

# Chapter 15

---

## Surface Modeling

### Learning Objectives

**After completing this chapter, you will be able to:**

- *Create an extruded surface.*
- *Create a revolved surface.*
- *Create a sweep surface.*
- *Create a blended surface.*
- *Create a swept blend surface.*
- *Create a helical sweep surface.*
- *Create a surface by blending boundaries.*
- *Create a surface using variable section sweep.*
- *Create surfaces using style environment.*
- *Understand surface editing tools.*

## SURFACE MODELING

Surface models are a type of three-dimensional (3D) models with no thickness. These models are widely used in industries like automobile, aerospace, plastic, medical, and so on.

Surface models should not be confused with thick models, that is, models having mass properties. Surface models do not have thickness whereas thick or solid models have a user-defined thickness. In Pro/ENGINEER, the surface modeling techniques and feature creation tools are the same as those used in solid modeling. A solid model of any shape that is created can also be created using the surface modeling techniques. The only difference between the solid model and the surface model is that the solid model has mass properties but the surface model does not. Sometimes, complex shapes are difficult to create using solid modeling. Such models can be easily created using surface modeling and then the surface model can be converted into the solid model. It becomes easy for a person to learn surface modeling if he is familiar with solid modeling feature creation tools.

## CREATING SURFACES IN Pro/ENGINEER WILDFIRE 4.0

In Pro/ENGINEER Wildfire 4.0, a sketch can be toggled between a solid model and a surface model. The two buttons that are used to toggle between the solid feature and a surface feature are available on dashboards.

### Creating an Extruded Surface

To create an extruded surface, choose the **Extrude** button from the **Base Features** toolbar; the **Extrude** dashboard will be displayed, as shown in Figure 15-1.



Figure 15-1 The *Extrude* dashboard



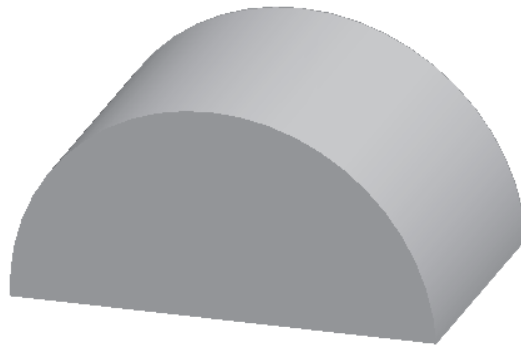
In this dashboard, the **Extrude as solid** button is chosen by default. Choose the **Extrude as surface** button to extrude the sketch and create a surface model. All the attributes of a solid model that were discussed in Chapter 4 are the same for a surface model also. Some of the examples of these attributes are sketching plane, both-side or one-side extrusion, depth of extrusion, and so on.

A surface model can be extruded with capped ends or with open ends. Figure 15-2 shows the open ends surface model and Figure 15-3 shows the capped ends surface model. Remember that to create the capped end surface model, the sketch should be a closed loop. Otherwise, a surface can be created with the open sketch.

To create a surface with capped ends, select the **Capped Ends** check box in the **Options** slide-down panel.



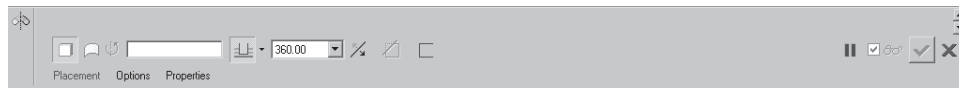
*Figure 15-2 Surface with open ends*



*Figure 15-3 Surface with capped ends*

## Creating a Revolved Surface

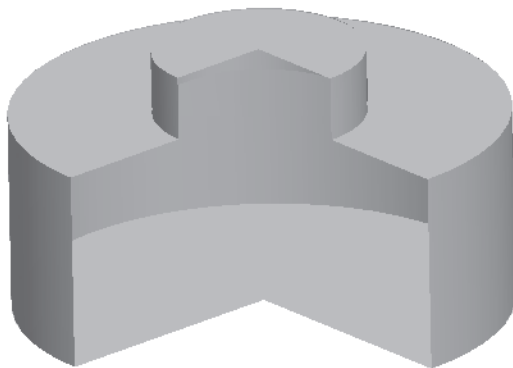
To create a revolved surface, choose the **Revolve** button from the **Base Features** toolbar; the **Revolve** dashboard will be displayed, as shown in Figure 15-4. This feature creation tool works in the same way as in the case of solid modeling.



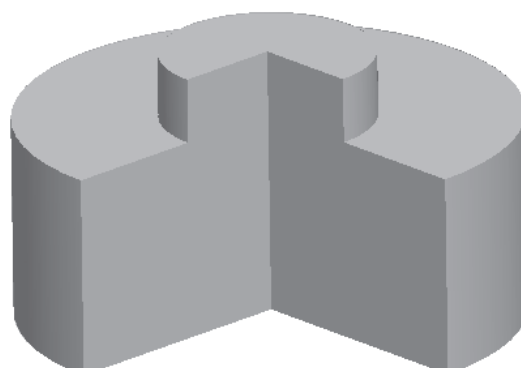
*Figure 15-4 The Revolve dashboard*



The **Revolve as solid** button is chosen by default. Choose the **Revolve as surface** button to create a revolve surface. You can create a revolved capped end surface or an open end surface. The **Capped Ends** check box in the **Options** slide-down panel is available only when the sketch is closed and the angle of revolution is less than 360-degree. Figure 15-5 shows the open end revolved surface and Figure 15-6 shows the capped end revolved surface.



*Figure 15-5 Revolved surface with open ends*

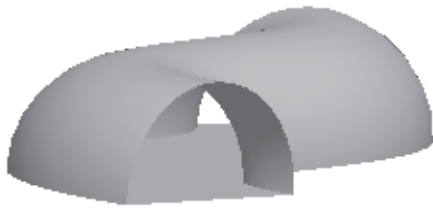


*Figure 15-6 Revolved surface with capped ends*

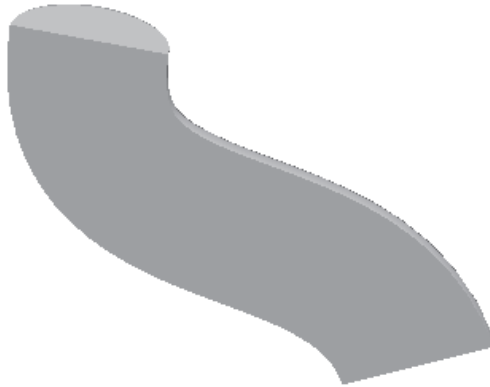
## Creating a Sweep Surface

To create a sweep surface feature, choose **Insert > Sweep > Surface** from the menu bar; the **SWEEP TRAJ** menu will be displayed. The method to create a surface sweep feature is the same as that to create a solid sweep feature. To create a solid sweep feature, refer to Chapter 8. The additional option of capping the ends that was available in the **Extrude** and **Revolve** options is also available in the **Sweep** option.

Figures 15-7 and 15-8 show the sweep surfaces with the open and capped ends, respectively.



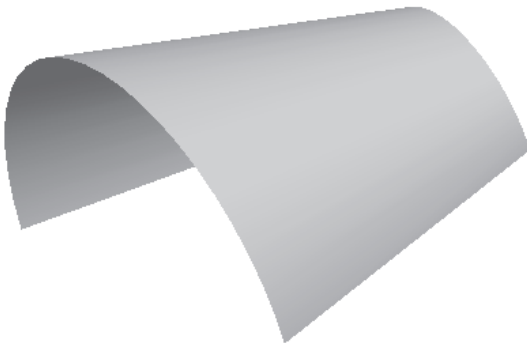
*Figure 15-7 Sweep surface with open ends created using a closed sketch*



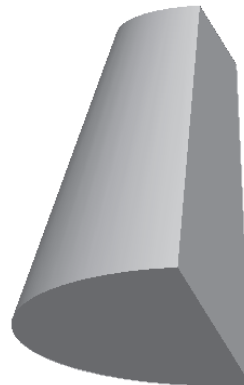
*Figure 15-8 Sweep surface with capped ends created using a closed sketch*

## Creating a Blended Surface

To create a surface blend, choose **Insert > Blend > Surface** from the menu bar. The **BLEND OPTS** menu will be displayed. The method to create a blended surface is the same as that to create a solid blend. To create a solid blend feature, refer to Chapter 8. Blended surfaces can be with open ends or capped ends. Figure 15-9 shows the blended surface with open ends and Figure 15-10 shows the blended surface with capped ends.



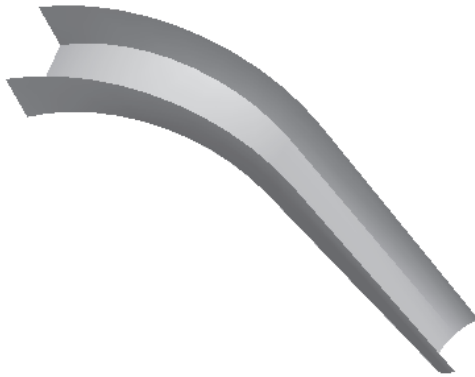
*Figure 15-9 Blended surface with open ends*



*Figure 15-10 Blended surface with capped ends*

### Creating a Swept Blend Surface

To create a swept blend surface, choose **Insert > Swept Blend** from the menu bar. The **Swept Blend** dashboard will be displayed. The method to create a swept blend surface is the same as that to create a solid swept blend feature. To create a solid swept blend feature, refer to Chapter 9. Figure 15-11 shows the swept blend with open ends and Figure 15-12 shows the swept blend with capped ends.



**Figure 15-11** Swept blend surface with open ends



**Figure 15-12** Swept blend surface with capped ends

### Creating a Helical Sweep Surface

To create a surface helical sweep, choose **Insert > Helical Sweep > Surface** from the menu bar. The **Surface** dialog box and the **ATTRIBUTES** menu will be displayed. The method to create a helical sweep surface feature is the same as that to create a solid helical sweep feature. For more information on creating solid helical sweep features, refer to Chapter 9. Figure 15-13 shows the helical sweep surface with open ends and Figure 15-14 shows the helical sweep surface with capped ends.



**Figure 15-13** Helical sweep surface with open ends created using an open sketch



**Figure 15-14** Helical sweep feature with capped ends created using the closed sketch



**Tip:** If you want to create a surface blend with capped ends, you need to create a closed sketch. Pro/ENGINEER does not accept an open sketch for a capped end blend surface.

If you want to create a surface blend with capped ends along with an open loop section, choose the **Open Ends** option and then sketch the open loop section. Give the blend depth and create the blended surface. Now, redefine the surface feature and modify the open ends attribute to capped ends. Choose **OK** from the **SURFACE** dialog box. The blended surface with the capped ends is created. This is also true with other features like extrude, revolve, sweep, and so on.

## Creating a Surface by Blending Boundaries



To create a surface by blending boundaries, datum curves, or points, choose the **Boundary Blend** button from the **Base Features** toolbar; the **Boundary Blend** dashboard will be displayed, as shown in Figure 15-15 and you will be prompted to select two or more curve chains to define a blended surface. The options and buttons in this dashboard are discussed next.



Figure 15-15 The *Boundary Blend* dashboard

### Curves Tab

When you choose the **Curves** tab, the slide-down panel will be displayed. Choose a curve from the drawing area; the curve will be highlighted in red. At the two ends of the curve, **T=0** will be displayed. Also, an arrow will be attached to the curve. When you modify the value of **T**, which is by default 0 to some higher value, then the curve will be extended from that end. Press CTRL+left mouse button to select the second curve; the second curve will also get highlighted, as shown in Figure 15-16. Both the selected curves are numbered as per the sequence of selection. The surface is created, as shown in Figure 15-17.

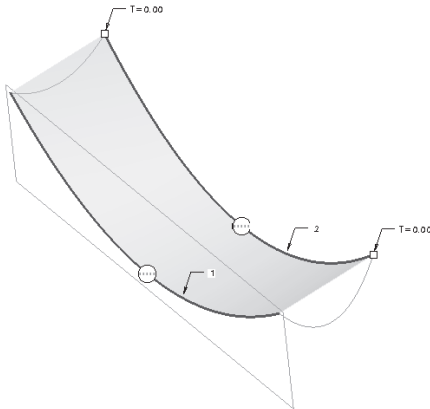


Figure 15-16 Curves selected to the boundary blend



Figure 15-17 Boundary blend surface

The collector present above the **Curves** tab shows **2 Chains**. This collector represents the **Curves** tab. The number of curves selected in the first direction will be displayed in this collector.

Now, invoke the **Curves** slide-down panel and select **2 Chain** from the **First direction** collector; the slide-down panel will be modified, as shown in Figure 15-18. The **Move up** and **Move down** buttons available in the slide-down panel can change the order of selection of the curves. The **Close blend** check box is used to close the surfaces.

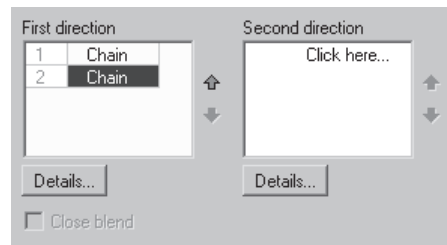


Figure 15-18 The **Curves** slide-down panel



**Tip:** To delete the curves from the collector, right-click the collector and choose the **Remove all** option from the shortcut menu; all the curves available in the collector will be deleted.

Figure 15-19 shows the surface created after selecting the three curves and Figure 15-20 shows the surface that is created by selecting the **Close blend** check box.

