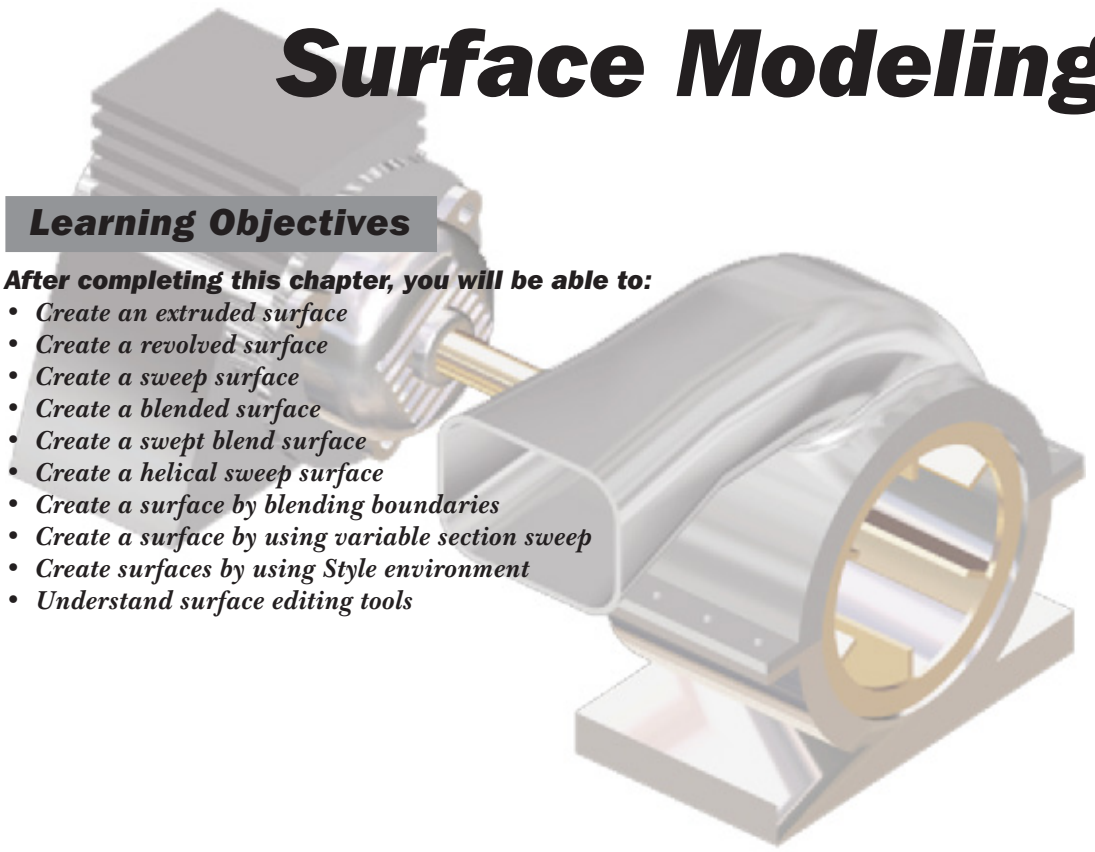
Chapter 15

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 by using variable section sweep*
- *Create surfaces by 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. Thick models are the models having mass properties. Surface models do not have thickness whereas thick or solid models have a user-defined thickness. In Creo Parametric, surface modeling techniques and feature creation tools are same as those used in solid modeling. A solid model created of any shape can also be created using the surface modeling techniques. Sometimes, it is difficult to create complex shapes using solid modeling. Such models can be easily created using surface modeling and then the surface model can be converted into a 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 Creo Parametric

In Creo Parametric, you can toggle between a solid feature and a surface feature while creating a protrusion. The two buttons that are used to toggle between the solid feature and a surface feature are available on dashboards.

Creating an Extruded Surface

Ribbon: Model > Shapes > Extrude



To create an extruded surface, choose the **Extrude** tool from the **Shapes** group; the **Extrude** dashboard will be displayed, as shown in Figure 15-1.

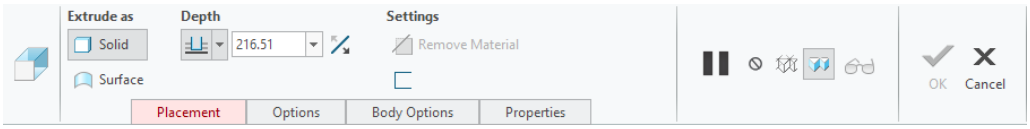


Figure 15-1 *Partial view of the* ***Extrude*** *dashboard*



In this dashboard, the **Solid** button is chosen by default. Choose the **Surface** button to extrude the sketch and create a surface model. All attributes of a solid model that were discussed in Chapter 4 are the same for a surface model. Some 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. However, 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.

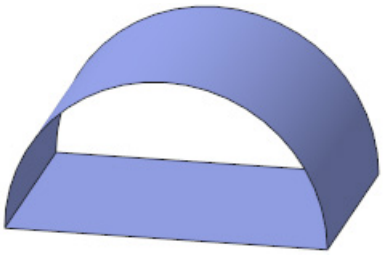


Figure 15-2 Surface with open ends

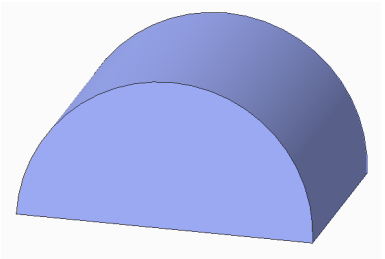


Figure 15-3 Surface with capped ends

Creating a Revolved Surface

Ribbon: Model > Shapes > Revolve

To create a revolved surface, choose the **Revolve** tool from the **Shapes** group; 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.

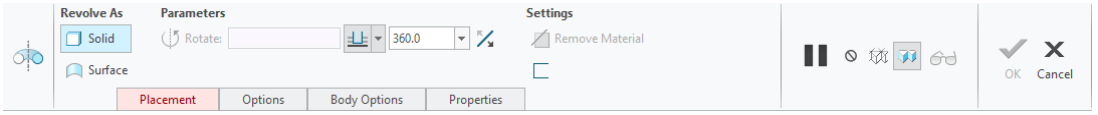


*Figure 15-4 Partial view of the **Revolve** dashboard*

In the **Revolve** dashboard, the **Solid** button is chosen by default. Choose the **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 degrees. Figure 15-5 shows the revolved surface with open ends and Figure 15-6 shows the revolved surface with capped ends.

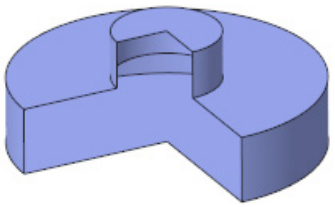


Figure 15-5 Revolved surface with open ends

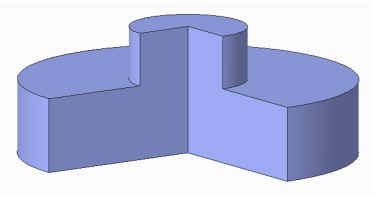


Figure 15-6 Revolved surface with capped ends

Creating a Sweep Surface

Ribbon: Model > Shapes > Sweep drop-down> Sweep

To create a sweep surface feature, choose the **Sweep** tool from the **Sweep** drop-down in the **Shapes** group of the **Model** tab in the **Ribbon**; the **Sweep** dashboard 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 9. The additional option of capping the ends that was available in the **Extrude** and **Revolve** tools is also available in the **Sweep** tool.

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

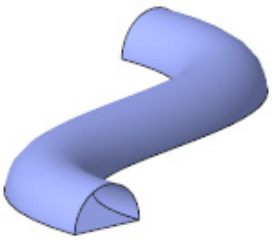


Figure 15-7 *Sweep surface with open ends created using a closed sketch*

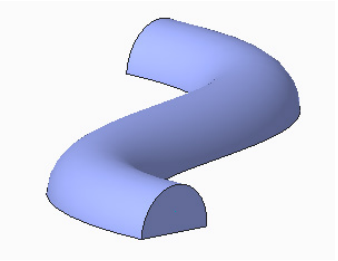


Figure 15-8 *Sweep surface with capped ends created using a closed sketch*

Creating a Blend Surface

Ribbon: Model > Shapes > Blend

To create a surface blend, choose the **Blend** tool from the expanded **Shapes** group in the **Ribbon**; the **Blend** dashboard will be displayed. In this dashboard, the **Solid** button is chosen by default. Choose the **Surface** button to blend the sketch and create a surface model. The method to create a blend surface is the same as that to create a solid blend. To create a solid blend feature, refer to Chapter 9. Blended surfaces can be with open ends or capped ends. Figure 15-9 shows the blend surface with open ends and Figure 15-10 shows the blend surface with capped ends.

Tip

If you want to create a surface blend with capped ends, you need to create a closed sketch. Creo Parametric does not accept an open sketch for a capped end blend surface.

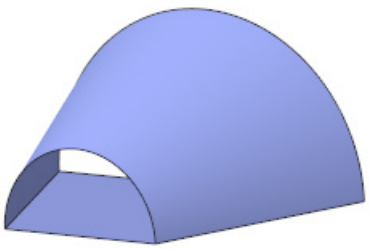


Figure 15-9 *Blend surface with open ends*

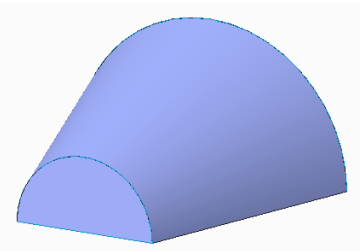


Figure 15-10 *Blend surface with capped ends*

Creating a Rotational Blend Surface

Ribbon: Model > Shapes > Rotational Blend

To create a rotational blend surface, choose the **Rotational Blend** tool from the expanded **Shapes** group in the **Ribbon**; the **Rotational Blend** dashboard will be displayed. In this dashboard, the **Solid** button is chosen by default. Choose the **Surface** button to create a surface model. The method to create a rotational blend surface is the same as that to create a solid rotational blend. To create a solid rotational blend feature, refer to Chapter 9. Rotational blend surface can be created with open ends or capped ends. Figure 15-11 shows the rotational blend surface with open ends and Figure 15-12 shows the rotational blend surface with capped ends.

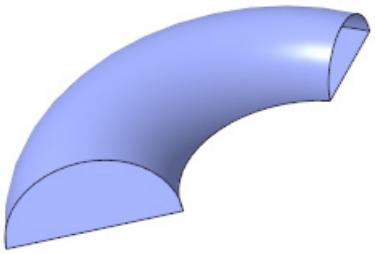


Figure 15-11 *Rotational blend surface with open ends created using a closed sketch*

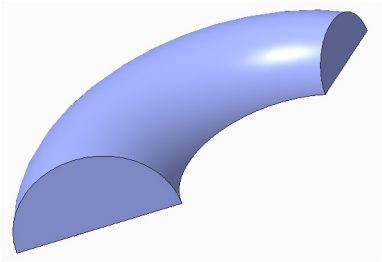


Figure 15-12 *Rotational blend surface with capped ends*

Creating a Swept Blend Surface

Ribbon: Model > Shapes > Swept Blend

To create a swept blend surface, choose the **Swept Blend** tool from the **Shapes** group in the **Ribbon**; the **Swept Blend** dashboard will be displayed. Choose the **Create a Surface** button to create a sweep surface. 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 15-13 shows the swept blend with open ends and Figure 15-14 shows the swept blend with capped ends.

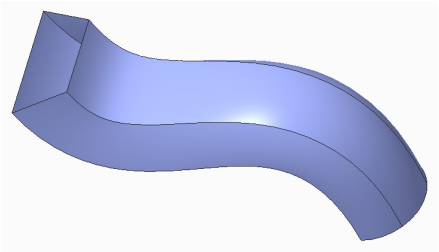


Figure 15-13 *Swept blend surface with open ends*

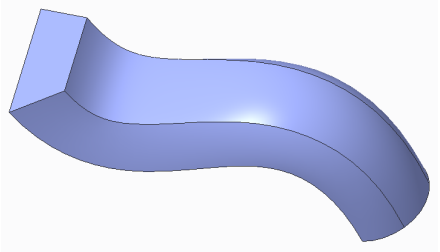


Figure 15-14 *Swept blend surface with capped ends*

Creating a Helical Sweep Surface

Ribbon: Model > Shapes > Sweep > Helical Sweep

To create a helical sweep surface, choose the **Helical Sweep** tool from the **Sweep** drop-down of the **Shapes** group in the **Ribbon**; the **Helical Sweep** dashboard will be displayed. Choose the **Surface** button to create a sweep surface. 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 15-15 shows the helical sweep surface with open ends and Figure 15-16 shows the helical sweep surface with capped ends.

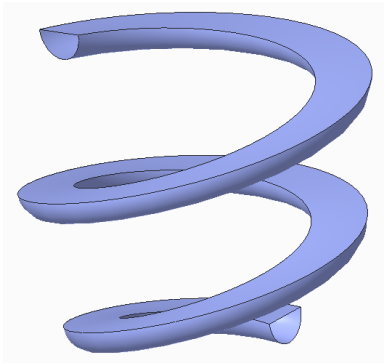


Figure 15-15 Helical sweep surface with open ends

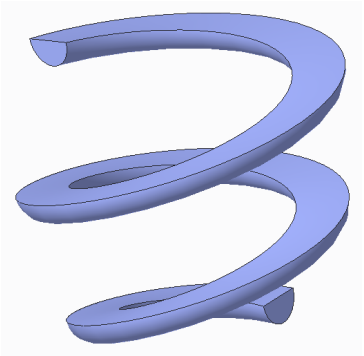


Figure 15-16 Helical sweep feature with capped ends

Creating a Surface by Blending Boundaries

Ribbon: Model > Surfaces > Boundary Blend

To create a surface by blending boundaries, datum curves, or points, choose the **Boundary Blend** tool from the **Surfaces** group in the **Ribbon**; the **Boundary Blend** dashboard will be displayed, as shown in Figure 15-17 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.

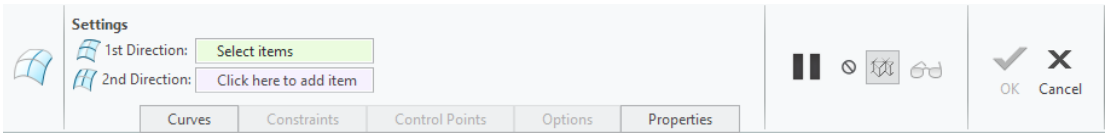


*Figure 15-17 Partial view of the **Boundary Blend** dashboard*

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 green and the value **0.00** will be displayed on both ends of the curve. When you modify this value, the curve will change accordingly at the corresponding end. Press CTRL+left mouse button to select the second curve; the second curve will also get highlighted, refer to Figure 15-18. These curves will be numbered as per the sequence of selection. The surface created by using these two blending boundaries is shown in Figure 15-19.

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

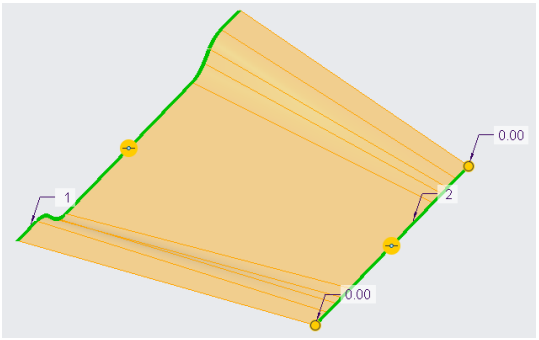


Figure 15-18 *Curves selected as the blending boundary*

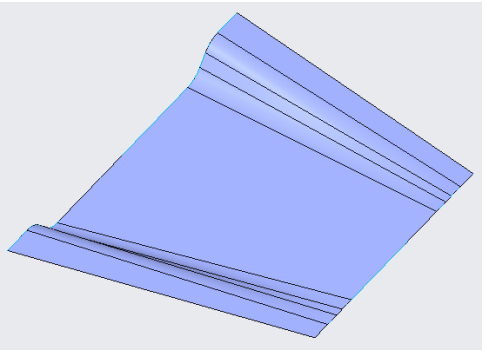


Figure 15-19 *Surface created by blending boundaries*

Now, invoke the **Curves** slide-down panel, refer to Figure 15-20 and select **2 Chain** from the **First direction** collector. The **Move up** and **Move down** buttons available in the slide-down panel are used to change the order of selection of the curves. The **Close blend** check box is used to close the surfaces.

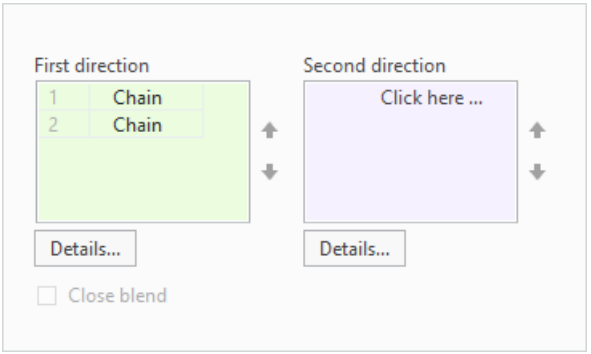


Figure 15-20 *The **Curves** slide-down panel*



Tip

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

Figure 15-21 shows the surface created by using four curves and Figure 15-22 shows the surface created by selecting the **Close blend** check box.

