Chapter 11
Advanced Surface Modeling

After completing this chapter, you will be able to:
• Create curves from bodies
• Create projected curves
• Create emboss sheet features
• Create face blend features
• Create fillet features
• Create bridge features

CREATING CURVES FROM BODIES

In NX, you can create various types of curves using the existing bodies. These curves are further used to create surface bodies. The methods to create different types of curves are discussed next.

Creating Intersection Curves

Ribbon: Home > Create > Curve Gallery > Intersection Curve
Menu: Insert > Derived Curve > Intersect

The **Intersection Curve** tool is used to create intersection curves between two sets of objects. The set of objects could be a solid, a sheet body, one or more faces, or a datum plane. To create the intersection curve, choose the **Intersection Curve** tool from the **Curve** gallery of the **Create** group in the **Home** tab, refer to Figure 11-1; the **Intersection Curve** dialog box will be displayed, as shown in Figure 11-2.

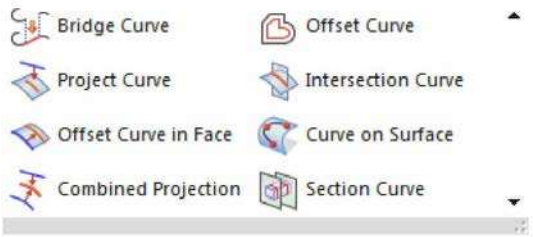


Figure 11-1 *Tools in the* ***Curve*** *gallery*

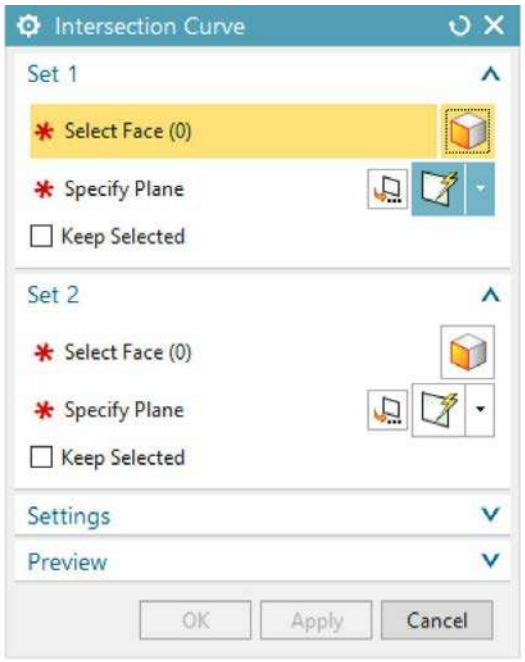


Figure 11-2 *The* ***Intersection Curve*** *dialog box*

Note

If the tools in the ***Curve*** *gallery are not visible by default, then you need to expand this gallery. To expand the* ***Curve*** *gallery, click on the down arrow available on its lower right corner.*

In this dialog box, by default, the **Face** button is chosen in the **Set 1** rollout. As a result, you are prompted to select the first set of faces to intersect. To select all the faces of the solid body, drag a box around it; all faces of the solid and sheet body will be selected, refer to Figure 11-3. Next, press the SHIFT key and select the sheet body to remove it from the selection and then release the SHIFT key. Choose the **Face** button from the **Set 2** rollout; you will be prompted to select the second set of faces to intersect. Select the sheet body, refer to Figure 11-3. The **Specify Plane** area that is available in both the **Set 1** and **Set 2** rollouts is used to create datum planes, which are selected as the first and second sets of intersection, respectively. However, after selecting the first set and second set of faces to intersect, this area will no longer will be available in the rollouts. Select the **Keep Selected** check box from the **Set 1** rollout to ensure that the first set of objects is automatically selected again to create the next intersection curve. Next, choose the **Apply** button. Similarly, the second set of objects can be selected automatically by

selecting the **Keep Selected** check box from the **Set 2** rollout.

You need to expand the **Settings** rollout for some additional options. The **Associative** check box in this rollout allows you to specify whether the intersection curve is associative or not. An associative intersection curve will update automatically when changes are made to its source objects.

An intersection curve resulting from the selections made in Figure 11-3 is shown in Figure 11-4.

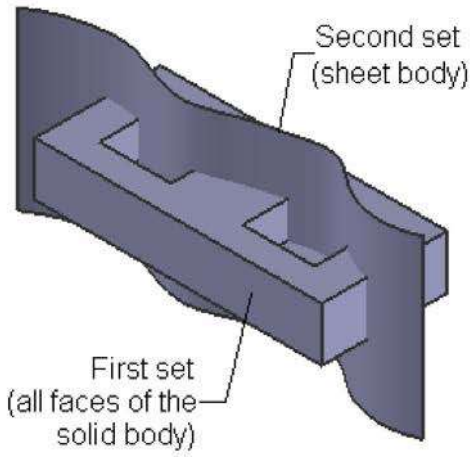


Figure 11-3 *Objects to be selected*

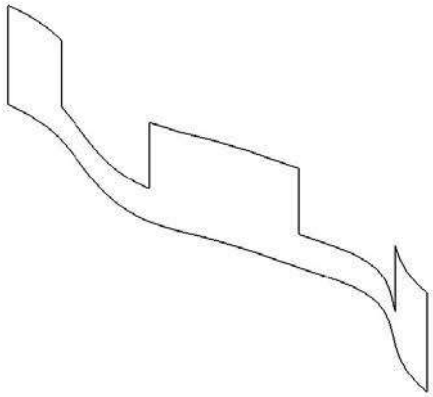


Figure 11-4 *Resulting intersection curve*

Creating Section Curves

Ribbon: Home > Create > Curve Gallery > Section Curve
Menu: Insert > Derived Curve > Section

