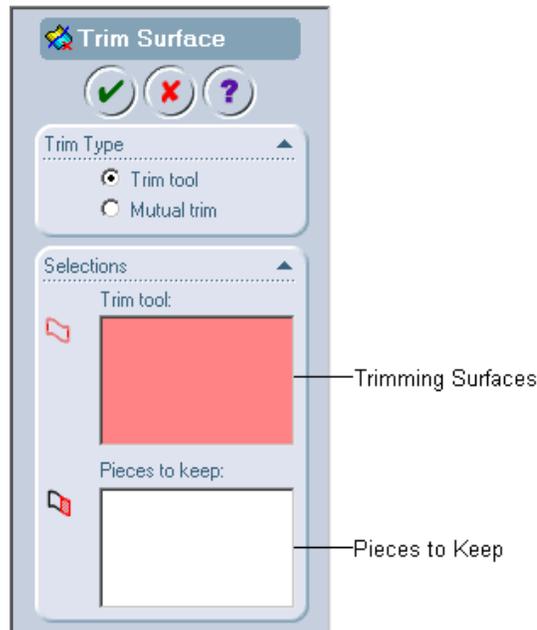
Figure 15-46 The Trim Surface PropertyManager

The **Trim tool** radio button is selected by default. The trimming of surfaces using this option is discussed next.

When you invoke this tool, you will notice that the cursor is replaced by the trim tool cursor. Select the trimming surface using this cursor. You can select a surface, sketch, or an edge. The selected entity is highlighted in green and the name of the trimming surface is displayed in the **Trimming Surfaces** display area. The selection mode in the **Pieces to keep** display area is active and you are prompted to select the pieces to keep. Also, the cursor is replaced by the surface body cursor. Move the cursor on the surface that is being trimmed, the pieces of the surface on which you place the cursor are displayed in blue. Select the piece or pieces of the surface to keep. The selected pieces will be displayed in yellow and their names will be displayed in the **Pieces to keep** display area. Choose the **OK** button from the **Trim Surface PropertyManager**.

Figure 15-47 shows the trimming surface and the piece to keep after trimming. Figure 15-48 shows the resultant trimmed surface. Figure 15-49 shows the sketch selected as a trimming surface and Figure 15-50 shows the resultant trimmed surface.

The other method of trimming a surface is known as **Mutual trim** method. In this method, you need to select two surfaces as trimming surfaces. To trim the surfaces using this method, invoke the **Trim PropertyManager** and then choose the **Mutual trim** radio button from the **Trim Type** rollout. You are prompted to select the surfaces to trim, followed by the pieces to keep. Select the trimming surfaces and then select the pieces to keep, refer to Figure 15-51 and choose the **OK** button from the **Trim PropertyManager**. Figure 15-52 shows the resultant trimmed surface.
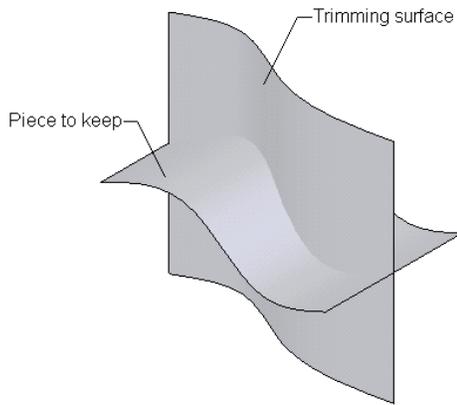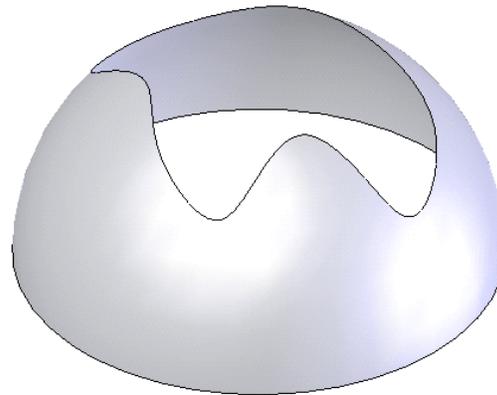
Figure 15-47 *Trimming surface and piece to keep after trimming the surface*
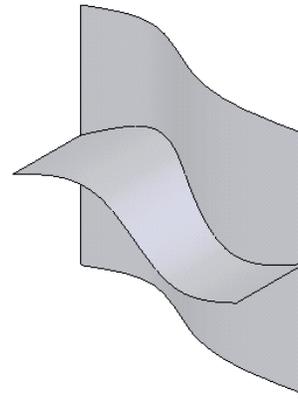
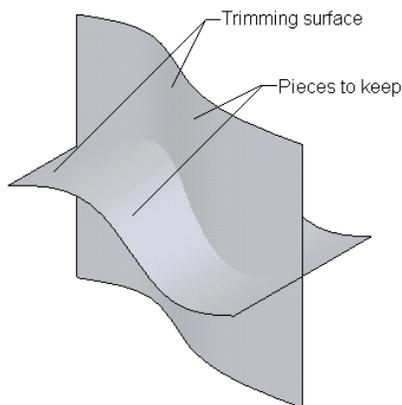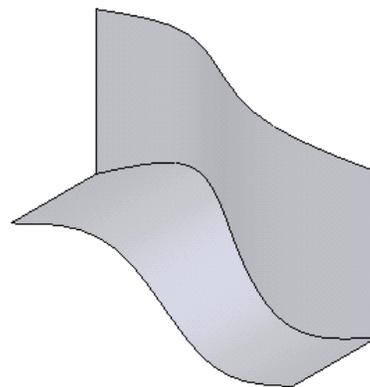*Figure 15-48* *Resultant trimmed surface*
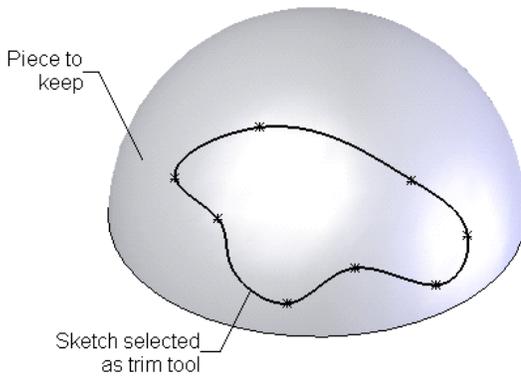
Figure 15-49 *Sketch selected as trimming surface*

Figure 15-50 *Resultant trimmed surface*

Figure 15-51 *Surfaces to trim and the pieces to keep*

Figure 15-52 *Resultant trimmed surface*

Untrimming the Surface

Toolbar:	Surfaces > Untrim Surface	(*Customize to Add*)
Menu:	Insert > Surface > Untrim	

The **Untrim Surface** tool is used to create a surface patch by extending the existing surfaces. Using this option, the trimmed portion of a surface is filled with a surface patch. To untrim a surface, choose **Insert > Surface > Untrim** from the menu bar. You can also use the **Untrim Surface** button from the **Surfaces** toolbar, but you first need to customize this button. As you invoke this tool, the **Untrim PropertyManager** is invoked as shown in Figure 15-53 and you are prompted to select surface bodies or edges of surface for the Untrim operation.
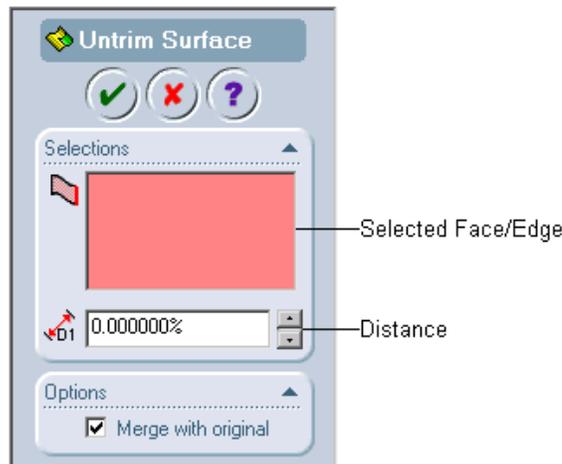

Figure 15-53 *The* ***Untrim Surface PropertyManager***

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

Untrimming by Selecting the Faces

In this method you will untrim the surface by selecting the face or faces of the surface to untrim. Invoke the **Untrim PropertyManager** and select the face of the surface that you need to untrim. The preview of the untrimmed surface with default settings is displayed in the drawing area. Additional options are also displayed in the **Options** rollout as shown in Figure 15-54. These options are discussed next.

Face untrim type
The **Face untrim type** area is used to specify the type of edges along which you need to untrim the surface. The options available in this area are discussed next.
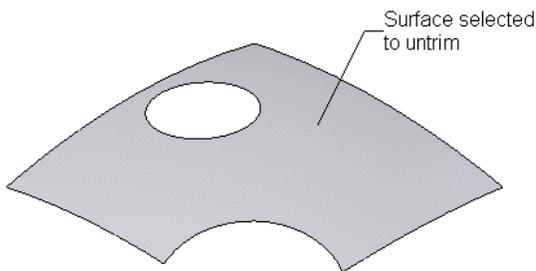
All edges
The **All edges** radio button is selected by default. Therefore, all the internal and
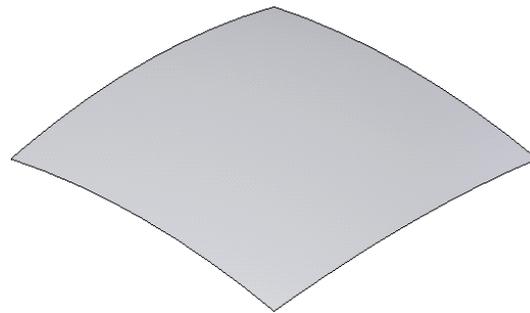
Figure 15-54 The **Options** *rollout*

external edges of the selected surface are extended to be untrimmed. Figure 15-55 shows the surface to be selected and Figure 15-56 shows the untrimmed surface using the **All edges** radio button selected.

Figure 15-55 Surfaces selected to untrim

Figure 15-56 Resultant untrimmed surface with the **All edges** *radio button selected*

Internal edges
The **Internal edges** radio button is used if you need to patch only the internal edges of the selected surface using the **Untrim Surface** tool. Figure 15-57 shows the untrimmed surface created with the **Internal edges** radio button selected.
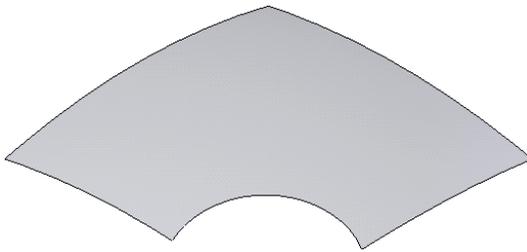
External edges
The **External edges** radio button is used if you need to patch only the external edges of the selected surface using the **Untrim Surface** tool. Figure 15-58 shows the untrimmed surface created with the **External edges** radio button selected.

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

You can also specify the percentage of distance to which you need to extend the surface depending on the edges type selected from the **Face untrim type** area of the **Options**

Figure 15-57 Resultant untrimmed surface
*with the **Internal edges** radio button selected*
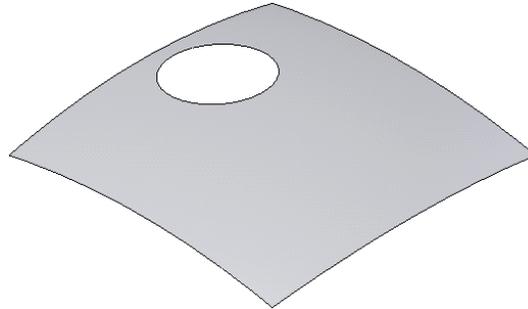
Figure 15-58 Resultant untrimmed surface
*with the **External edges** radio button selected*

rollout. The **Distance** spinner is used to define the percentage distance of extension of surface. The preview of the surface extension is displayed in the drawing area.

Untrimming by Selecting the Edges

You can also patch a trimmed surface using the **Untrim Surface** tool by selecting the edges of the trimmed portion of the surface. Invoke the **Untrim Surface PropertyManager** and select the edge of the surface along which you need to patch the trimmed surface. Now, select the edge, additional options in the **Options** rollout are displayed as shown in Figure 15-59. The options available in this rollout are discussed next.

*Figure 15-59 The **Options** rollout displayed when the **Edge** is*
*selected to be patched using the **Untrim Surface** tool*

Edge untrim type

The **Edge untrim type** area is used to specify the options to patch the trimmed surface using the selected edges. The options available in this rollout are discussed next.
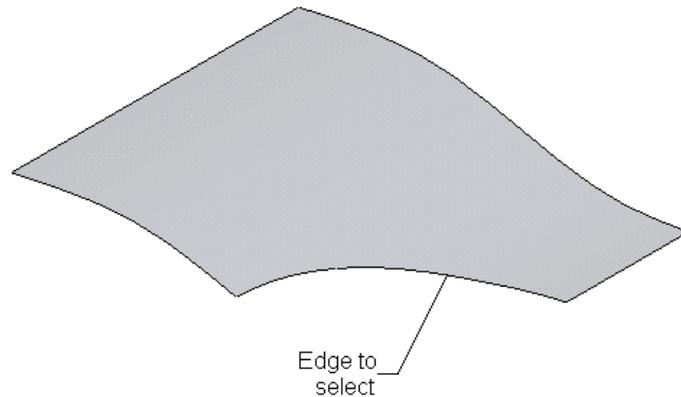
Extend edges

The **Extend edges** radio button is selected by default and it extends the edge to create a corner to untrim the trimmed surface.
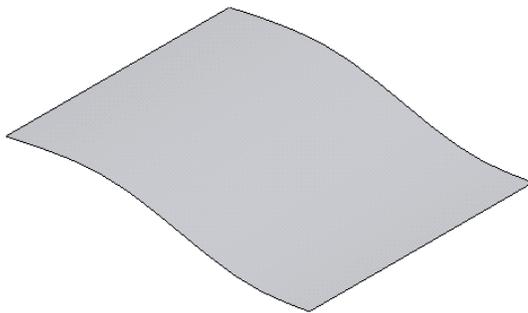
Connect endpoints

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

Figure 15-60 shows the edge to be selected for untrimming a surface. Figure 15-61 shows an untrimmed surface created using the **Extend edges** check box selected. Figure 15-62 shows the untrimmed surface created using the **Connect endpoints** check box selected.