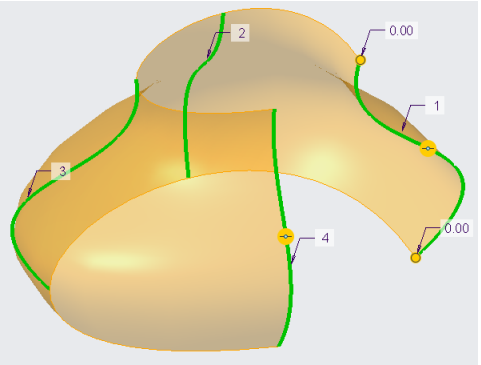


Figure 15-21 *Surface created after selecting the curves*

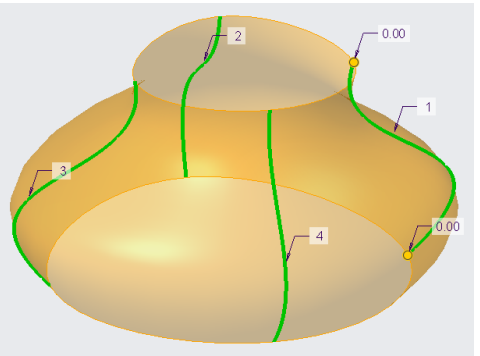


Figure 15-22 *Surface created by selecting the* ***Close blend*** *check box*

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 15-23 shows the first and second direction curves and Figure 15-24 shows the surface created after selecting the curves shown in Figure 15-23.

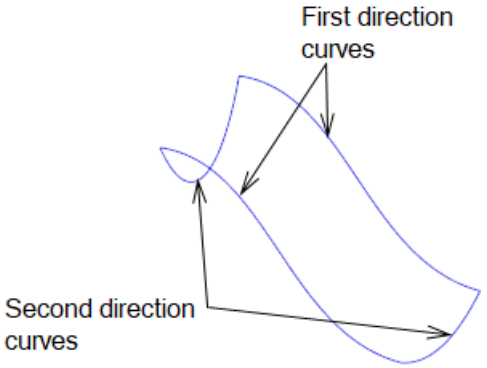


Figure 15-23 *Datum curves*

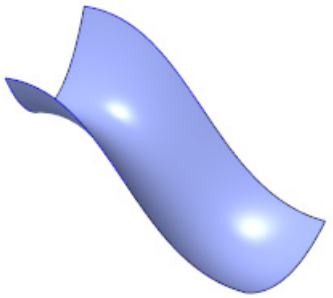


Figure 15-24 *Surface created by using the first and second direction curves*

Creating a Variable Section Sweep Surface by Using the Sweep Tool

Ribbon: Model > Shapes > Sweep

To create the variable section sweep surface, choose the **Sweep** tool from the **Shapes** group in the **Ribbon**; the **Sweep** dashboard will be displayed. Choose the **Surface** button from the dashboard. To learn more about the variable section sweep, refer to Chapter 9. The procedure of creating a variable section sweep surface is similar to the procedure for creating variable section sweep as was discussed in Chapter 9.

Figure 15-25 shows the section and trajectories 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.

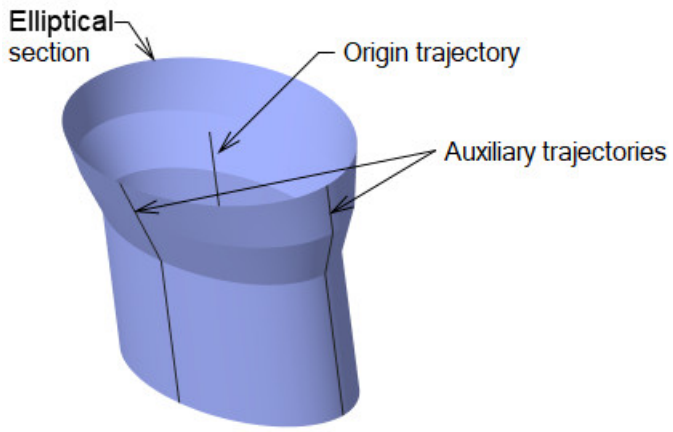


Figure 15-25 *Variable section sweep surface with open ends*

Creating a Curve Using the Project Tool

Ribbon: Model > Editing > Project

The **Project** tool is used to project a selected or sketched entity on one or more planar or non-planar surfaces or datum planes. The projected datum curve forms a true projection of the selected or sketched entity on the specified surfaces. The dimensions of the original entity may distort while projecting.

Choose the **Project** tool from the **Editing** group; the **Projected Curve** dashboard is displayed, as shown in Figure 15-26. The tools and options in this dashboard are discussed next.

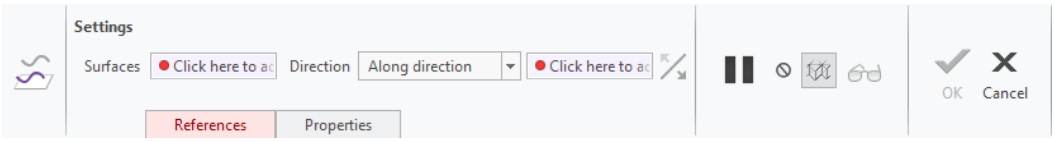


Figure 15-26 *The **Projected Curve** dashboard*

Surfaces Collector

The **Surfaces** collector is used to select the surface on which you need to project the sketched or selected datum curve.

Direction Drop-down List

The **Direction** drop-down list is used to specify the method of projection of the datum curve on the receiving surface or plane. The two options available in this drop-down list are: **Along direction** and **Normal to surface**, and are discussed next.

Along direction

If you use this option, then the datum curve is projected in the direction indicated by the pink arrow. To specify the direction of projection, select the datum plane, edge, or surface. Figure 15-27 shows the top view of the projected datum curves that are overlapping. The datum curve that is selected to project on the receiving surface and the datum curve after

projection are of same geometry. This is because, using the **Along direction** option, the true geometry is obtained after projection. Figure 15-28 shows the sketched datum curve after it has been projected on a receiving surface. The curve is sketched on a datum plane that is parallel to the bottom face of the model.

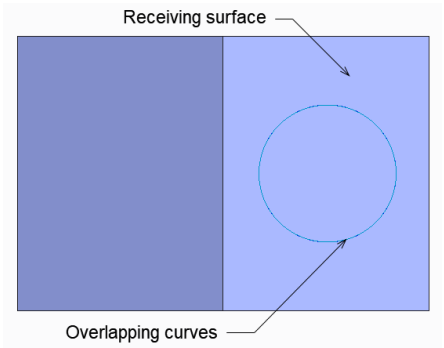


Figure 15-27 *Top view of the projected datum curve*

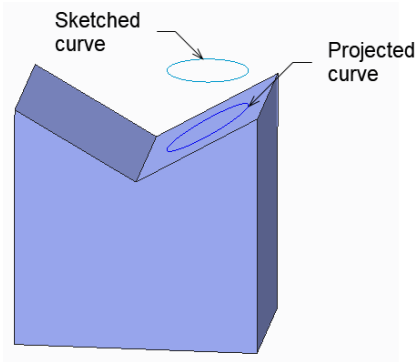


Figure 15-28 *Projecting a datum curve along the specified direction*

Normal to surface

This option of projecting a datum curve projects the datum curve normal to the receiving surface. Figure 15-29 shows a datum curve selected to be projected on a receiving surface and the datum curve after projection. Figure 15-30 shows the sketched datum curve after the projection on the receiving surface.

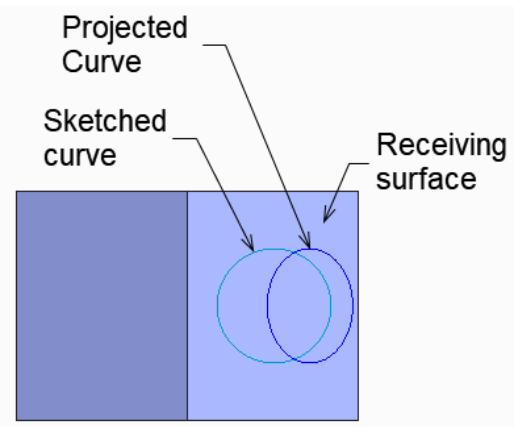


Figure 15-29 *Top view of the projected datum curve*

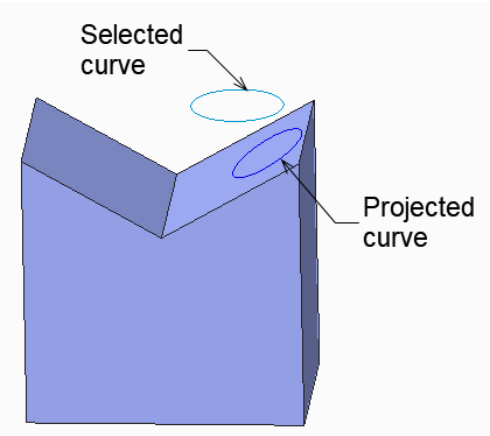


Figure 15-30 *Projecting a datum curve*

References Tab

This tab is used to define the references for projection. If you choose the **References** tab, a slide-down panel is displayed, as shown in Figure 15-31. This slide-down panel is used to select the datum curve to be projected or sketch a datum curve to project. The drop-down list in the slide-down panel lists three options: **Project chains**, **Project a sketch**, and **Project a cosmetic sketch**. Using these options, you can project multiple chains or a sketch on a surface or face. These options are discussed next.

Project chains

The **Project chains** option is used when the datum curve to be projected exists. Click in the **Chains** collector to make it active and then select the datum curve that you need to project. After selecting the datum curve, click in the **Surfaces** collector and select the receiving surface (plane or surface to project on to). Then you need to specify the direction of projection. Remember that if you are using the **Normal to surface** option, then you do not need to specify the direction of projection. The **Direction Reference** collector is displayed while selecting the **Along direction** option from the dashboard, whereas it does not appear while selecting the **Normal to surface** option.

Project a sketch

When you select the **Project a sketch** option from the drop-down list in the **References** slide-down panel, a slide-down panel appears, as shown in Figure 15-32.

Click on the **Sketch** collector and then select a curve or sketch to be projected or create a sketch that can be projected on the sketch. To create a new sketch, choose the **Define** button to invoke the **Sketch** dialog box and to select the sketch plane for the curve to be projected. After sketching, exit the Sketcher environment and select the surface or plane on which the curve will be projected. If you are using the **Along direction** option to project the curve then you need to click in the **Direction Reference** collector to specify the direction of projection.

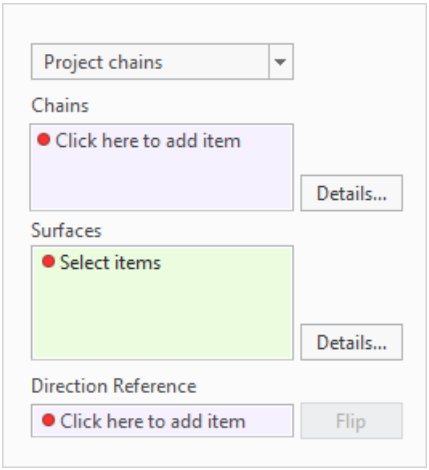


*Figure 15-31 The **References** slide-down panel with the **Project chains** option selected*

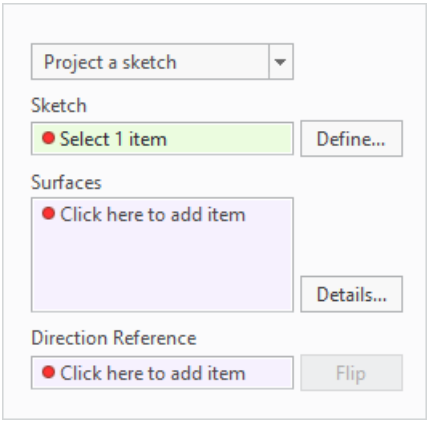


*Figure 15-32 The **References** slide-down panel with the **Project a sketch** option selected*

Project a cosmetic sketch

When you select the **Project a cosmetic sketch** option from the drop-down list of the **References** slide-down panel, a slide-down panel will be displayed which is similar to the **Project a sketch** slide-down panel. Click on the **Sketch** collector and then select a cosmetic sketch to be projected or create a sketch that can be projected on the sketch. To create a new cosmetic sketch, choose the **Define** button to invoke the **Sketch** dialog box and then select the sketch plane for sketching the curve. After sketching, exit the Sketcher environment and select the surface or plane on which the curve needs to be projected.

Creating a Curve Using the Wrap Option

Ribbon: Model > Editing > Wrap

This option is used to create a datum curve by wrapping a sketched entity around a solid or a quilt. Choose the **Wrap** tool from the expanded **Editing** group, the **Wrap** dashboard will be displayed, as shown in Figure 15-33. The tools and options in this dashboard are discussed next.

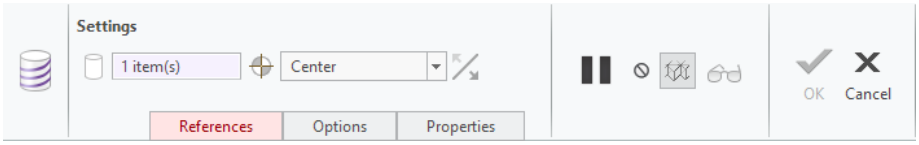


Figure 15-33 *Partial view of the* ***Wrap*** *dashboard*

References Tab

When you choose the **References** tab, a slide-down panel will be displayed, as shown in Figure 15-34. The **Define** button available in this slide-down panel is used to invoke the **Sketch** dialog box.

The **Destination** collector in the slide-down panel is used to select the object on which you need to wrap the curve. Generally, if there is a single feature in the drawing area, then you do not need to select the object to wrap on. Creo Parametric automatically wraps the selected curve or the sketched curve on the object. However, if you want to select a different object to wrap on then you can click in this collector to make it active and then select the object.

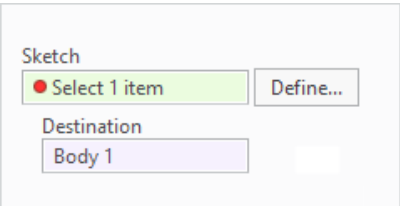


Figure 15-34 *The* ***References*** *slide-down panel*

Options Tab

On choosing the **Options** tab, a slide-down panel will be displayed, as shown in Figure 15-35. There are two options available in this panel. The **Ignore intersection surface** check box when selected, ignores intersection surface, if any and wraps the selected curve on the destination object.

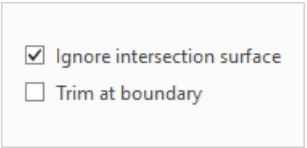


Figure 15-35 *The* ***Options*** *slide-down panel*

The **Trim at boundary** check box when selected, trims the extra portion of the curve that is beyond the boundary of the destination object.

Follow the steps given below to sketch and wrap a curve on the rectangular block, as shown in Figure 15-36. It is assumed that the rectangular block of dimension 20x20x50 exists.

1. Choose **Wrap** tool from the **Editing** group to invoke the **Wrap** dashboard.

2. Choose the **References** tab to invoke the slide-down panel.

3. Choose the **Define** button to invoke the **Sketch** dialog box.

4. Choose the datum plane that is passing through the center of the rectangular block as the sketching plane and then choose a reference for orienting the sketching plane.

5. After entering the Sketcher environment, draw a line that starts from the bottom of the rectangular block. Its start point is aligned with the center of the rectangular block and with the bottom edge. The endpoint of the line is at a distance of 1000 and at a height of 50 from the bottom of the rectangular block, refer to Figure 15-37.

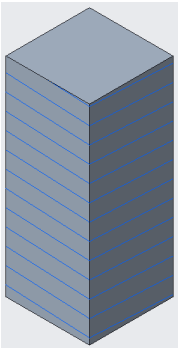


Figure 15-36 *Curve wrapped on the rectangular block*

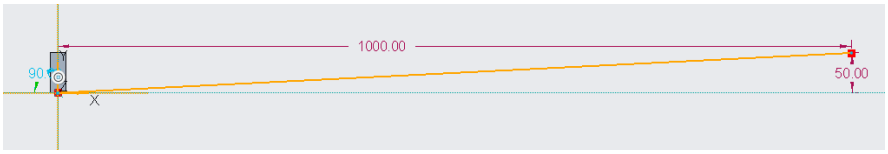


Figure 15-37 *Sketch of the curve to be wrapped*

6. Place a user-defined coordinate system at the start point of the line.

7. Exit the Sketcher environment; the sketched curve will be automatically wrapped on the circumference of the cylinder.

8. Choose the **OK** button to exit the dashboard.

CREATING SURFACES BY USING THE STYLE ENVIRONMENT OF Creo Parametric

Ribbon: Model > Surfaces > Style

Style is an environment available in Creo Parametric and 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 Creo Parametric surfaces. They can have the parent-child relationship among themselves and also with Creo Parametric features.

To enter the **Style** environment, choose the **Style** tool from the **Surfaces** group. Figure 15-38 shows the appearance of the **Style** environment.

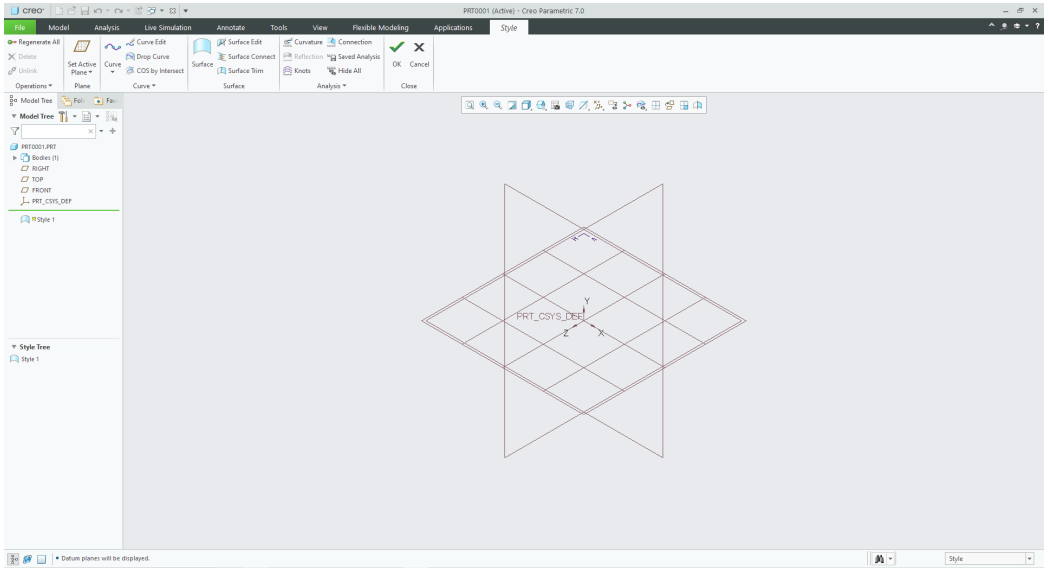


*Figure 15-38 The **Style** environment*

Style Dashboard

Figure 15-39 shows the **Style** dashboard in the **Style** environment. The options in this dashboard are discussed next.