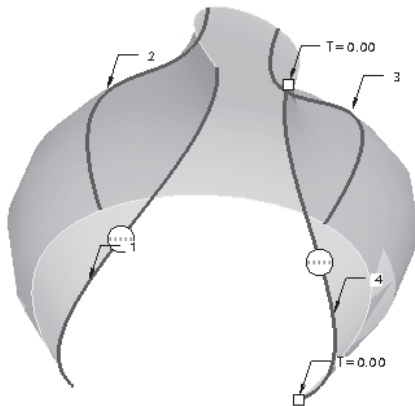


Figure 15-19 Surface created after selecting the curves

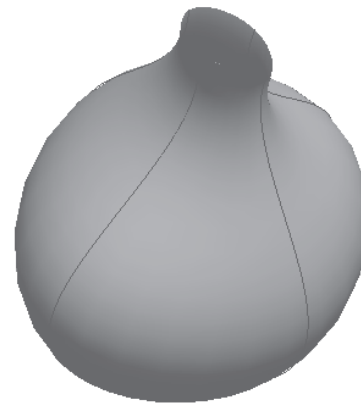
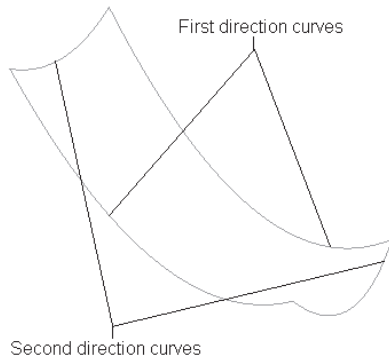


Figure 15-20 Surface created after selecting the **Close blend** check box

The **Second direction** collector in the **Curves** slide-down panel is used to select curves in the second direction. The second direction curves are usually drawn in a direction other than that of the first direction. Figure 15-21 shows the first and second direction curves and Figure 15-22 shows the surface created after selecting the curves shown in Figure 15-21.



**Figure 15-21** Datum curves

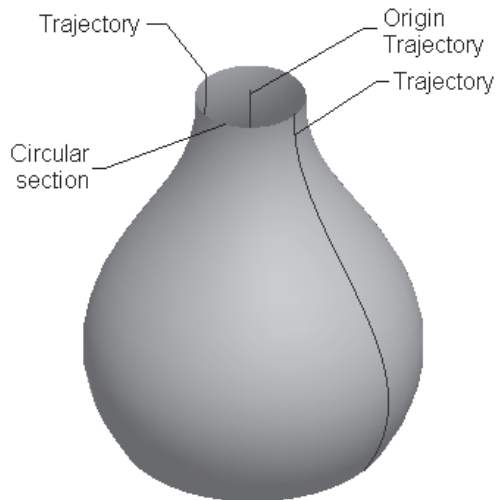


**Figure 15-22** Surface created by selecting the curves in two directions

### Creating a Surface Using the Variable Section Sweep

To create a surface by using the variable section sweep, choose **Insert > Variable Section Sweep** from the menu bar. The **Variable Section Sweep** dashboard will be displayed. To learn more about the variable section sweep, refer to Chapter 9. The procedure of creating a variable section sweep feature or surface is the same as was discussed in Chapter 9.

Figure 15-23 shows the section and trajectories that are used to create the variable section sweep surface. You have an option to keep the ends open or capped. This option is available in the **Options** slide-down panel.



**Figure 15-23** Variable section sweep surface with open ends



## CREATING SURFACES USING THE STYLE ENVIRONMENT OF Pro/ENGINEER WILDFIRE 4.0

Style is an environment available in Pro/ENGINEER that is used to draw free style curves and create surfaces by joining them. The surfaces created using the **Style** environment are called Super features. This is because these features can contain any number of curves or surfaces. The Style surfaces can be joined with the Pro/ENGINEER surfaces. They can have the parent-child relationship among themselves and also with Pro/ENGINEER features.



To enter the **Style** environment, choose the **Style** button from the **Base Features** toolbar or choose **Insert > Style** from the menu bar. Figure 15-24 shows the appearance of the **Style** environment.

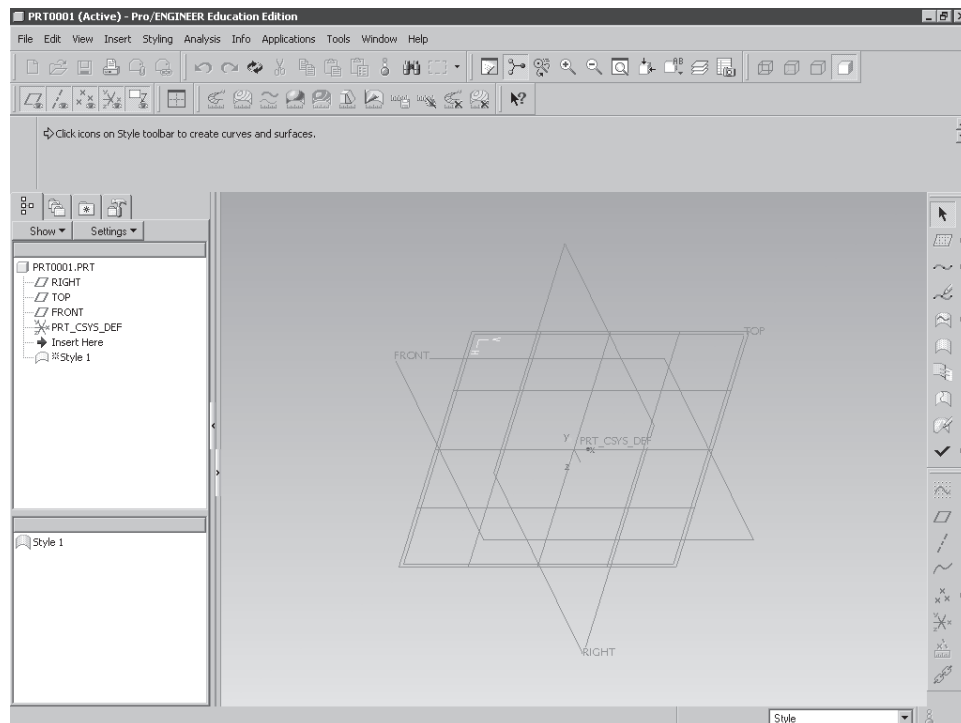


Figure 15-24 The *Style* environment

### Style Tools Toolbar

Figure 15-25 shows the **Style Tools** toolbar in the **Style** environment. The buttons in this toolbar are discussed next.

#### Select Button



This button is used to select the surfaces, curves, planes, and so on in the **Style** environment. If you are in middle of a feature creation tool, you can choose the **Select** button to exit that tool.

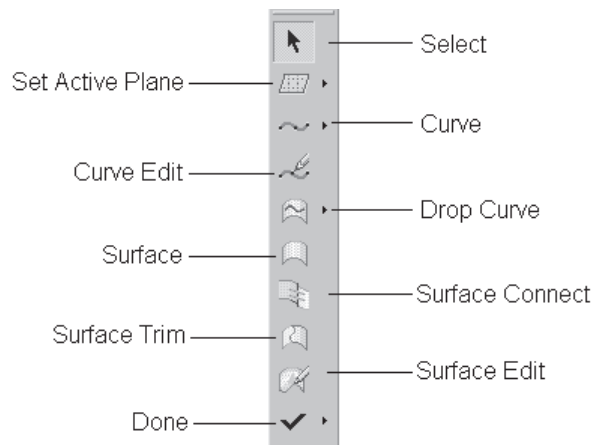


Figure 15-25 The Style Tools toolbar

### Set Active Plane Button



This button is used to select the datum plane on which the drawing or the editing operation needs to be performed. The datum plane that you select is highlighted by a mesh.

### Internal Plane Button



This button is chosen by selecting the black arrow on the right of the **Set Active Plane** button. When you select this arrow, a flyout will be displayed. Choose the **Internal Plane** button from this flyout to create an internal datum plane in the **Style** environment. When you choose this button, the **DATUM PLANE** dialog box will be displayed. This dialog box is used to create a datum plane in a similar procedure that was discussed in Chapter 5. The datum planes are named as DTM1, DTM2, and so on.

It should be noted that the datum planes created using this button will be displayed in the drawing area only when you are in the **Style** environment. Once you exit the **Style** environment, the datum plane becomes invisible. Any feature created in the **Style** environment will be displayed in the **Model Tree** as a style feature.

### Curve Button



This button is used to draw curves. When you choose this button, the **Curve** dashboard will be displayed, as shown in Figure 15-26. The options in this dashboard are discussed next.

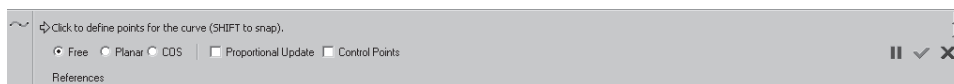


Figure 15-26 The Curve dashboard

**Free Radio Button**

When the **Curve** dashboard is displayed, the **Free** radio button will be selected by default and you will be prompted to define the points for the creation of the curve. To create the curve, click on the screen. A yellow point will be displayed at the location where you clicked. Now, again click to define the second point of the curve. The two points are joined. When you click to define the location of the third point, you will notice that the curve that you are drawing is defined by a spline. After defining the points, press the middle mouse button to create the curve. While specifying a point, if you press the SHIFT key, then the point is snapped to the entity already present on the screen.

Remember that the curve drawn using the **Free** option is created on the active datum plane. To draw a 3D curve, you need to snap the point on the existing entity. You can also draw a 3D curve by choosing the **Show All Views** button from the **Style** toolbar. When you choose this button, the display is turned into four windows. In Pro/ENGINEER, this type of display is called a 4-view display mode. The 4-view display mode shows the top, default, right-side, and front views. You can select a point in one window and then select the second point in the other window. By specifying points in different windows, the 3D curve can be drawn. To switch back to the single window display mode, choose the **Show All Views** button.



**Tip:** To undo the last operation, choose the **Undo** button from the **Edit** toolbar. The shortcut for undo is CTRL+Z.

**Planar Radio Button**

This radio button, when selected, allows you to create the curve on the datum plane that is highlighted by the mesh. This datum plane is called the active plane. The active plane can be selected before invoking the **Curve** dashboard by choosing the **Set Active Plane** button from the **Style Tools** toolbar.



**Tip:** Using the **Planar** option, you can project a point of an existing entity on the active datum plane. This can be done by selecting the point on the entity using the SHIFT key. The selected point is projected on the active datum plane.

**COS Radio Button**

This radio button is used to draw curves on surfaces. The points that you define on a surface are constrained to that surface. When you click to define the location of the first point of the curve, the point is placed. Now, this surface will be selected and the points placed hereafter should lie on the same surface. If you click outside this surface, then the point is not placed on the surface. After the curve is drawn, press the middle mouse button. The red curve is converted to a white curve indicating that the curve is completed. The curve drawn on the surface is the child of the surface.

**Control Points Check Box**

While drawing the curve, if this check box is selected, then while editing the curve the control points will be displayed.



**Tip:** Using the **Free** option, you can draw a curve on a surface. To draw a curve on a surface, press **SHIFT** to select a point on the surface. The surface will be highlighted as you select a point on it and then the point will be placed on it. This method of selecting points on a surface can be used to draw curves that join points on two separate surfaces.

## Curve Edit Button



This button is used to edit the curves that are created as style features. If you choose this button, the **Curve Edit** dashboard will be displayed and you will be prompted to select a curve. When you select a curve to edit, the **Curve Edit** dashboard appears, as shown in Figure 15-27.

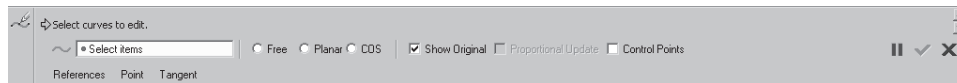


Figure 15-27 The **Curve Edit** dashboard

The options in the **Edit curve** dashboard are discussed next.

### Curve Collector

When you select a curve to edit, the id of the curve will be displayed in this collector.

### Free Radio Button

If the curve selected for editing is drawn using the **Free** option, then this radio button will be selected by default.

### Planar Radio Button

If the curve selected for editing is drawn using the **Planar** option, then this radio button will be selected by default.

### COS Radio Button

If the curve selected for editing is drawn using the **COS** option, then this radio button will be selected by default.

### Proportional Update Check Box

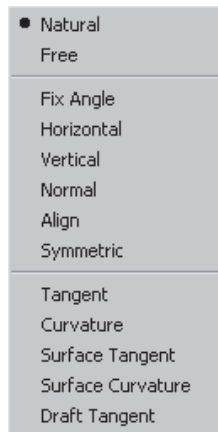
If the curve selected for editing is drawn using the **Proportional Update** option, then the curve will be edited proportionately with the points.

### Control Points Check Box

If the curve that for editing is drawn using the **Control Points** option, then the control points will be displayed on the curve. Using these control points, you can modify the shape of the curve.

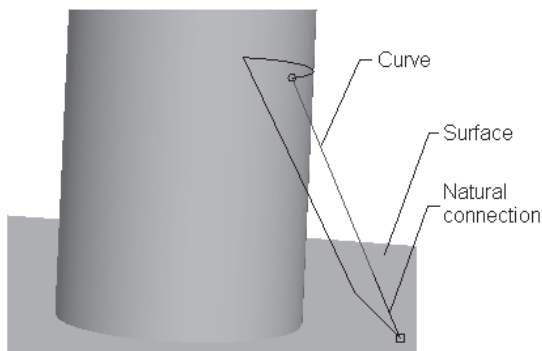
### Shortcut Menu Options

When one of the end control points of the curve is selected, the tangent vector of the curve is highlighted in yellow color. Right-click the yellow vector to display the shortcut menu, as shown in Figure 15-28.