The **Section Curve** tool is used to create the section curves between specified planes and solid bodies, surfaces, or curves. The output curve can be associative. The section curves can be created in four ways: by using **Selected Planes**, **Parallel Planes**, **Radial Planes**, and **Planes Perpendicular to Curve** methods. The selection steps for each method are different. The methods for creating section curves are discussed next.

Creating Section Curves by Using the Selected Planes Method

This method is used to create section curves by specifying solid or sheet bodies and one or more section planes. To create section curve by selecting planes, choose the **Section Curve** tool from the **Curve** gallery of the **Create**

group in the **Home** tab; the **Section Curve** dialog box will be displayed, as shown in Figure 11-5.

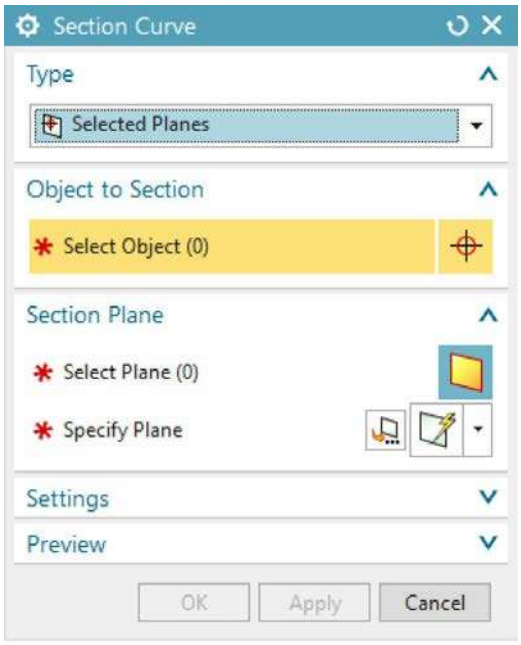


Figure 11-5 *The* ***Section Curve*** *dialog box*

By default, the **Selected Planes** option is selected inthe drop-down list in the **Type** rollout, and the **Object** button is chosen in the **Object to Section** rollout. As a result, you will be prompted to select the object to be sectioned. Select the solid body, refer to Figure 11-6. Choose the **Plane** button from the **Section Plane** rollout; you will be prompted to select the plane for section. Select the plane, refer to Figure 11-6. Expand the **Settings** rollout to use some additional options. By default, the **Associative** check box is selected in the **Settings** rollout. As a result, the section curve will be associative to the source object. Clear this check box; the **Non-associative Settings** sub-rollout will be displayed. In this sub-rollout, the **Group Objects** check box allows you to automatically group the output curves and the points that are created for each plane. In the **Non-associative Settings** sub-rollout, the **Output Sampled Points** check box is clear. If you select this check box, instead of curve, the points will be created in the resultant model. You can specify the distance between these points by using the **Sample Distance** edit box. The **Join Curves** drop-down list allows you to join a chain of curves to create a single B-spline curve. The resultant spline is either a polynomial cubic spline,

a general spline, or a polynomial quintic. After specifying the required parameters, choose the **OK** button from the **Section Curve** dialog box; the section curves will be created, refer to Figure 11-7.

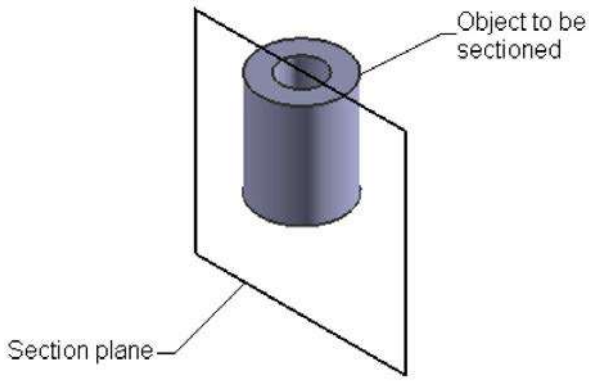


Figure 11-6 *Objects to be selected*

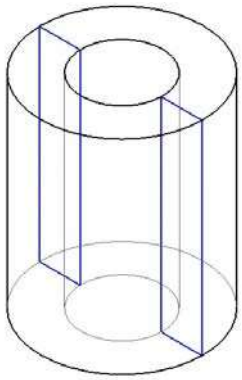


Figure 11-7 *Section curves created*

Creating Section Curves by Using the Parallel Planes Method

This method is used to create the section curves by specifying the base plane, step distance, start distance, and end distance. A series of parallel planes are spaced at the equal distances and are further used to create the section of the selected object. To create the section curves using this method, choose the **Section Curve** tool from the **Curve** gallery of the **Create** group in the **Home** tab; the **Section Curve** dialog box will be displayed. Select the **Parallel Planes** option from the drop-down list in the **Type** rollout; you will be prompted to select the objects to be sectioned. Select the solid body, refer to Figure 11-8.