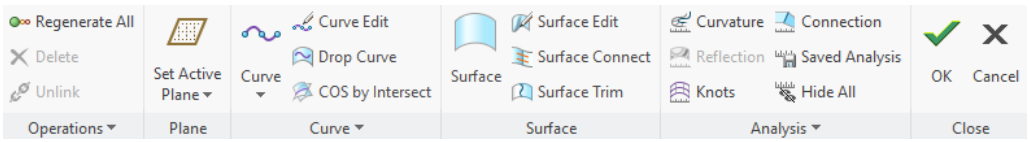


*Figure 15-39 The **Style** dashboard*

Set Active Plane

Ribbon: Style > Plane > Set Active Plane drop-down > Set Active Plane

The **Set Active Plane** tool is available in the **Set Active Plane** drop-down in the **Plane** group of the **Style** dashboard in the **Ribbon**. This tool 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. Also, horizontal and vertical directions are shown on the selected plane.

Internal Plane

Ribbon: Style > Plane > Set Active Plane drop-down > Internal Plane

The **Internal Plane** tool is available in the **Set Active Plane** drop-down in the **Plane** group of the **Style** dashboard. This tool is used to create an internal datum plane in the **Style** environment. When you choose this tool, 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 by using this tool will be displayed in the drawing area only when you are in the **Style** environment. Once you exit the **Style** environment, these datum plane become invisible. Any feature created in the **Style** environment will be displayed in the **Style Tree** as a style feature.

Curve

Ribbon: Style > Curve > Curve drop-down > Curve

The **Curve** tool is available in the **Curve** drop-down of the **Curve** group in the **Style** dashboard. This tool is used to draw curves. When you choose this tool, the **Style: Curve** dashboard will be displayed, as shown in Figure 15-40. The options in this dashboard are discussed next.

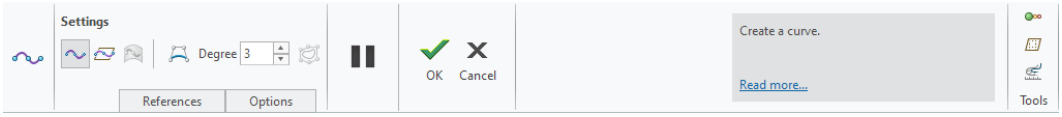


Figure 15-40 *The* ***Style: Curve*** *dashboard*

Create a Free Curve

In the **Style:Curve** dashboard, the **Create a Free Curve** button is chosen by default. As a result, you will be prompted to define the points for creating the curve. To create the curve, click on the screen. A green point will be displayed at the location where you have clicked. Now, again click to define the second point of the curve. The two points are joined by a line. 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 **Create a Free Curve** button will lie 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 **Graphics** toolbar and then selecting a point each from any of the two different windows. When you choose the **Show All Views** button, the four-window view is displayed. In Creo Parametric, this type of display is called a 4-view display mode. The 4-view display mode shows the top, default, right, 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 again.

Tip

To undo the last operation, choose the ***Undo*** *button from the* ***Quick Access*** *toolbar or press CTRL+Z keys.*

Create a Planar Curve

This tool, 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 after invoking the **Style: Curve** dashboard by choosing the **Set Active Plane** button from the **Style: Curve** dashboard. The **Set Active Plane** button is available on the right of the dashboard.



Tip

Using the Planar curve 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.

Create a Curve On Surface



This tool 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 here after 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 orange curve is converted to a black curve indicating that the curve is completed. The curve drawn on the surface is the child of the surface.

Edit this curve using control points



If this toggle button is chosen while drawing the curve, then on editing the curve the control points will be displayed.

Proportional Update

If you create a curve, when this check box is selected from **Options** tab, then the curve created can be edited proportionally.



Tip

Using the Free Curve 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.

Circle

Ribbon: Style > Curve > Curve drop-down> Circle

The **Circle** tool is used to create circles. To create circles, choose **Circle** from the **Curve** drop-down in the **Curve** group of the **Style** dashboard; the **Style: Circle** dashboard will be displayed. Click anywhere in the drawing area; a circle with default radius will be placed at that location. The circle thus created can be free or planar circle. By default, the **Create a Free Curve** button is chosen in the dashboard, and therefore the curve created will be a free curve. You can adjust the radius of the circle by dragging the handle or enter the radius value in the edit box. The circles can also be edited similar to curves. You can also select the **Proportional Update** check box from **Options** tab to edit the circle proportionally.

Similarly, you can create arcs by choosing the **Arc** button from the **Curve** drop-down.

Curve Edit

Ribbon: Style > Curve > Curve Edit



The **Curve Edit** button is used to edit the curves that are created as style features. If you choose this button, the **Style: Curve Edit** dashboard will be displayed, as shown in

Figure 15-41 and you will be prompted to select a curve. Select the curve; the curve will be displayed in the collector of the **Curve Edit** dashboard.

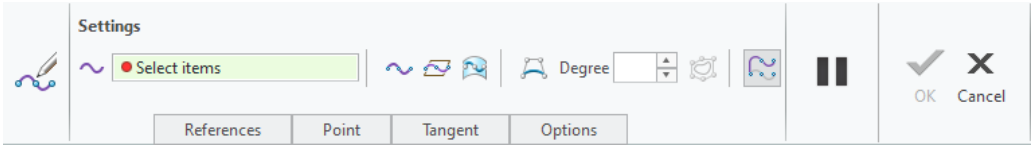


Figure 15-41 *Partial view of the* ***Style: Curve Edit*** *dashboard*

The options in the **Style: Curve Edit** dashboard are discussed next.

Curve Collector

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

Change to a Free Curve

The **Change to a Free Curve** button is used to change the selected curve into a free curve.

Change to a Planar Curve

The **Change to a Planar Curve** button is used to change the selected curve into a planar curve.

Change to a Curve on Surface

If the curve selected for editing is drawn using the **Curve On Surface** option, then the **Change to a Curve on Surface** button will be selected by default.

Proportional Update Check Box

If the curve selected for editing is drawn by selecting the **Proportional Update** check box from **Options** tab, then the curve will be edited proportionately with the points.

Edit this curve using control points

If the curve that for editing is drawn using the **Edit this curve using control points** button, then the control points will be displayed on the curve. Using these control points, you can modify the shape of the curve. You can also choose this button to display the control points on a curve drawn without choosing this button.

Tangent

Choose the **Tangent** tab to define the nature of contact between the curve and the adjacent surface. Alternatively, select one of the end control points of the curve, the tangent vector of the curve will be highlighted in green color. Now, right-click on the green vector to display the shortcut menu, as shown in Figure 15-42.

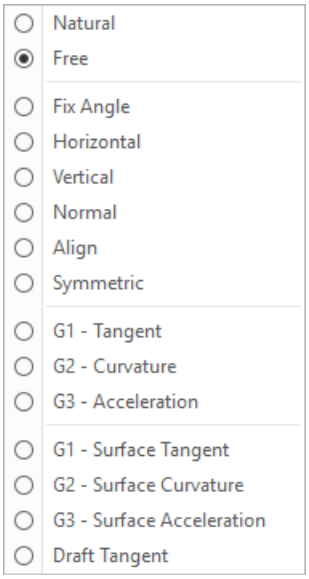


Figure 15-42 *Shortcut menu for the end control point of the curve*

By default, a curve has a natural contact with the adjacent surface. This is evident from the circle mark on the left of the **Natural** option in the shortcut menu. Figure 15-43 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 15-44 shows the curve whose contact type is changed to the **Surface Tangent** option by choosing it from the shortcut menu.

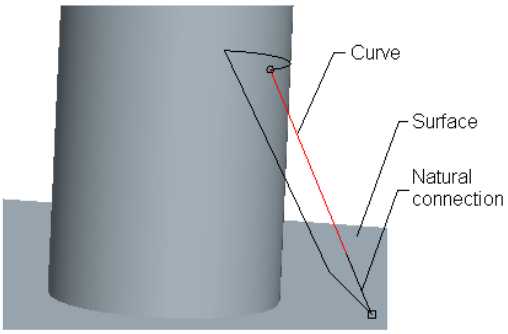


Figure 15-43 *Curve joining the two surfaces*

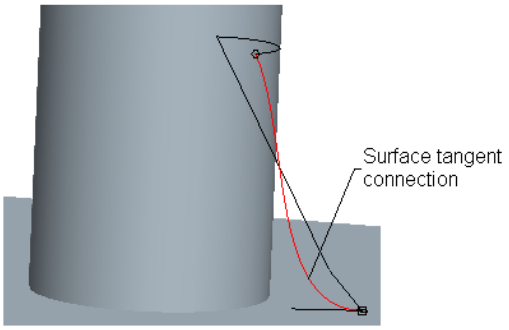


Figure 15-44 *Curve joining the base surface tangentially*

Drop Curve

Ribbon: Style > Curve > Drop Curve

Using the **Drop Curve** tool, a curve created in the **Style** environment can be projected onto the selected surface.

To project a curve on a selected surface, choose the **Drop Curve** tool from the **Curve** group of the **Style** dashboard to create COS (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; 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.

Curve on Surface by Intersect

Ribbon: Style > Curve > COS by Intersect

The **COS by Intersect** tool is used to create a curve at the intersection of two sets of surfaces. To do so, choose **COS by Intersect** from the **Curve** group in the **Style** dashboard; you will be prompted to select the first set of surfaces. Select the required surfaces. Next, click the middle mouse button to accept the selection and then select the second set of surfaces or planes and choose the **OK** button to create the COS at the intersection.

Surface

Ribbon: Style > Surface > Surface

The **Surface** tool is used to create a surface among a closed boundary of curves. Choose the **Surface** tool from the **Surface** group in the **Style** dashboard; the **Style: Surface** dashboard will be displayed and you will be prompted to select boundary curves to define a surface. For example, to create surface using four curves, as shown in Figure 15-45, select these four curves with the CTRL key pressed. After selecting these four curves, press the middle mouse button; the surface will be created, as shown in Figure 15-46.

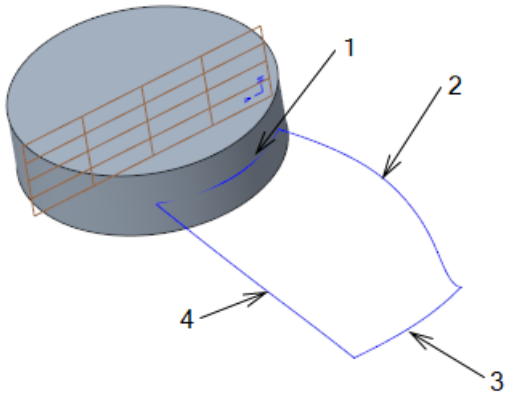


Figure 15-45 *Four curves*

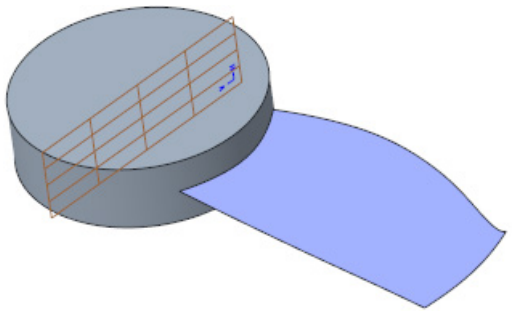


Figure 15-46 *Surface created using the curves*

Surface Connect

Ribbon: Style > Surface > Surface Connect

The **Surface Connect** tool is used to connect two Style surfaces. Style surfaces can also be connected to Creo Parametric surfaces using this tool. When you choose this tool, the **Style: Surface Connection** dashboard will be displayed and you will be prompted to select two surfaces. By pressing the CTRL key, select the two surfaces, as shown in Figure 15-47, and then press the middle mouse button; the connections will be applied to the two surfaces. These

connections may be of two types: curvature and tangent. 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 15-48 shows the two surfaces where the tangent connection as well as the curvature connection has been applied.

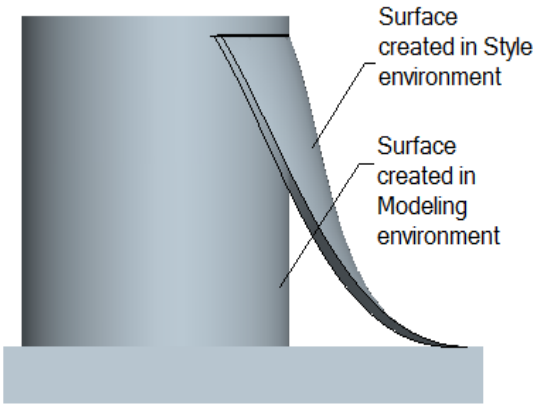


Figure 15-47 The two surfaces

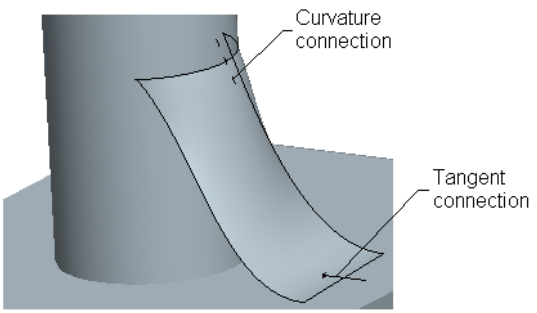


***Figure 15-48** Surfaces after applying connections*

Choose the **Surface Connect** tool from the **Surface** group of the **Style** dashboard; the **Style: Surface Connection** dashboard will be displayed, as shown in Figure 15-49, 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 15-50 shows the style surface when it is connected by using curvature connection and Figure 15-51 shows the surface when it is connected tangentially.

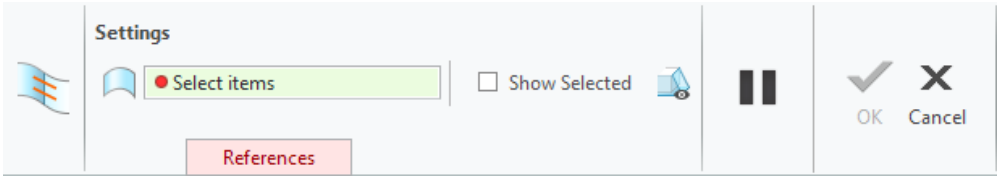


***Figure 15-49** Partial view of the **Style: Surfaces Connection** dashboard*

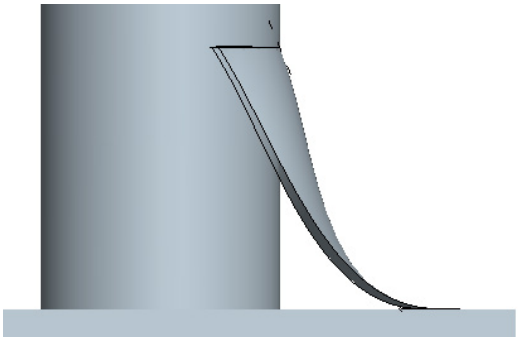


***Figure 15-50** Surface connected at top by using curvature connection*

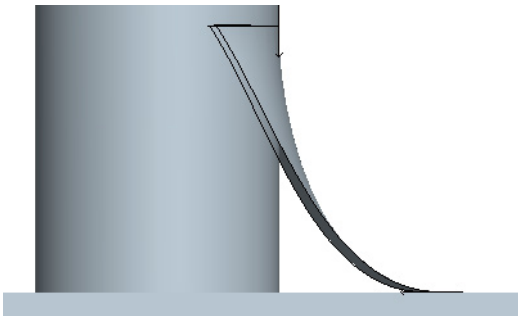


***Figure 15-51** Surface connected at top by using the tangent connection*

Note
To delete a curve, select the curve and when it turns green, press the DELETE key.

Surface Trim

Ribbon: Style > Surface > Surface Trim

The **Surface Trim** tool is used to trim a surface. When you choose this tool, the **Style: Surface Trim** dashboard will be displayed and you will be prompted to select the surface(s) to trim. Select the surface to trim and then press the middle mouse button; 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 **OK** button from the dashboard to exit the trim tool.

Figure 15-52 shows the surface and the curve that are selected for trimming. This figure also show the surface divided into two portions. The portion defined by the curve will be selected to delete. Figure 15-53 shows the surface after trimming.

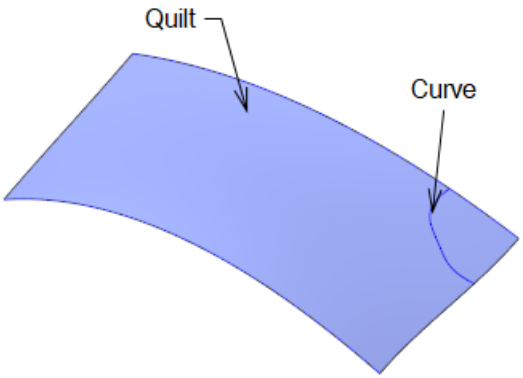


Figure 15-52 *Surface and curve selected for trimming*

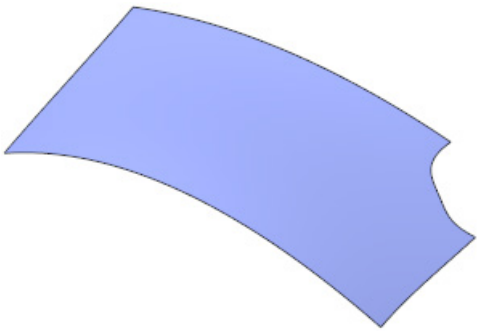


Figure 15-53 *Surface after trimming*

Note
*After completing the Style feature creation, choose the **OK** button from the **Close** group in the **Style** dashboard to exit the **Style** environment.*

MINI TOOLBAR

In Creo, when you select an item, a context-sensitive toolbar is displayed. This toolbar contains all tools that are relevant to the selection made. When you select a curve, surface, or datum, the mini toolbar appears in the graphics window.