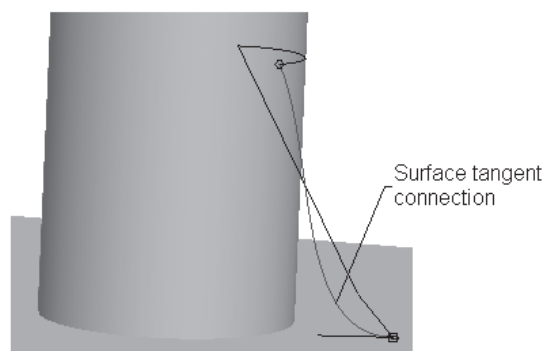


**Figure 15-28** Shortcut menu for end control point of the curve

By default, a curve has a natural contact with the adjacent surface. This is evident from the check mark on the left of the **Natural** option in the shortcut menu. Figure 15-29 shows the curve that is connected to the adjacent surface using the **Natural** option. The curve is drawn using the **Free** option. The point on the cylindrical surface will be selected by using SHIFT+left mouse button and similarly another point will be selected on the surface at the base. Figure 15-30 shows the curve whose contact type is changed to the **Surface Tangent** option by choosing it from the shortcut menu.



**Figure 15-29** Curve joining the two surfaces



**Figure 15-30** Curve joining the base surface tangentially

### Drop Curve Button



Using this button, a curve created in the **Style** environment can be projected onto the selected surface.

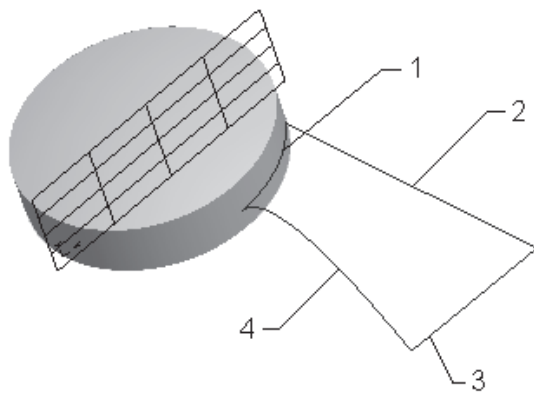
Choose the **Drop Curve** button from the **Style Tools** toolbar to create COS's (Curve On Surface); you will be prompted to select the curve that you need to drop onto the surface. Select the curve and press the middle mouse button. Now, you will be prompted to select the

surface on which you need to drop the curve. Select the surface and press the middle mouse button; you will be prompted to select the plane normal to which the curve will be dropped. Select the plane normal to which the curve will be projected and exit the dashboard.

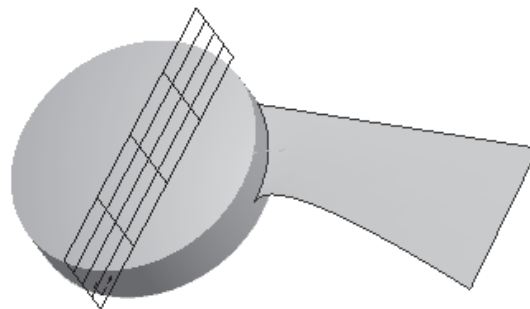
### Surface Button



This button is used to create a surface among a closed boundary of curves. When you choose this button, the **Surfaces** dashboard will be displayed and you will be prompted to select three or four boundary curves to define a surface. Select the four curves, as shown in Figure 15-31. After selecting the four curves, press the middle mouse button. The surface is created, as shown in Figure 15-32.



*Figure 15-31 Four curves*



*Figure 15-32 Surface created using the curves*

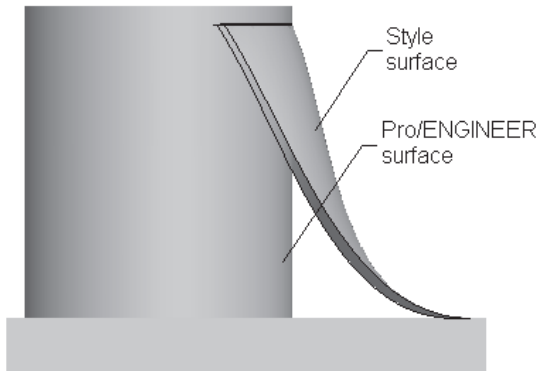
### Surface Connect Button



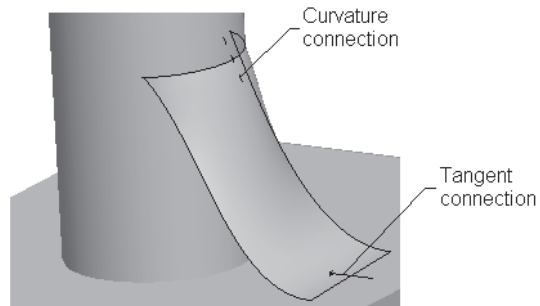
When you choose this button, the **Surface Connect** dashboard will be displayed and you will be prompted to select two surfaces. The Style surface can be connected to the Pro/ENGINEER surface. When you select these two surfaces, as shown in Figure 15-33, and press the middle mouse button, the connections are automatically applied to the two surfaces. These connections may be of two types: curvature connection represented by a dashed line and the tangent connection represented by an arrow. If the tangent connection is applied, then the arrow will be displayed and if the curvature connection is applied, then a dashed line will be displayed on the surfaces. Figure 15-34 shows the two surfaces where the tangent connection as well as the curvature connection has been applied.

Choose the **Surface Connect** button; the **Surface Connect** dashboard will be displayed, as shown in Figure 15-35, and you will be prompted to select the two surfaces. Select the surfaces. To apply the connection, click on any one end of the dashed line; the dashed line will be converted to an arrow, indicating that the two surfaces are connected. To remove the connection, use SHIFT+left click on the arrow.

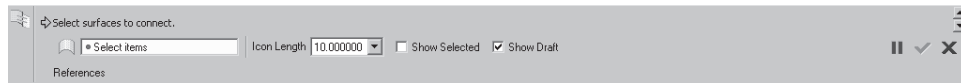
Figure 15-36 shows the style surface when it is connected using curvature connection and Figure 15-37 shows the surface when it is connected tangentially.



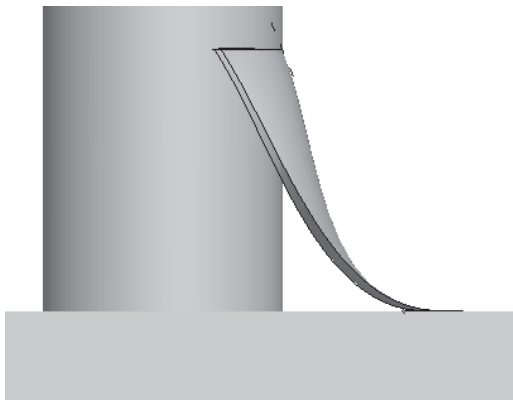
**Figure 15-33** The two surfaces



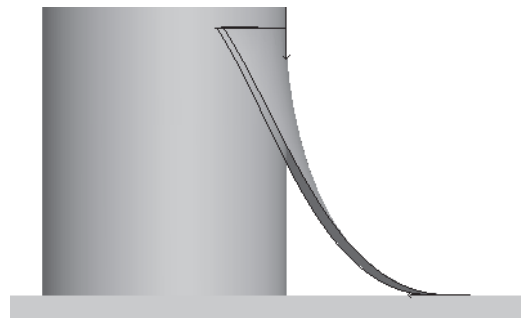
**Figure 15-34** Arrow and dashed line



**Figure 15-35** The Surfaces Connect dashboard



**Figure 15-36** Surface connected at top by curvature connection



**Figure 15-37** Surface connected at top by tangent connection



#### Note

The **Icon Length** dimension box on the **Surface Connect** dashboard is used to increase the length of the arrow and the dashed line.

To delete a curve, select the curve and when it turns red, press the **DELETE** key.

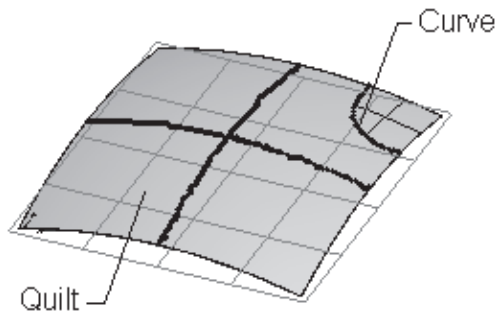
### Surface Trim Button



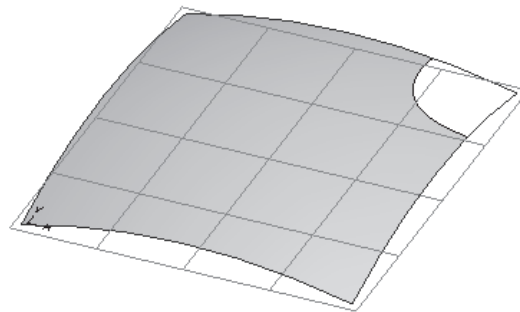
This button is used to trim a surface. When you choose this button, the **Surface Trim** dashboard will be displayed and you will be prompted to select the surface(s) to trim. Select the surface so that it turns pink in color and then press the middle mouse

button. Now, you will be prompted to select the curve that can be used to trim the surface. Select the curve and press the middle mouse button. The selected surface will be highlighted in two portions. Select the portion to delete. Choose the green check mark to exit the trim tool.

Figure 15-38 shows the surface and the curve that are selected for trimming. This figure also shows the surface divided into two portions. The portion defined by the curve will be selected to delete. Figure 15-39 shows the surface after trimming.



**Figure 15-38** Surface and Curve selected for trimming



**Figure 15-39** Surface after trimming



#### Note

After completing the Style feature creation, choose the **Done** button to exit the **Style** environment.

## SURFACE EDITING TOOLS

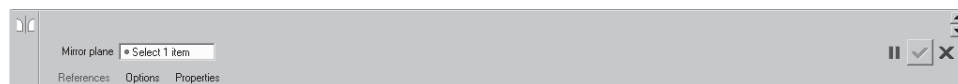
The surface editing tools help in decreasing the modeling time. They also help in creating complex surface models. The surface editing tools that you will be learning in the next section are as follows:

- |           |              |                 |
|-----------|--------------|-----------------|
| 1. Mirror | 5. Intersect | 9. Vertex Round |
| 2. Merge  | 6. Offset    |                 |
| 3. Trim   | 7. Thicken   |                 |
| 4. Fill   | 8. Solidify  |                 |

### Mirroring the Surfaces



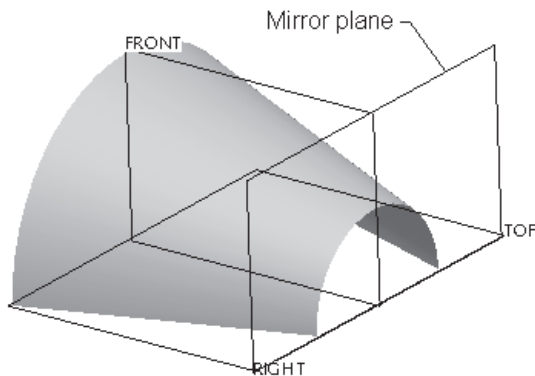
The **Mirror** button is used to mirror the surface about a plane. This button is available in the **Edit Features** toolbar only when a surface is selected. When you choose this button, the **Mirror** dashboard will be displayed, as shown in Figure 15-40.



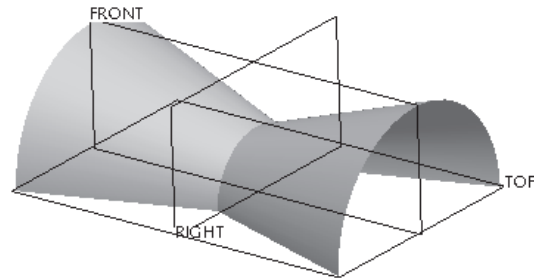
**Figure 15-40** The **Mirror** dashboard



Using the **References** tab, you can choose the mirroring plane. The **Copy as dependent** check box in the **Options** tab is selected by default. This makes sure that the parent-child relationship is maintained between the mirrored and original surfaces. Figure 15-41 shows the mirror plane and Figure 15-42 shows the mirrored feature.



**Figure 15-41** Mirror plane and the surface to be mirrored



**Figure 15-42** Surfaces after mirroring and keeping the original surface

## Merging Surfaces



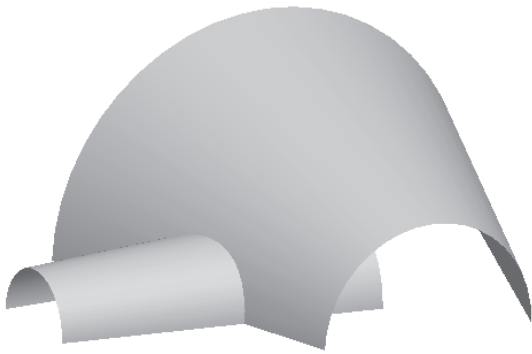
The **Merge** button is used to merge two surfaces and make them a single surface. A surface is also known as a quilt. To convert a surface to a solid, it is necessary that the surfaces are merged. While merging the surfaces, this tool also trims them. This button is available in the **Edit Features** toolbar only when the two surfaces to be merged are selected. When you choose the **Merge** button, the **Merge** dashboard will be displayed, as shown in Figure 15-43.