Edge to
select

Figure 15-60 Edge selected for untrimming the surface

*Figure 15-61 Untrimmed surface created with **Extend edges** radio button selected*
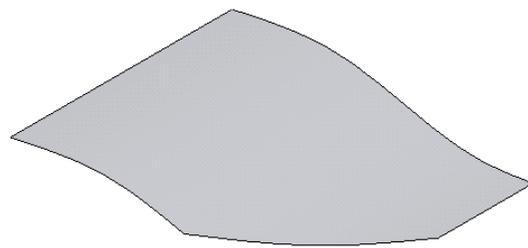
*Figure 15-62 Untrimmed surface created with **Connect endpoints** radio button selected*

Extending the Surfaces

Toolbar:	Surfaces > Extended Surface
Menu:	Insert > Surface > Extend

The **Extended Surface** tool is used to extend the surface along the selected edge or the selected face. To extend the surface, choose the **Extended Surface** button from the **Surfaces** toolbar. The **Extend Surface PropertyManager** is displayed as shown in Figure 15-63 and you are prompted to select a face or edge(s) and set the properties. There are two options of extending a surface, both of which are discussed next.
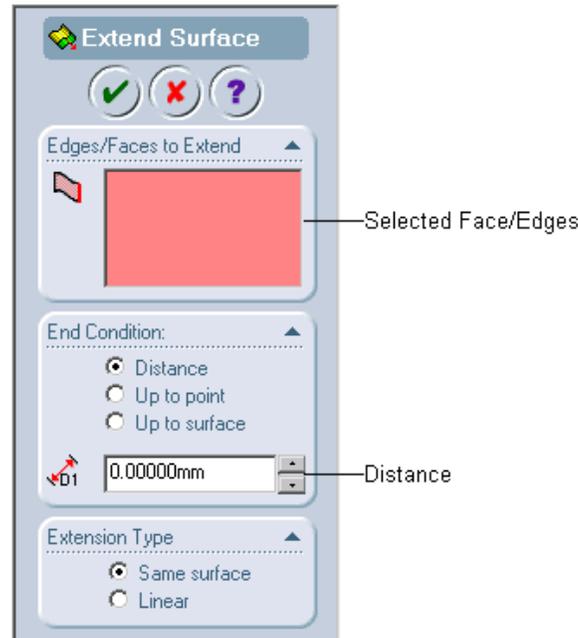
*Figure 15-63 The **Extend Surface PropertyManager***

Extending using Same surface Option

When you extend a surface using the **Same surface** option, the surface is extended by maintaining the curvature. To extend the surface using the **Same surface** option, invoke the **Extend Surface PropertyManager**. You will observe that the **Same surface** radio button is automatically selected in the **Extension Type** rollout. Now, you need to select the edge or the face that you need to extend. Note that when you select the face to extend the surface, the surface will extend equally in all the directions. Using the drag handle, extend the surface dynamically or set the extending distance in the **Distance** spinner. You can also use the other feature termination options available in the **End Condition** rollout.

Figure 15-64 shows the edge selected to extend the surface. Figure 15-65 shows the preview of the surface being extended by selecting the edge with the **Same surface** radio button selected. Figure 15-66 shows the face selected to extend the surface. Figure 15-67 shows the preview of the surface being extended by selecting the face with the **Same Surface** radio button selected.

Note

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

Extending the Surface in the Linear Direction to An Existing Surface

To extend the surface in the linear direction to an existing surface by maintaining the tangency, invoke the **Extend Surface PropertyManager** and select the **Linear** radio button from the **Extension Type** rollout. Now, select the face or the edge along which you need to extend the surface and specify the feature termination using the **End Condition** rollout.
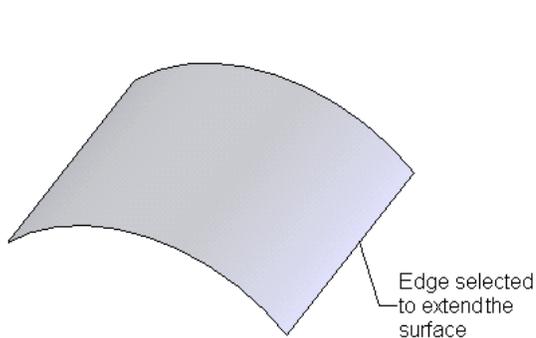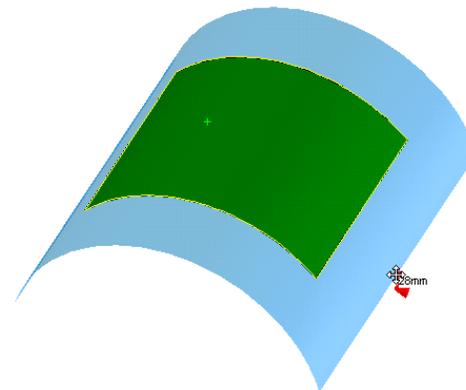
Figure 15-64 Edge selected to extend the surface
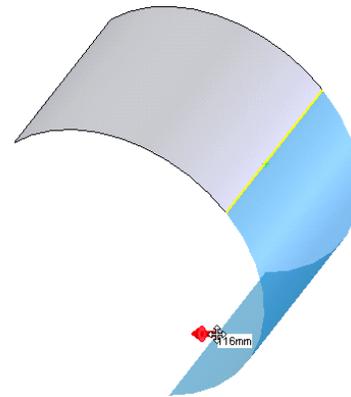
*Figure 15-65 Preview of the extended surface with the **Same surface** radio button selected*
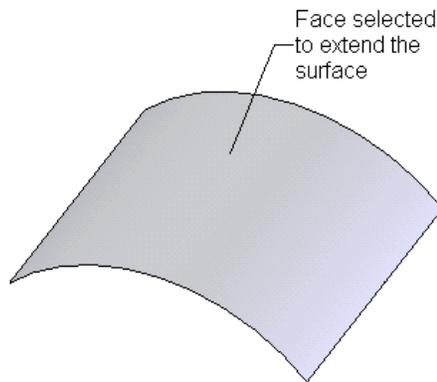
Figure 15-66 Face selected to extend the surface

*Figure 15-67 Preview of the extended surface with the **Same surface** radio button selected*

Figure 15-68 shows the edge selected to extend the surface and Figure 15-69 shows the preview of the surface being extended with the **Linear** radio button selected. Figure 15-70 shows the face selected to extend the surface and Figure 15-71 shows the preview of the surface being extended with the **Linear** radio button selected.
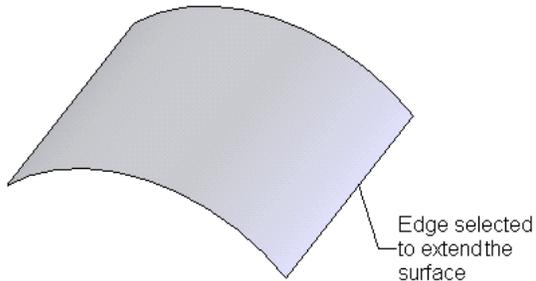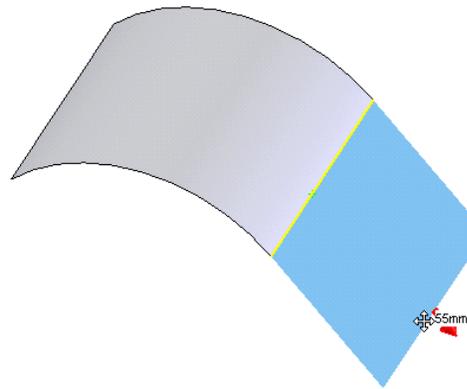
Figure 15-68 *Edge selected to extend the surface*

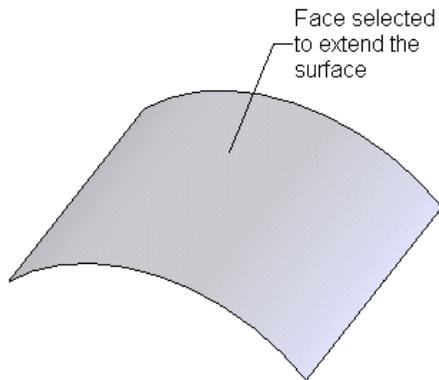***Figure 15-69*** *Preview of the extended surface*

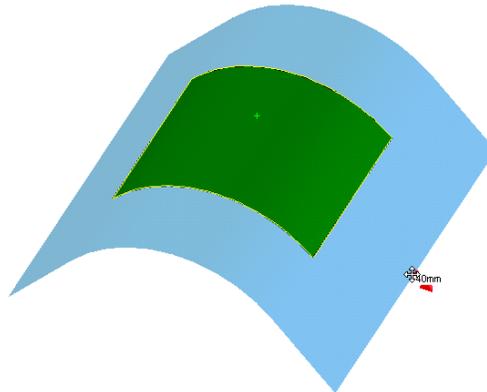***Figure 15-70*** *Face selected to extend the surface*

Figure 15-71 *Preview of the extended surface with the* ***Linear*** *radio button selected*

Knitting the Surfaces

Toolbar:	Surfaces > Knit Surface
Menu:	Insert > Surface > Knit

Using the **Knit Surface** tool you can knit the surfaces together to create a single surface. You can also knit a surface with the faces of a solid body. The surfaces that you need to knit together must have contact with each other. You cannot knit the disjoint surfaces or faces. The knit surface option is widely used while extracting core and cavity while designing a mold. To knit the surfaces, choose the **Knit Surface** button from the **Surfaces** toolbar, the **Knit Surface PropertyManager** is invoked as shown in Figure 15-72.

You are prompted to select the surfaces to knit. Select the surfaces that you need to knit, the names of the surfaces are displayed in the **Surfaces and Faces to Knit** display area. After selecting all the surfaces and faces choose the **OK** button from the **Knit Surface PropertyManager**.
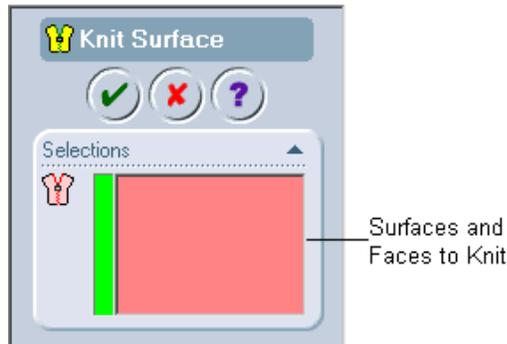
*Figure 15-72 The **Knit Surface PropertyManager***

Filleting the Surfaces

Toolbar:	Features > Fillet
Menu:	Insert > Surface > Fillet/Round

Filleting the surfaces is the same as discussed earlier while filleting the solid models. All the options are the same except some exceptions. These exceptions are discussed next.

1. You cannot create a variable radius on the fillet edges.

2. While applying the face fillet to the surface, you need to define the direction in which you need to add the fillet.

3. You cannot use the **Keep features** option while filleting a surface.

4. You cannot select the surface using the **FeatureManager Design Tree** to fillet all the edges in a surface.

Note
You can only fillet the edge of the surface which is created using the intersection of the two surfaces. You need to make sure that if the edge is created using two surfaces then you must knit them before filleting them. You will learn more about knitting the surfaces later in this chapter.

Creating a Mid-Surface

Toolbar:	Surfaces > Mid-Surface
Menu:	Insert > Surface > Mid Surface

The **Mid-Surface** tool is used to create a surface between the two offset faces of a solid model. You can define the placement position of the surface in terms of percentage from the face selected first. The faces to be selected to create a mid surface should be offset from each other, such as two parallel faces or two concentric curved faces. To create a mid surface, choose the **Mid-Surface** button from the **Surfaces** toolbar. The **MidSurface1 PropertyManager** is displayed as shown in Figure 15-73.
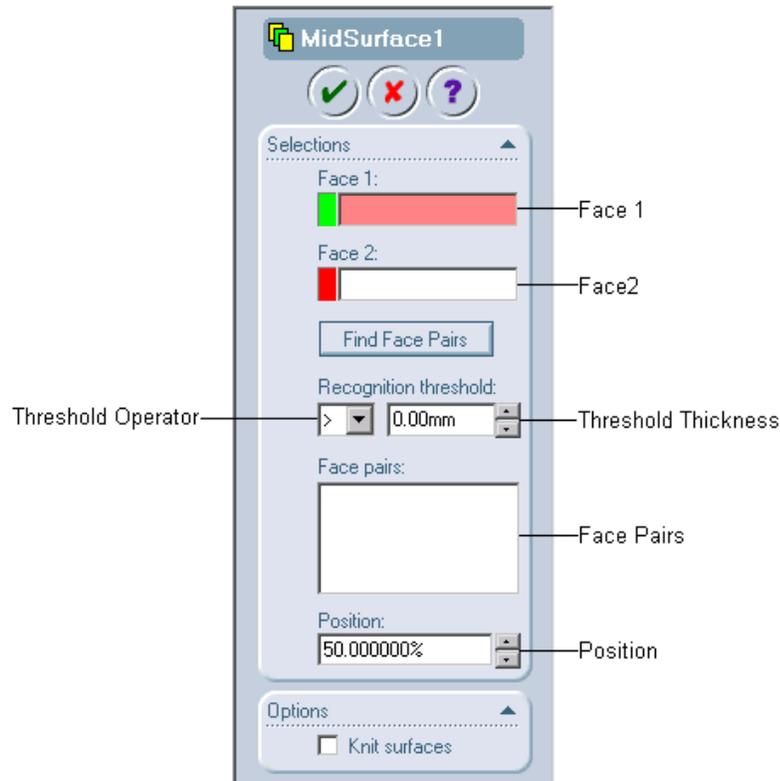
Figure 15-73 The MidSurface1 PropertyManager

You are prompted to select face pairs manually or use **Find Face Pairs** to automatically recognize face pairs. Select the face between which you need to create the surface. The first selected face will be highlighted in green and the second selected face will be highlighted in red and the names of the selected faces will be displayed in the **Face pairs** display area. You can also set the position of the mid surface. By default, the mid surface is placed in the middle of the selected faces. You can also define the percentage distance of the placement of the mid surface using the **Position** spinner. The position of the mid surface is defined from the first selected surface. After setting all the parameters, choose the **OK** button from the **Mid-Surface1 PropertyManager**.

Figure 15-74 shows the offset faces being selected and Figure 15-75 shows the mid surface created in the middle of the selected faces.