Benefits of Mini Toolbar

The mini toolbar appears close to the pointer when you select a curve, surface, or datum. As a result, you can quickly access the required tool just by clicking. The tools in the mini toolbar are displayed based on the types of item you select in the graphics window or **Style Tree**.

You can access mini toolbar either from the graphics window or the **Style Tree**. The tools in the mini toolbar can be customized by choosing the **Customize** option in the shortcut menu. In Figure 15-54 part (a), (b), and (c) show the mini toolbar for surface, curve, and datum items, respectively.

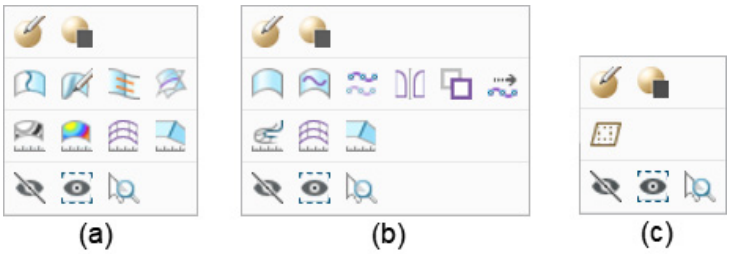


Figure 15-54 *The* ***Mini Toolbar*** *for surface, curve and datum items*

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

Mirroring the Surfaces

Ribbon: Model > Editing > Mirror

The **Mirror** tool is used to mirror the surface about a plane. This tool is available in the **Editing** group only when a surface is selected. When you choose this tool, the **Mirror** dashboard will be displayed, as shown in Figure 15-55.

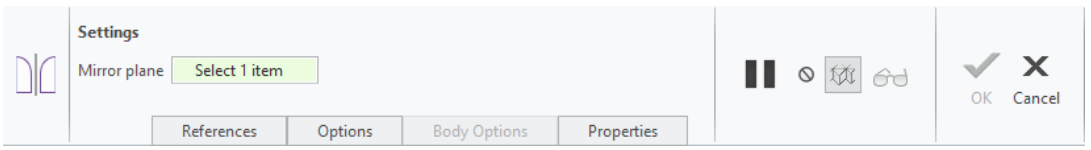


Figure 15-55 *Partial view of the* ***Mirror*** *dashboard*

Click in the **Mirror plane** collector, you can choose the mirroring plane from drawing area. The **Dependent Copy** 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 15-56 shows the mirror plane and Figure 15-57 shows the mirrored feature.

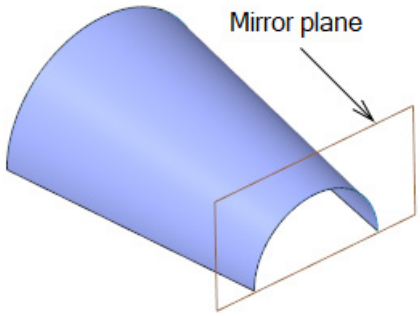


Figure 15-56 *The mirror plane and the surface to be mirrored*

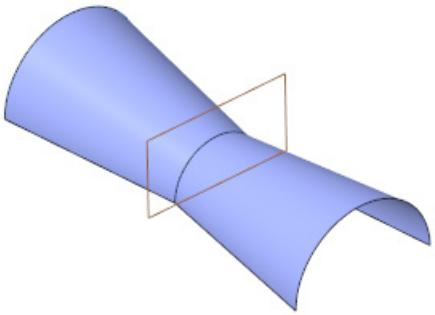


Figure 15-57 *Surfaces after mirroring and keeping the original surface*

Merging the Surfaces

Ribbon: Model > Editing > Merge

The **Merge** tool is used to merge two surfaces and make them a single surface. To convert a surface into a solid, it is necessary that the surfaces are merged. While merging the surfaces, this tool also trims them. This tool will be enabled in the **Editing** group only when the two surfaces to be merged are selected. When you choose the **Merge** tool, the **Merge** dashboard will be displayed, as shown in Figure 15-58.

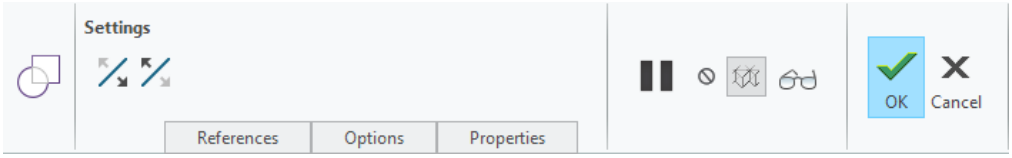


Figure 15-58 *Partial view of the* ***Merge*** *dashboard*

The following steps explain the procedure to merge the surfaces, as shown in Figure 15-59.

1. Select the **Quilts** option from the **Filter** drop-down list. Select the two surfaces and when they are highlighted, choose the **Merge** tool; the **Merge** dashboard will be displayed. While merging the two surfaces, the part of the surfaces that will be retained after the two surfaces are merged is highlighted by dots on it. The arrows point to the direction in which the surfaces are retained. The direction of arrow can be toggled by clicking on them or by using the **Change side of first quilt to keep** and **Change side of second quilt to keep** buttons of the **Merge** dashboard. You can also click on the arrows to flip the direction in which the surfaces are retained.

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 dots, as shown in Figure 15-60.

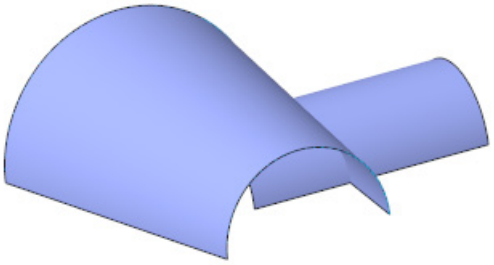


Figure 15-59 *Two surfaces to be merged*

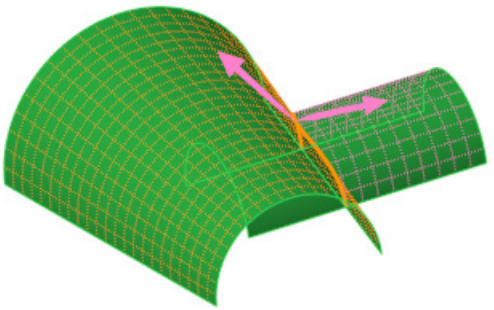


Figure 15-60 *Arrows showing the part of the surface to retain*

3. Choose the **Preview** button to verify the desired trim and then exit the dashboard by choosing the **OK** button. The resulting merged surface is shown in Figure 15-61. This merged surface is a single surface and now can be converted to a solid feature.

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.



Note

While working with more than two surfaces or quilts, it should be checked that the selection of quilts in the ***References*** *tab is in proper order.*

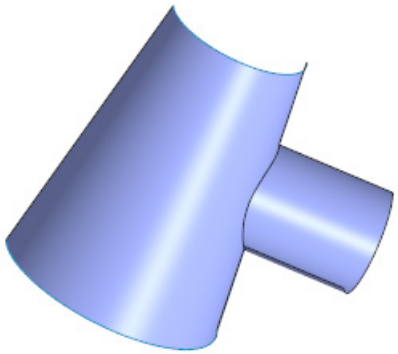


Figure 15-61 *Resulting merged surface*

Trimming the Surfaces

Ribbon: Model > Editing > Trim



As the name suggests, the **Trim** tool is used to trim the selected surfaces by using a trimming object. Select the surface that you need to trim and then choose the **Trim** tool from the **Editing** group; the **Surface Trim** dashboard will be displayed, as shown in Figure 15-62. Also, you will be prompted to select the trimming object. This trimming object can be a curve, plane, edge, or a surface.

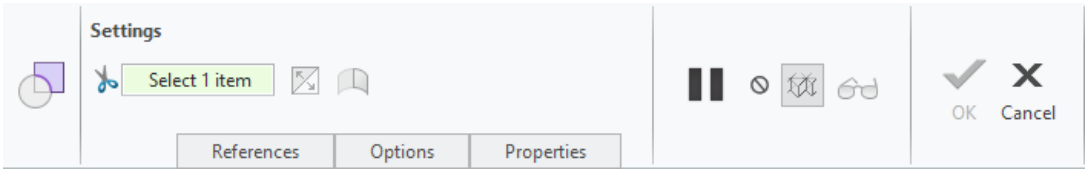


Figure 15-62 *Partial view of the* ***Surface Trim*** *dashboard*

The part of the surface that is to be retained is highlighted with dots. Also, an arrow is displayed, pointing 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 from the dashboard or click on the arrow to toggle the direction. By default, the trimming surface

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 15-63 shows the surface selected as the trimming object, the trimming surface, and the arrow. It is evident from this figure that the arrow is pointing toward the right; therefore, the right portion of the surface will be retained after trimming. Figure 15-64 shows the surface obtained after trimming.

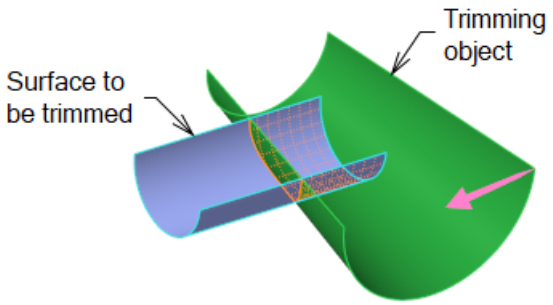


Figure 15-63 *Surface and object to be trimmed*

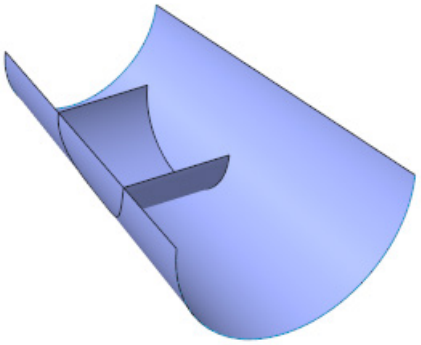


Figure 15-64 *Surface obtained after trimming*

Creating the Fill Surfaces

Ribbon: Model > Surfaces > Fill

The **Fill** tool is used to create a planar surface by sketching its boundaries. When you choose this option from the **Surfaces** group of the **Ribbon**, the **Fill** dashboard will be displayed, as shown in Figure 15-65.



Figure 15-65 *Partial view of the* ***Fill*** *dashboard*

Choose the **References** tab; a slide-down panel will be displayed. Choose the **Define** button from it to select the sketching plane and draw the sketch. Figure 15-66 shows the sketch plane and Figure 15-67 shows the surface that is created by using the **Fill** tool.

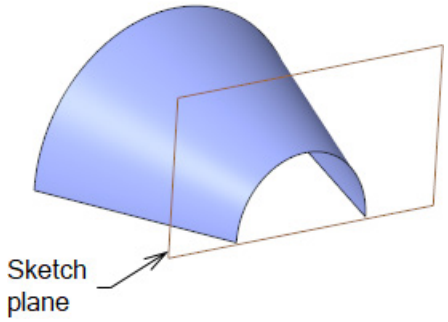


Figure 15-66 The sketch plane for creating the fill surface

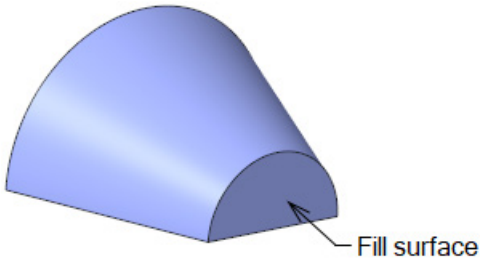


*Figure 15-67 Fill surface created using the **Fill** tool*

Creating the Intersect Curves

Ribbon: Model > Editing > Intersect

The **Intersect** tool is used to create a curve at the intersection of two items. The intersection can be the combination of any item either a face of solid model or a datum plane or a quilt surface. Note that you cannot create a datum curve at the intersection of two datum planes or two solid model faces. The curve created can be used with various tools. The **Intersect** tool is available in the **Editing** group only when you select one or more items from the graphics window. When you choose this tool from the **Editing** group; the **Surface Intersection** dashboard will be displayed, as shown in Figure 15-68.

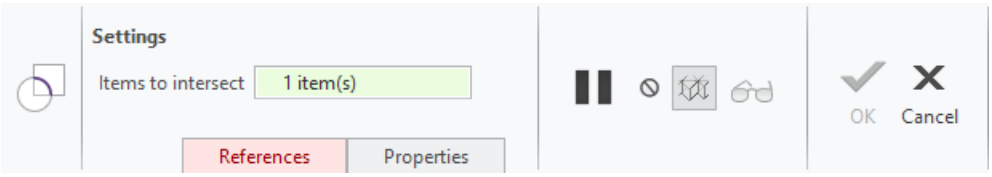


*Figure 15-68 Partial view of the **Surface Intersection** dashboard*

When you select the second surface, the intersecting curve will be created, as shown in Figure 15-69. 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 15-70 and 15-71.

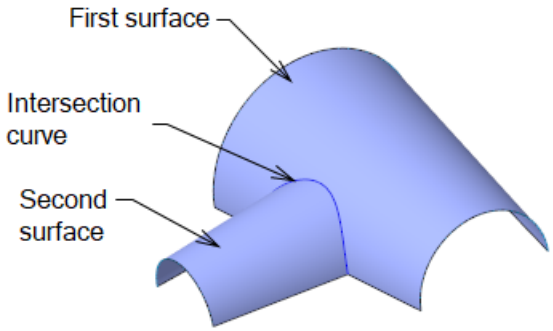


Figure 15-69 Surfaces selected to create the intersecting curve

In Figure 15-70, the intersecting curve is copied at a distance of 1250. The **Boundary Blend** tool is used to create the surface shown in Figure 15-71. 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.

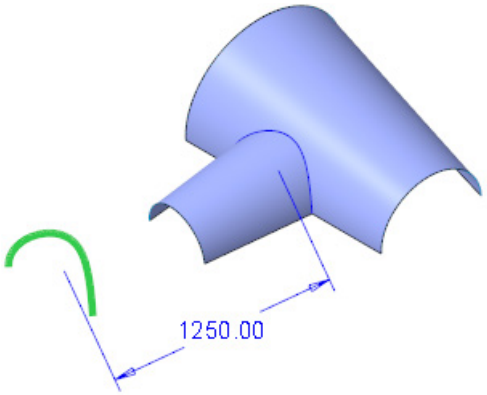


Figure 15-70 *Copied curve*

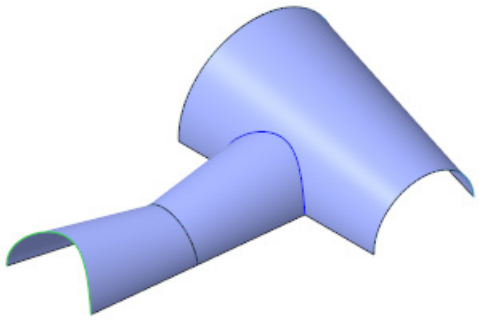


Figure 15-71 *Boundary blend created using the intersecting curve*

Creating the Offset Surfaces

Ribbon: Model > Editing > Offset

A surface can be copied to an offset distance. To offset a surface, select the surface to offset and then choose the **Offset** tool from the **Editing** group in the **Ribbon**; the **Offset** dashboard will be displayed, as shown in Figure 15-72. The **Offset** tool will be available only when you select a surface to offset.

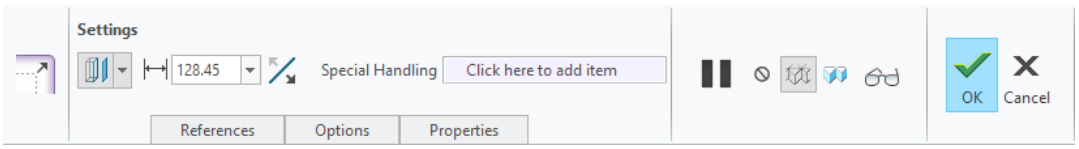


Figure 15-72 *Partial view of the **Offset** dashboard*

In Creo Parametric, 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 Creo Parametric are as follows: Standard Offset Feature, With Draft Feature, Expand Feature, and Replace Surface Feature.

Standard Offset Feature



The **Standard Offset Feature** tool 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 Creo Parametric 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 15-73 shows the original surface and the offset surface.

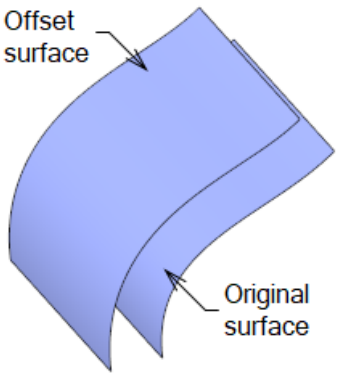


Figure 15-73 *The original and offset surfaces*

With Draft Feature



The **With Draft Feature** tool is available in the drop-down that appears by selecting the black arrow present on the right of the Standard Offset Feature tool. Using this tool, 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 of the draft surface. Figure 15-74 shows the draft offset surface with the **Straight** radio button selected from the **Options** slide-down panel. The section drawn on the sketching plane is circular. Similarly, Figure 15-75 shows the draft offset surface with the **Tangent** radio button selected from the **Options** slide-down panel.

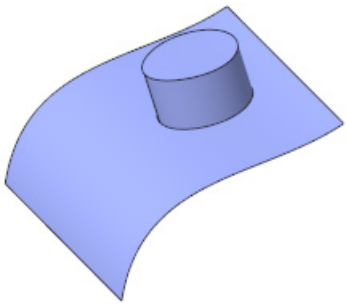


Figure 15-74 *Draft offset surface with straight profile*

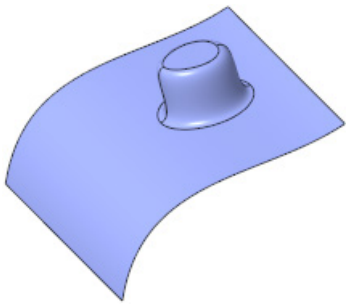


Figure 15-75 *Draft offset surface with tangent profile*

Expand Feature



The **Expand Feature** tool can be chosen from the list of tools that appear by selecting the down arrow present on the right of Standard Offset Feature tool. Using this option, you can sketch the section and then choose whether to offset the inside or the outside of the sketch. For this purpose, you need to choose the **Change direction of offset to the other side** button from the dashboard. Alternatively, click on the arrow displayed on the model. Figure 15-76 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 15-77 shows the draft offset surface when the outside of the sketch is selected to offset.