Now, choose the **Inferred** button from the **Base Plane** rollout; you will be prompted to select the objects to define a plane. Select the plane, refer to Figure 11-8. Next, enter the **Start**, **End**, and **Step** distance values in their corresponding edit boxes. The step distance is the distance between two parallel planes. The start and the end distances are measured from the base plane. The software generates as many planes as possible between the start and end distances by maintaining a step distance between the consecutive planes.

Next, choose the **OK** button to create curves. The section curves created using the selections made in Figure 11-8 are shown in Figure 11-9.

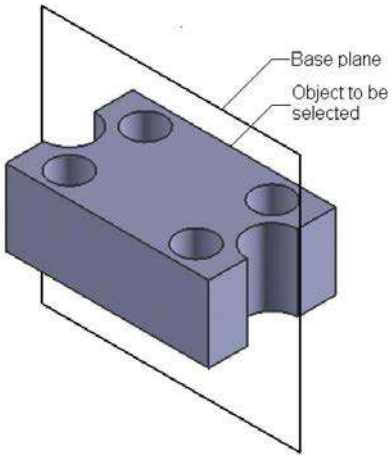


Figure 11-8 *Objects to be selected*

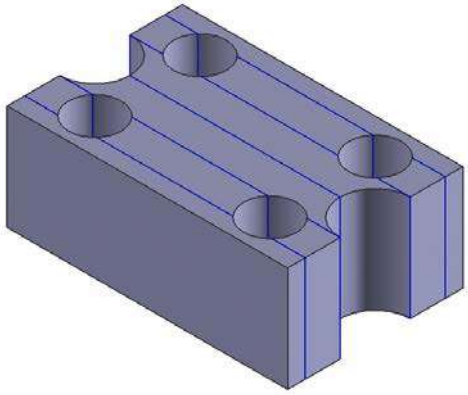


Figure 11-9 *Section curves created by using the Parallel Planes method*

Creating Section Curves by Using the Radial Planes Method

This method is used to specify the planes spaced at equal angles, which are further used to section the selected bodies. The planes are pivoted around a common axis. To create the section curves using this method, choose the **Section Curve** tool from the **Curve** gallery of the **Create** group in the **Home** tab; the **Section Curve** dialog box will be displayed. Select the **Radial Planes** option from the drop-down list in the **Type** rollout; you will be prompted to select the objects to be sectioned. Select the solid body, refer to Figure 11-10. Next, choose the **Inferred Vector** button from the **Radial Axis** rollout; you will be prompted to select objects to infer vector. Select the edge of the object, refer to Figure 11-10. Alternatively, you can use the **Vector Dialog** button to create a vector. Choose the **Inferred Point** button from the **Point on Reference Plane** rollout; you will be prompted to select the object to infer point. Select the radial reference point, refer to Figure 11-10. Next, enter the values for the **Start**, **End**, and **Step** angles in their corresponding edit boxes. The step angle is the angle between two radial planes. The start and end angles are measured from the base plane. The base plane passes through the radial axis and the point on the reference plane. The software generates as many planes as possible between the start and end angles by maintaining the step distance between the consecutive planes.

The section curves resulted from the selections made in Figure 11-10 are shown in Figure 11-11.