The **Find Face Pairs** button is used to find the faces that are adjacent to the selected face. The options available 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, you can set the mathematical operator such as >, <, =, and so on. Using the **Threshold Thickness** spinner, you can specify the threshold thickness.
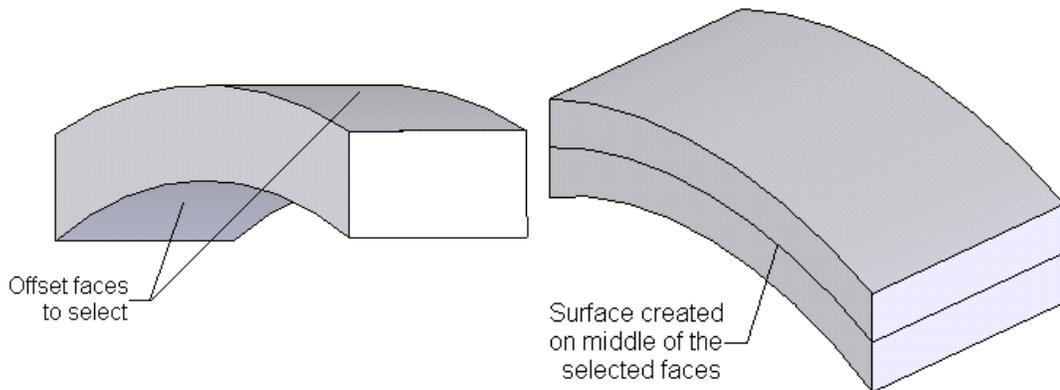
Figure 15-74 Offset faces selected

Figure 15-75 Resultant mid surface created

Deleting the Holes in the Surfaces

Using this option, you can delete the closed contours that cut the surface. The closed contours can be in any shape. For deleting a contour, select any one edge of the contour from the drawing area and invoke the shortcut menu. Choose the **Delete** option from the shortcut menu, the **Choose Option** dialog box is displayed as shown in Figure 15-76.
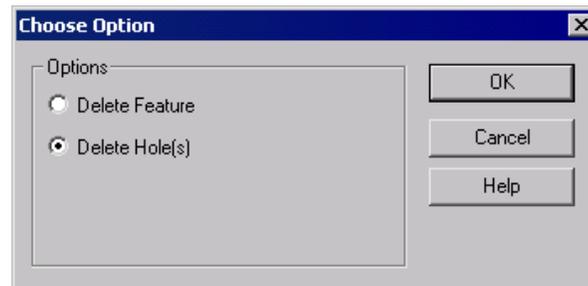
*Figure 15-76 The **Choose Option** dialog box*

The **Delete Hole(s)** radio button is selected by default. Using this option, the empty area of the selected contour is patched and it also maintains the tangency and curvature with the surrounding surfaces. You will notice that the icon of the **DeleteHole** is displayed in the **FeatureManager Design Tree**. If you select the **Delete Feature** radio button, the feature associated with that contour is deleted. After selecting this option, when you choose the **OK** button from the **Choose Option** dialog box, the **Confirm Delete** dialog box is displayed and you are informed about the name of the feature to be deleted and the names of the dependent item that will also be deleted.

Figure 15-77 shows the edge of the closed contour to be selected. Figure 15-78 shows the hole deleted using the delete feature option.
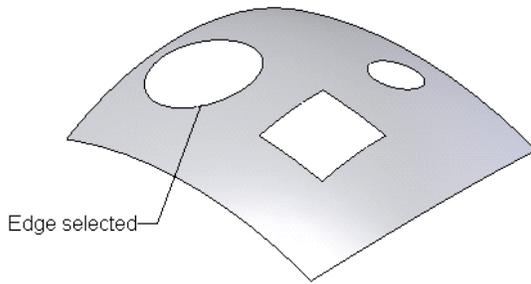
Edge selected

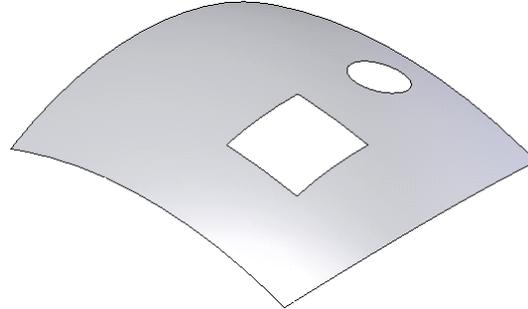*Figure 15-77 Edge to be selected to delete the hole*   *Figure 15-78 Resultant surface*

Replacing the Faces

Toolbar:	Surfaces > Replace Face
Menu:	Insert > Face > Replace

In SolidWorks, you can also 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 will retain the shape of the replacement surface by adding or subtracting the material from the solid body. To use this option, choose the **Replace Face** button from the **Surfaces** toolbar, the **Replace Face1 PropertyManager** is displayed as shown in Figure 15-79.
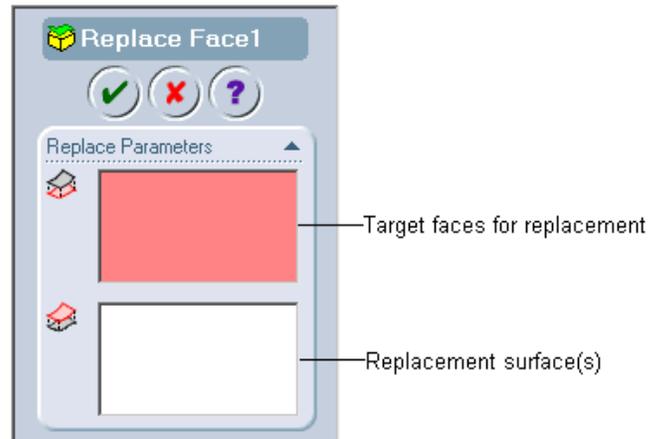
*Figure 15-79 The **Replace Face1 PropertyManager***

Select the face that you need to replace, the name of the selected face is displayed in the **Target faces for replacement** area. Now, click once in the **Replacement surface(s)** display area to invoke the selection environment in this area. Now, select the replacement surface, the name of the replacement surface is displayed in the **Replacement surface(s)** display area. Choose the **OK** button from the **Replace Face1 PropertyManager**.

Figure 15-80 shows the target face that is to be replaced and the replacement surface. Figure 15-81 shows the resultant replaced face. Figure 15-82 shows the solid body after hiding the surface body. For hiding the surface body, select the surface body and invoke the shortcut menu. Choose the **Hide Surface Body** option from the shortcut menu.
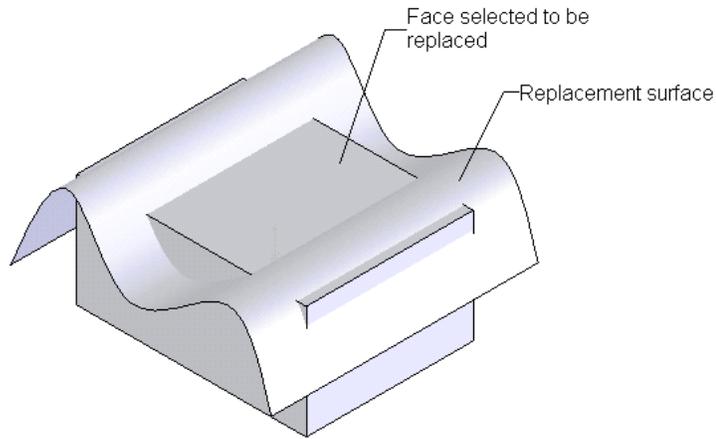
Figure 15-80 Target face and the replacement surface
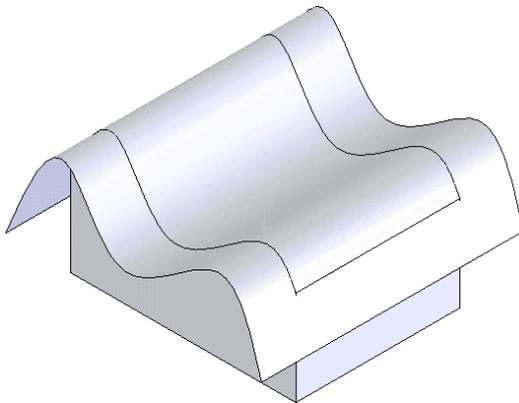
Figure 15-81 Resultant replaced face
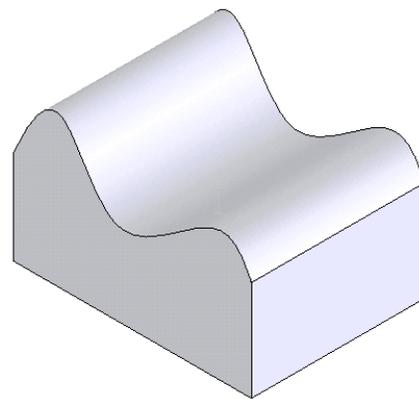
Figure 15-82 Model after hiding the surface

Deleting the Faces

Toolbar:	Surfaces > Delete Face
Menu:	Insert > Face > Delete

The **Delete Face** tool is used to delete the faces of the selected surface or solid body. When you delete the face of a solid body, it is converted into a surface body. If you patch the deleted face, then the solid body is not converted into a surface body. To

delete a face, choose the **Delete Face** button from the **Surfaces** toolbar. The **Delete Face PropertyManager** is displayed as shown in Figure 15-83.
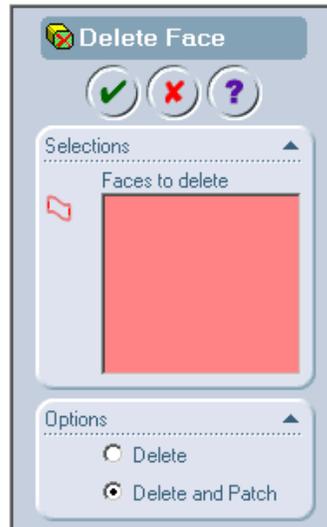

*Figure 15-83 The **Delete Face PropertyManager***

Now, you need to select the face or faces to be deleted. The name of the selected face is displayed in the **Faces to delete** display area. Since the **Delete and Patch** radio button is selected by default in the **Options** rollout, the preview of the patch to be created after deleting the faces is displayed. If the preview of the patch is not displayed, it is confirmed that deleting and patching the selected face is not possible. In this case, you need to select the **Delete** radio button from the **Options** rollout so that the face is only deleted and not patched. After setting all the parameters, choose the **OK** button from the **Delete Face PropertyManager**.

Figure 15-84 shows the face selected to be deleted. Figure 15-85 shows the face deleted with the **Delete and Patch** radio button selected. Figure 15-86 shows the face deleted with the **Delete** radio button selected and it forms a surface body.

Tip. *You can also delete a face by selecting a face and then invoking the shortcut menu. Choose the **Delete Face** option from the shortcut menu. The **Delete Face PropertyManager** is displayed.*

Moving and Coping the Surfaces

You can move and copy the surface bodies using the **Move/Copy Bodies** tool. Moving and coping the surface bodies using the **Move/Copy Bodies** tool is similar to that discussed in moving and coping the solid bodies.
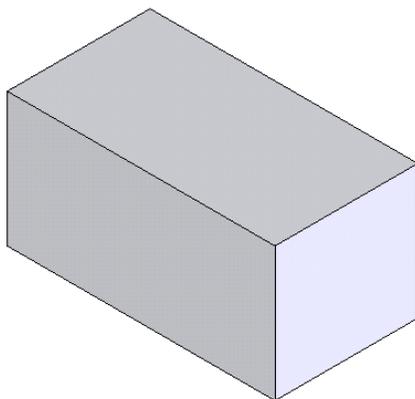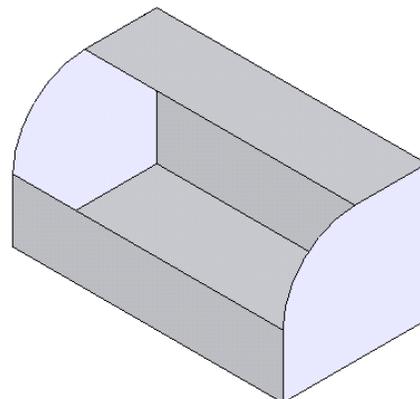
Note
*You can also delete the surface bodies using the **Delete Body PropertyManager**. The procedure of deleting the surface bodies is similar to that discussed for deleting the solid bodies.*
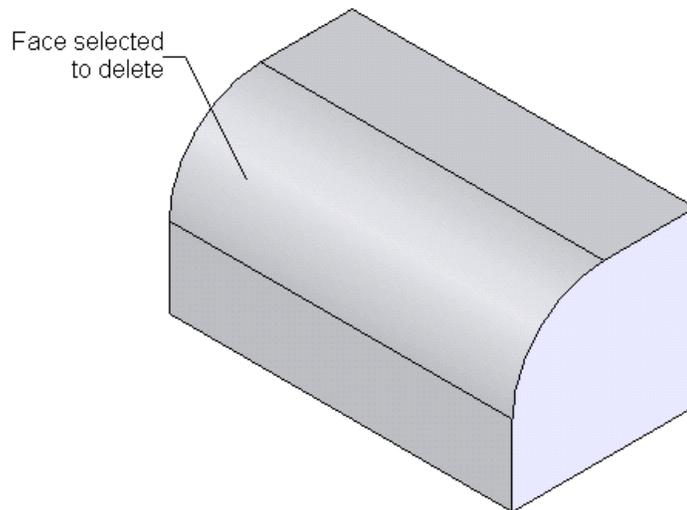
Figure 15-84 *Target face and the replacement surface*

Figure 15-85 *Face deleted with the **Delete and Patch** radio button selected*

Figure 15-86 *Face deleted with the **Delete** radio button selected and it forms a surface body*

Mirroring the Surface Bodies

You can also mirror the surface bodies using the **Mirror** tool. The procedure of mirroring the surface bodies is the same as that discussed for mirroring the solid bodies.
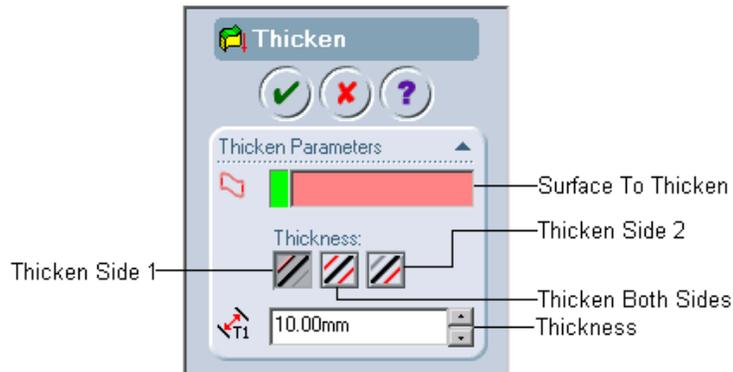
Adding Thickness to the Surface Bodies