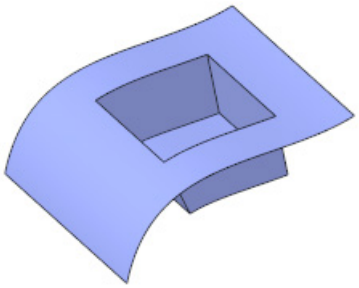


Figure 15-76 *Inside of the sketch selected to offset*

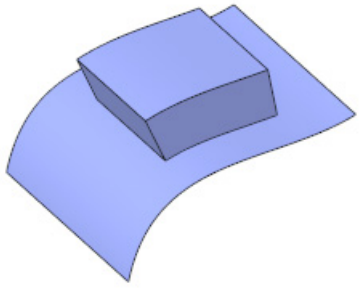


Figure 15-77 *Outside of the sketch selected to offset*

Replace Surface Feature



The **Replace Surface Feature** tool is used to replace a specified face of a solid with a quilt or datum plane. It can add or remove material from the solid. To create a replacement surface, select the face to be replaced and then choose the **Offset** tool from the **Editing** group of the **Ribbon**; the **Offset** dashboard will be displayed. Choose the down arrow present on the right of the Standard Offset Feature tool; a list of tools will be displayed. Choose the **Replace Surface Feature** button; you will be prompted to select the surface or datum plane to replace. Select the replacement surface; the preview of the modified solid feature will be displayed. Choose the **OK** button to accept the preview and create the feature. Figure 15-78 shows the face of the solid selected to be offset and the replacement surface. Figure 15-79 shows preview of the modified solid feature.

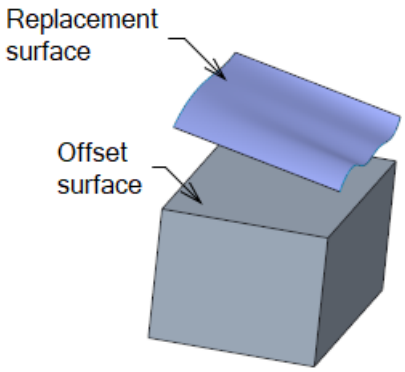


Figure 15-78 *The offset and replacement surfaces*

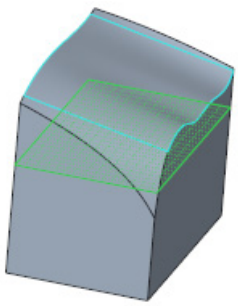


Figure 15-79 *Preview of the modified solid feature*

Adding Thickness to a Surface

Ribbon: Model > Editing > Thicken

To add thickness to a quilt or a surface, select the quilt and choose the **Thicken** option from the **Editing** group; the **Thicken** dashboard will be displayed, as shown in Figure 15-80.

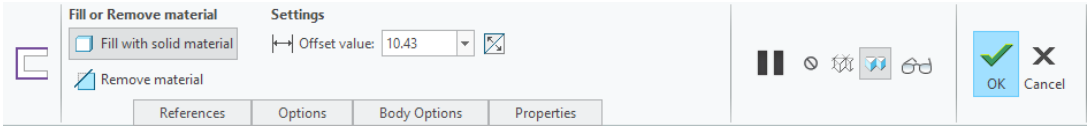


Figure 15-80 *Partial view of the* ***Thicken*** *dashboard*

Drag the handle to set thickness of the quilt or enter the thickness value in the **Offset value** dimension edit box. You can even remove material from the quilt by choosing the **Remove material** 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. Creo Parametric automatically scale the surface along the axes 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 15-81 and 15-82 show the surfaces after adding thickness by using the **Normal to surface** and **Automatic fit** options, respectively.

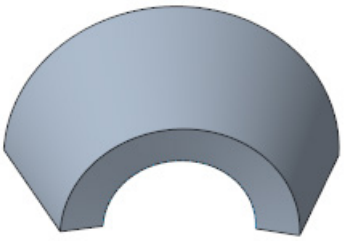


Figure 15-81 *Thickening the surface by using the* ***Normal to surface*** *option*

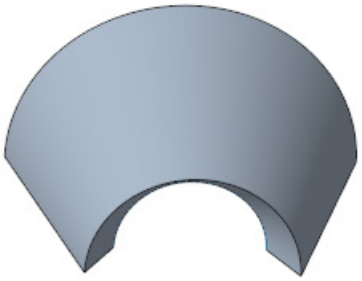


Figure 15-82 *Thickening the surface using the* ***Automatic fit*** *option*

Converting a Surface into a Solid

Ribbon: Model > Editing > Solidify

You can convert a closed surface into a solid by choosing the **Solidify** tool from the **Editing** group of the **Ribbon**. 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

Ribbon: Model > Surfaces > Vertex Round

The vertices of a surface or quilt can be rounded by using the **Vertex Round** option. To do so, choose **Vertex Round** from the expanded **Surfaces** group of the **Ribbon**; the **Vertex Round** dashboard will be displayed, as shown in Figure 15-83. Also, you will be prompted to select a vertex to add round.



Figure 15-83 *Partial view of the* ***Vertex Round*** *dashboard*

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 15-84. After selecting the vertices, enter the radius value in edit box available in the dashboard. Choose the **OK** button from the **Vertex Round** dashboard; the vertices will be rounded, as shown in Figure 15-85.

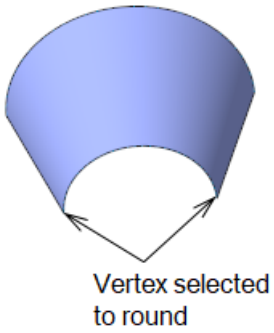


Figure 15-84 *Vertices selected to be rounded*

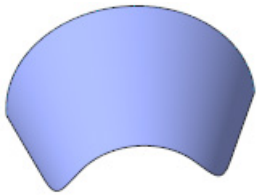


Figure 15-85 *Vertices after creating round*

FREESTYLE MODELING ENVIRONMENT

Ribbon: Model > Surfaces > Freestyle



The Freestyle Modeling is a surfacing environment which encompasses qualities of both the NURBS and polygon surfaces. NURBS surfaces are smooth surfaces controlled by CV points and the polygon surfaces give the flexibility to extrude a specific area of the surface for detailing. Features created in the Freestyle Modeling environment are called Freestyle features. These features can be created at the beginning of modeling in Creo Parametric. You can invoke the Freestyle Modeling environment by choosing the **Freestyle** tool from the **Surfaces** group of the **Model** tab in the **Ribbon**.

Freestyle Dashboard

When you invoke the **Freestyle** tool from the **Surfaces** group of the **Model** tab in the **Ribbon**, the **Freestyle** dashboard is displayed, as shown in Figure 15-86. The options in this dashboard are discussed next.

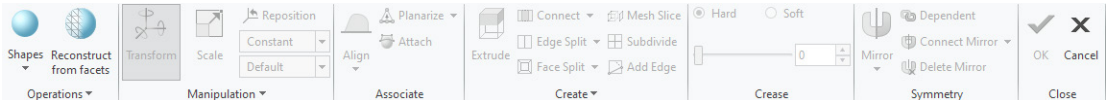


Figure 15-86 *Partial view of the* ***Freestyle*** *dashboard*

Primitives

A primitive is the first feature required to proceed further to create a model in the Freestyle Modeling envir onment. All operations available in the Freestyle Modeling environment are performed on the primitive. Primitives are available in the **Shapes** drop-down of the **Operations** group in the **Freestyle** dashboard, refer to Figure 15-87. It is important to note that you can create only one primitive at a time in the Freestyle Modeling environment. If you want to create more primitives then you need to invoke the Freestyle environment again. There are two categories of primitives available in the **Shapes** drop-down: **Open Primitives** and **Closed Primitives**. The **Open Primitives** category contains two-dimensional primitives that are created in the XY plane. The **Closed Primitives** category contains three-dimensional primitives which are created with coordinate system at its center. To create a primitive, click on the desired primitive icon; the primitive will be created in the drawing area. You can change the reference coordinate system by using the **Freestyle Options** dialog box. This dialog box is displayed when you choose the **Options** button from the drop-down of the **Operations** group, refer to Figure 15-88.

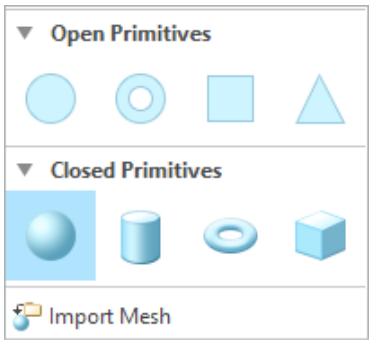


Figure 15-87 *The* ***Shapes*** *drop-down*

To change the coordinate system, click in the **Reference CSYS** collector in the **References** area and select the desired reference coordinate system. You can also change the increment value of the 3D Dragger in the linear or angular direction.

Transform

The **Transform** tool in the **Manipulation** group of the **Freestyle** dashboard is used to transform the primitive. This tool will be activated only when you select any face, edge, or vertex of the primitive. Figure 15-89 shows a rectangle primitive with a vertex selected. Figure 15-90 shows the deformed primitive after dragging the 3D Dragger along the Y direction. You can rotate and drag the 3D Dragger along any of the direction available.

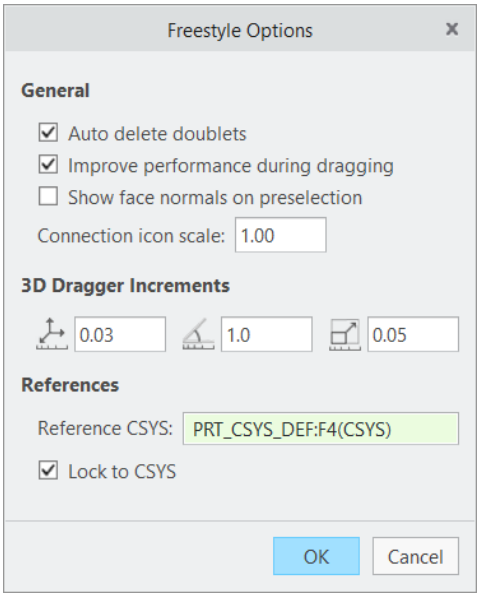


Figure 15-88 *The* ***Freestyle Options*** *dialog box*



Figure 15-89 *A rectangle primitive with a vertex selected*

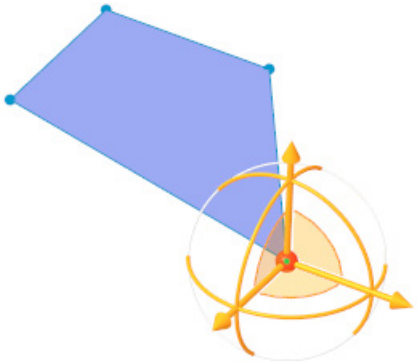


Figure 15-90 *The deformed primitive after dragging the 3D Dragger along Y direction*

Scale

The **Scale** tool in the **Manipulation** group of the **Freestyle** dashboard will be activated only when you select any face, edge, or vertex of a primitive. Note that the scaling operation cannot be performed on a vertex. To apply scaling, you need to select an edge or a face. When you select an edge or a face and choose the **Scale** tool from the dashboard, the 3D Dragger is displayed without the circular handles. Select any of the axes or planes of the 3D Dragger to scale the primitive.

Reposition

The **Reposition** tool is activated in the **Manipulation** group of the **Freestyle** dashboard only when you select any face, edge, or vertex of a primitive. Using the **Reposition** tool, you can reposition the 3D Dragger by rotating or translating it along any of the available axes.

Align

Using the **Align** tool, you can align the primitives along any of the datum planes or planar surfaces. This tool will be activated in the **Associate** group of the **Freestyle** dashboard only when you select any face or edge of a primitive. To align a primitive along a datum plane, select the primitive, choose the **Align** tool from the dashboard, and then select the datum plane along which you want the primitive to be aligned.

Planarize

The **Planarize** tool is used to make all the selected vertices lie in a single plane. This tool will be activated in the **Manipulation** group of the **Freestyle** dashboard only when you select any face, edge, or vertex of a primitive. When you invoke this tool after selecting the primitive, a default plane is displayed in which the vertices will lie. You can press the middle mouse button to accept the default plane or you can select a plane along which the vertices will be planarized.

Increment

The **Increment** tool is activated in the **Manipulation** group of the **Freestyle** dashboard only when you select any face, edge, or vertex of a primitive. When the **Increment** toggle button is selected, the increment value will also be displayed during the manipulation of the primitive.

Extrude

The **Extrude** tool is available in the **Create** group of the **Freestyle** dashboard and will be activated only when you select any face or edge of a primitive. The **Extrude** tool available in the Freestyle environment works in the same way as in the Modeling environment. When you select any face and then choose the **Extrude** tool from the dashboard, all the edges of the selected face are converted into walls. Figure 15-91 shows extruded feature of a rectangle primitive.

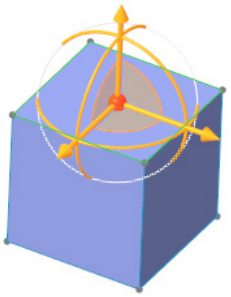


Figure 15-91 *The extruded feature of a rectangle primitive*

Edge Split

The **Edge Split** drop-down list is available in the **Create** group of the **Freestyle** dashboard and will be activated only when you select an edge of a primitive. After selecting an edge, click on the down arrow at the right of the **Edge Split** button, a drop-down list will be displayed, as shown in Figure 15-92. Now, select the desired option, the selected edge will split into desired number of equal pieces.

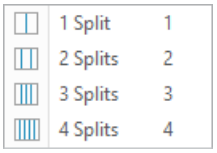


Figure 15-92 *The **Edge Split** drop-down list*

Face Split

The **Face Split** drop-down list is available in the **Create** group of the **Freestyle** dashboard and will be activated only when you select a face of a primitive. After selecting the face, click on the down arrow at the right of the **Face Split** button, a drop-down list will be displayed, as shown in Figure 15-93. Select the desired option from the menu; the selected face will split into five pieces of selected offset percentage.

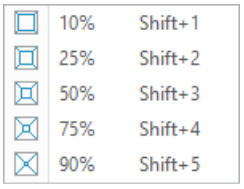


Figure 15-93 *The **Face Split** drop-down list*

Connect

The **Connect** tool is used to join two or more edges. This tool is available in the **Create** group of the **Freestyle** dashboard and will be activated only when you select two or more faces or edges of a primitive. Note that you can select more than one edge or face by holding the CTRL key while selecting the entities. After selecting two or more faces or edges, choose the **Connect** tool from the **Create** group of the dashboard; the faces or edges will be joined together and the resultant surface will be displayed. Figure 15-94 shows a deformed rectangle with two edges to be selected for connecting. Figure 15-95 shows the output after connecting the edges.

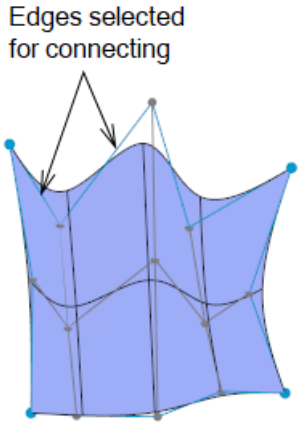


Figure 15-94 *A deformed rectangle with two edges to be selected for connecting*

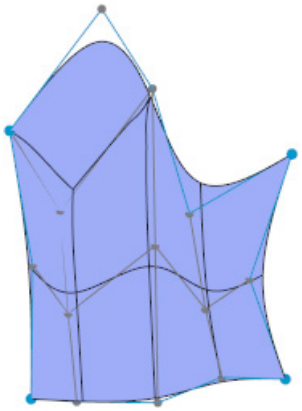


Figure 15-95 *The output after connecting the edges*

Mirror

The **Mirror** tool is available in the **Mirror** drop-down of the **Mirror** drop-down of the **Symmetry** group in the **Freestyle** dashboard and will be activated only when you select an edge or a face of the primitive. After selecting an edge or a face, choose the **Mirror** tool; you will be prompted to select the datum plane or planar surface as a mirror plane. Select the mirror plane, the edge or face will be mirrored along with the control points.

Connect Mirror

The **Connect Mirror** tool is available in the **Symmetry** group of the **Freestyle** dashboard and will be activated only when you select a face of the surface mirrored earlier. Figure 15-96 shows a mirrored surface and the face to be selected for the mirror connect operation. Figure 15-97 shows the output surface.

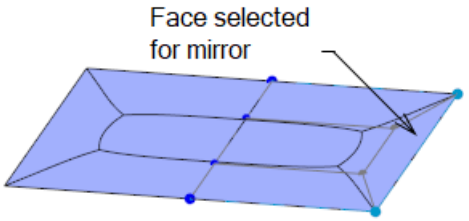


Figure 15-96 *A mirrored surface and the face to be selected for the mirror connect operation*

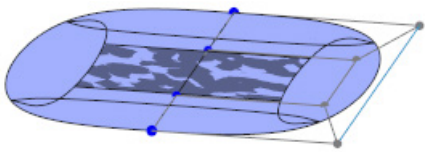


Figure 15-97 *The output surface*

Dependent

The **Dependent** toggle button is available in the **Symmetry** group of the dashboard. When this toggle button is selected, the control mesh of mirrored feature is dependent on the original feature. But when this button is not selected, the mirrored feature is controlled by its own control mesh. Figure 15-97 shows the default output surface created by using the **Connect Mirror** tool which is a dependent mirrored surface and Figure 15-98 shows the independent mirrored surface.