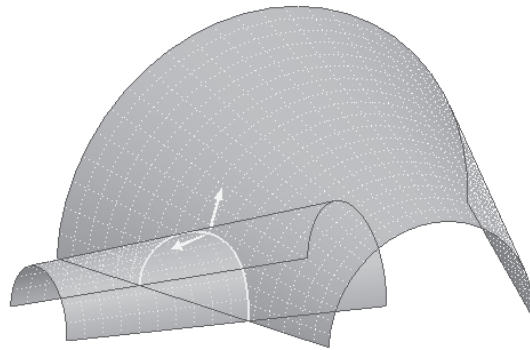
**Figure 15-43** The Merge dashboard

The following steps explain the procedure to merge the surfaces, as shown in Figure 15-44.

1. Select the **Quilts** option from the **Filter** drop-down list. Select the two surfaces and when they turn pink in color, choose the **Merge** button; the **Merge** dashboard will be displayed. While merging two surfaces, the part of the surfaces that will be retained after the two surfaces are merged is highlighted by yellow dots on it. The yellow arrows point to the direction in which the surfaces are retained. The direction of yellow arrow can be toggled by using the **Change side of first quilt to keep** and the **Change side of second quilt to keep** buttons in the **Merge** dashboard.
2. Choose the **Change side of first quilt to keep** button and then choose the **Change side of second quilt to keep** button. Notice that the outer side of the surfaces are highlighted with yellow dots, as shown in Figure 15-45.

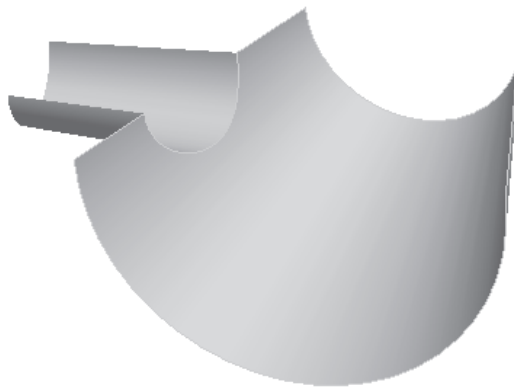


**Figure 15-44** Two surfaces to merge



**Figure 15-45** Arrows showing the part of the surface to retain

3. Choose the **Preview** button and then exit the dashboard. The resulting merged surface is shown in Figure 15-46. This merged surface is a single surface and now can be converted to a solid feature.



**Figure 15-46** Resulting merged surface

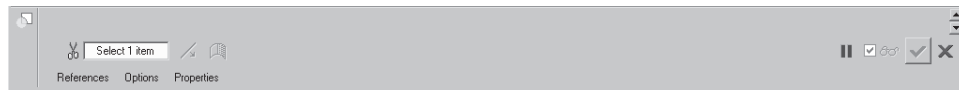
The **Reference** tab of the **Merge** dashboard shows the selected quilts. In the **Options** tab, you can select between **Intersect** and **Join** options. The **Join** option can be used when the edge of one quilt lies on the other quilt.

## Trimming Surfaces



As the name suggests, the **Trim** button is used to trim the selected surfaces using a trimming object. Select the surface that you need to trim and then choose the **Trim** button from the **Edit Features** toolbar; the **Trim** dashboard will be displayed, as shown in Figure 15-47. Also, you will be prompted to select the trimming object. This trimming object can be a curve, plane, edge, or a surface.

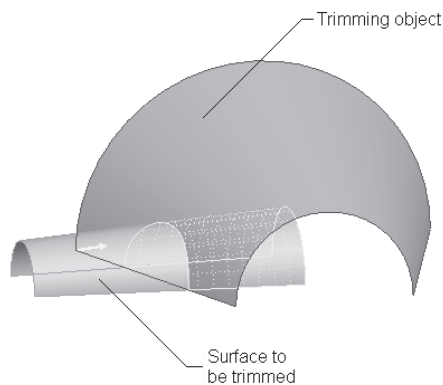
The part of the surface that is to be retained is highlighted with yellow dots. A yellow arrow



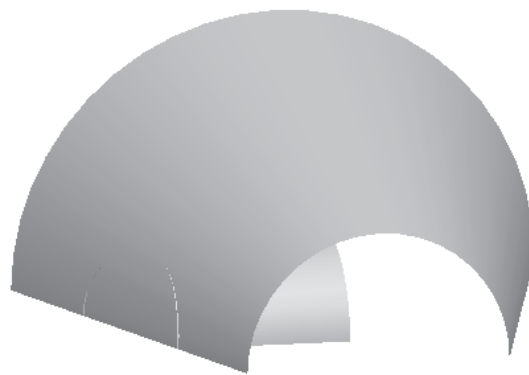
*Figure 15-47 The **Trim** dashboard*

points in the direction of the surface to be retained after trimming. You can choose the **Flip between one side, other side, or both sides of trimmed surface to keep** button to toggle the direction of the yellow arrow. By default, the trimming object is deleted after the surfaces are trimmed. If you need to keep the trimming object, select the **Keep trimming surface** check box from the **Options** slide-down panel.

Figure 15-48 shows the surface selected as the trimming object, the trimming surface, and the yellow arrow. From this figure it is evident that the arrow is pointing toward the right; therefore the right portion of the surface will be retained after trimming. Figure 15-49 shows the surface obtained after trimming.



*Figure 15-48 Surface and object to be Trimmed*



*Figure 15-49 Surface obtained after trimming*

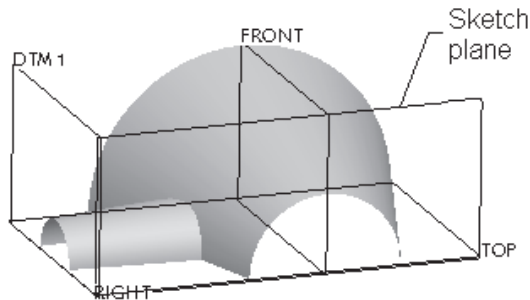
## Creating the Fill Surfaces

The **Fill** option is used to create a planar surface by sketching its boundaries. When you choose this option from the **Edit** menu in the menu bar, the **Fill** dashboard will be displayed, as shown in Figure 15-50.

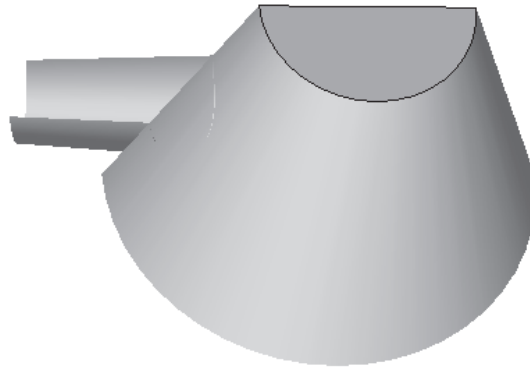


*Figure 15-50 The **Fill** dashboard*

From the **References** slide-down panel, choose the **Define** button to select the sketching plane and draw the sketch. Figure 15-51 shows the sketch plane and Figure 15-52 shows the surface that is created using the **Fill** option.



**Figure 15-51** The sketch plane for creating the fill surface



**Figure 15-52** Fill surface

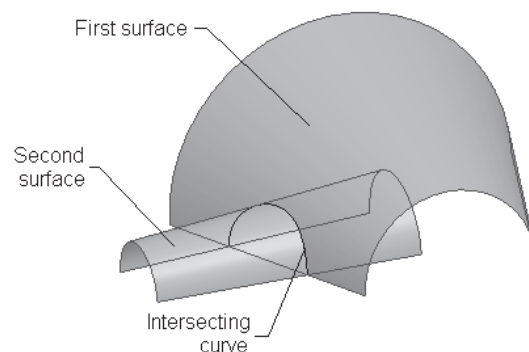
## Creating the Intersect Curves

The **Intersect** option is used to create a curve at the intersection of two surfaces. The intersect curve can then be used for various purposes. The **Intersect** option is available in the **Edit** menu only when you have selected a surface. When you choose this option from the **Edit** menu, the **Intersect** dashboard will be displayed, as shown in Figure 15-53.



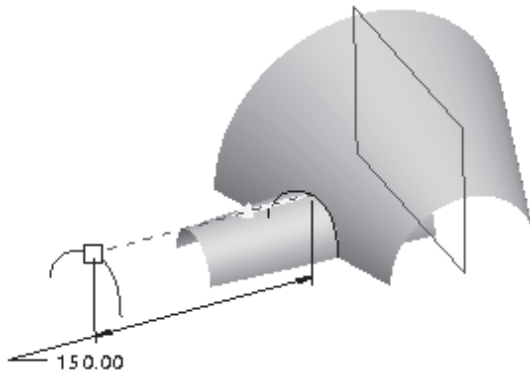
**Figure 15-53** The **Intersect** dashboard

When you select the second surface, the intersecting curve is created, as shown in Figure 15-54. Make sure to select the second surface by holding the CTRL key. The curve created can be copied, moved, and so on. One of the uses of the intersect curve is shown in Figures 15-55 and 15-56.

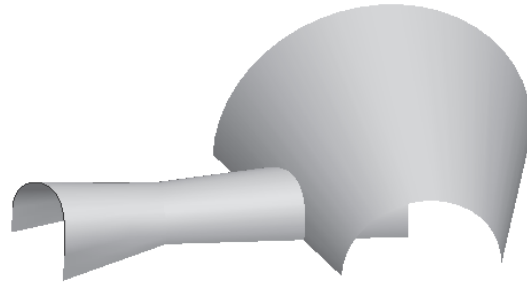


**Figure 15-54** Surfaces selected to create the intersecting curve

In Figure 15-55, the intersecting curve is copied at a distance of 150. To create the surface shown in Figure 15-56, the **Boundary Blend** tool is used. To create the boundary blend, the intersecting curve will be selected and then the curve edge of the surface will be selected. Both the curves are blended and the tangency is increased by dragging the handles.



*Figure 15-55 Copied curve*



*Figure 15-56 Boundary blend created using the intersecting curve*

## Creating the Offset Surfaces

A surface can be copied to an offset distance. To offset a surface, select the surface to offset and choose **Edit > Offset** from the menu bar. The **Offset** dashboard will be displayed, as shown in Figure 15-57. The **Offset** option is available only when you have selected a surface to offset.



*Figure 15-57 The Offset dashboard*

In Pro/ENGINEER, there are three methods to offset a surface. These methods are as follows:

1. Create the offset of the whole surface using the **Standard Offset Feature** option.
2. Sketch a section and offset the area inside the section with the draft using the **With Draft Feature** option.
3. Sketch a section and offset the area inside or outside the section using the **Expand Feature** option.

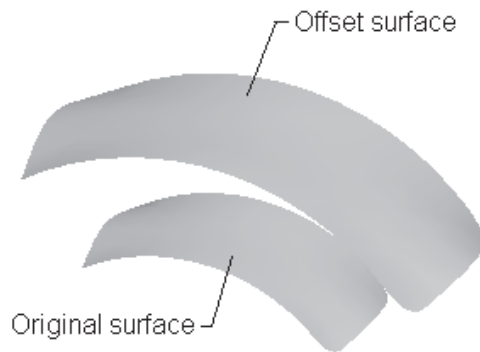
In the **Offset** dashboard, first you need to specify the type of offset surface you need to create. The types of offset that can be created in Pro/ENGINEER Wildfire 4.0 are as follows:

1. Standard Offset Feature
2. With Draft Feature
3. Expand Feature

### Standard Offset Feature

The **Standard Offset Feature** button is present at the upper-left corner of the **Offset** dashboard and is chosen by default. You can enter the offset value in the dimension edit box.

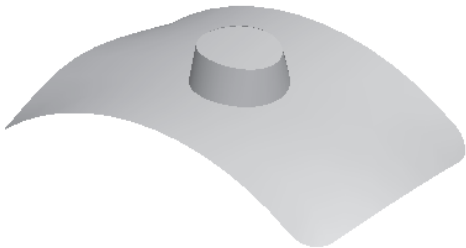
This option can be used to offset the surface as a whole. From the drop-down list in the **Options** slide-down panel, the **Normal to Surface** option is selected by default. This option allows you to offset a surface normal to the surface. If you select the **Automatic Fit** option from the drop-down list, then Pro/ENGINEER automatically fits the surface, or controls the direction of the offset in the X, Y, and Z axes. If you select the **Controlled Fit** option from the drop-down list in the **Options** slide-down panel, you need to select a coordinate system and specify the direction to offset. You can also join the offset surface with the side surfaces by selecting the **Create side surface** check box. Figure 15-58 shows the original surface and the offset surface.



*Figure 15-58 Original and the offset surfaces*

### With Draft Feature

The **With Draft Feature** button can be chosen from a list of buttons that appear by selecting the black arrow present on the right of the **Standard Offset Feature** button. Using this button, you can sketch the section and then give a draft angle to the side surfaces. Choose the **Define** button from the **References** slide-down panel to define a sketching plane and to create the sketch. Figure 15-59 shows the draft offset surface with the **Straight** radio button selected from the **Options** slide-down panel. The section that was drawn on the sketching plane was circular. Similarly, Figure 15-60 shows the draft offset surface with the **Tangent** radio button selected from the **Options** slide-down panel.