Menu:	Insert > Boss/Base > Thicken

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

Adding Thickness to the Surface Body

To add thickness to a surface body choose **Insert > Boss/Base > Thicken** from the menu bar. The **Thicken PropertyManager** is invoked as shown in Figure 15-87. You are prompted to select surface to thicken. Select the surface that you need to thicken. The preview of the thickened body with default values is displayed in the drawing area. Using the buttons available in the **Thickness** area, you can specify the side on which you need to thicken the surface. Using the **Thickness** spinner, you can specify the wall thickness. After setting all the parameters, choose the **OK** button from the **Thicken PropertyManager**.
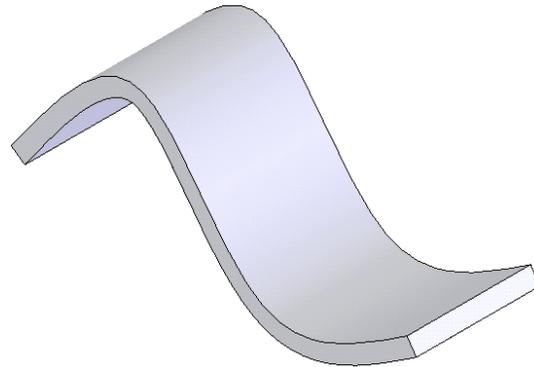

*Figure 15-87 The **Thicken PropertyManager***

Figure 15-88 shows the surface body and Figure 15-89 shows the model after adding thickness to the surface body.
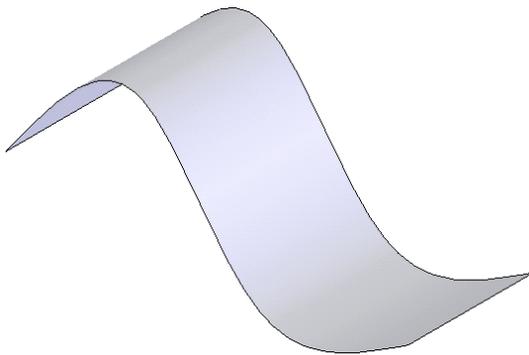
Figure 15-88 Surface to thicken

Figure 15-89 Model after thickening the surface

Tip. *If the surface model that you need to thicken consists of multiple joined surface bodies, then you first need to knit the surfaces together and then add thickness to them.*

Solidifying a Closed Surface Body

To solidify a closed surface body, the surface body should be exempted from any type of gap and all the surfaces should be stitched together using the **Knit** tool. To solidify the surface body, invoke the **Thicken PropertyManager** and select the closed surface body. The **Create solid from enclosed volume** check box is displayed. Select this check box, the option available in the **Thickness** area and the **Thickness** spinner are no more available. Choose the **OK** button from the **Thicken PropertyManager**. The solid volume from the closed surface is created.

Creating a Thicken Surface Cut

Menu:	Insert > Cut > Thicken

In SolidWorks, you can create a thicken surface cut. To create a thicken surface cut, choose **Insert > Cut > Thicken** from the menu bar. The **Cut-Thicken PropertyManager** is displayed as shown in Figure 15-90.
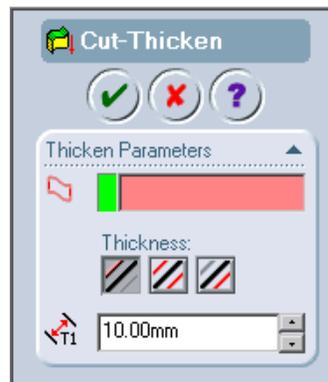

*Figure 15-90 The **Cut-Thicken PropertyManager***

You are prompted to select surface to thicken. Select the surface that you need to use as the cutting tool and then specify the parameters for defining the side in which you need to add the thickness and the thickness of cut. The preview is displayed in the drawing area. Now, choose the **OK** button from the **Cut-Thicken PropertyManager**. If the thicken cut results in creation of multiple bodies, the **Bodies to Keep** dialog box as displayed. Using this dialog box, you can define the bodies that you need to keep.

Figure 15-91 shows the surface selected for creating thicken cut. Figure 15-92 shows the resultant thicken cut.

Creating a Surface Cut

Menu:	Insert > Cut > With Surface

In SolidWorks, you can also cut a solid body using a surface. To create this type of surface cut, choose **Insert > Cut > With Surface** from the menu bar. The **SurfaceCut PropertyManager** is displayed as shown in Figure 15-93.

Surface selected
for creating a
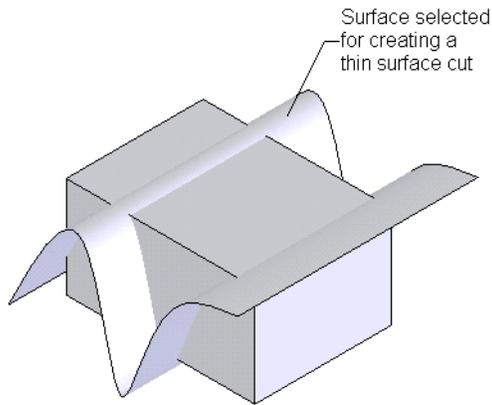thin surface cut

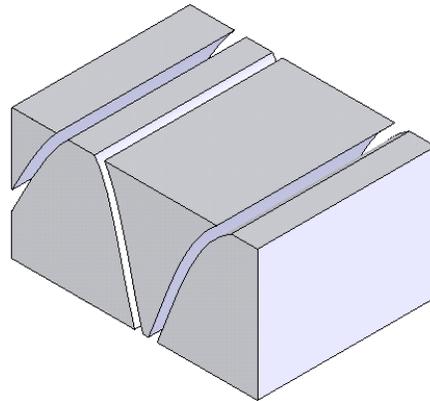***Figure 15-91*** *Surface to be selected*   ***Figure 15-92*** *Resultant thicken surface cut*
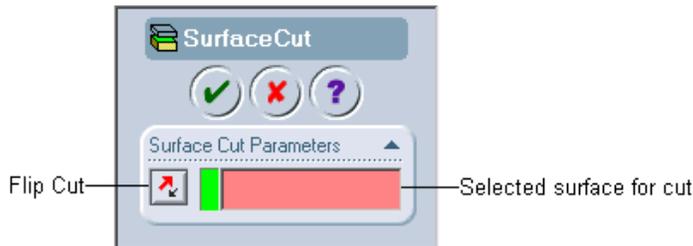
Figure 15-93 *The **SurfaceCut PropertyManager***

You are prompted to select cutting surface. Select the cutting surface, the name of the selected surface is displayed in the **Selected surface for cut** display area. An arrow is also displayed that indicates the direction of material removal. Using the **Flip Cut** button, you can flip the direction of material removal. Now, choose the **OK** button from the **SurfaceCut PropertyManager**. Figure 15-94 shows the surface selected to create a surface cut. Figure 15-95 shows the resultant surface cut after hiding the surface body.

Surface selected
for cut
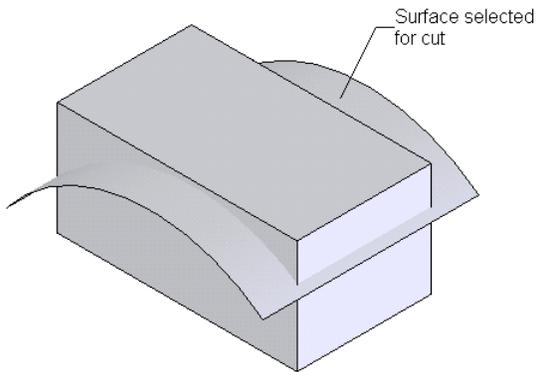
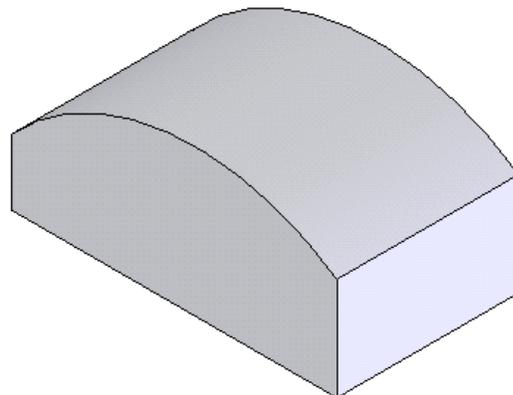***Figure 15-94*** *Surface selected to create the*   ***Figure 15-95*** *Resultant surface cut after hiding*
surface cute   *the surface body*

Tip. *If the surface cut results in the creation of multiple bodies, then the **Bodies to Keep** dialog box is displayed after exiting the **SurfaceCut PropertyManager**.*

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

TUTORIALS

Tutorial 1

In this tutorial, you will create the model shown in Figure 15-96. Create this model using the surface modeling tools available in SolidWorks and then add the wall thickness to the surface model. The views and dimensions of the model are shown in Figure 15-97.

(Expected time: 1hr)
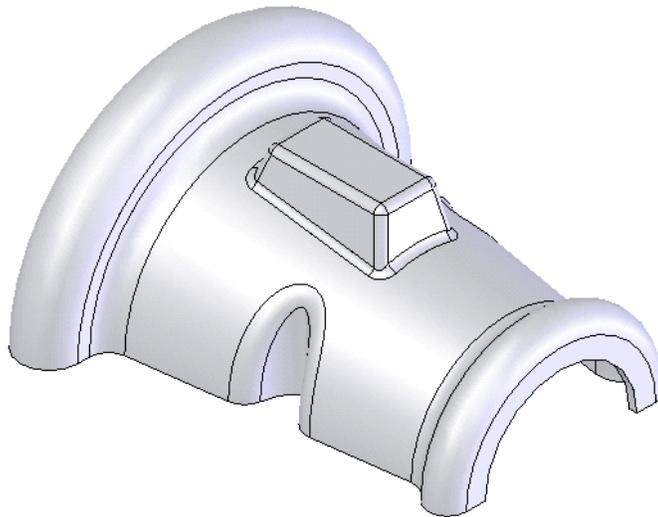
Figure 15-96 Model for Tutorial 1

The steps to be followed to complete this tutorial are discussed next.

a.   Create the base surface of the model by revolving the sketch using the **Mid Plane** option to 180-degree, refer to Figures 15-98 and 15-99.
b.   Create the second feature, which is an extruded surface feature located at the sides of the base feature, refer to Figures 15-100 and 15-101.
c.   Using the trim tool, trim the extruded surface, refer to Figures 15-102 and 15-103.
d.   Add the fillet to the trimmed base surface, refer to Figures 15-104 and 15-105.
e.   Create a plane at an offset distance of **40** from the **Top** plane.
f.   Create a lofted surface by lofting the section created on the newly created plane and the curve projected on the base surface, refer to Figures 15-106 through 15-110.
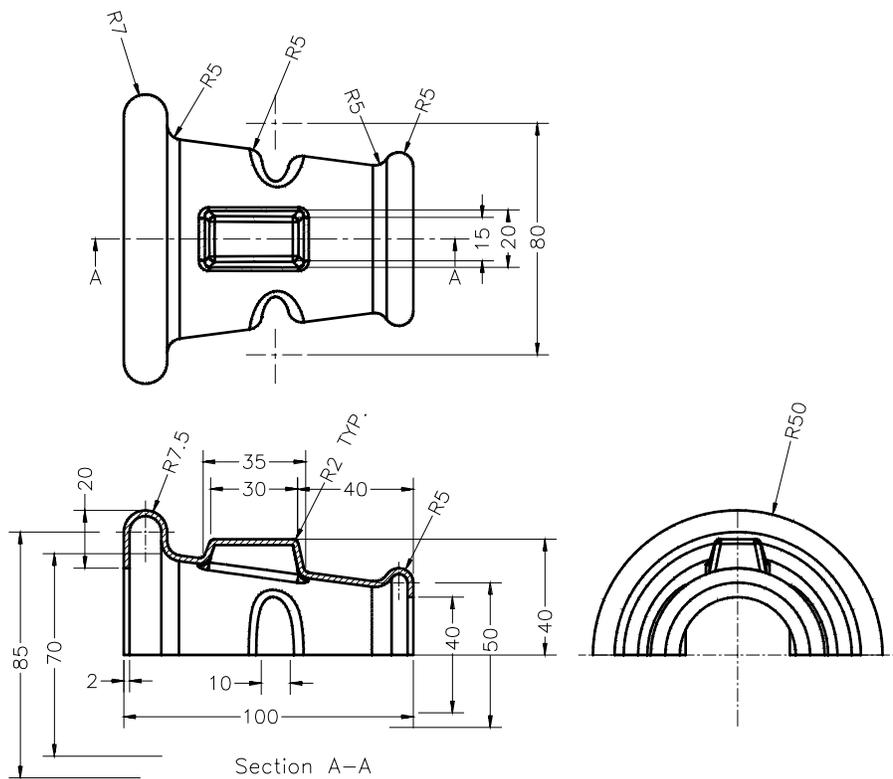g.   Create a planar surface on the top of the lofted feature and trim the base feature using

Figure 15-97 Views and dimensions for the model of Tutorial 1

the lofted feature, Figures 15-111 and 15-112.

h.   Trim and knit all the surfaces together, refer to Figure 15-113.

i.   Add the fillets to the surface model, refer to Figure 15-114.

j.   Add the thickness to the knitted surface, Figure 15-115.

Creating the Base Surface

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

1.   Start SolidWorks and create a new SolidWorks document in the **Part** mode.

2.   Invoke the sketcher environment using the **Front** plane as the sketching plane and create the sketch of the base surface as shown in Figure 15-98.

3.   Choose the **Revolve Surface** tool from the **Surfaces** toolbar. The **Surface-Revolve PropertyManager** is invoked.

4. 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 15-99 shows the base feature after revolving the sketch created on the **Front** plane.
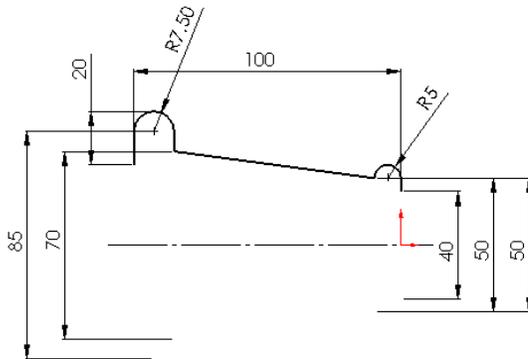

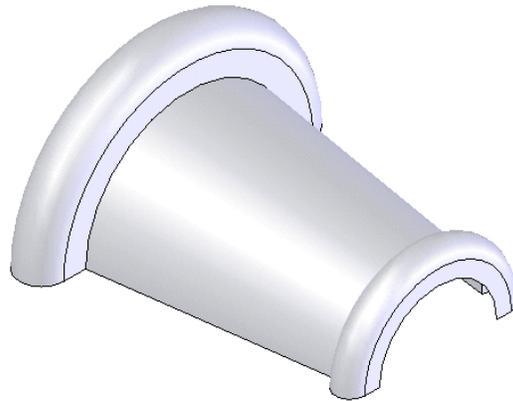*Figure 15-98 Sketch of the base surface*   *Figure 15-99 Base revolved surface*

Creating the Second Surface Feature

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

1. Using the **Top** plane, invoke the sketching environment.

2. Create the sketch of the second surface feature as shown in Figure 15-100. You may have to apply **Horizontal** or **Vertical** relations between the points of the ellipse.