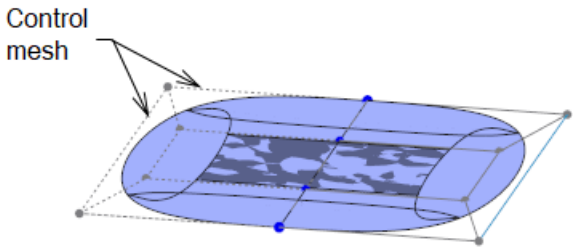


Figure 15-98 *An independent mirrored surface*

Delete Mirror/Change Plane

You can delete the mirrored surface by selecting the **Delete Mirror** option available in the **Symmetry** group. You can change the mirror plane by selecting the **Change Plane** option available in the **Mirror** drop-down.

TUTORIALS

Tutorial 1

In this tutorial, you will create the surface model shown in Figure 15-99. The orthographic views of the surface model are shown in Figure 15-100. **(Expected time: 45 min)**

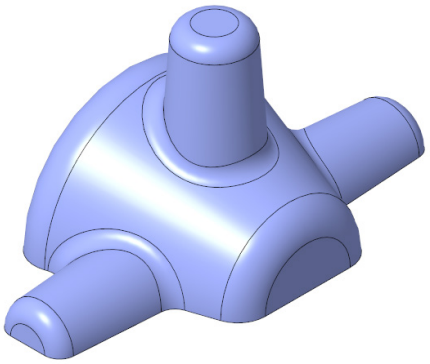


Figure 15-99 *The surface model*

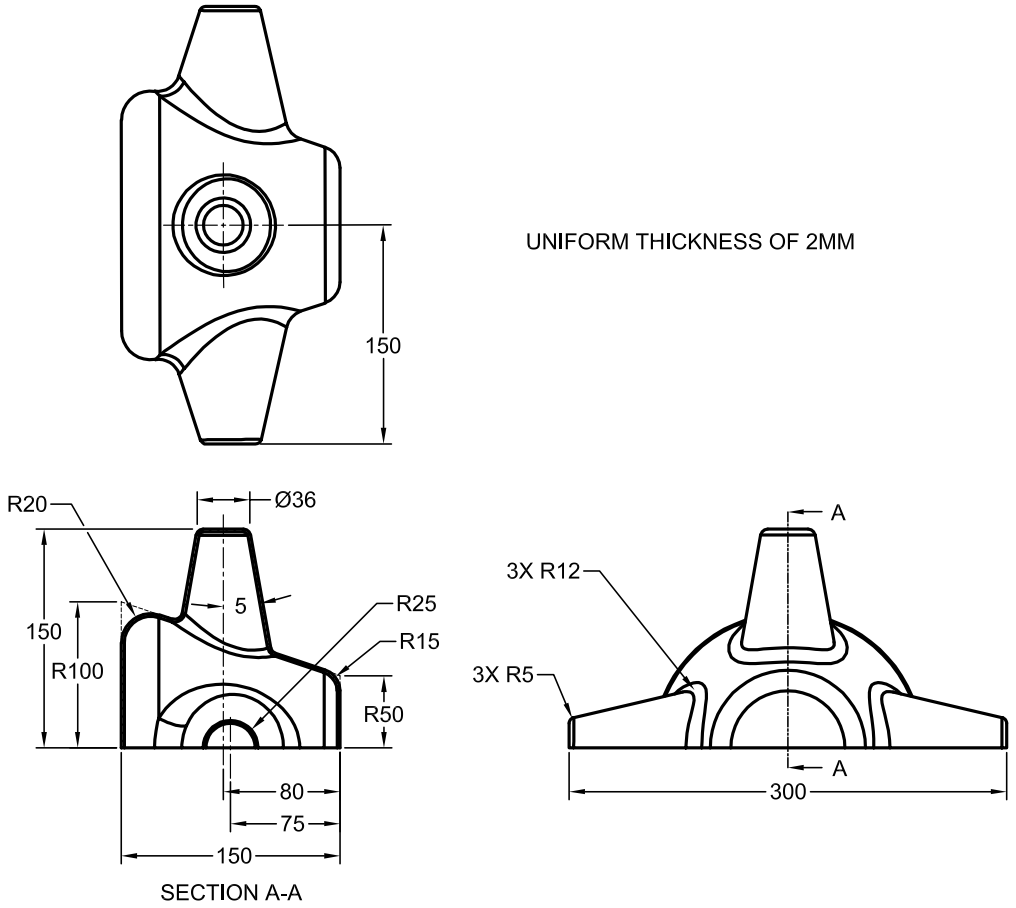


Figure 15-100 *Top, front, and right views of the surface model*

Examine the model and determine the number of features in it, refer to Figure 15-99.

The following steps are required to complete this tutorial:

a. Create the base feature, which is a blend surface.
b. Create the second feature, which is a blend feature.
c. The third feature is a mirror feature. It will be created by mirroring the second feature about a plane that passes through the center.
d. Create the fourth feature, which is also a blend feature.
e. Select the surfaces individually and merge them.
f. Remaining features are the fill features and they will be used to create surfaces on the blend features.
g. Create rounds on the edges and save the model.

Starting a New Object File

1. Start a new part file and name it *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 need to create the base feature by using the **Blend** option.

1. Choose the **Blend** tool from the expanded **Shapes** group; the **Blend** dashboard is displayed.

2. Choose the **Surface** button from the dashboard. Choose the **Options** tab and select the **Straight** radio button. Choose the **Sections** tab from the dashboard; a slide down panel is displayed. Select the **Sketched sections** radio button and choose the **Define** button from the slide down panel to define the sketching plane.

3. Select the **RIGHT** datum plane as the sketching plane.

4. Select the **TOP** datum plane as reference plane and then select the **Top** option from the **Orientation** drop-down list. Choose the **Sketch** button to enter the sketcher environment.

5. Draw an arc and dimension it, as shown in Figure 15-101.

6. After drawing the first arc, choose the **OK** button to exit the sketcher environment. Choose the **Insert** button from the **Section** tab; an offset value edit box with a default value will be available in the dashboard and in the **Sections** tab. Enter **-150** in offset value edit box for depth of second section. Choose the **Sketch** button from the **Sections** tab to enter the sketcher environment.

7. Draw the second arc and dimension it, as shown in Figure 15-102.

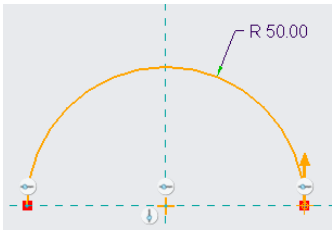


Figure 15-101 *Sketch of the first arc*

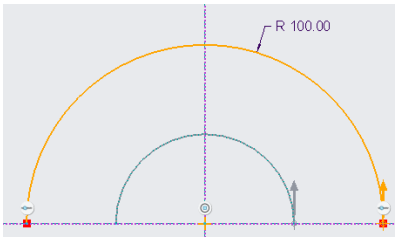


Figure 15-102 *Sketch of the second arc*

8. After drawing the second arc, choose the **OK** button to exit the sketcher environment. Choose the **OK** button from the dashboard to exit it. The model, similar to the one shown in Figure 15-103, is displayed in the drawing area.

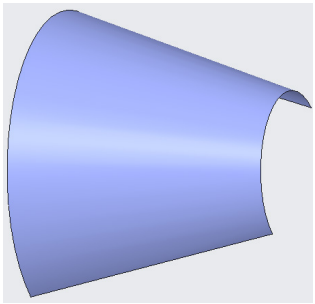


Figure 15-103 *Trimetric view of the base feature*

Creating the Second Feature

The second blend feature will be created on the **FRONT** datum plane and it will pass through the center of the base feature.

1. Choose the **Blend** tool from the expanded **Shapes** group; the **Blend** dashboard is displayed.

2. Choose the **Surface** button from the dashboard. Choose the **Options** tab and select the **Straight** radio button. Choose the **Sections** tab from the dashboard; a slide down panel is displayed. Next, select the **Sketched sections** radio button and then choose the **Define** button to define the sketching plane.

3. Select the **FRONT** datum plane as the sketching plane.

4. Select the **TOP** datum plane as the reference plane and then select the **Top** option from the **Orientation** drop-down list. Choose the **Sketch** button to enter the sketcher environment.

5. Sketch an arc of radius **35**; for the remaining dimensions refer to Figure 15-100 and then choose the **OK** button to exit the sketcher environment. Choose the **Insert** button from the **Section** tab and enter **150** in the offset value edit box for depth of second section. Choose the **Sketch** button from the **Sections** tab to enter the sketcher environment.

6. Draw the second arc of radius **25**, as shown in Figure 15-104.

7. Choose the **OK** button to exit the sketcher environment. Next, choose the **OK** button from the dashboard to exit it. The model, similar to the one shown in Figure 15-105, is displayed in the drawing area.

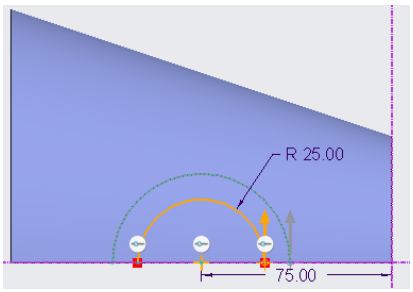


Figure 15-104 *Sketch of the second feature*



Figure 15-105 *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 blend feature will be used to create the third feature.

1. Select the second feature and then choose the **Mirror** tool; 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 **OK** button. A mirror copy of the second feature is created, as shown in Figure 15-106.

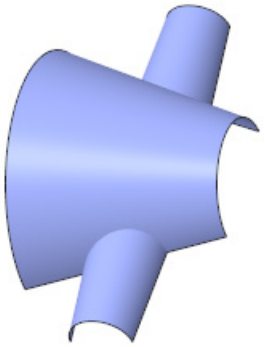


Figure 15-106 *Surface model after creating the third feature*

Creating the Fourth Blend Feature

The fourth blend feature will be created on the **TOP** datum plane and will pass through the center of the base feature.

1. Choose the **Blend** tool from the expanded **Shapes** group; the **Blend** dashboard is displayed.

2. Choose the **Surface** button. Choose the **Options** tab and then select the **Straight** radio button. Also, select the **Capped Ends** check box.

3. Choose the **Sections** tab from the dashboard to open the slide-down panel. Choose the **Sketched sections** radio button and then choose the **Define** button to define the sketching plane.

4. Select the **TOP** datum plane as the sketching plane.

5. Select the **RIGHT** datum plane as the reference plane and then select the **Top** option from the **Orientation** drop-down list. Choose the **Sketch** button to enter the sketcher environment.

6. Sketch a circle of diameter 72; for the remaining dimensions refer to Figure 15-100 and then choose **OK** to exit the sketcher environment. Choose the **Insert** button from the **Section** tab and enter **150** in the offset value edit box for depth of second section. Choose the **Sketch** button from the **Sections** tab to enter the sketcher environment.

7. Draw second circle of diameter 36, as shown in Figure 15-107.

8. Choose the **OK** button to exit the sketcher environment. Choose the **OK** button from the dashboard to exit it. The model, similar to the one shown in Figure 15-108, is displayed in the drawing area.

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 will join. For this purpose, the surfaces need to be merged.

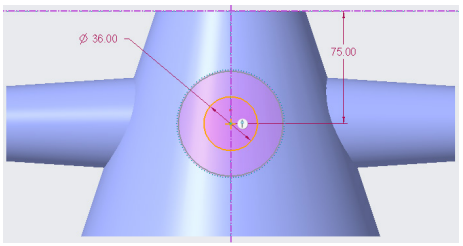


Figure 15-107 *Sketch of the fourth blend feature*

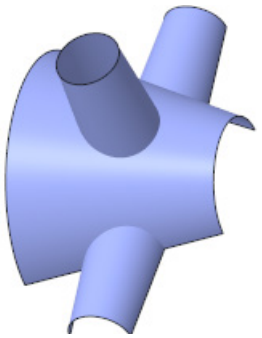


Figure 15-108 *Model after creating the fourth blend surface*

Note

1. It's 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, indicating that the surfaces are selected.*

2. You can also choose the ***Quilts*** *option from the* ***Filter*** *drop-down list to select the surfaces. The* ***Filter*** *drop-down list is available in the* ***Status Bar*** *at the right corner below the drawing area.*

1. Select the blend surface on the left, press the CTRL key, and then select the blend in the middle. When the two surfaces are highlighted, choose the **Merge** tool; the **Merge** dashboard is displayed with two arrows showing the portion that will be retained after merging.

Note

The ***Merge*** *button will be available only when two surfaces are selected for merging.*

2. If required, choose the **Change side of first quilt to keep** button from the dashboard; the direction of the arrow changes.

3. If required, choose the **Change side of second quilt to keep** button from the dashboard; the direction of the arrow changes. The portion of the surface that is now highlighted will be retained after merging the surfaces.

4. Exit the dashboard by choosing the **OK** button. The model after merging the two surfaces is shown in Figure 15-109.

Using the above procedure, merge the blend surface on the right with the merged surface. Next, merge the top blend surface with the merged surface. Figure 15-110 shows the surface model after merging all surfaces and forming a quilt.

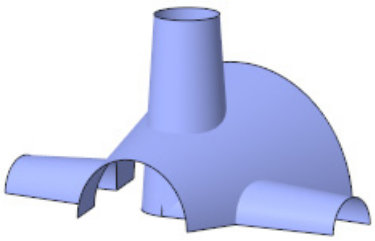


Figure 15-109 *Surface model after merging the two surfaces*

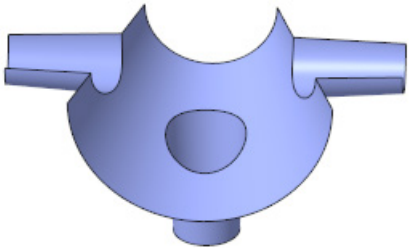


Figure 15-110 *Surface model after merging all the surfaces*

Creating the Fill Surfaces

You need to create four surfaces to cap the ends of the blend surfaces. First, you will cap the left blend surface using the **Fill** tool.

1. Choose the **Fill** tool from the **Surfaces** group of the **Model** tab in the **Ribbon**; 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** tool from the **Datum** group of the **Model** tab; the **Datum Plane** dialog box is displayed.

4. Select the two vertices of the left blend surface. To select the second vertex, hold down the CTRL key and then select the **FRONT** datum plane.

5. Select **FRONT** from the **Datum Plane** dialog box; a drop-down list appears in the row where you have clicked. Choose the **Parallel** option from the drop-down list and then choose the **OK** button from the **Datum Plane** dialog box.

   The datum plane is created and an arrow pointing in the direction of viewing the sketch is displayed.

6. Choose the **Sketch** button to close the **Sketch** dialog box to enter the Sketcher environment.

7. Choose the **Project** tool from the **Sketching** group and select the smaller semicircular edge of the blend surface. Next, close the sketch by drawing a horizontal line between the two ends, as shown in Figure 15-111.

8. Exit the Sketcher environment and then exit the **Fill** dashboard by choosing the **OK** button; the fill surface is created, as shown in Figure 15-112.

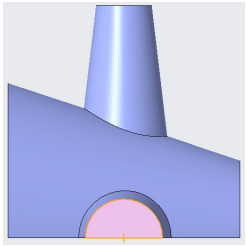


Figure 15-111 *Sketch for the fill surface*

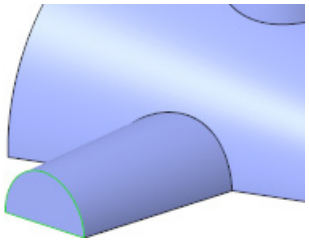


Figure 15-112 *Surface after creating the fill surface*

Similarly, create the fill surfaces to cap the ends of the middle surface blend feature. Mirror the left fill surface to the right. Figure 15-113 shows the surface model after capping all the ends of the blend surfaces.

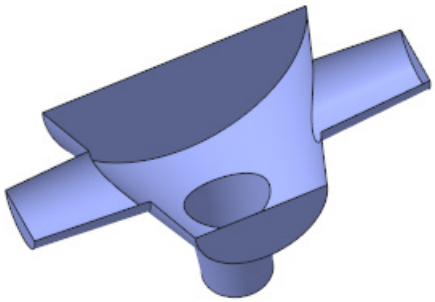


Figure 15-113 *Surface model after creating the fill surfaces*

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.

1. Hold the CTRL key to select the fill surface that is on the left and the blend surface in the middle. When the two surfaces turn green in color, choose the **Merge** tool to display the **Merge** dashboard.

2. Exit the dashboard by choosing the **OK** button.

Using the same procedure, merge the remaining fill surfaces individually with the blend surface in the middle. To check whether all the surfaces have been merged, select the surface model. If the whole surface model is highlighted with green border then it indicates that all the surfaces have been merged and they form a quilt.

Creating Rounds

When all the surfaces are merged, edges are formed at the intersection of two surfaces. These edges need to be 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** group.

2. Select the edges having radius value **12**; rounds of radii **12** get created.

   Remember that to select more than one edge, you need to hold down the CTRL key.

3. Choose the **Sets** tab to display the slide-down panel.

4. Click on **New set**; a set with the name **Set2** is added. Select the three edges that have radius of **5**; rounds of radii **5** get created.

5. Simlarly, create **Set3** and **Set4** with radius **20** and **15**, refer to Figure 15-100. After creating the rounds, exit the **Round** dashboard.

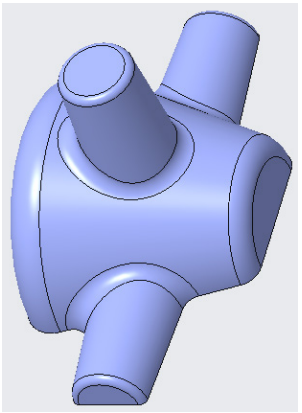


Figure 15-114 *Surface model after creating rounds*

   The surface model after creating the rounds is shown in Figure 15-114.