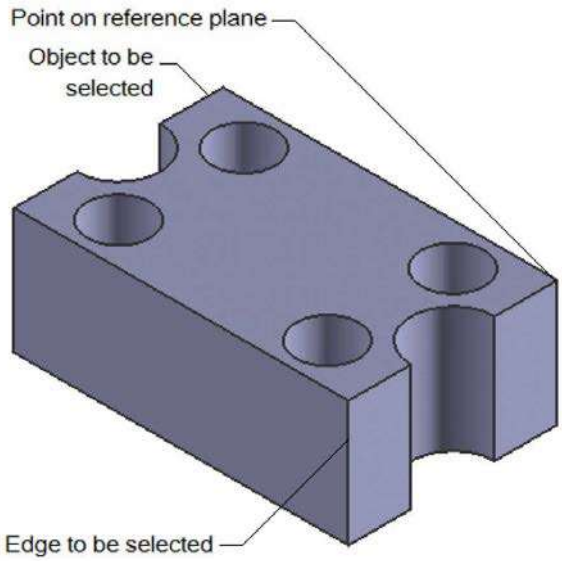


Figure 11-10 *Objects to be selected*

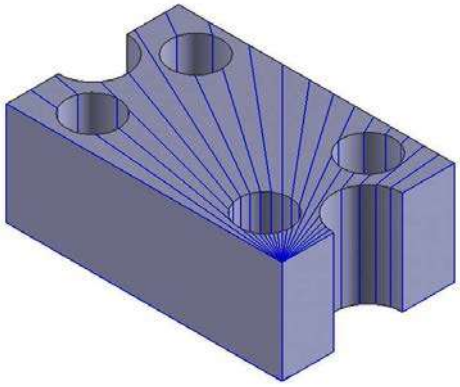


Figure 11-11 *Section curves created by using the Radial Planes method*

Creating Section Curves by Using the Planes Perpendicular to Curve Method

This method is used to create section curves along the planes perpendicular to the selected curve. You need to specify the solid or the sheet body, curve, and the spacing method. To create section curves using this method, choose the **Section Curve** tool from the **Curve** gallery of the **Create** group in the **Home** tab; the **Section Curve** dialog box will be displayed. Select the **Planes Perpendicular to Curve** option from the drop-down list in the **Type** rollout; you will be prompted to select the objects to be sectioned. Select the solid body, refer to Figure 11-12. Next, choose the **Curve or Edge** button from the **Curve or Edge** rollout; you will be prompted to select the curve or the edge. Select the curve along which the perpendicular planes will be created, refer to Figure 11-12.

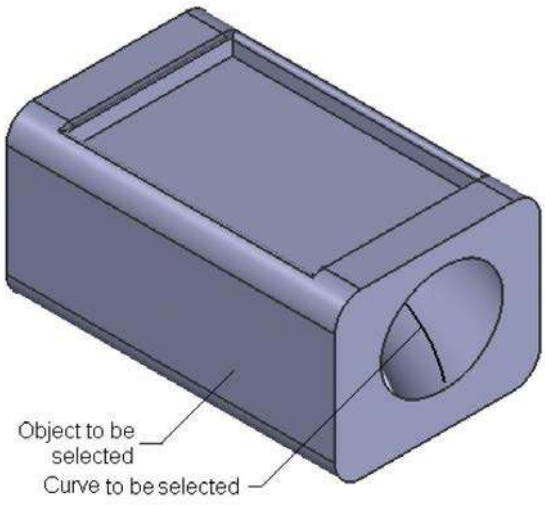


Figure 11-12 *Objects to be selected*

Next, you need to select the spacing method from the **Spacing** drop-down list in the **Plane Location** rollout. The section planes will be placed perpendicular to the curve. The distance between the planes will be determined by the spacing method. You can use one of the following spacing methods:

Equal Arc Length

This option allows you to create sections using the planes at equal arc lengths along the curve. You need to enter the values for the number of section planes on the curve. Similarly, enter the start and end percentage values relative to the arc length of the curve.

Equal Parameters

This option allows you to create sections using the planes based on the parameterization of the curve. You need to enter the values for the number of section planes on the curve. Similarly, enter the start and the end percentage values relative to the arc length of the curve.

Geometric Progression

This option allows you to create sections using the planes based on a geometric ratio. You need to enter the values for the number of section planes on the curve as well as the start and the end percentage values relative to the arc length of the curve. Similarly, enter a value in the **Ratio**

edit box to determine the mathematical ratio for spacing the planes between the start and end percentage points.

Chordal Tolerance
This option allows you to create sections using the planes based on a chordal tolerance. In this case, you need to enter the value for the chordal tolerance.

Incremental Arc Length
This option allows you to create sections using the planes placed at increments along the curve. In this case, you need to enter the value for the arc length.

Select the **Equal Arc Length** option from the **Spacing** drop-down list of the **Plane Location** rollout. Next, enter the values for the number of copies, start percentage, and end percentage in their corresponding edit boxes. Choose the **OK** button from the dialog box. The section curves are created, as shown in Figure 11-13.

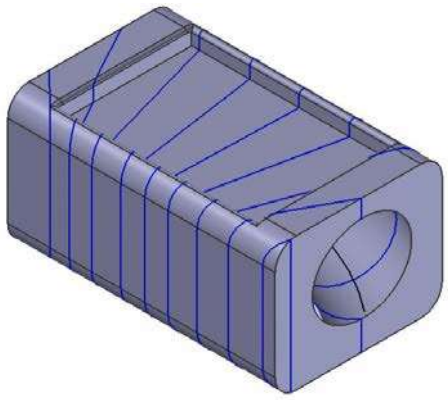


Figure 11-13 *Section curves created by using the Planes Perpendicular to Curve method*

Creating Isoparametric Curves

Ribbon: Curve > Derived Curve > Derived Curve Gallery > Isoparametric Curve *(Customize to add)*
Menu: Insert > Derived Curve > Isoparametric Curve

The isoparametric curves are created along the U/V parameters on a face. To create isoparametric curves, choose the **Isoparametric Curve** tool from

Menu > Insert > Derived Curve in the **Top Border Bar**; the **Isoparametric Curve** dialog box will be displayed, as shown in Figure 11-14. In this dialog box, by default, the **Face** button is chosen in the **Face** rollout. As a result, you will be prompted to select a face of the model. Select the face of the model to create isoparametric curves on it. Next, you need to specify the parameters using the options in the **Iso Curve** rollout. The options in this rollout are discussed next.

Direction

The options in this drop-down list are used to specify the direction of the curves. You can specify the direction of the curves using the **U**, **V**, and **U and V** options.

Location

The options in this drop-down list are used to define the location of the curves and these are discussed next.

Uniform

This option is selected by default. As a result, the curves are created uniformly on the selected surface. Figure 11-15 shows curves created uniformly along the U and V directions.