3. Choose the **Extruded Surface** button from the **Surfaces** toolbar. The **Surface-Extrude PropertyManager** is displayed.

4. Set the value of the **Depth** spinner to **40** and choose the **OK** button from the **Surface-Extrude PropertyManager**.

   Surface created after extruding the sketch is displayed in Figure 15-101.

## Trimming the Base Surface using the Extruded Surface

Next, you need to trim the unwanted portion of this surface model. You will trim the base surface using the extruded surface.

1. Choose the **Trimmed Surface** button from the **Surfaces** toolbar to invoke the **Trim Surface PropertyManager**.

2. Select the **Mutual trim** radio button from the **Trim Type** rollout.
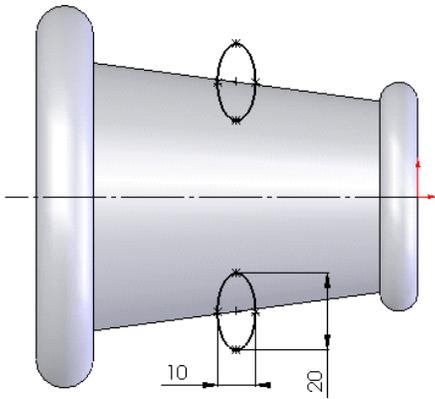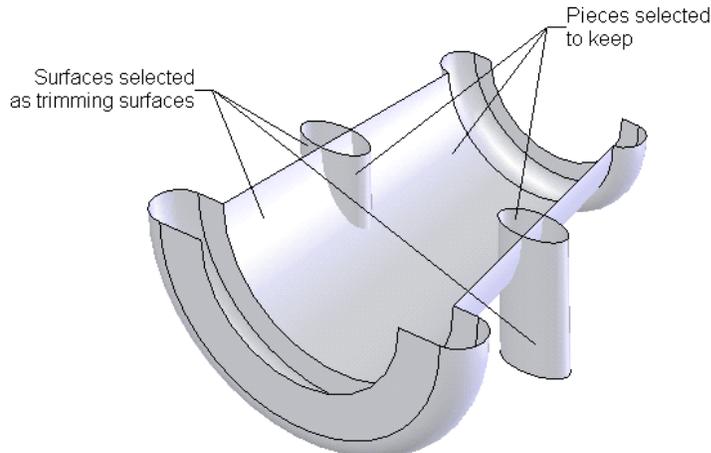
Figure 15-100 *Sketch of the second surface feature*
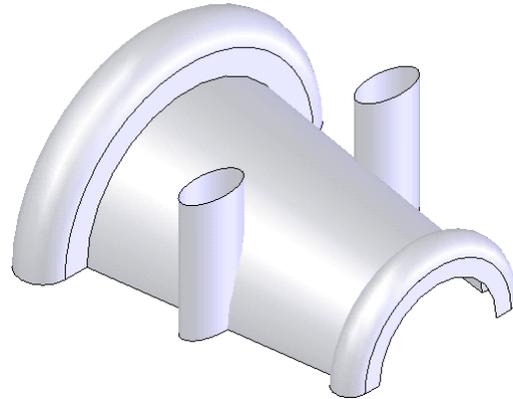
Figure 15-101 *Extruded surface*

3. Select the surfaces shown in Figure 15-102 as the trimming surfaces.

4. Now, click once in the **Pieces to Keep** display area to invoke the selection mode. Select the pieces to keep as shown in Figure 15-102.

***Figure 15-102*** *Surfaces and pieces to keep to be selected*

5. Choose the **OK** button from the **Trim Surface PropertyManager**. The model after trimming the base surface is displayed in Figure 15-103.

Filleting the Edges of the Trimmed Surface

Next, you need to fillet the edges created at the intersection of the base surface and the extruded surface after trimming the surfaces.
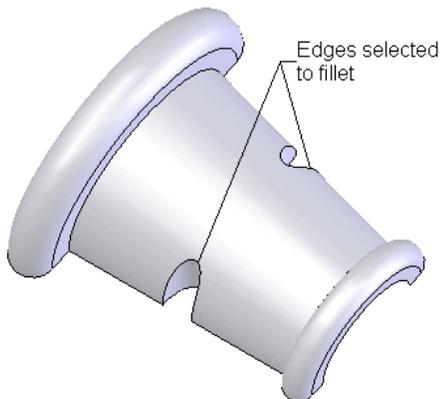
1. Choose the **Fillet** button from the **Features** toolbar.
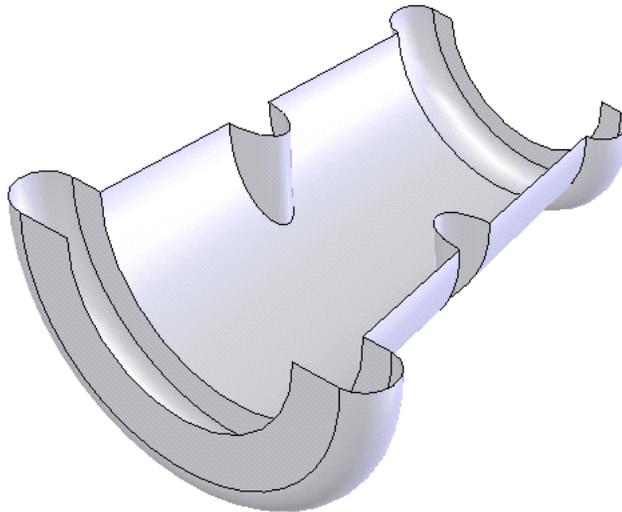
Figure 15-103 Model after trimming the surfaces

2. Select the edges to fillet as shown in Figure 15-104.

3. Set the value of the **Radius** spinner to **5** and choose the **OK** button from the **Fillet PropertyManager**. Figure 15-105 shows the model after adding the fillet.

Edges selected to fillet

Figure 15-104 Edges selected to fillet   *Figure 15-105 Resultant fillet feature*

Creating a Lofted Surface

Next, you need to create a lofted surface. The lofted surface will be created using a sketch drawn on a plane at an offset distance from the **Top** plane and a curve projected on the base surface. Before proceeding further you first need to create a plane at an offset distance from the **Top** plane.

1. Create a plane at an offset distance of **40** from the **Top** plane.

2. Invoke the sketching environment using the newly created plane as the sketching plane.

3. Create the sketch as shown in Figure 15-106 and exit the sketching environment.

    Next, you need to create a projected curve. First, the sketch of the curve will be created on the newly created plane and then you need to project this curve on the base surface.

4. Invoke the sketching environment with the newly created plane as the sketching plane and create the sketch as shown in Figure 15-107 and exit sketching environment.
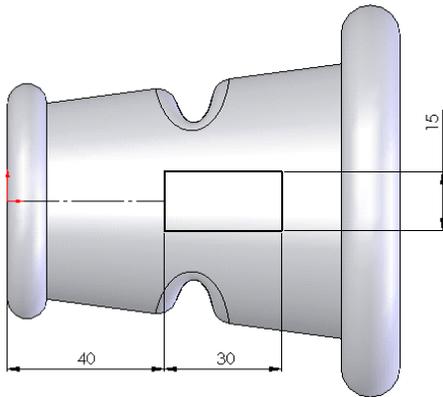

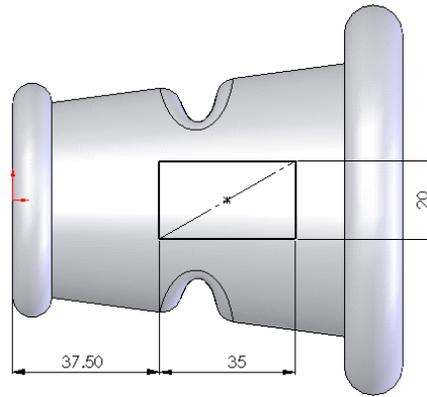*Figure 15-106 First sketch for loft surface*   *Figure 15-107 Sketch to create projected curve*

5. Choose **Insert > Curve > Projected** from the menu bar to invoke the **Projected Curve PropertyManager**.

6. Select the **Sketch onto Face(s)** option from the **Projection Type** drop-down list.

7. Select the second sketch created on the newly created plane and then click once in the **Projection Faces** display area to invoke the selection mode.

8. Select the base surface from middle and select the **Reverse Projection** check box. Choose the **OK** button from the **Projected Curve PropertyManager**.

    Figure 15-108 shows the model after creating the sketch and the projected curve.

9. Choose the **Lofted Surface** button from the **Surfaces** toolbar. The **Surface-Loft PropertyManager** is displayed.

10. Select the loft section as shown in Figure 15-109 and choose the **OK** button from the **Surface-Loft PropertyManager**.

    Figure 15-110 shows the model after creating the lofted surface.
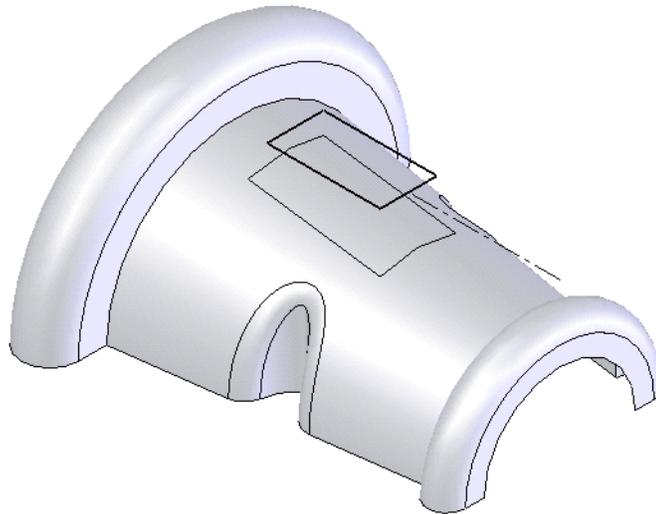
Figure 15-108 Model after creating the sketch and the projected curve
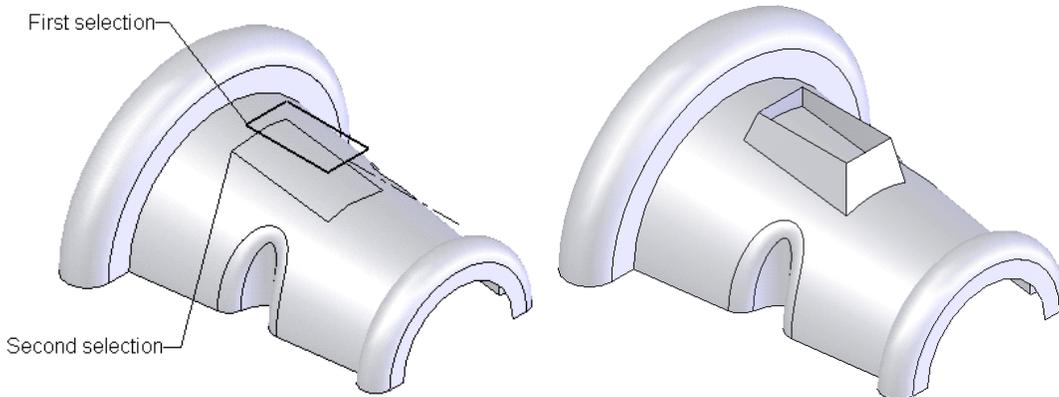
Figure 15-109 Section selected for lofted surface

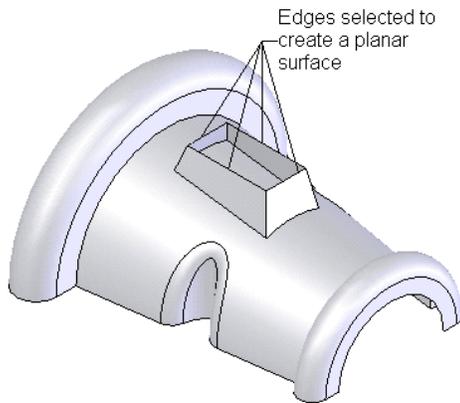*Figure 15-110 Resultant lofted surface*

Creating the Planar Surface

Next, you need to create a planar surface using the top edges of the lofted surface.
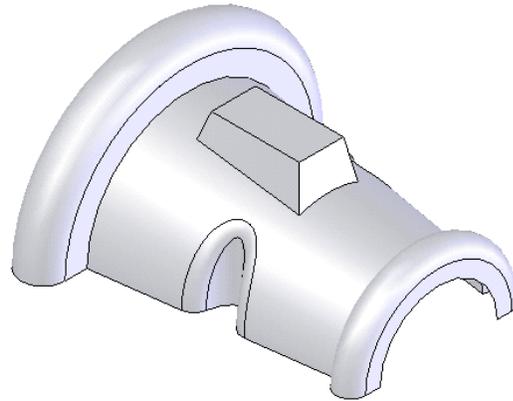
1. Choose the **Planar Surface** button from the **Surfaces** toolbar to invoke the **Planar Surface** toolbar.

2. Select the edges as shown in Figure 15-111 to create the planar surface.

    The name of the selected edges is displayed in the **Bounding Entities** display area.

3. Choose the **OK** button from the **Planar Surface PropertyManager**. Figure 15-112 shows the resultant planar surface.

*Figure 15-111* *Edges selected to create the planar surface*

Figure 15-112 *Resultant planar surface*

Trimming the Base Surface using the Lofted Surface

Next, you need to trim the base surface using the lofted surface. You need the trimming tool to trim the base surface.