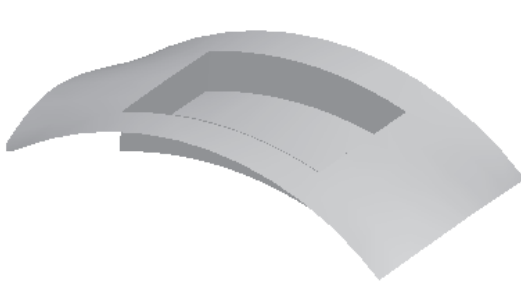
*Figure 15-59 Draft offset surface with straight profile*



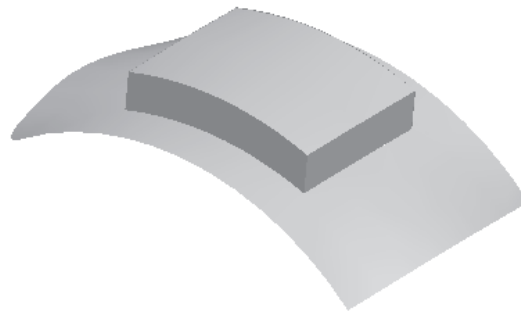
*Figure 15-60 Draft offset surface with tangent profile*

### Expand Feature

The **Expand Feature** button can be selected from the list of buttons that appear by selecting the black arrow present on the right of **Standard Offset Feature** button. Using this option, you can sketch the section and then choose to offset the inside or the outside of the sketch. For this purpose, you need to choose the **Flip the material sides of sketch** button from the dashboard. Figure 15-61 shows the offset surface when the inside of the sketch is selected to offset. The section that was drawn on the sketching plane was rectangular. Choose the **Define** button from the **Options** slide-down panel to define the sketching plane and create the sketch. Figure 15-62 shows the draft offset surface when the outside of the sketch is selected to offset.



**Figure 15-61** Inside of the sketch selected to offset



**Figure 15-62** Outside of the sketch selected to offset

### Giving Thickness to a Surface

To add thickness to a quilt or a surface, select the quilt and choose the **Thicken** option from the **Edit** menu; the **Thicken** dashboard will be displayed, as shown in Figure 15-63.

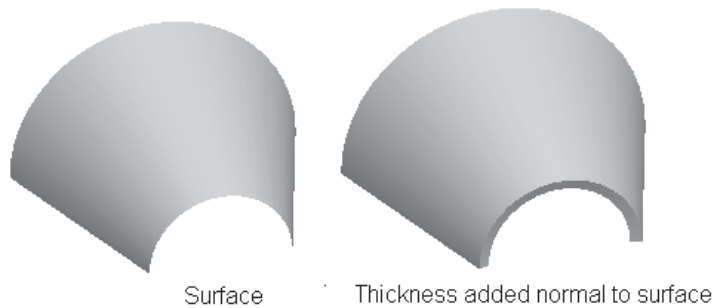


**Figure 15-63** The **Thicken** dashboard

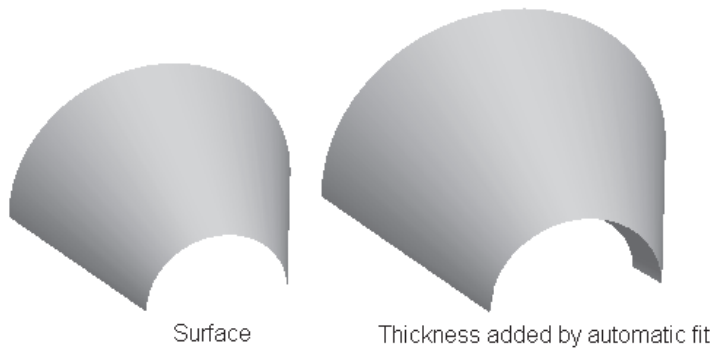
Drag the handle to set thickness of the quilt or enter the thickness value in the dimension edit box. You can even remove material from the quilt by choosing the **Removes material from inside thickened quilt** button from the dashboard.

Using the drop-down list in the **Options** slide-down panel, you can give thickness to the quilt normal to the surface, allow Pro/ENGINEER to automatically scale the surface along the axes, or scale and fit the original surface with respect to the coordinate system. If you select the **Controlled Fit** option from the drop-down list in the **Options** slide-down panel, you need to select a coordinate system and specify the direction to scale.

Figures 15-64 and 15-65 show the surfaces after adding thickness by controlling thickness using the **Normal to surface** option and the **Automatic fit** option, respectively.



**Figure 15-64** Thickening the surface using the *Normal to surface* option



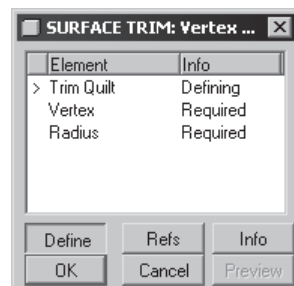
**Figure 15-65** Thickening the surface using the *Automatic fit* option

## Converting a Surface to a Solid

You can convert a closed surface into a solid by choosing **Edit > Solidify** from the menu bar. This option is available only when a closed surface with capped ends is selected. This option fills the hollow surface with material.

## Creating a Round at the Vertex of a Surface

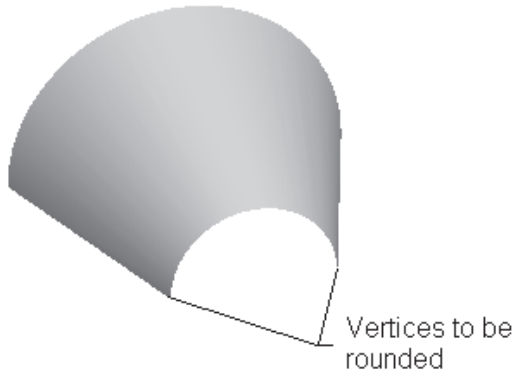
The vertices of a surface or quilt can be rounded using the **Vertex Round** option. Choose **Insert > Advanced > Vertex Round** from the menu bar. The **SURFACE TRIM: Vertex Round** dialog box will be displayed, as shown in Figure 15-66. You will be prompted to select



**Figure 15-66** The *SURFACE TRIM: Vertex Round* dialog box



the datum quilt to intersect. Select the surface; now you are prompted to select the corner vertex(s) to be rounded. Select the first vertex and then press the CTRL key to select the other vertex, as shown in Figure 15-67. After selecting the vertices, press the middle mouse button. The **Message Input Window** will be displayed. Type the radius of round and press ENTER. Choose the **OK** button from the **SURFACE TRIM: Vertex Round** dialog box. The vertices are rounded, as shown in Figure 15-68.



*Figure 15-67 Vertices selected to be rounded*

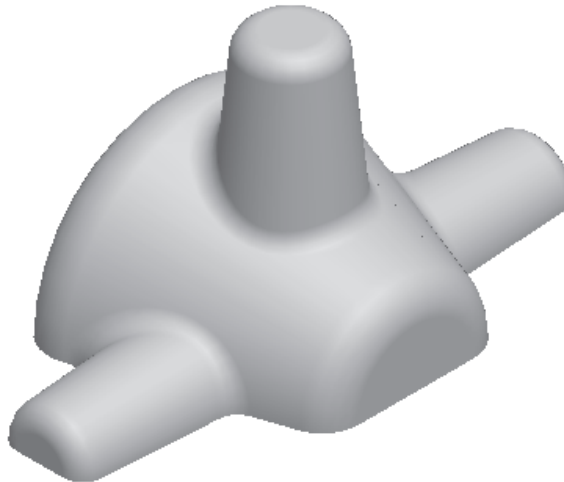


*Figure 15-68 Vertices after creating round*

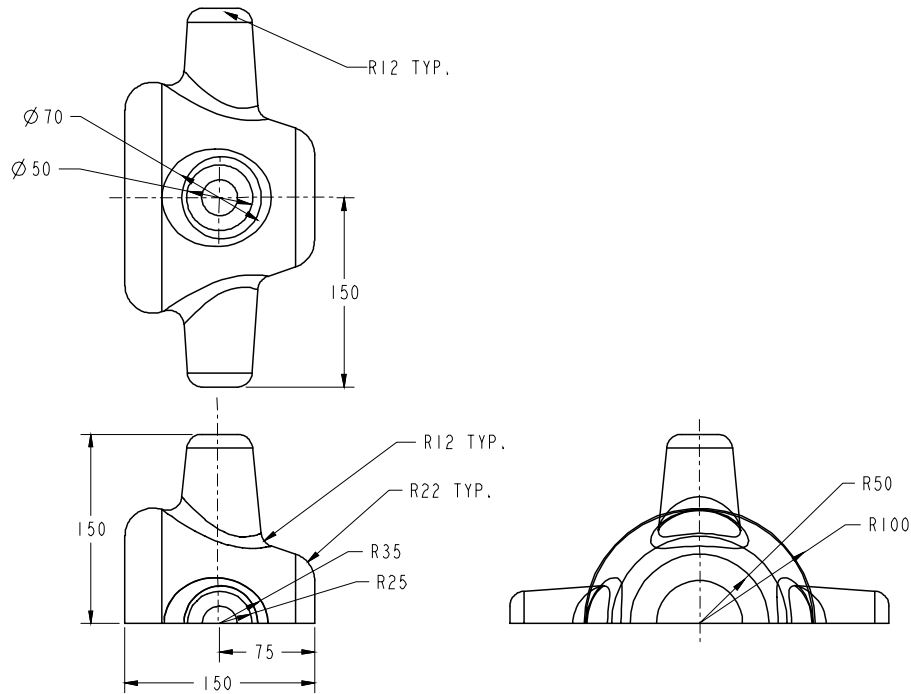
## TUTORIALS

### Tutorial 1

In this tutorial, you will create the surface model shown in Figure 15-69. The orthographic views of the surface model are shown in Figure 15-70. **(Expected time: 45 min)**



*Figure 15-69 Isometric view of the surface model*



**Figure 15-70** Top view, front view, and right-side view of the surface model

The following steps are required to complete this tutorial:

- Examine the model and determine the number of features in it, refer to Figure 15-70.
- Create the base feature, which is a blend surface, refer to Figures 15-71 through 15-73.
- Create the second feature, which is a blend feature, refer to Figures 15-74 and 15-75.
- The third feature is a mirror feature that will be created by mirroring the second feature about a plane passing from the center, refer to Figure 15-76.
- Create the fourth feature which is also a blend feature, refer to Figures 15-77 and 15-78.
- Next, select the surfaces individually and merge them, refer to Figures 15-79 and 15-80.
- Remaining features are the fill features and they will be used to create surfaces on the blend features, refer to Figures 15-81 through 15-83.
- Create rounds on the edges, refer to Figure 15-84.

### Starting a New Object File

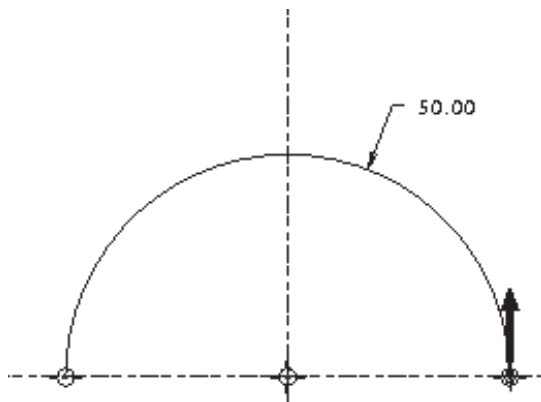
- Start a new part file and name it as *c15tut1*.

The three default datum planes are displayed in the drawing area. The **Model Tree** is also displayed in the drawing area. Close the **Model Tree** by clicking on the sash present on its right edge.

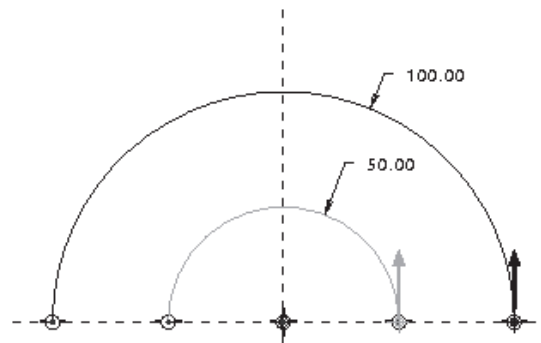
### Creating the Base Feature

You will use the menu bar present on the top of the screen to invoke the **Blend** option. The **Blend** option will be used to create the base feature.

1. Choose **Insert > Blend > Surface** from the menu bar; the **BLEND OPTS** menu is displayed.
2. Choose **Parallel > Regular Sec > Sketch Sec > Done** from the **BLEND OPTS** menu; the **SURFACE** dialog box and the **ATTRIBUTES** menu is displayed.
3. Choose **Straight > Open Ends > Done** from the **ATTRIBUTES** menu; you are prompted to select the sketch plane.
4. Select the **RIGHT** datum plane; the **DIRECTION** menu is displayed.
5. Choose **Okay** from the **DIRECTION** menu; the **SKET VIEW** menu is displayed.
6. Select the **Top** option and then choose the **TOP** datum plane.
7. Draw the arc and dimension it, as shown in Figure 15-71.
8. After drawing the first arc, press and hold down the right mouse button and choose the **Toggle Section** option from the shortcut menu.
9. Draw the second arc and dimension it, as shown in Figure 15-72.