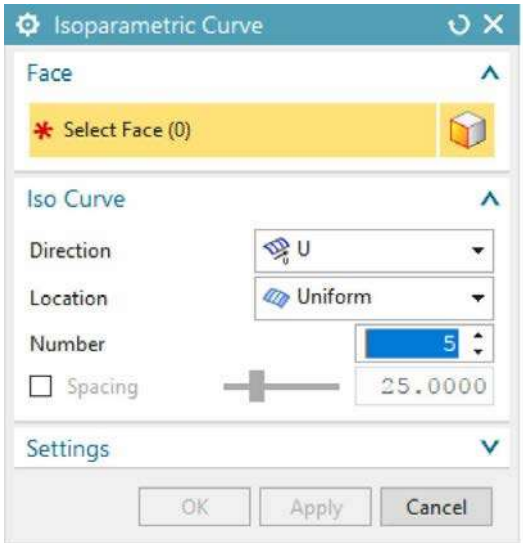


Figure 11-14 *The* ***Isoparametric Curve*** *dialog box*

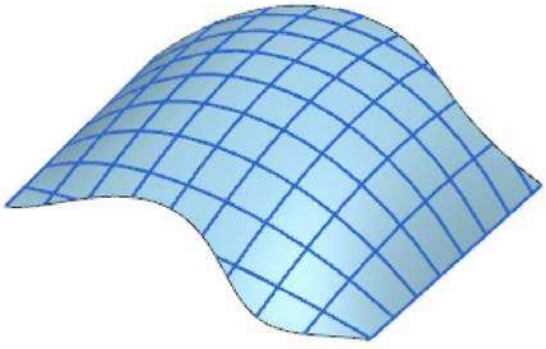


Figure 11-15 *Curves created uniformly along the U and V directions*

Through Points

On selecting this option, the **Specify Point** area is activated in the **Iso Curve** rollout. As a result, you will be prompted to select points to insert iso curves. Specify the points on the selected face; the curves will be created passing through the specified points, as shown in Figure 11-16.

Between Points

This option is used to create curves between two specified points on the face of a model. Select both the points on the face and then enter the number of curves to be created in the **Number** edit box; the specified number of curves will be created between the selected points, as shown in Figure 11-17.

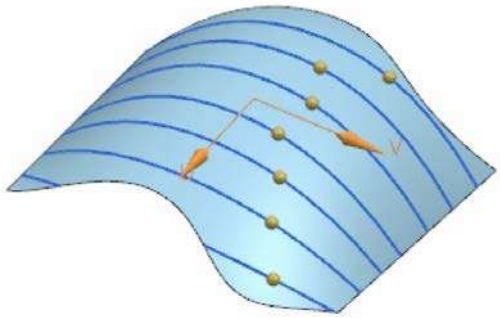


Figure 11-16 *Curves passing through specified points*

Figure 11-17 *Curves created between two points*

Projecting Curves

Ribbon: Curve > Derived Curve > Project Curve
Menu: Insert > Derived Curve > Project

You can project a closed or an open curve onone or more than one planar or curved faces. To do so, first create a feature and then draw a curve. Note that the curve and the feature should not be on the same plane. Next, choose the **Project Curve** tool from the **Derived Curve** group of the **Curve** tab; the **Project Curve** dialog box will be displayed, as shown in Figure 11-18. Select the curve from the drawing area and press the middle mouse button; the **Select Object** area in the dialog box will be activated and you will be prompted to select a face, facet body, or a datum plane onto which the curve is to be projected. Select the face on which you want the curve to be projected and then press the middle mouse button; the **Specify Vector** area will be activated in the **Projection Direction** rollout as the **Along Vector** option is

selected by default in the **Direction** drop-down list. Also, you will be prompted to select the objects to infer vector and a vector triad will be displayed in the drawing area. Next, click on the required handle of the triad to specify the vector direction. You can also reverse the direction of the specified vector by using the **Reverse Direction** button available in the **Projection Direction** rollout. Next, choose the **OK** button from the dialog box; the selected curve will be projected on the specified face. Figure 11-19 shows the curve and the face to be selected. Figure 11-20 shows the resulting projected curve. The rollouts available in the **Project Curve** dialog box are discussed next.