1. Choose the **Trimmed Surface** button from the **Surfaces** toolbar to invoke the **Trim Surface PropertyManager**.

2. Select the lofted surface as the trim tool and select the base surface as pieces to keep.

3. Choose the **OK** button from the **Trim Surface PropertyManager**.

Figure 15-113 shows the model after trimming the base surface using the lofted surface.
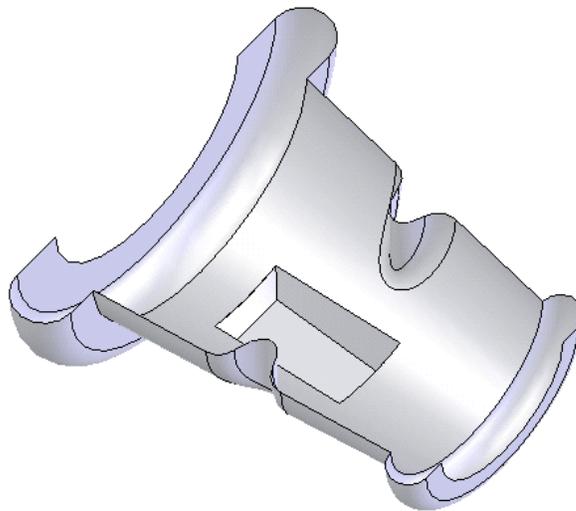

Figure 15-113 *Model after trimming the base surface using the lofted surface*

Knitting all the Surfaces

Next, you need to knit the base surface, lofted surface, and the planar surface using the **Knit Surface** tool. This is done because, for creating fillets on the edges created at the intersection of the two surfaces, the two surfaces should be knitted first. If you need to add the wall thickness to the surface model created using multiple surfaces, all the surfaces should be knitted first.

1. Choose the **Knit Surface** button from the **Surfaces** toolbar to invoke the **Knit Surface PropertyManager**.

2. Select the base surface, lofted surface, and the planar surface. The name of the selected surfaces is displayed in the **Surfaces and Faces to Knit** display area.

3. Choose the **OK** button from the **Surface Knit PropertyManager**.

Next you need to add the required fillets. The final surface model after adding fillets is shown in Figure 15-114.
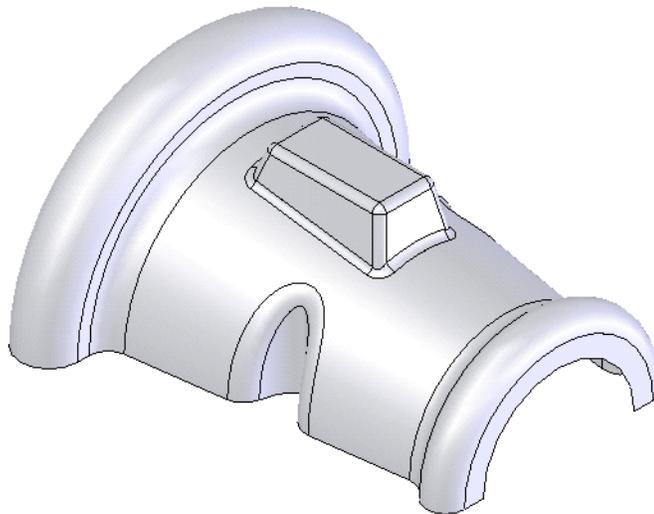

Figure 15-114 Final surface model

## Adding Thickness to the Surface Model

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

1. Choose **Insert > Boss/Base > Thicken** from the menu bar to invoke the **Thicken PropertyManager**.

2. Select the surface model and set the value of the **Thickness** spinner to **2**.

3. Choose the **OK** button from the **Thicken PropertyManager**.

Figure 15-115 shows the final model after thickening the surface. Figure 15-116 shows the **FeatureManager Design Tree** of the model.
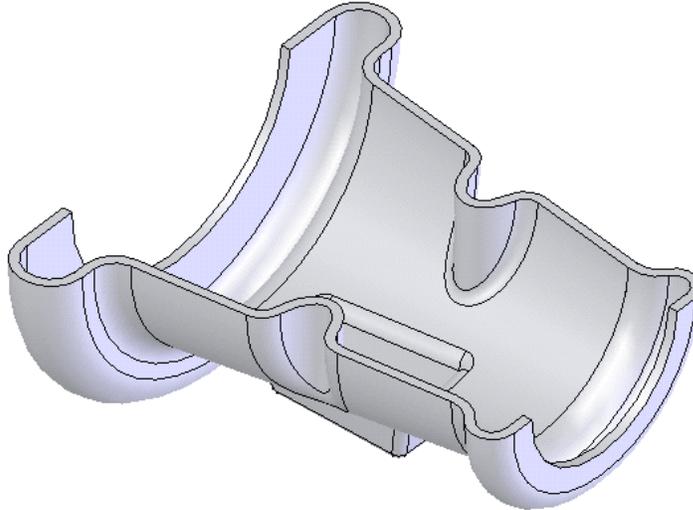
Figure 15-115 *Final model*
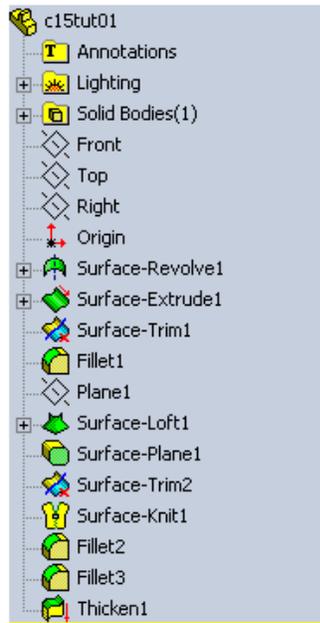

Figure 15-116 **FeatureManager Design Tree**

Saving the Model

1. Choose the **Save** button from the **Standard** toolbar and save the drawing with the name given below and close the file.

   \My Documents\SolidWorks\c15\c15-tut01.SLDPRT.

Tutorial 2

In this tutorial you will create the cover of the hair dryer shown in Figure 15-117. You will create this model using surfaces and then you will thicken it. The views and dimensions of the hair drye are displayed in Figure 15-118. **(Expected time: 1.5 hr)**
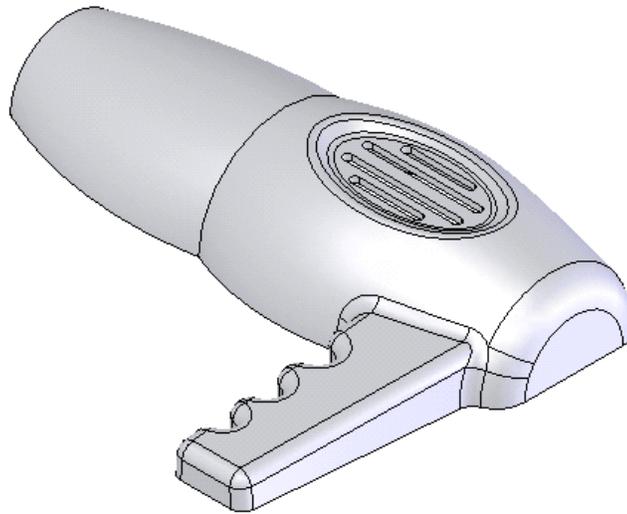
Figure 15-117 Cover of hair dryer

The steps to be followed to complete this tutorial are discussed next.

a. First, you need to create the base surface. The base surface is created by lofting the open sections along the guide curves, refer to Figures 15-119 and 15-120.
b. Create a planar surface to close the right face of the base surface, refer to Figures 15-121 and 15-122.
c. Next, you will create the basic structure of the handle of the hair dryer cover by creating a lofted surface between two open sections, refer to Figures 15-123 through 15-125.
d. Next, you will trim the unwanted portion of the lofted surface used to create the handle, refer to Figures 15-126 and 15-127.
e. You will create a planar surface to close the front face of the handle, refer to Figure 15-128.
f. Extrude the elliptical sketches to create the grips of the handle and then trim the unwanted surfaces, refer to Figures 15-129 through 15-132.
g. Create dip on the top surface of the hair dryer, refer to Figures 15-133 throught 15-138.
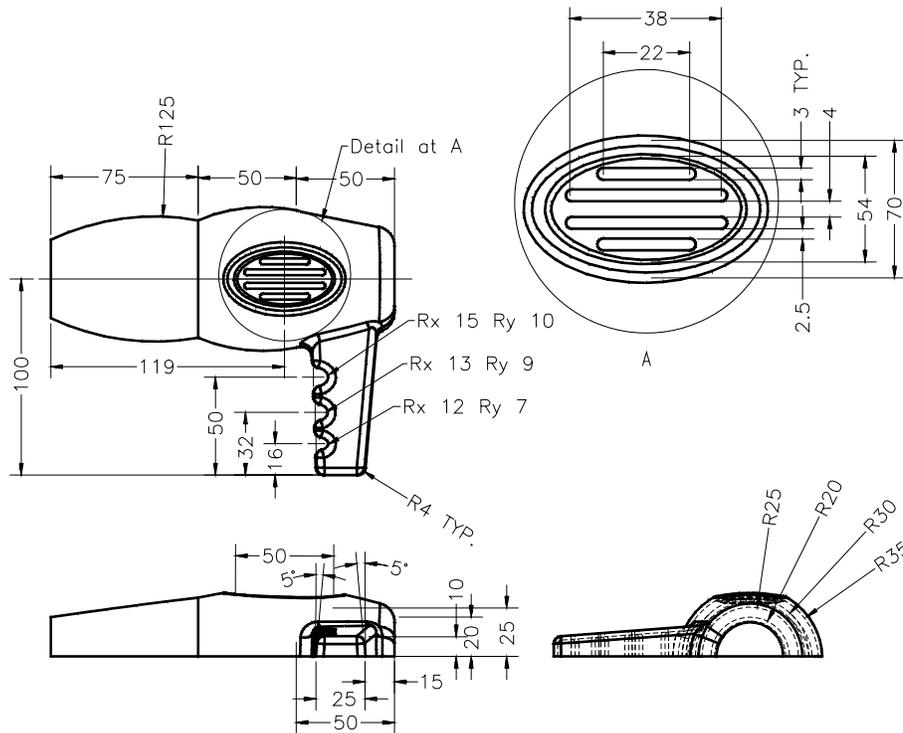h. Trim the surface to create the air vents, refer to Figure 15-139.

Figure 15-118 Views and dimensions for the model of Tutorial 2

i. Knit all the surfaces together and add the required fillets to the model, refer to Figure 15-140.
j. Thicken the surface, refer to Figure 15-141.

Creating the Base Surface

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

1. Create a new SolidWorks document in the **Part** mode.

2. Create three planes at an offset distance from the **Right** plane, refer to Figure 15-119.

3. Next you need to create the sections and the guide curves to create a lofted surface as shown in Figure 15-119.

4. Using the **Lofted Surface** tool create the lofted surface as shown in Figure 15-120.
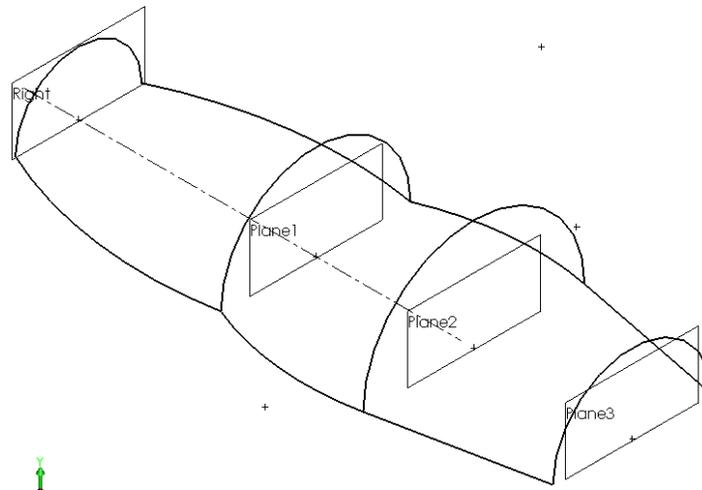
*Figure 15-119 Sections and guide curves to create a lofted surface*
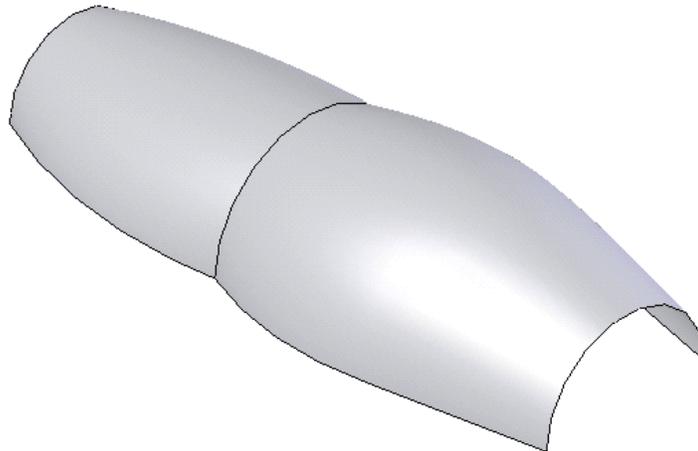
Figure 15-120 Resultant lofted surface

5.   Next, you will create a planar surface to close the right face of the base surface.

6.   Invoke the sketching environment by selecting the **Plane3** as the sketching plane.

7.   Create a closed sketch to create the planar surface as shown in Figure 15-121.

8.   Invoke the **Planar Surface PropertyManager** and choose the **OK** button from this **PropertyManager** to create the planar surface as shown in Figure 15-122.
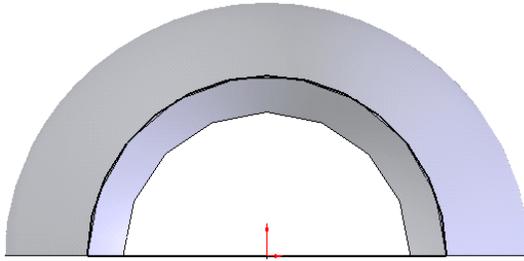
Figure 15-121 *Sketch for creating the planar surface*
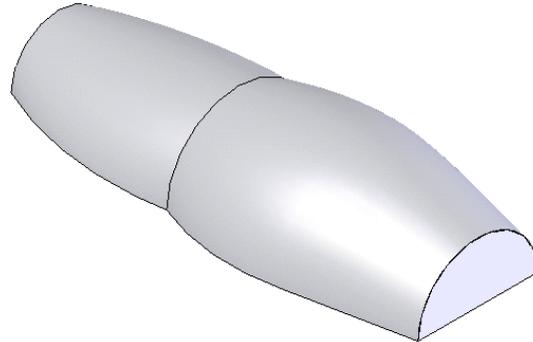
Figure 15-122 *Resultant planar surface*
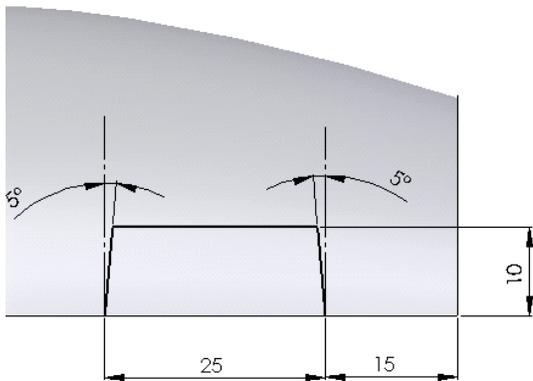
Creating the Base Surface for the Handle

Next, you need to create the base surface for the handle. The base surface for the handle is 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** 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 15-123 and exit the sketching environment.