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

Tutorial 2

In this tutorial, you will create the surface model shown in Figure 15-115. The front and right views of the surface model are shown in Figure 15-116. **(Expected time: 45 min)**

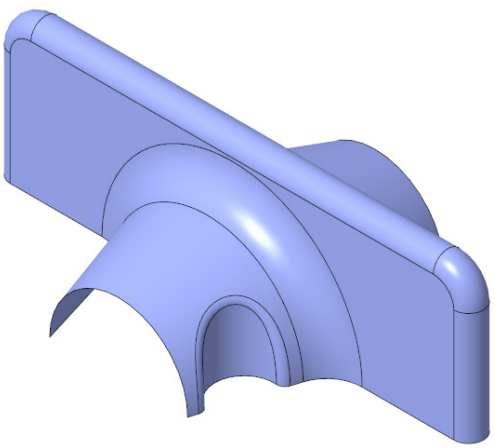


Figure 15-115 *Isometric view of the surface model*

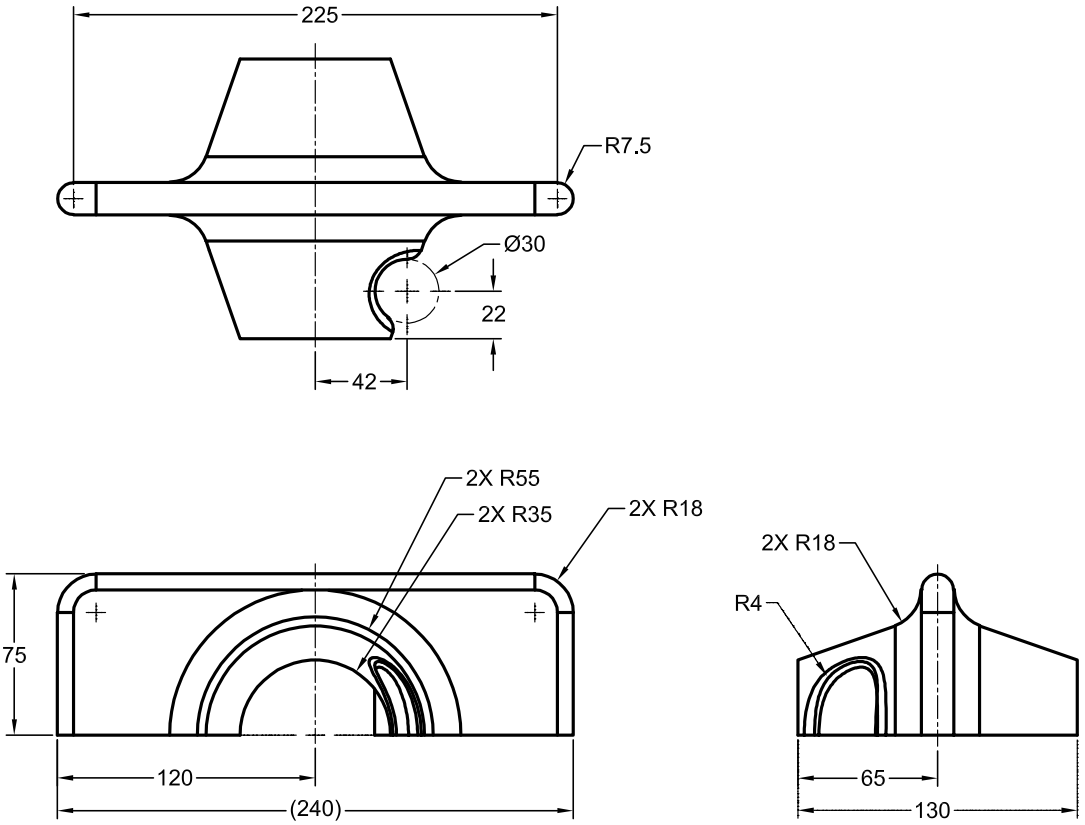


Figure 15-116 *Orthographic views of the surface model*

First, examine the model and determine the number of features in it, refer to Figure 15-115.

The following steps are required to complete this tutorial:

a. Create the base feature, which is an extruded surface with open ends.
b. Create the second feature, which is a blend feature created at an offset distance of 65 from the **RIGHT** datum plane.
c. The third feature is a mirror copy of the second feature.
d. The fourth feature is the cylindrical surface.
e. Create the two fill surfaces that will cap the ends of the base surface.
f. Merge all surfaces by selecting them individually.
g. Create the round features and save the model.

Starting a New Object File

1. Start a new part file and name it *c15tut2*.

Creating the Base Feature

The base feature for this model is a surface between two blend surfaces. It will be created on the **RIGHT** datum plane.

1. Choose the **Extrude** tool from the **Shapes** group.

2. Select the **Surface** button from the **Extrude** dashboard. Select the **RIGHT** datum plane as the sketch plane.

3. Select the **TOP** datum plane as reference 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. Create the sketch of the base feature and apply dimensions, as shown in Figure 15-117.

6. Choose the **OK** button to exit the Sketcher environment.

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

7. Enter **240** in the dimension box present in the **Extrude** dashboard. Choose the **OK** button from the **Extrude** dashboard.

   The base feature is created and the default trimetric view of this feature is shown in Figure 15-118.



Figure 15-117 *Sketch of the base surface*



Figure 15-118 *Base surface with open ends*

Creating the Blend Feature

The second feature is a blend surface. It will be created on the **FRONT** datum plane and passes through the center of the base feature.

1. Choose the **Blend** tool from the expanded **Shapes** group; the **Blend** dashboard is displayed.

2. Choose the **Surface** button from the dashboard. Choose the **Options** tab and select the **Straight** radio button. Choose the **Sections** tab from the dashboard; a slide down panel is displayed. Select the **Sketched sections** radio button and choose the **Define** button to define the sketching plane.

3. Select the **FRONT** datum plane as the sketching plane.

4. Select the **TOP** datum plane as the reference plane and then select the **Top** option from the **Orientation** drop-down list. Choose the **Sketch** button to enter the Sketcher environment.

5. Draw an arc of diameter **55**; for the remaining dimensions refer to Figure 15-116 and then choose **OK** to exit the Sketcher environment. Choose the **Insert** button from the **Section** tab; an offset value edit box with a default value will be available in the dashboard and in the **Sections** tab. Enter **65** in the offset value edit box for specifying the depth of second section. Next, choose the **Sketch** button from the **Sections** tab to enter the Sketcher environment.

6. Now, draw another arc of diameter **35**, as shown in Figure 15-119.

7. After drawing the second arc, choose the **OK** button to exit the Sketcher environment. Choose the **OK** button from dashboard to exit it. The model, similar to the one shown in Figure 15-120, is displayed in the drawing area.

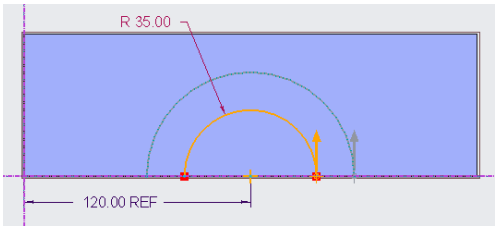


Figure 15-119 *Sketch of the blend surface*

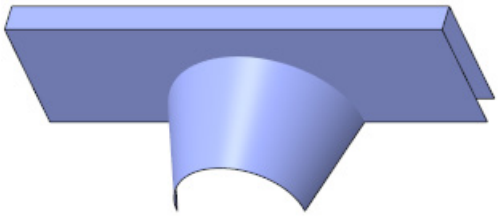


Figure 15-120 *Blend surface*

Mirroring the Blend Surface

Next, you need to mirror the blend surface about the **FRONT** datum plane.

1. Select the blend surface and then choose the **Mirror** tool from the **Editing** group; 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 15-121.

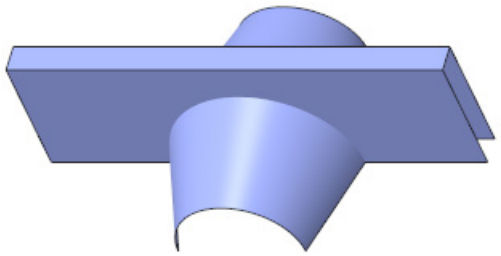


Figure 15-121 *Model after creating the mirror copy of the blend surface*

Creating the Cylindrical Surface

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

1. Choose the **Extrude** tool from the **Shapes** group to display the **Extrude** dashboard.

2. Choose the **Surface** button from the **Extrude** dashboard.

3. Select the **TOP** datum plane as the sketching plane.

4. After entering the Sketcher environment, draw a circle and dimension it, as shown in Figure 15-122.

5. Exit the Sketcher environment and extrude the sketch to some appropriate depth, refer to Figure 15-123.

The model after creating the surface extrusion is shown in Figure 15-123.

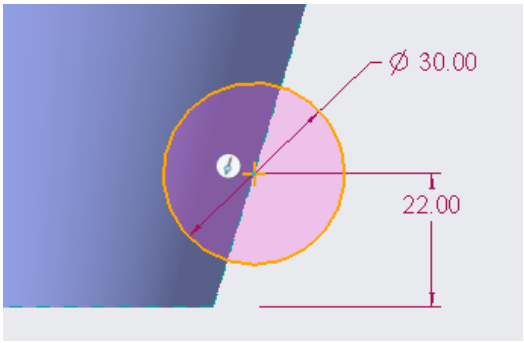


Figure 15-122 *Sketch of the cylindrical surface*

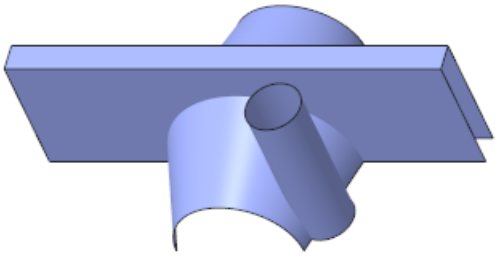


Figure 15-123 *Cylindrical surface*

Creating the Fill Surface

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

1. Choose the **Fill** tool from the **Surfaces** group of the **Model** tab in the **Ribbon**; 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.

4. Select the **TOP** datum plane and select the **Top** option from the **Orientation** drop-down list. Choose the **Sketch** button to enter the Sketcher environment.

5. Choose the **Project** button and select the edges of the base feature. Create a line to join the ends of base feature; the sketch is completed, as shown in Figure 15-124.

6. Exit the Sketcher environment and then exit the **Fill** dashboard by choosing the **OK** button. On doing so, the fill surface is created, as shown in Figure 15-125.

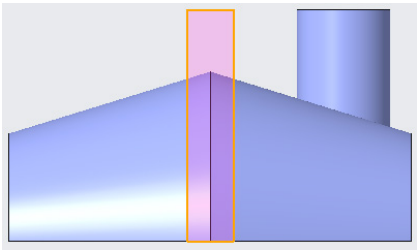


Figure 15-124 Sketch of the fill surface

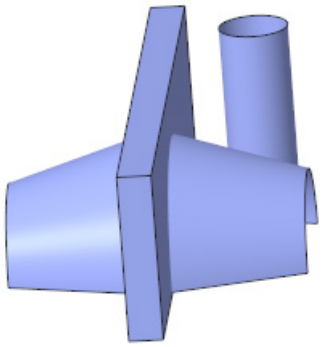


Figure 15-125 Model after creating the fill surface

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 mirror copy of the fill surface is created, the other end of the base feature also gets capped, as shown in Figure 15-126.

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 the blend surface by pressing the CTRL key.

2. Choose the **Merge** button from the **Editing** group; 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** or **Change side of second quilt to keep** button to change the direction of the pink arrow, if required.

4. Exit the **Merge** dashboard by choosing the **OK** button. The model after merging the two surfaces is shown in Figure 15-127.

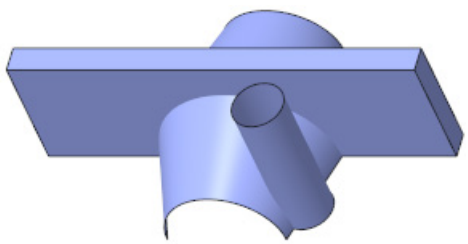


Figure 15-126 Model after creating the mirror copy of the fill surface

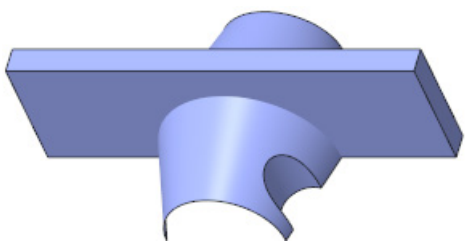


Figure 15-127 Model after merging the surfaces

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 **Editing** group; 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** and **Change side of second quilt to keep** buttons to change the direction of the pink arrow, if required.

4. You can see the preview of the blend surface by choosing the **Preview** button in the dashboard. Exit the **Merge** dashboard by choosing the **OK** button. The model after merging the two surfaces is shown in Figure 15-128.

5. Similarly, merge the mirrored feature and the base feature. The surface model after merging the mirrored surfaces is shown in Figure 15-129.

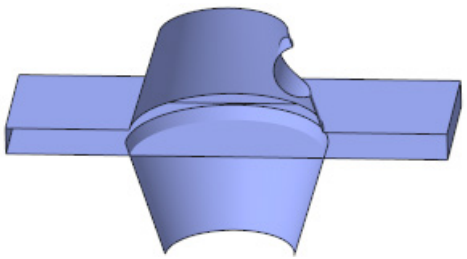


Figure 15-128 *Model after merging the blend surface with the base surface*

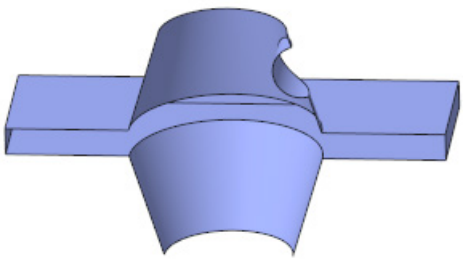


Figure 15-129 *Model after merging the mirror copy of the blend surface with the base surface*

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. After merging the surfaces, you need to round the edges.

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



Note

1. It is better to select surfaces to be merged from the ***Model Tree****.*

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

2. Choose the **Merge** button from the **Editing** group and then choose the **OK** button to merge both the surfaces.

3. Similarly, merge the mirror copy of the first fill surface with the base surface.

Creating Rounds

You need to create rounds on the cylindrical slot or edges where the two blend surfaces merge and on the edges of the base surface.

1. Choose the **Round** tool from the **Engineering** group. Select the edge of the cylindrical slot, refer to Figure 15-130; a preview of the round is highlighted on the selected edge.

2. Enter **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 having radius **18**.

   Among these edges, the two edges are those that are formed by merging the two blend surfaces with the base surface and the other two edges are the top corners of the base surface, refer to Figure 15-130.

5. Choose the **OK** button from the **Round** dashboard to create rounds. The surface model after creating the rounds is shown in Figure 15-131.

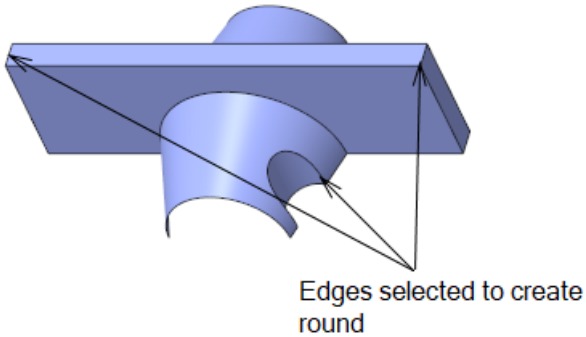


Figure 15-130 *Edges selected to create rounds*

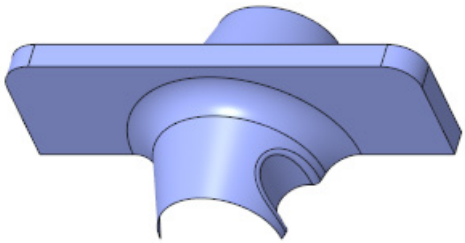


Figure 15-131 *Resulting model after creating rounds on the selected edges*

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.

1. Choose the **Round** tool from the **Engineering** group.

2. Select the two faces, front and back, of the base surface.

3. Invoke the slide-down panel by choosing the **Sets** tab. The selected surfaces are displayed in the **References** collector available in the slide-down panel. The **Full Round** button is chosen by default in the slide-down panel.

   Now, you need to select the surface to be removed.

4. Select the top face of a base surface; the preview of the round is highlighted on the selected surfaces. Exit the **Round** dashboard by choosing the **OK** button. The round is created, as shown in Figure 15-132.

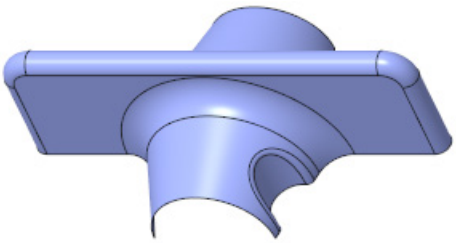


Figure 15-132 *Completed surface model*

5. Choose the **Save** button from the **File** menu and save the model.

Tutorial 3

In this tutorial, you will create the surface model of a table, as shown in Figure 15-133, using the Freestyle Modeling environment. **(Expected time: 45 min)**

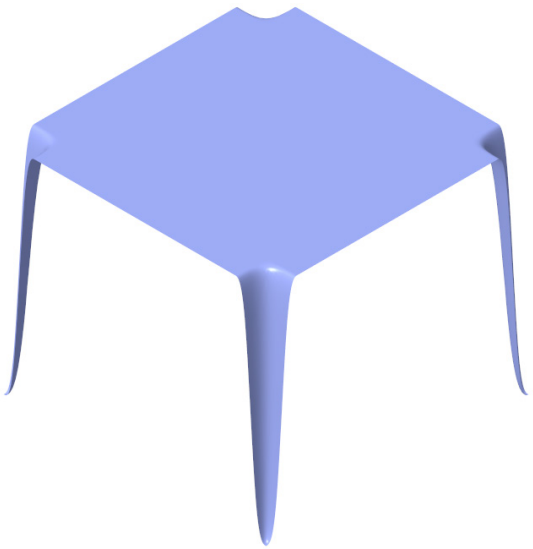


Figure 15-133 *The surface model of a table*

First, examine the model and determine the primitive to be used.