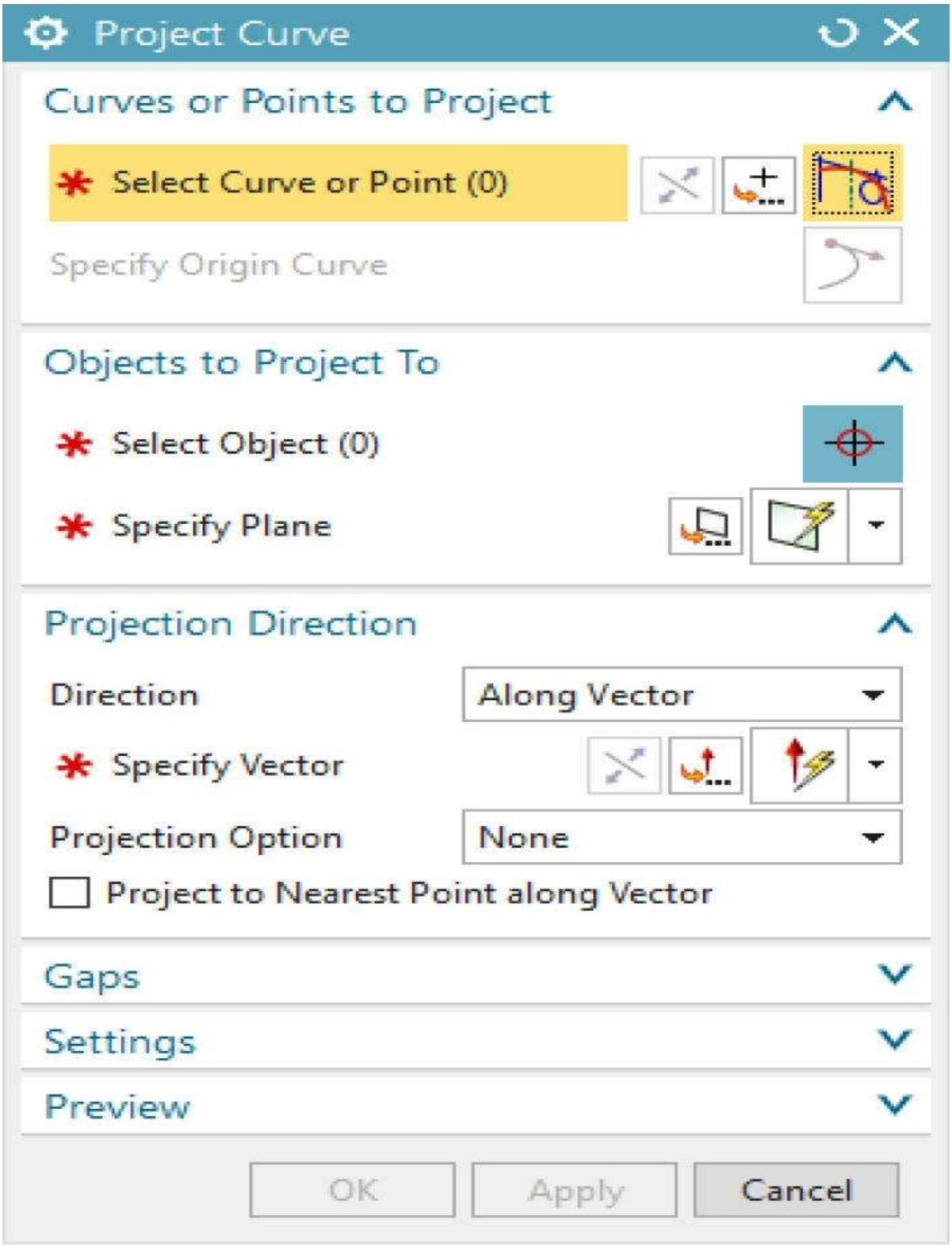


Figure 11-18 *The* ***Project Curve*** *dialog box*

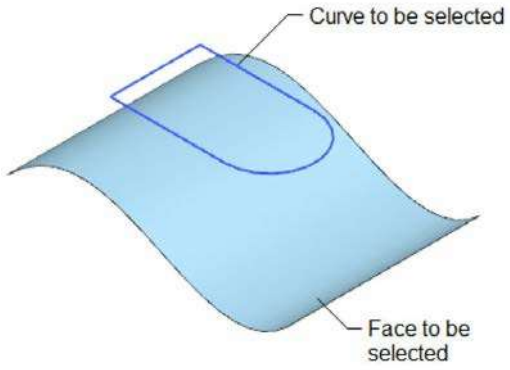


Figure 11-19 Curve and face to be selected

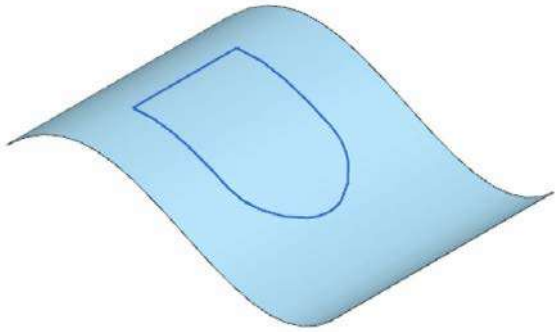


Figure 11-20 Resulting projected curve

Objects to Project To Rollout

The options in this rollout are used to select the objects onto which the selected curves will be projected. If you want to project the curve onto a plane, click in the **Specify Plane** area in the dialog box; you will be prompted to select objects to define a plane. You can also create a plane by choosing the **Plane Dialog** button in the **Specify Plane** area.

Projection Direction Rollout

The **Direction** drop-down list in this rollout contains the options to define the direction of curves to be projected and are discussed next.

Along Face Normal

On selecting this option, the selected curve will be projected along a direction normal to the selected face.

Toward Point

On selecting this option, you are prompted to select a point toward which the curve will be projected. Select a point from the graphics window, refer to Figure 11-21; the curve will be projected toward the selected point.