4. Now, invoke the sketching environment using the **Front** plane as the sketching plane.

5. Create an open sketch as shown in Figure 15-124 and exit the sketching environment.

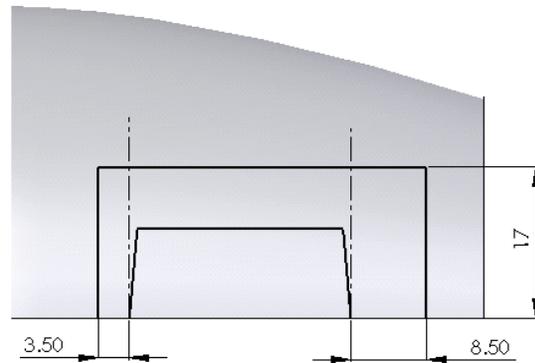6. Using the **Lofted Surface** tool create the lofted surface as shown in Figure 15-125.

Trimming the Unwanted Lofted Surface of Handle

After creating the lofted surface for the handle, if you rotate the model you will observe that a portion of the lofted surface needs to be trimmed. The method of trimming the unwanted portion of the lofted surface is discussed next.
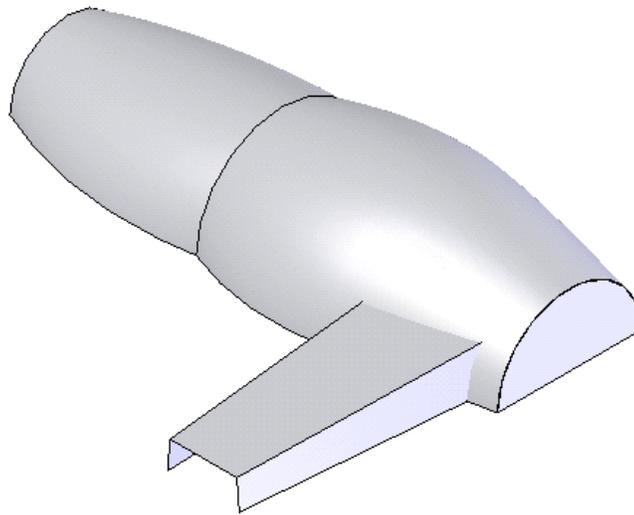
1. Invoke the **Trim Surface PropertyManager** and choose the **Mutual trim** radio button from the **Trim Type** rollout.

Figure 15-123 Sketch of the first section for creating the lofted surface of the handle

Figure 15-124 Sketch of the second section for creating the lofted surface of the handle

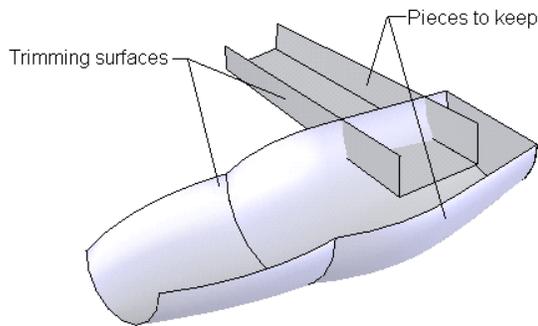*Figure 15-125 Resultant lofted surface*

2. Select the surfaces as trimming surfaces and the pieces to keep as shown in Figure 15-126 and choose the **OK** button from the **Trim Surface PropertyManager**.

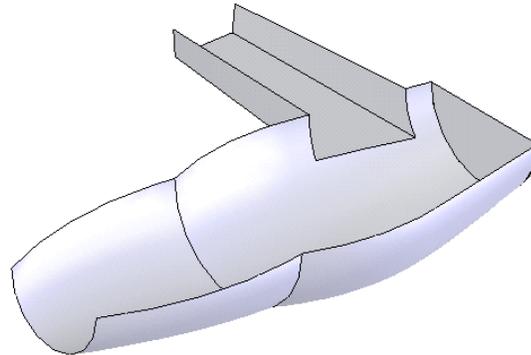   Figure 15-127 shows the resultant trimmed surface.

Creating a Planar Surface to Close the Front Face of the Handle

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

1. Invoke the sketching environment with the plane created at an offset distance from the **Front** plane as the sketch plane.

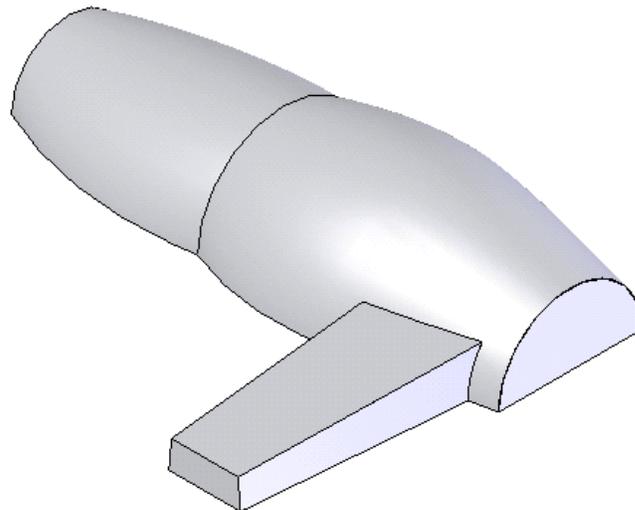2. Create a closed sketch to create a planar surface.

Figure 15-126 *Trimming surfaces and the pieces to keep*

Figure 15-127 *Resultant trimmed surface*

3. Invoke the **Planar Surface PropertyManager** to create the planar surface.

   Model after creating the planar surface is shown in Figure 15-128.

Figure 15-128 *Model after creating the planar surface*

Creating the Grips on the Handle

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

1. Invoke the sketching environment using the **Top** plane as the sketching plane.

2. Create the sketch for extruding the surface to create the grips as shown in Figure 15-129.

3. Invoke the **Surface-Extrude PropertyManager** and extrude the sketch to the depth of **25mm**. The extruded surface is displayed in Figure 15-130.
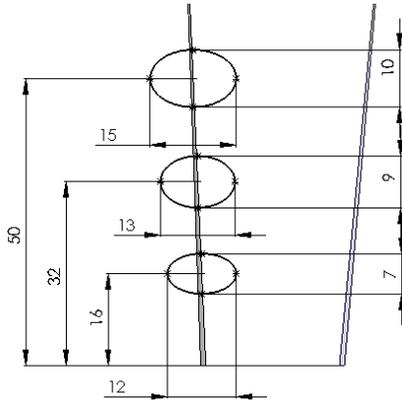

Figure 15-129 Sketch created for extruding the surface
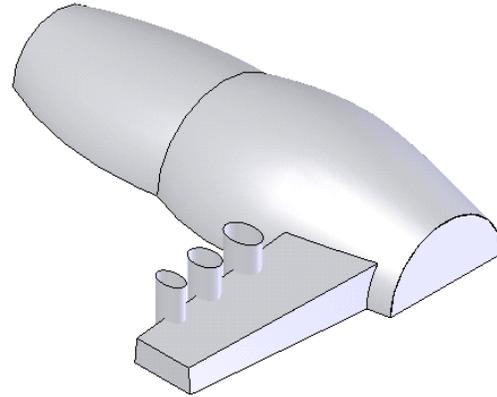
Figure 15-130 Resultant extruded surface

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

4. Invoke the **Trim Surface PropertyManager** and choose the **Mutual trim** radio button from the **Trim Type** rollout.

5. Select the trimming surfaces and the pieces to keep as shown in Figure 15-131.

6. Choose the **OK** button from the **Trim Surface PropertyManager**. Figure 15-132 shows the resultant trimmed surface.
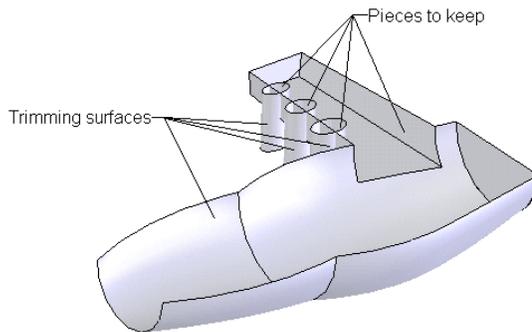

Figure 15-131 Trimming surfaces and the pieces to keep
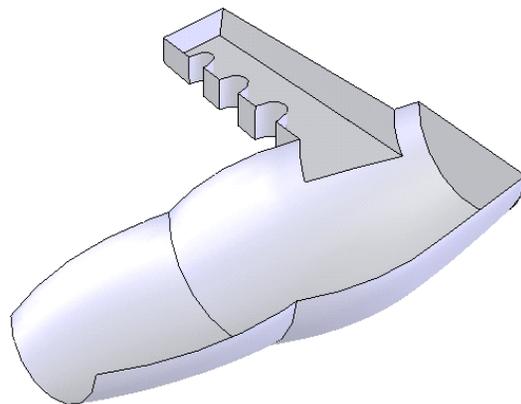

Figure 15-132 Model after trimming the surfaces

Creating the Dip on the Upper Surface of the Base Surface

Next, you need to create the dip on the base surface. To create this, you need several tools. These include offsetting the planes, creating lofted surface, trimming and creating planar surface.

1. Create a plane at an offset distance of **35** from the **Top** plane.

2. Invoke the sketching environment using the newly created plane as the sketching plane.

3. Create the sketch as shown in Figure 15-133 and exit the sketching environment.

4. Choose **Insert > Curve > Projected** from the menu bar and project the newly created sketch on the base surface. The model, after projecting the surface, is displayed in Figure 15-134.
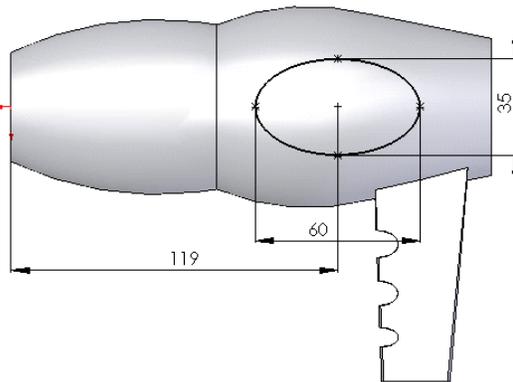

*Figure 15-133 Sketch to be projected*
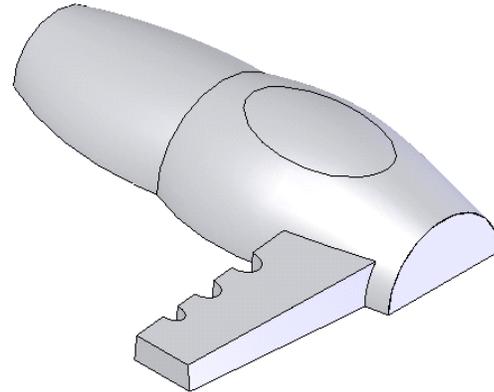

Figure 15-134 Resultant projected curve

5. Create a plane at an offset distance of **6** from the newly created plane at an offset distance in the downward direction.

6. Now, invoke the sketching environment using the newly created plane as the sketching plane and create the sketch as shown in Figure 15-135. Exit the sketching environment.

7. Invoke the **Lofted Surface** tool and create the lofted surface using the sketch and the projected curve created earlier.

8. The lofted surface after hiding the base surface is displayed in Figure 15-136. To hide the surface bodies, expand the **Surface Bodies** folder and select the surface. Now, right-click to invoke the shortcut menu. Choose the **Hide Surface Body** option from the shortcut menu.

9. Using the **Trim Surface** tool, trim the base surface using the newly created lofted surface. The model, after trimming the surface, is displayed in Figure 15-137. Next, using the **Planar Surface** tool, create a planar surface as shown in Figure 15-138.
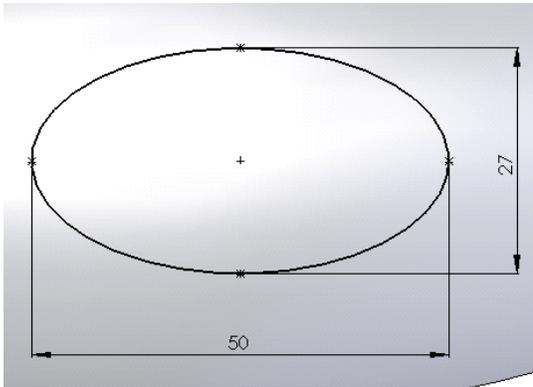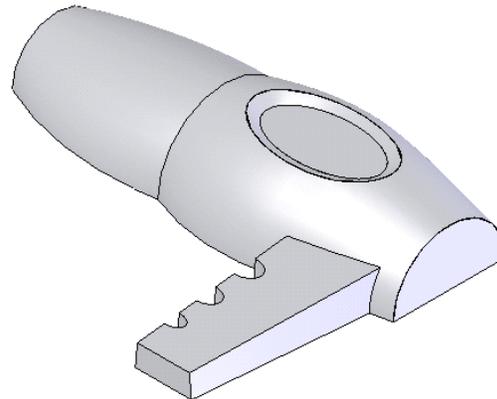
Figure 15-135 Sketch for lofting
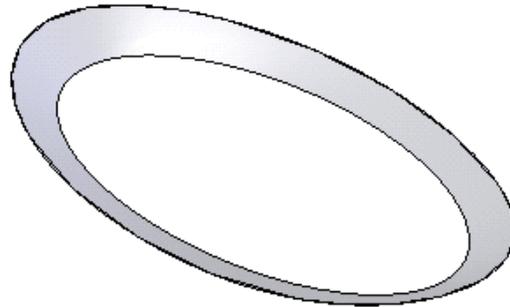
Figure 15-136 Lofted surface
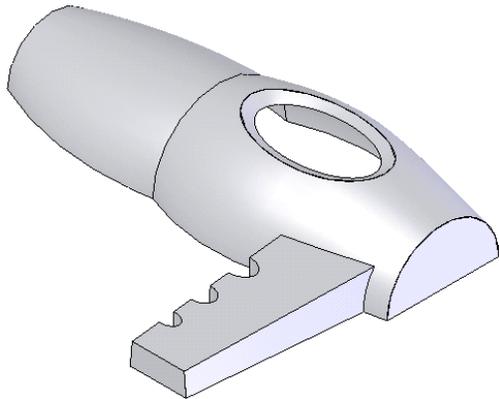
Figure 15-137 Model after trimming the surface

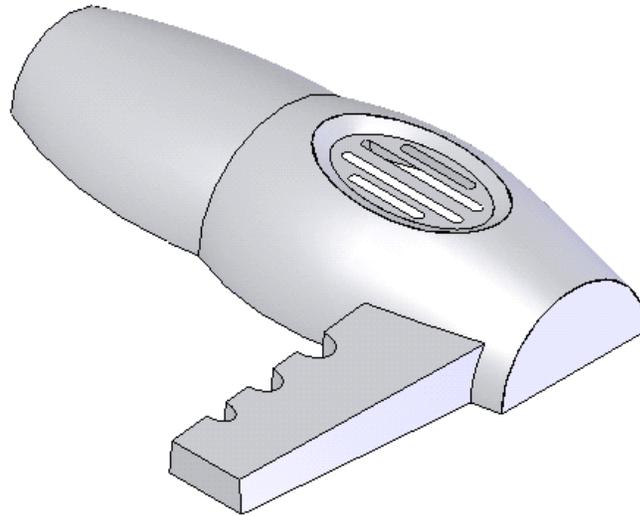*Figure 15-138 Planar surface*

Creating the Air Vents

Next, you need to create the air vents on the newly created planar surface. The air vents are created by drawing the sketch on the planar surface and then using the trimming tool to trim the surface.