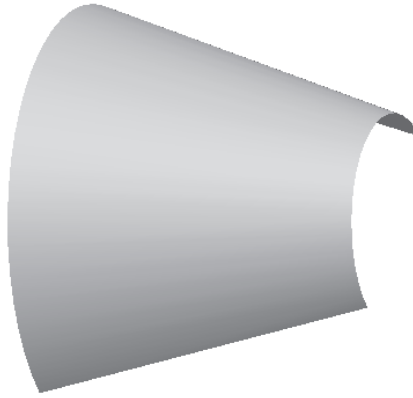
*Figure 15-71 Sketch of the first arc*



*Figure 15-72 Sketch of the second arc*

10. After drawing the sketch, choose the **Done** button to exit the **Sketcher** environment; the **DEPTH** menu is displayed.
11. Choose **Blind > Done** from the **DEPTH** menu; the **Message Input Window** is displayed.
12. Enter the value **150** and press ENTER.

13. Choose **OK** from the **SURFACE** dialog box; the base feature is created, as shown in Figure 15-73.



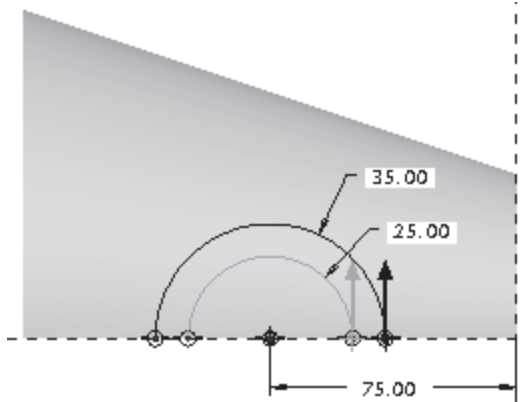
*Figure 15-73 Trimetric view of the base feature*

### Creating the Second Feature

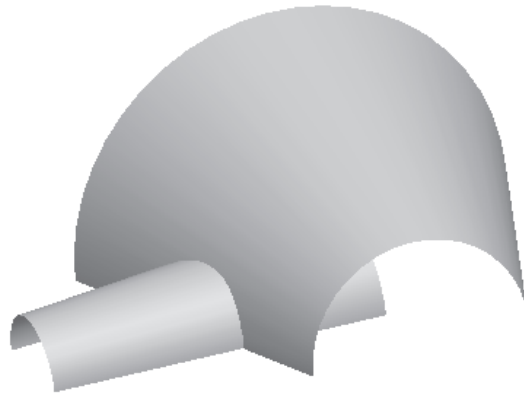
To create the second blend feature, you need to create a datum plane that is at a distance of 150 from the **FRONT** datum plane and passes through the center of the base feature.

1. Choose **Insert > Blend > Surface** from the menu bar.
2. Choose **Parallel > Regular Sec > Sketch Sec > Done** from the **BLEND OPTS** menu.
3. Choose **Straight > Open Ends > Done** from the **ATTRIBUTES** menu; you are prompted to select the sketch plane.
4. Choose the **Make Datum** option to display the **DATUM PLANE** menu. Select the **Offset** option and then select the **FRONT** datum plane.
5. Press the middle mouse button to confirm the selection. Now, select the **Enter Value** option from the **OFFSET** menu to display the **Message Input Window** and enter 150 in it.
6. Select the **Done** option from the **DATUM PLANE** menu and then select **Okay** from the **DIRECTION** menu.
7. Set the orientation of the sketching plane by selecting the **TOP** datum plane to be at top while sketching.
8. Sketch the first arc of radius 25, and then after toggling the sketch, draw the second arc, as shown in Figure 15-74.
9. Exit the **Sketcher** environment; the **DEPTH** menu is displayed.

10. Choose **Thru Until > Done** from the **DEPTH** menu.
11. Select the **FRONT** datum plane. Choose **OK** from the **SURFACE** dialog box. The blend surface is extruded up to the selected datum plane, as shown in Figure 15-75.




*Figure 15-74 Sketch of the second feature*

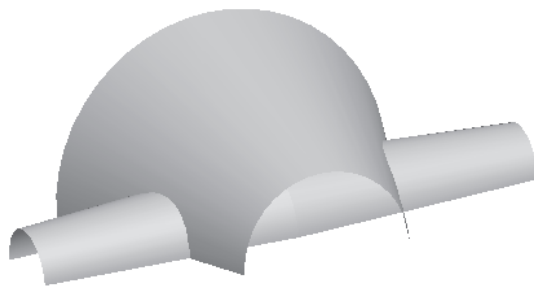


*Figure 15-75 Second feature created*

### Creating the Mirror Copy of the Second Feature

The third blend feature is the same as the second blend feature. Therefore, a mirror copy of the second feature will be used to create the third feature.

1. Select the second feature and then choose the **Mirror** button; the **Mirror** dashboard is displayed and you are prompted to select a plane to mirror about. 
2. Select the **FRONT** datum plane and exit the **Mirror** dashboard by choosing the **Build Feature** button. The mirror copy of the second feature is created, as shown in Figure 15-76.



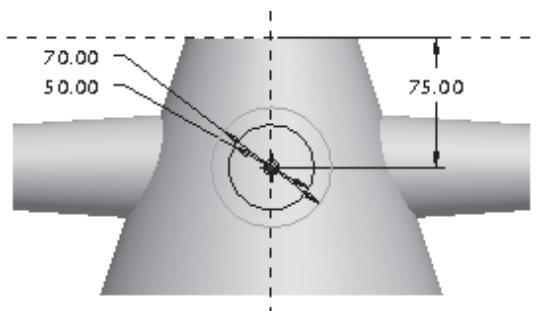
*Figure 15-76 Surface model after creating the third feature*

### Creating the Fourth Blend Feature

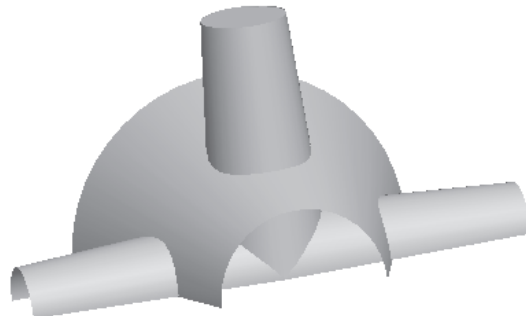
The fourth blend feature will be created on the top of the base feature. To create the

blend feature, you will need to create a datum plane that is at a distance of 150 from the bottom of the base feature.

1. Choose **Insert > Blend > Surface** from the menu bar.
2. Choose **Parallel > Regular Sec > Sketch Sec > Done** from the **BLEND OPTS** menu.
3. Choose **Straight > Capped Ends > Done** from the **ATTRIBUTES** menu; you are prompted to select the sketch plane.
4. Choose the **Make Datum** option to display the **DATUM PLANE** menu. Select the **Offset** option and create a datum plane at a distance of 150 from the **TOP** datum plane.
5. Set the orientation of the sketching plane by selecting the **RIGHT** datum plane to be at the top while sketching.
6. Sketch the first circle of diameter 50 and dimension it. Toggle the sketch and then draw the second circle of diameter 70, as shown in Figure 15-77.
7. Exit the **Sketcher** environment; the **DEPTH** menu is displayed.
8. Choose **Thru Until > Done** from the **DEPTH** menu.
9. Select the **TOP** datum plane. Choose **OK** from the **SURFACE** dialog box. The blend surface is extruded up to the selected datum plane, as shown in Figure 15-78.



**Figure 15-77** Sketch of the fourth blend feature



**Figure 15-78** Model after creating the fourth blend surface

### Merging the Surfaces to Create a Quilt

To create a round on the edges, it is necessary to create a common edge where the two surfaces join. For this purpose, the surfaces are merged.

**Note**

*It is easier to select two surfaces for merging from the **Model Tree**. You should remember that to select more than one surface, you need to press the CTRL key. When you select the surfaces from the **Model Tree**, their boundaries are highlighted in red, indicating that the surface is selected. To select a surface directly from the drawing area, you need to select it thrice. The third time when you select the surface, it turns pink.*

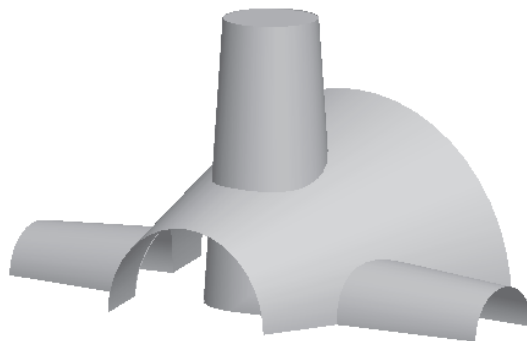
*You can also select the **Quilt** option from the **Filter** drop-down list to select the surfaces. The **Filter** drop-down list is available in the **Status Bar** at the bottom right corner of the main window.*

1. Select the blend surface from the left and then select it in the middle. When the two surfaces are highlighted, choose the **Merge** button; the **Merge** dashboard is displayed and the two arrows show the portion that will be retained after merging.

**Note**

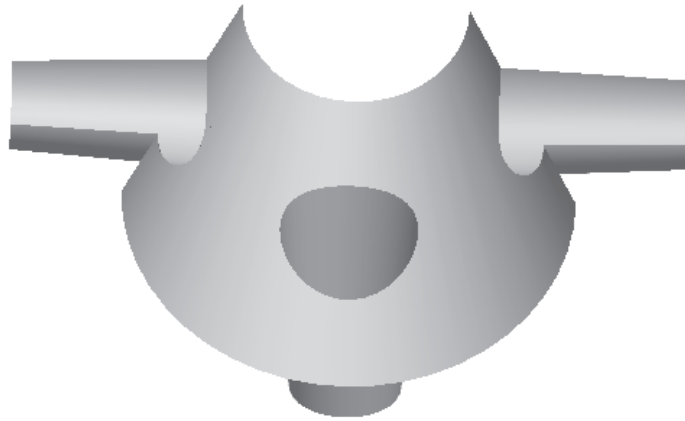
*The **Merge** button is available only when the two surfaces are selected for merging.*

2. Choose the **Change side of first quilt to keep** button from the dashboard. The direction of the yellow arrow changes.
3. Choose the **Change side of second quilt to keep** button from the dashboard. The direction of the yellow arrow changes. The portion of the surface that is now highlighted will be retained after merging.
4. Exit the dashboard by choosing the **Build Feature** button. The model after merging the two surfaces, is shown in Figure 15-79.



**Figure 15-79** Surface model after merging the two surfaces

Using the same procedure, merge the blend surface on the right with the blend surface at the middle. After that, merge the top blend surface with the middle blend surface. Figure 15-80 shows the surface model after merging all surfaces and forming a quilt.



*Figure 15-80 Surface model after merging the surfaces*

### Creating the Fill Surfaces

Four surfaces will be created to cap the ends of the blend surfaces. First, the left blend surface will be capped using the **Fill** option.

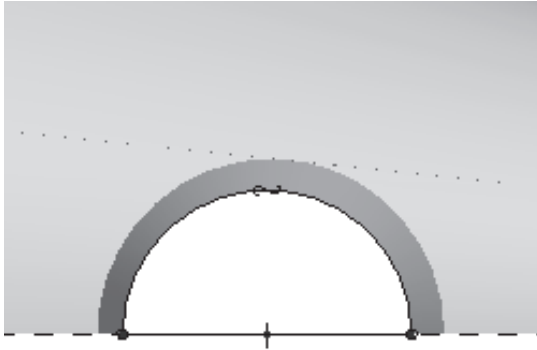
1. Choose the **Fill** option from the **Edit** menu; the **Fill** dashboard is displayed.
2. From the **References** slide-down panel, choose the **Define** button; the **Sketch** dialog box is displayed and you are prompted to select the sketching plane.
3. Choose the **Plane** button from the **Datum** toolbar.
4. Select the two vertices of the left blend surface. To select the second vertex hold down the CTRL key. Then holding down the CTRL key, select the **FRONT** datum plane.
5. Select **FRONT** from the **DATUM PLANE** dialog box. The drop-down list appears in the row where you clicked. From the drop-down list, select the **Parallel** option. Choose the **OK** button from the **DATUM PLANE** dialog box.

The datum plane is created and a yellow arrow points in the direction of viewing the sketch.

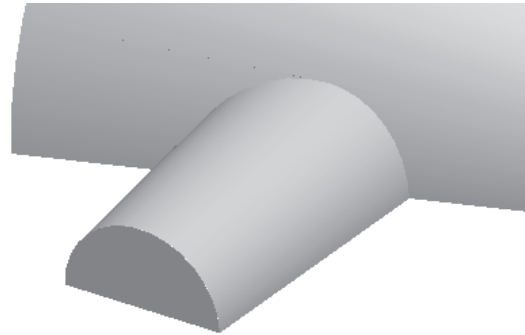
6. Choose the **Sketch** button to close the **Sketch** dialog box to enter the **Sketcher** environment.
7. Choose the **Use** button from the **Sketcher tools** toolbar and select the smaller semicircular edge of the blend surface; the sketch is completed, as shown in Figure 15-81.



- Exit the **Sketcher** environment and then exit the **Fill** dashboard by choosing the **Build Feature** button. The Fill surface is created, as shown in Figure 15-82.