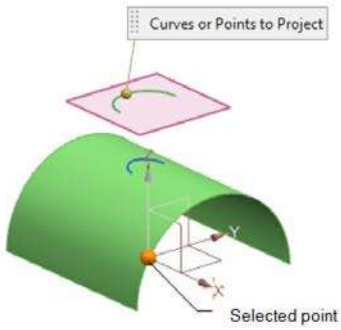


Figure 11-21 *Curve projected toward the selected point*

Toward Line

On selecting this option, you are prompted to select a line. Select a line from the drawing window, refer to Figure 11-22; the curve will be projected toward the selected line along the vector perpendicular to it.

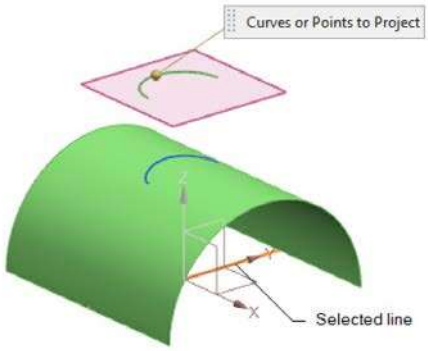


Figure 11-22 *Curve projected along the vector perpendicular to the selected line*

Along Vector

This option is used to project a curve along the selected vector, refer to Figure 11-23.

Angle to Vector

On selecting this option, you need to specify a vector using the vector triad displayed. Next, enter an angle value in the **Angle to Vector** edit box. As a result, the curve will be projected at an angle to the selected vector, as shown in Figure 11-24.

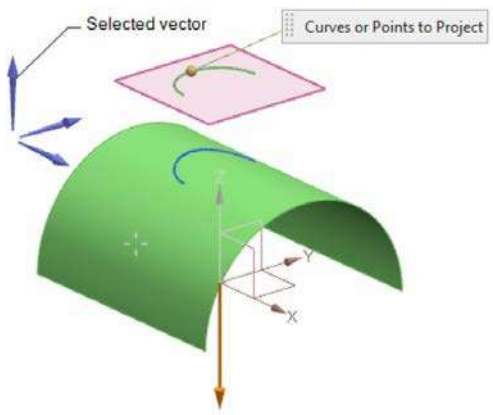


Figure 11-23 Curve projected along the selected vector

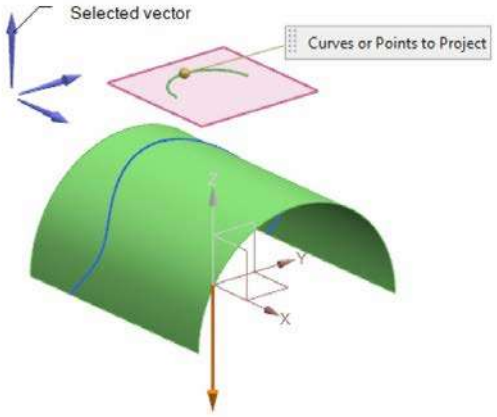


Figure 11-24 Curve projected at a specified angle to the specified vector

Project to Nearest Point along Vector

This check box is available only when you select the **Along Vector** option in the **Direction** drop-down list. If you select this check box, the curve will be projected to the surface, which is nearest to the input curve. Figures 11-25 and 11-26 show the curves projected with the **Project to Nearest Point along Vector** cleared and selected, respectively.

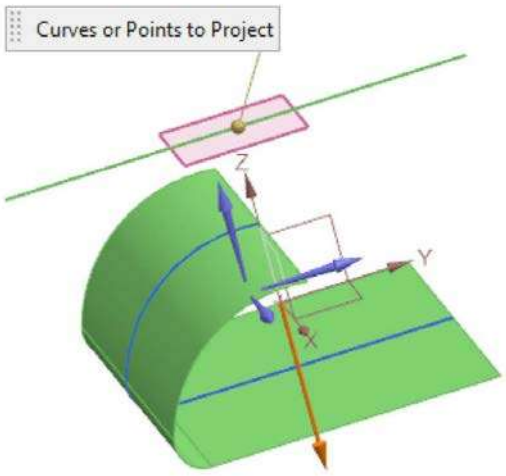


*Figure 11-25 Projected curve with **Project to Nearest Point along Vector** cleared*

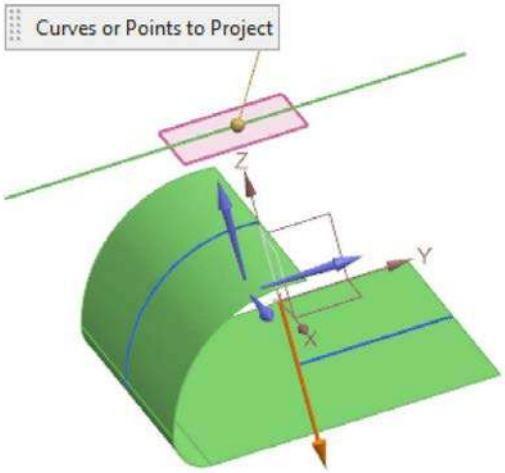


*Figure 11-26 Projected curve with **Project to Nearest Point along Vector** selected*

Gaps Rollout

The options in this rollout are used to bridge the gaps between two segments of a projected curve. To bridge the gap, select the **Create Curves to Bridge Gaps** check box; the **Maximum Bridged Gap Size** edit box will be activated. Next, enter a value in this edit box; the gaps between the curves

will be bridged. Note that the input value should be greater than the gap length. The gap lengths of the individual gaps are displayed in the **Gap List** table. Figures 11-27(a) and 11-27(b) shows the projected curve with the **Create Curves to Bridge Gaps** check box cleared and selected, respectively.