The following steps are required to complete this tutorial:

a. Create a new coordinate system.
b. Create the primitive.
c. Create the edge split feature.
d. Create the third feature which is transformation of faces and save the model.

Starting a New Object File

1. Start a new part file and name it *c15tut3*.

Creating a New Coordinate System

It is evident from the Figure 15-133, the model is created by using the square shape primitive on the **Top** plane. Therefore, you need to change the default sketching plane for the placement of primitive.

1. Choose the **Coordinate System** tool from the **Datum** group of the **Model** tab in the **Ribbon**; the **Coordinate System** dialog box is displayed.

2. Now, select the **TOP** plane from the **Model Tree** as a reference and the **FRONT** and **RIGHT** planes as offset references and then choose the **OK** button from the dialog box.

Starting the Freestyle Modeling Environment

To start with the Freestyle Modeling environment, you need to follow the step given next.

1. Choose the **Freestyle** tool from the **Surfaces** group in the **Model** tab of the **Ribbon** while working in the Modeling environment; the **Freestyle** dashboard is displayed and Freestyle Modeling environment is activated.

Changing the Sketching Plane

You need to change the sketching plane. To do so, a coordinate system is required.

1. Click on the down arrow in the **Operations** group in the **Freestyle** dashboard; a drop down list is displayed.

2. Choose the **Options** button from this drop-down list; the **Options** dialog box is displayed, as shown in Figure 15-134.

3. Select the new coordinate system created earlier and choose the **OK** button from the dialog box.

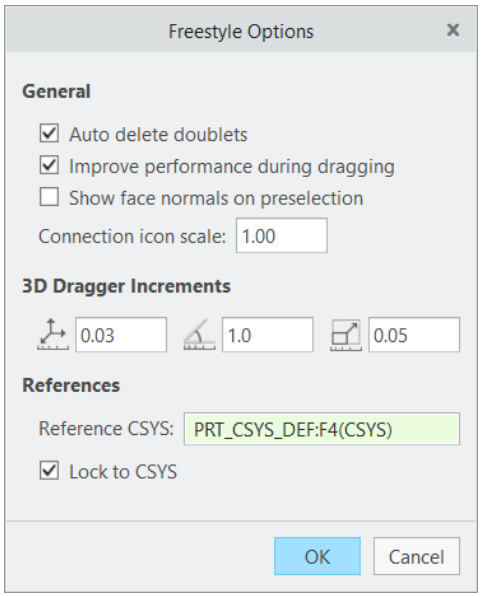


Figure 15-134 *The* ***Options*** *dialog box*

Creating the Primitive

After examining the model, you can guess that the basic primitive required for this model is a square surface.

1. Choose the **Choose a square initial shape** option from the **Open Primitives** area of the **Shapes** drop-down list; a square surface is created in the XY plane, as shown in Figure 15-135.

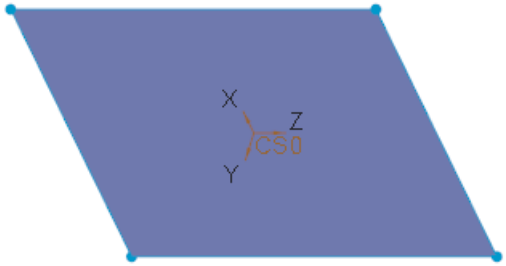


Figure 15-135 *Square surface created*

Creating the Edge Split Feature

To create the legs of the table, you need to split the edges of the surface.

1. Select any of the four edges and click on the down arrow next to the **Edge Split** button in the **Create** group of the dashboard; a drop-down list is displayed.

2. Select the **4 Splits** option from the drop-down list; the surface is displayed, as shown in Figure 15-136.

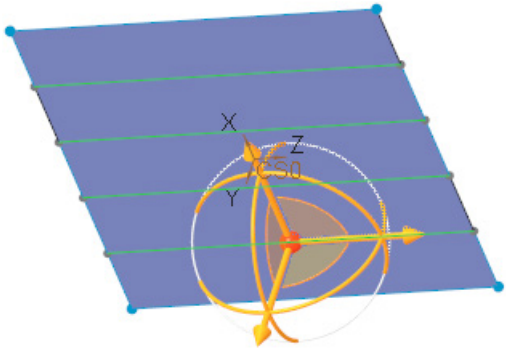


Figure 15-136 *Surface created after using the* ***4 Splits*** *option*

3. Select the two corner edges, as shown in Figure 15-137, and then choose the **4 Splits** option again; the surface is split, as shown in Figure 15-138.

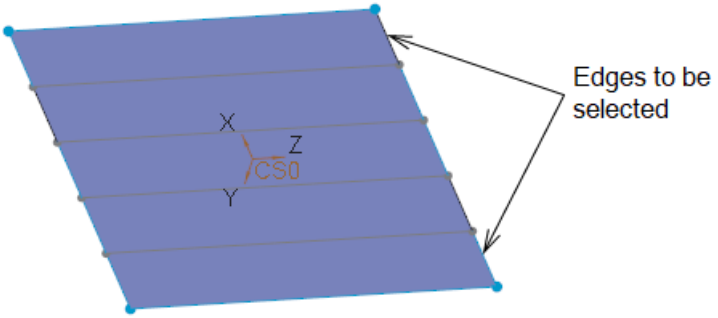


Figure 15-137 *Surface with corners edges to be selected*

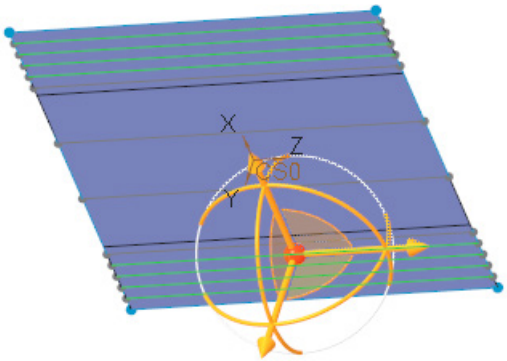


Figure 15-138 *Surface with corner edges split by using the* ***4 Split*** *option*

Similarly, split one edge perpendicular to the earlier selected edge so that the surface is displayed, as shown in Figure 15-139.

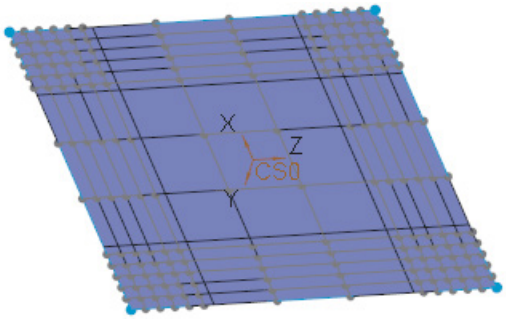


Figure 15-139 *Surface model after splitting is completed*

Transforming the Faces

Now, you need to transform the corner squares to create legs of the table.

1. Select the four small squares, as shown in Figure 15-140, and then drag the orange handle in z-direction such that the surface model looks like the one shown in Figure 15-141.

2. Choose the **OK** button and choose the **Save** button from the **File** menu to save the model.

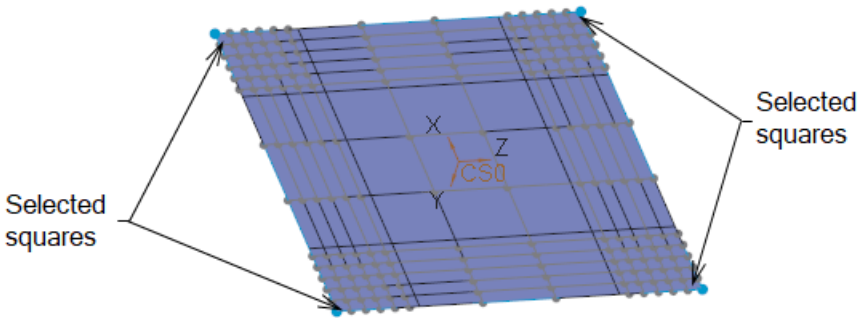


Figure 15-140 *Surface with the squares to be transformed*

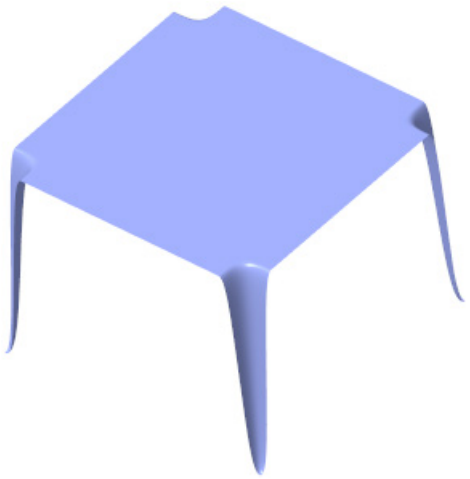


Figure 15-141 *Completed surface model*

Self-Evaluation Test

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

1. A feature created in the **Style** environment is displayed in the **Model Tree** as a _____ feature.

2. You can enter the **Style** environment by choosing _____ tool from the **Surface** group in the **Model** tab.

3. The _____ tool is used to merge two surfaces and form an edge.

4. In the **Style** environment, the _____ tool is used to draw curves.

5. A quilt is a _____ feature.

6. You can create a surface with capped ends by drawing an open sketch. (T/F)

7. Surface models have no thickness. (T/F)

8. Style features have the parent-child relationship among themselves as well as with the corresponding solid features created in the **Style** window. (T/F)

9. In the **Style** environment, when you press the SHIFT key and select a point on a surface by using the Free option, the point is snapped on that surface. (T/F)

10. The procedure to create a helical sweep surface is the same as that of a solid helical sweep feature. (T/F)

Review Questions

Answer the following questions:

1. Which of the following tools is used to create straight or smooth feature profile?

(a) **Sweep** (b) **Blend**
(c) **Extrude** (d) None of these

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 these

3. What is the minimum number of sections required for creating a blend feature?

(a) one (b) two
(c) three (d) None of these

4. Which of the following editing tools forms an edge between two intersecting surfaces?

(a) **Merge** (b) **Intersect**
(c) **Trim** (d) None of these

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 of these

6. The **Intersect** option is used to create an intersect curve. (T/F)

7. In the **Style** environment, the **Curve Edit** tool 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** tool is used to select at least three or four curves and create a surface. (T/F)

10. To undo the last operation performed in the **Style** environment, choose the **Undo** button from the **Quick Access** toolbar. (T/F)

EXERCISES

Exercise 1

In this exercise, you will create the surface model shown in Figure 15-142. The orthographic views of the surface model are shown in Figure 15-143. **(Expected time: 40 min)**

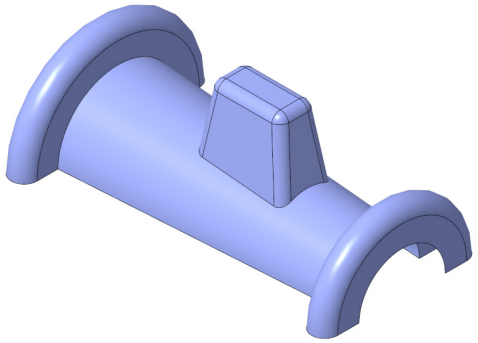
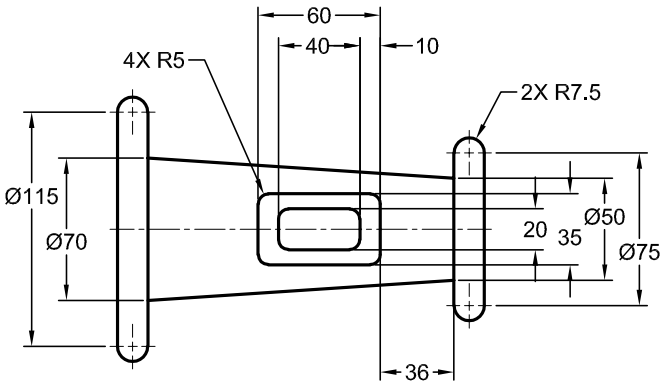


Figure 15-142 *Isometric view of the surface model*



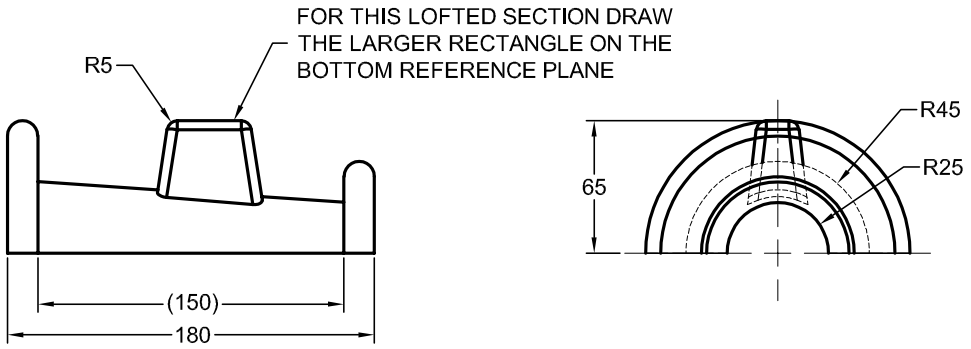


Figure 15-143 *Orthographic views of the surface model*

Exercise 2

In this exercise, you will create the surface model shown in Figure 15-144. The orthographic views and the detailed view of the surface model are shown in Figure 15-145.

(Expected time: 55 min)

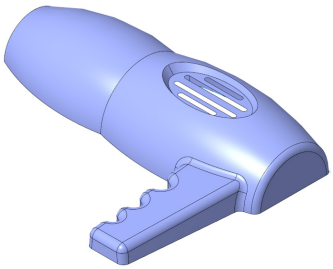


Figure 15-144 *Surface model*

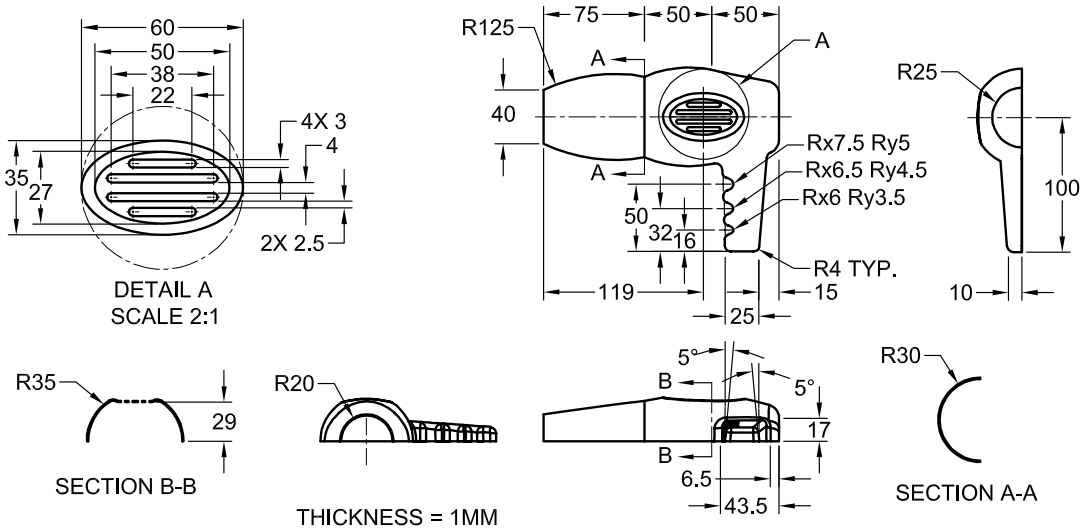


Figure 15-145 *Top, front, right, and detailed views of the surface model*

Exercise 3

In this exercise, you will create the surface model by using the sphere primitive shown in Figure 15-146 in the Freestyle modeling environment. **(Expected time: 55 min)**

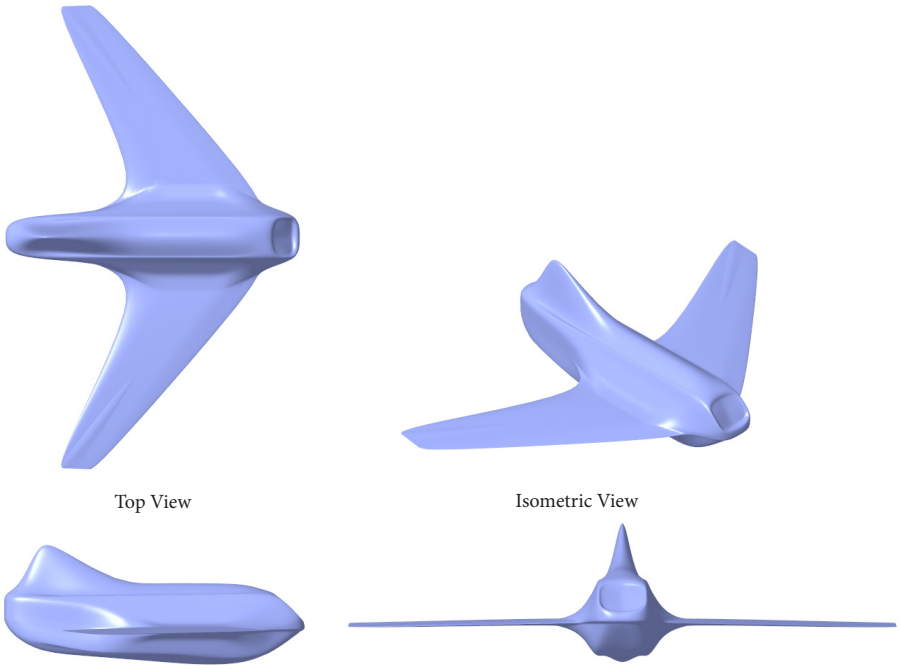


Figure 15-146 *Top, front, right, and isometric views of the surface model*

Answers to Self-Evaluation Test

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

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