1. Select the newly created planar surface as the sketching plane and invoke the sketching environment.

2. Create the sketch of the air vents, refer to Figure 15-118.

3. Using the **Trim Surface** tool, trim the planar surface in order to create the air vents.

    The surface model after creating the air vents is displayed in Figure 15-139.

Knitting All the Surfaces

After creating all the surfaces, you need to knit all the surfaces together so that you can add fillets to the surfaces and also thicken the model.

Figure 15-139 Surface model after trimming the planar surface

1. Choose the **Knit Surface** button from the **Surfaces** toolbar. The **Knit Surface PropertyManager** is invoked and you are prompted to select the surfaces to knit.

2. Invoke the **FeatureManager Design Tree** flyout and expand the **Surface Bodies** folder.

3. Select all the surface bodies from this folder.

4. Choose the **OK** button from the **Knit Surface PropertyManager**.

5. Add all the required fillets to the surface model. The model after adding the fillets is displayed in Figure 15-140.
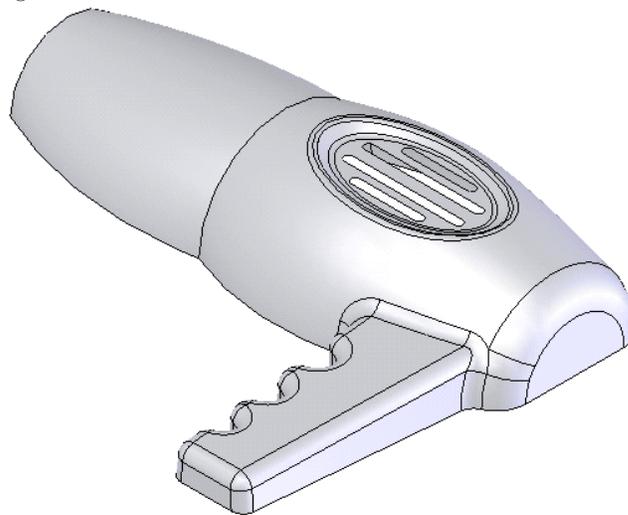

Figure 15-140 Surface model after adding fillets

Adding Thickness to the Surface Model

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

1. Choose **Insert** > **Boss/Base** > **Thicken** from the menu bar. The **Thicken PropertyManager** is invoked and you are prompted to select surface to thicken.

2. Set the value of the **Thickness** spinner to **2** and select the surface model from the drawing area. The preview of the thickened model is displayed in the drawing area.

3. Choose the **OK** button from the **Thicken PropertyManager**. Save the model.

The final model is displayed in Figure 15-141. The **FeatureManager Design Tree** of the model is shown in Figure 15-142.
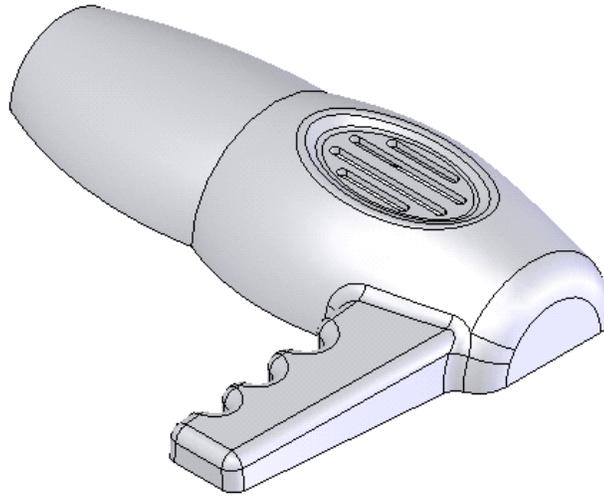
Figure 15-141 Final model

Figure 15-142 FeatureManager Design Tree

SELF-EVALUATION TEST

Answer the following questions and then compare your answers with the answers given at the end of this chapter.

1. In SolidWorks the _____ tool is provided to extrude a closed or an open sketch to create an extruded surface.

2. The _____ **PropertyManager** is used to create a revolved surface.

3. The _____ tool is used to create a surface patch by extending the existing surfaces.

4. The _____ tool is used to offset the selected surface or surfaces to a given distance.

5. The _____ **PropertyManager** is used to create a lofted surface.

6. You cannot patch the faces of the solid model deleted using the **Delete Face** tool. (T/F)

7. Using the **Stitch Surface** tool, you can knit the surfaces together to create a single surface. (T/F)

8. You cannot create a filled surface by selecting a 3D sketch as patch boundary. (T/F)

9. The **Curvature Control** drop-down list is used to define the type of curvature you need to apply while creating the fill surface. (T/F)

10. The **SurfaceCut PropertyManager** is used to create a surface cut. (T/F)

REVIEW QUESTIONS

Answer the following questions:

1. The _____ **PropertyManager** is used to add thickness to a surface body.

2. To extend the surface in the linear direction to an existing surface, invoke the **Extend Surface PropertyManager** and select the _____ radio button from the **Extension Type** rollout.

3. The _____ check box is used to solidify a closed surface model.

4. The _____ tool is used to delete the faces of a surface or a solid body.

5. The **Mid-Surface** tool is used to create a surface between the selected faces.

6. Which **PropertyManager** is used to created a fillet surface?

    (a) **Fill Surface**   (b) **Surface Fill**

    (c) **Fillet**   (c) None of these

7. Which button from the **Surfaces** toolbar is used to invoke the **Replace Face1 PropertyManager**?

    (a) **Face Replace**   (b) **Replace Face**

    (c) **Offset Surface**   (d) **Fillet Surface**

8. The face or surfaces selected to offset are highlighted in green. In which display area are their names displayed?

    (a) **Surface or Faces to Offset**   (b) **Options**

    (c) **Patch Boundaries**   (d) **Offset**

9.   Which rollout is used to define the constraint curves while creating the fill surface?

      (a) **Define Constraint Curves**   (b) **Constraint Curves**

      (c) **Patch Boundaries**   (d) None of these

10. In which display area of the **Replace Face PropertyManager**, the name of the face selected to be replaced is displayed?

      (a) **Target faces for replacement**   (b) **Faces for replacement**

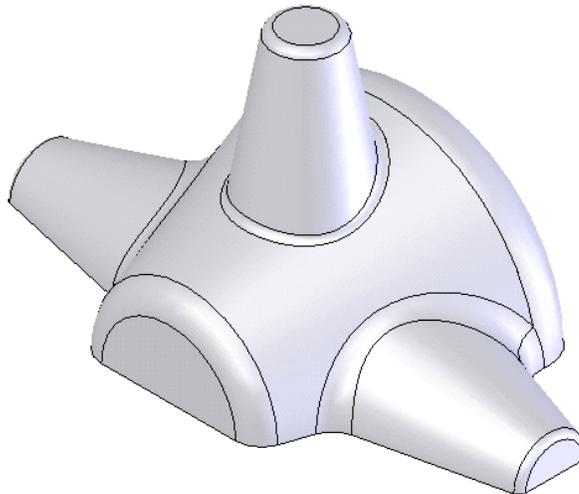   (c) **Faces**   (d) **Faces and Surfaces to Replace**

EXERCISES

Exercise 1

In this exercise, you will create the model shown in Figure 15-143. You need to create this model using surfaces. After creating and knitting all the surfaces you will add the required thickness to the model. The views and dimensions of the model are shown in Figure 15-144.

(Expected time: 1hr)

Note

In this model you will have to knit the surfaces first and then fillet the edges.

Figure 15-143 Model for Exercise 1
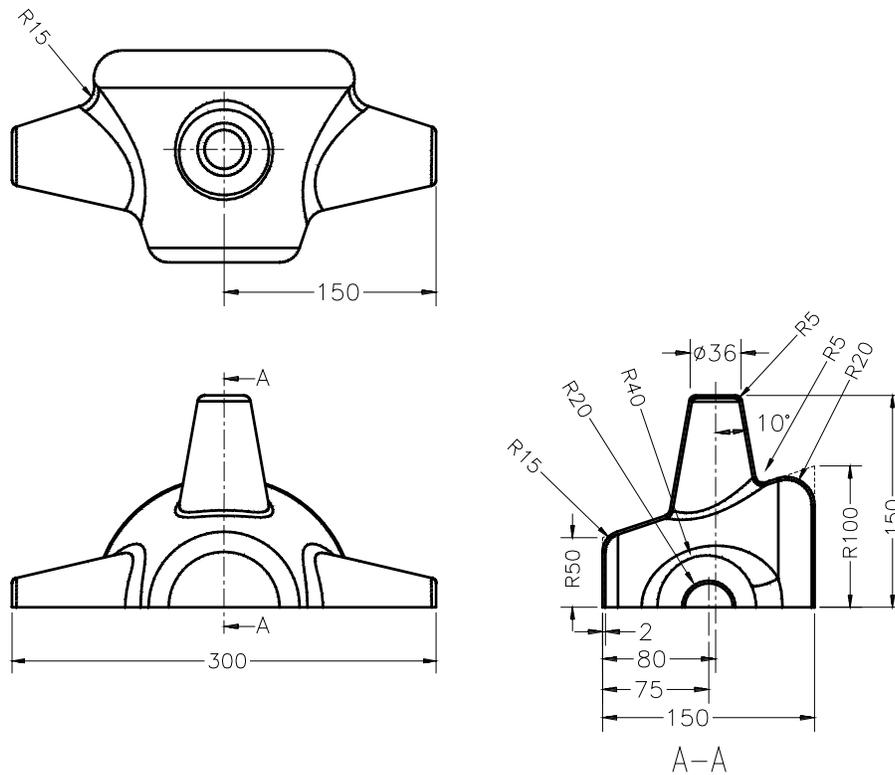
Figure 15-144 *Views and dimensions for Exercise 1*

Exercise 2

In this exercise you will create the model of the binoculars shown in Figure 15-145. You need to create this model using surfaces. You need to create a closed surface model and then knit all the surfaces together and then solidify it. The views and dimensions of the model are shown in Figure 15-146. **(Expected time: 1hr)**
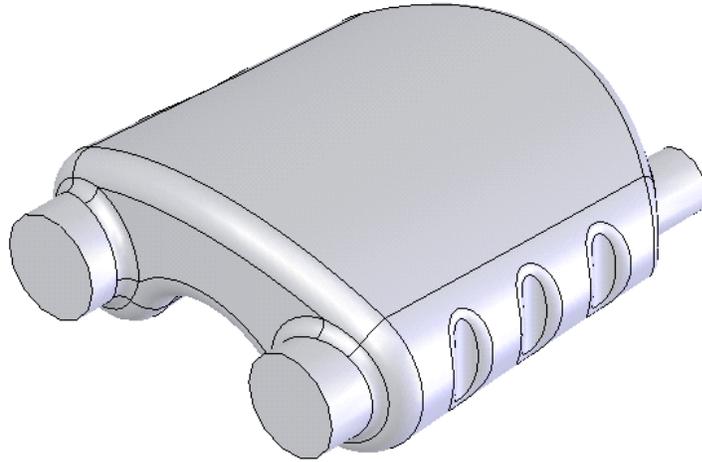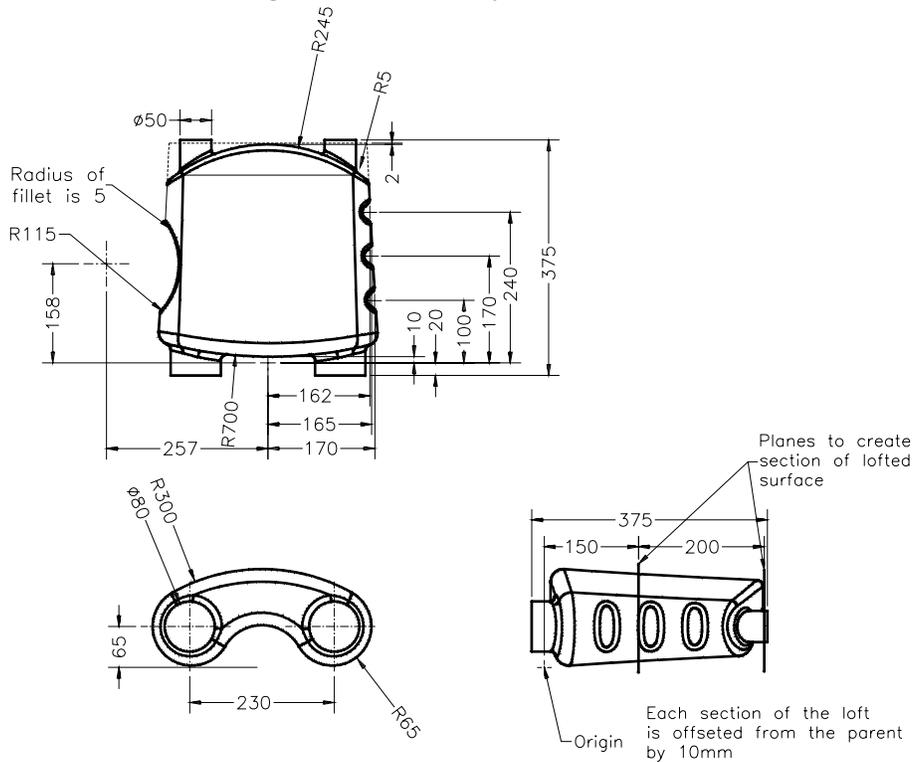
Figure 15-145 *Model for Exercise 2*

Figure 15-146 *Views and dimensions of 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.** F, **8.** F, **9.** T, **10.** T