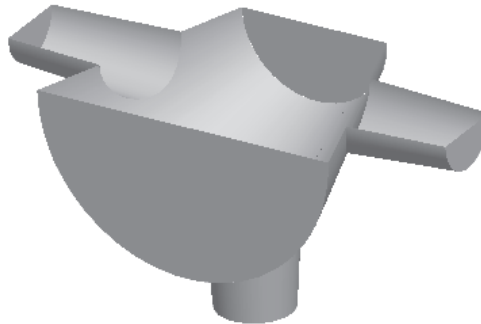


*Figure 15-81 Sketch for the fill surface*



*Figure 15-82 Surface after creating the fill surface*

Similarly, create the fill surfaces to cap the ends of the middle surface blend feature. Mirror the fill surface to create the fill surface on the right blend surface. Figure 15-83 shows the surface model after capping all the ends of the blend surfaces.



*Figure 15-83 Surface model after creating the fill surfaces*

### Merging the Fill Surfaces

The fill surfaces that you have created should be merged with the other surfaces in order to create a round on their edges.

- Hold the CTRL key to select the fill surface that is on the left and then the blend surface in the middle. When the two surfaces turn pink in color, choose the **Merge** button to display the **Merge** dashboard.
- Exit the dashboard by choosing the **Build Feature** button.

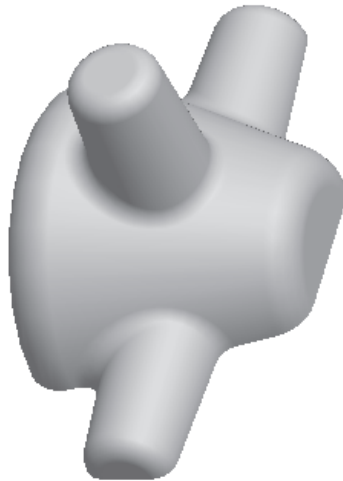
Using the same procedure, merge the remaining fill surfaces individually with the blend surface in the middle. To check whether all the surfaces are merged, select the surface model thrice. If the whole surface model is highlighted in pink color then all the surfaces are merged and they form a quilt.

### Creating Rounds

When all the surfaces are merged, then the edges are obtained at the intersection of two surfaces. These edges can be easily rounded. In the given surface model, note that there are rounds that have two different radius values. Therefore, you need to create two sets to define two values of rounds.

1. Choose the **Round** button from the **Engineering Features** toolbar.
2. Select the edges that have a radius value of 12. Remember that to select more than one edge, you need to hold down the CTRL key.
3. After creating the rounds of radii 12, choose the **Sets** tab to display the slide-down panel.
4. Click on **New set**; you have added a set that is named **Set2**.
5. Select the two edges that have a radii of 22. After creating the rounds of radii 22, exit the **Round** dashboard.

The surface model, after creating the rounds, is as shown in Figure 15-84.

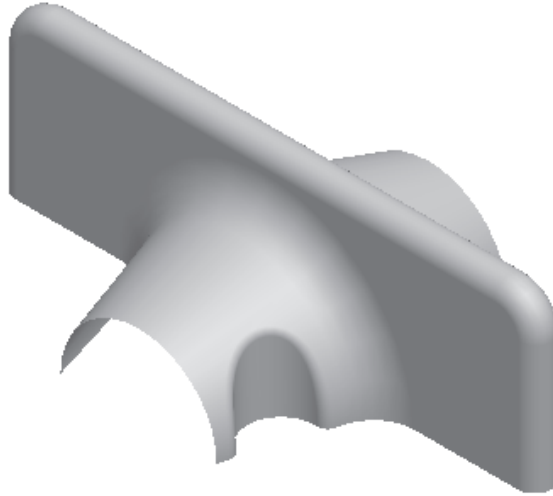


*Figure 15-84 Surface model after creating rounds*

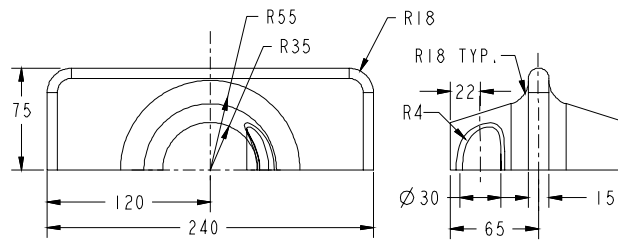
6. Choose the **Save** button from the **File** toolbar and save the model.

## Tutorial 2

In this tutorial, you will create the surface model shown in Figure 15-85. The front and the right-side views of the surface model are shown in Figure 15-86. **(Expected time: 45 min)**



*Figure 15-85 Isometric view of the surface model*



*Figure 15-86 Front view and right-side view of the surface model*


The following steps are required to complete this tutorial:

- First, examine the model and determine the number of features in it, refer to Figure 15-86.
- Create the base feature, which is an extruded surface with open ends, refer to Figures 15-87 and 15-88.
- Create the second feature, which is a blend feature created at an offset distance of 65 from the **RIGHT** datum plane, refer to Figures 15-89 and 15-90.

- d. The third feature is a mirror copy of the second feature, refer to Figure 15-91.
- e. The fourth feature is the cylindrical surface, refer to Figures 15-92 and 15-93.
- f. Create the two fill surfaces that will cap the ends of the base surface, refer to Figures 15-94 through 15-96.
- g. Merge all surfaces by selecting them individually, refer to Figures 15-97 through 15-99.
- h. Create the round features, refer to Figures 15-100 through 15-102.

### Creating the Base Feature

The base feature is a surface between the two blend surfaces. The base feature is created on the **RIGHT** datum plane.

1. Choose the **Extrude** button from the **Base Features** toolbar.
2. Select the **Extrude as surface** button from the **Extrude** dashboard. Select the **RIGHT** datum plane as the sketch plane. 
3. Select the **TOP** datum plane from the drawing area and then select the **Top** option from the **Orientation** drop-down list.
4. Choose the **Sketch** button to enter the **Sketcher** environment.
5. Once you enter the **Sketcher** environment, create the sketch of the base feature and apply dimensions, as shown in Figure 15-87.
6. After the sketch is complete, choose the **Done** button to exit the **Sketcher** environment.

The **Extrude** dashboard reappears above the drawing area. The **Extrude from sketch plane by specified depth value** button is chosen by default.

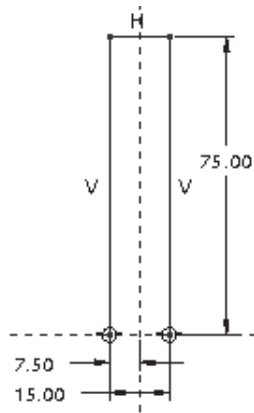
7. Enter a depth of **240** in the dimension box present in the **Extrude** dashboard. Choose the **Build feature** button from the **Extrude** dashboard.

The base feature is completed and the default trimetric view is shown in Figure 15-88.

### Creating the Blend Feature

The second feature is the blend surface and it will be created on the datum plane that is at an offset distance of 65 from the **FRONT** datum plane.

1. Choose **Insert > Blend > Surface** from the menu bar.
2. Choose **Parallel > Regular Sec > Sketch Sec > Done** from the **BLEND OPTS** menu.
3. Choose **Straight > Open Ends > Done** from the **ATTRIBUTES** menu; you are prompted to select the sketch plane.
4. Choose the **Make Datum** option to display the **DATUM PLANE** menu. Select the **Offset** option and create a datum plane at a distance of 65 from the **FRONT** datum plane.

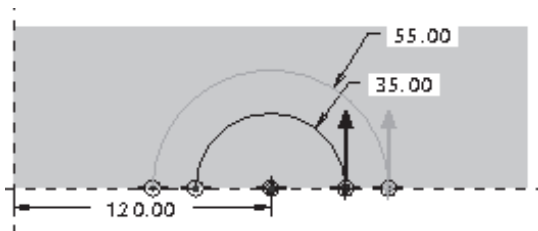


*Figure 15-87 Sketch of the base surface*



*Figure 15-88 Base surface with open ends*

5. Set the orientation of the sketching plane by selecting the **TOP** datum plane to be at top while sketching.
6. After you enter the **Sketcher** environment.
7. Sketch the first arc of diameter 35, dimension it, and then draw the second arc of diameter 55, as shown in Figure 15-89.
8. Exit the **Sketcher** environment; the **DEPTH** menu is displayed.
9. Choose **Thru Until > Done** from the **DEPTH** menu.
10. Select the **FRONT** datum plane. Choose **OK** from the **SURFACE** dialog box. The blend surface is extruded up to the selected datum plane, as shown in Figure 15-90.



*Figure 15-89 Sketch of the blend surface*

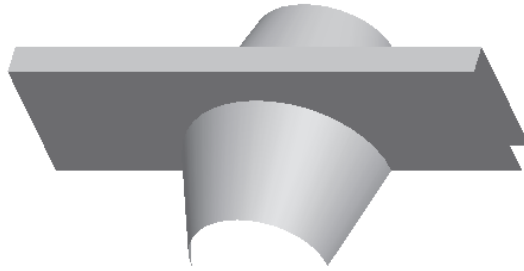


*Figure 15-90 Blend surface*

### Mirroring the Blend Surface

The blend surface that you created earlier should be mirrored about the **FRONT** datum plane.

1. Select the blend surface and then choose the **Mirror** button from the **Edit Features** toolbar; the **Mirror** dashboard is displayed.
2. Select the **FRONT** datum plane and exit the dashboard. The blend surface is mirrored about the selected datum plane, as shown in Figure 15-91.



*Figure 15-91 Model after creating the mirror copy of the blend surface*

### Creating the Cylindrical Surface

The cylindrical surface will be created on the **TOP** datum plane.

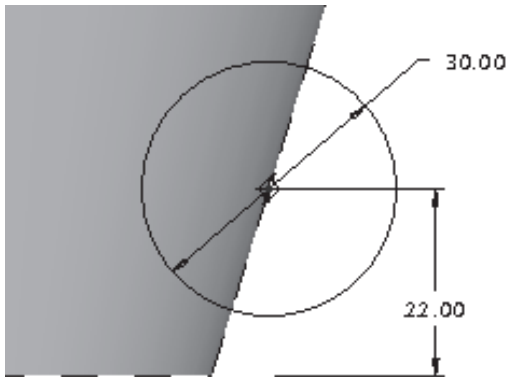
1. Choose the **Extrude** button from the **Base Features** toolbar.
2. From the **Extrude** dashboard, choose the **Extrude as surface** button.
3. Select the **TOP** datum plane as the sketching plane.
4. After entering the **Sketcher** environment, draw the circle and dimension it, as shown in Figure 15-92.
5. Exit the **Sketcher** environment and extrude the sketch to some appropriate depth, refer to Figure 15-93.

The model, after creating the surface extrusion, is shown in Figure 15-93.

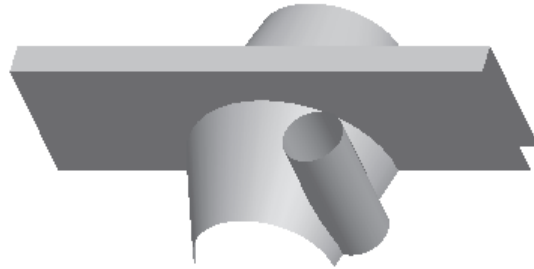
### Creating the Fill Surface

The fill surface will be created to cap the ends of the base feature.

1. Choose **Edit > Fill** from the menu bar; the **Fill** dashboard is displayed.
2. Choose the **Define** button from the **References** slide-down panel; the **Sketch** dialog box is displayed and you are prompted to select the sketching plane.
3. Select the **RIGHT** datum plane as the sketching plane. Choose the **Flip** button.

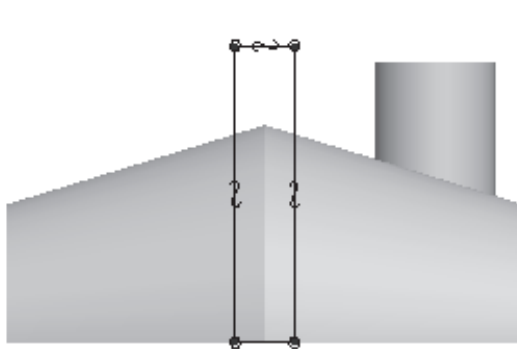


*Figure 15-92 Sketch of the cylindrical surface*

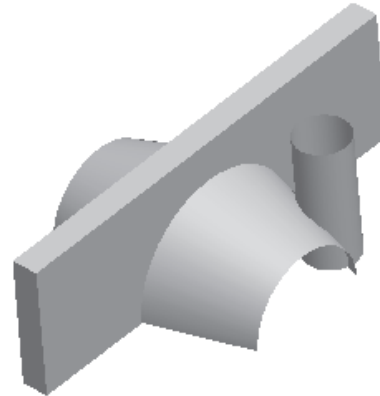


*Figure 15-93 Cylindrical surface*

4. Select the **Top** option from the **Orientation** drop-down list and select the **TOP** datum plane. Choose the **Sketch** button to enter the **Sketcher** environment.
5. Choose the **Use** button and select the edges of the base feature. The sketch is completed, as shown in Figure 15-94.
6. Exit the **Sketcher** environment and then exit the **Fill** dashboard by choosing the **Build Feature** button. The fill surface is created, as shown in Figure 15-95.



*Figure 15-94 Sketch of the fill surface*



*Figure 15-95 Model after creating the fill surface*

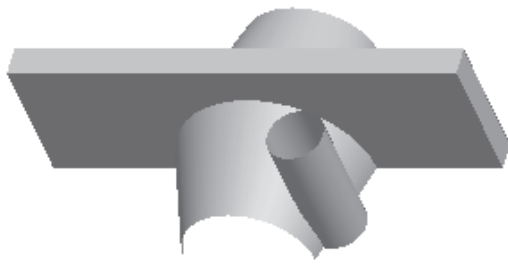
7. Mirror the fill surface about the datum plane on-the-fly. This datum plane is at an offset distance of 120 from the **RIGHT** datum plane.

After creating the mirror copy of the fill surface, the other end of the base feature is also capped, as shown in Figure 15-96.

### Merging the Blend Surface with the Cylindrical Surface

The blend surface that was the second feature and the cylindrical surface will be merged to get the required circular slot.

1. Select the cylindrical surface and then select the blend surface.
2. Choose the **Merge** button from the **Edit Features** toolbar; the **Merge** dashboard is displayed and the surface that will be retained after merging is highlighted.
3. Choose the **Change side of second quilt to keep** button to change the direction of the yellow arrow.
4. Exit the **Merge** dashboard by choosing the **Build Feature** button. The model, after merging the two surfaces, is as shown in Figure 15-97.



*Figure 15-96 Model after creating the mirror copy of the fill surface*



*Figure 15-97 Model after merging the surfaces*

### Merging the Blend Surface and the Extruded Surface

The blend surface and the extruded surface will be merged to build a single surface.

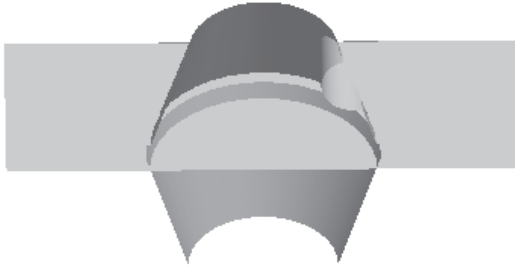
1. Select the base feature and then select the **Merge 1** feature from the **Model Tree**.
2. Choose the **Merge** button from the **Edit Features** toolbar. The **Merge** dashboard is displayed and the surface that will be retained after merging is highlighted.
3. Choose the **Change side of first quilt to keep** button to change the direction of the yellow arrow.
4. Exit the **Merge** dashboard by choosing the **Build Feature** button. The model, after merging the two surfaces, is shown in Figure 15-98.
5. Similarly, merge the mirrored feature and the base feature. The surface model, after merging the mirrored surfaces, is shown in Figure 15-99.

### Merging the Fill Surfaces with the Base Surface

The fill surfaces that you have created should be merged with the base surface in order to create a single quilt or a single surface. When the surfaces are merged, you will use the edge formed by the merge feature to create rounds.

1. Select the fill surface and then select the base surface.





**Figure 15-98** Model after merging the blend surface with the base surface



**Figure 15-99** Model after merging the mirror copy of the blend surface with the base surface

**Note**

*It is easier to select surfaces from the **Model Tree**.*

*To merge two surfaces, it is necessary that they intersect each other.*

2. Choose the **Merge** button from the **Edit Features** toolbar and then choose the **Build feature** button to merge both the surfaces.
3. Similarly, merge the mirror copy of the first fill surface with the base surface.

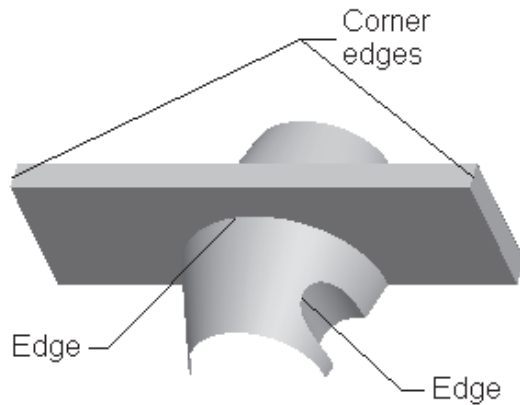
**Creating Rounds**

The rounds that you need to create are on the cylindrical slot, edges where the two blend surfaces are merged, and on the edges of the base surface.

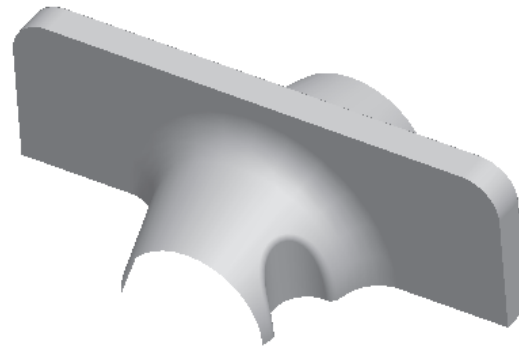
1. Choose the **Round** button from the **Engineering Features** toolbar. Select the edge of the cylindrical slot, see Figure 15-100. The preview of the round is highlighted on the selected edge.
2. Enter the value **4** in the dimension edit box for the radius of the round.
3. Click on **New set** in the **Sets** slide-down panel to add a second set named **Set2**.
4. Select the four edges that have a radii of 18. The two edges are the edges that are formed by merging the two blend surfaces with the base surface and the two edges are the top corners of the base surface, see Figure 15-100.
5. Choose the **Build feature** button from the **Round** dashboard to create the rounds. The surface model, after creating the rounds, is shown in Figure 15-101.

**Creating a Full Round**

A full round will be created by selecting the two surfaces. These surfaces are the front and back faces of the base surface.



**Figure 15-100** Edges selected to create rounds



**Figure 15-101** Resulting model after creating rounds on the selected edges

1. Choose the **Round** button from the **Engineering Features** toolbar.
2. Select the two faces, front and back, of the base surface.
3. Invoke the slide-down panel by choosing the **Sets** tab. Select the two surfaces; the selected surfaces are displayed in the **References** collector available in the slide-down panel. The **Full Round** button is chosen by default. Now, you need to select the surface to be removed.
4. Select the top face of the base surface. The preview of the round is highlighted on the selected surfaces. Exit the **Round** dashboard by choosing the **Build Feature** button. The round is created, as shown in Figure 15-102.
5. Choose the **Save** button from the **File** toolbar and save the model.



**Figure 15-102** Completed surface model

### Self-Evaluation Test

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

1. You can create a surface with capped ends by drawing an open sketch. (T/F)
2. Surface models have no thickness. (T/F)
3. Style features have the parent-child relationship among themselves as well as with the Pro/ENGINEER features. (T/F)
4. In the **Style** environment, when you press the SHIFT key and select a point on a surface using the **Free** option, the point is selected on that surface. (T/F)
5. To create a helical sweep surface, the procedure to follow is the same as in the case of creating a solid helical sweep feature. (T/F)
6. Any feature created in the **Style** environment is displayed in the **Model Tree** as a \_\_\_\_\_ feature.
7. To enter the **Style** environment, choose the \_\_\_\_\_ button in the **Base Features** toolbar.
8. The \_\_\_\_\_ button is used to merge two surfaces and form an edge.
9. In the **Style** environment, the \_\_\_\_\_ button is used to draw curves.
10. A Quilt is a \_\_\_\_\_ feature.

### Review Questions

Answer the following questions:

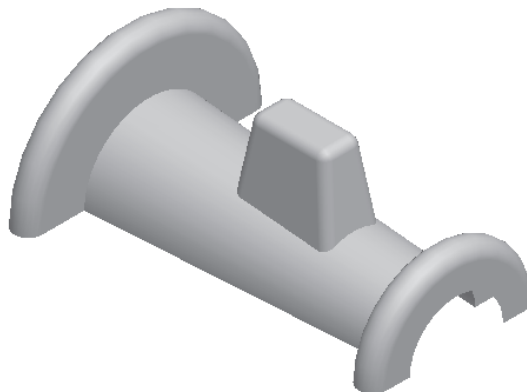
1. Which of the following feature creation tools contains the options like parallel, rotational, and general?
  - (a) **Sweep**
  - (b) **Blend**
  - (c) **Extrude**
  - (d) **None**
2. Which of the following editing tools is used to create a flat surface by drawing a sketch?
  - (a) **Trim**
  - (b) **Copy**
  - (c) **Fill**
  - (d) **None of the above**

3. What is the minimum number of sections required for a blend feature?
- (a) one (b) two  
(c) three (d) None of the above
4. Which of the following editing tools forms an edge between two intersecting surfaces?
- (a) **Merge** (b) **Intersect**  
(c) **Trim** (d) None
5. In which one of the following types of blend, sections are translated and rotated about the x, y, and z-axes?
- (a) **Parallel** (b) **Rotational**  
(c) **General** (d) None
6. The **Intersect** option is used to create an intersect curve. (T/F)
7. In the **Style** environment, the **Curve Edit** button is used to project curves on surfaces. (T/F)
8. Surface models are the 3D models with no thickness. (T/F)
9. In the **Style** environment, the **Surface** button is used to select at least three or four curves and create a surface. (T/F)
10. To undo the last operation, choose the **Undo** button from the **Style** toolbar. (T/F)

## Exercises

### Exercise 1

In this exercise, you will create the surface model shown in Figure 15-103. The orthographic views of the surface model are shown in Figure 15-104. **(Expected time: 40 min)**



*Figure 15-103 Isometric view of the surface model*

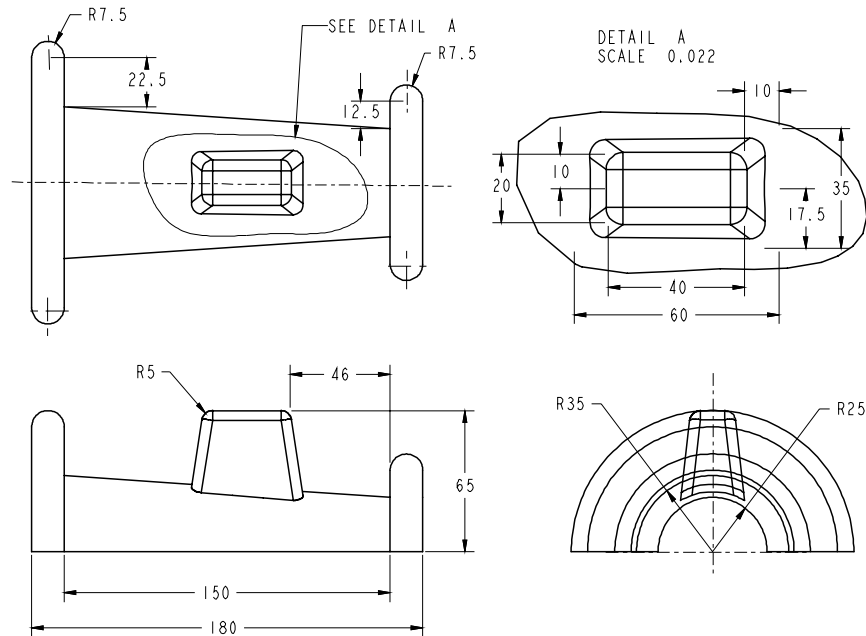


Figure 15-104 Top, front, right-side, and detailed views of the surface model



**Note**

Create the base feature using the **Blend** option and the ends as revolve features.

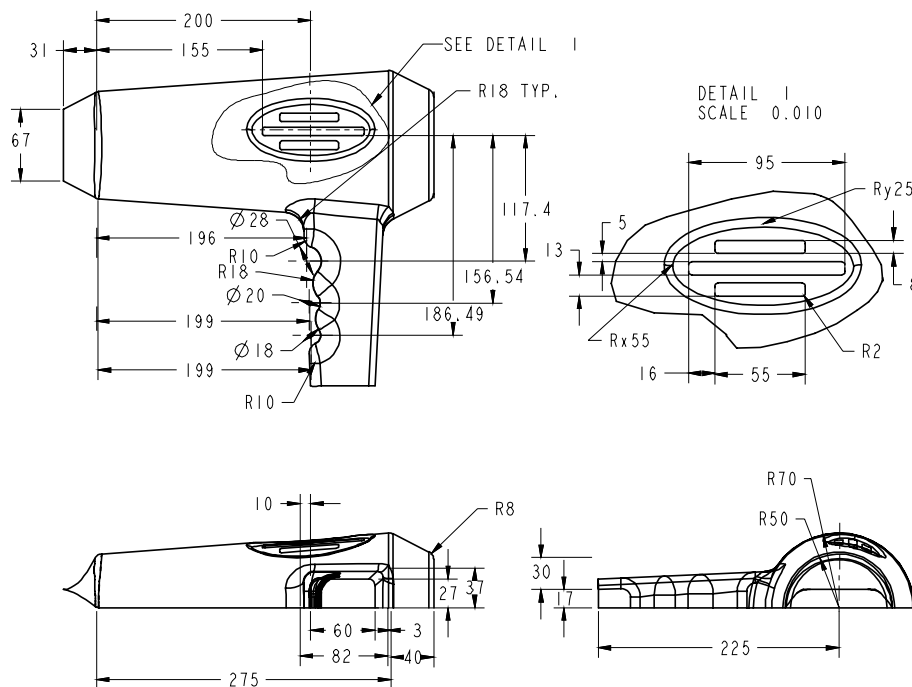
## Exercise 2

In this exercise, you will create the surface model shown in Figure 15-105. The orthographic views and the detailed view of the surface model are shown in Figure 15-106.

(Expected time: 55 min)



**Figure 15-105** Surface model



**Figure 15-106** Top, front, right-side, and detailed views of the surface model

## Answers to Self-Evaluation Test

**1. F, 2. T, 3. T, 4. T, 5. T, 6. Style, 7. Style, 8. Merge, 9. Curve, 10. surface.**