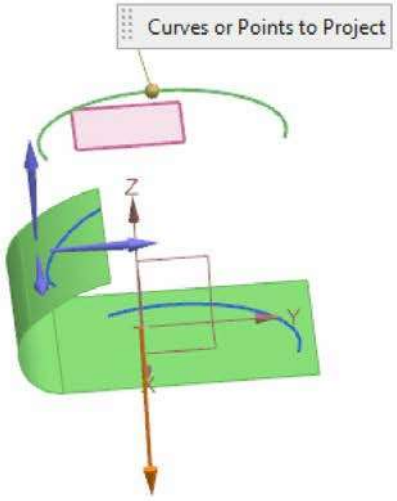


Figure 11-27(a) *Projected curve with the* ***Create Curves to Bridge Gaps*** *check box cleared*

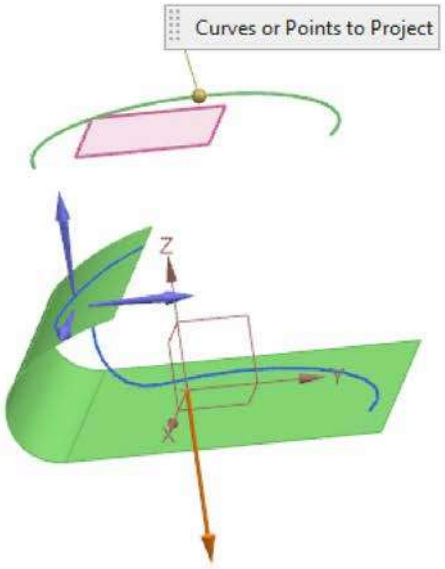


Figure 11-27(b) *Projected curve with the* ***Create Curves to Bridge Gaps*** *check box selected*

Settings Rollout

The **Associative** check box in this rollout is used to specify whether the projected curve is associative or not. Note that an associative projected curve is updated automatically when changes are made to its input curve. The options in the **Input Curves** drop-down list are used to keep or hide the input curves.

ADVANCED SURFACE MODELING TOOLS

These tools are used to create the basic and advanced surfaces and are discussed next.

Creating Emboss Body on a Sheet or Solid Body

Ribbon: Feature > Feature > More Gallery > Combine > Emboss Body

(Customize to Add) Menu: Insert > Combine > Emboss Body

The **Emboss Body** tool is used to emboss the shape of one solid body onto another solid or sheet body. To create the emboss body on a sheet, choose **Menu > Insert > Combine > Emboss Body** from the **Top Border Bar**; the **Emboss Body** dialog box will be displayed, as shown in Figure 11-28.

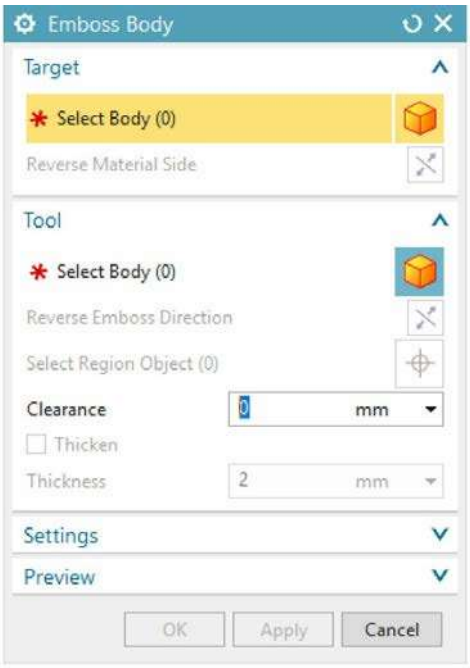


***Figure 11-28** The **Emboss Body** dialog box*

By default, the **Target** button is chosen in the **Target** rollout of this dialog box. As a result, you will be prompted to select the target body. Select the sheet as the target body, as shown in Figure 11-29; the **Tool** button in the **Tool** rollout will be activated and you will be prompted to select the tool bodies. Select a solid body from the drawing window as the tool body, as shown in Figure 11-29.

By default, the **Keep Target** and **Keep Tool** check boxes are clear in the **Settings** rollout. If you select the **Keep Target** check box, the target body will not be discarded from the resultant model. If you select the **Keep Tool** check box, the tool body will not be discarded from the resultant model.

Choose the **Show Result** button from the **Preview** rollout to preview the resultant model. Next, choose the **OK** button from the **Emboss Body** dialog

box; the emboss body feature will be created. Figure 11-30 shows the emboss sheet body created by embossing the solid body. Note that the side of tool body is reversed in order to get this result.

Figure 11-29 Objects to be selected

Figure 11-30 Resulting emboss body feature

Creating Face Blend Features

Ribbon: Home > Create > Blend Gallery > Face Blend
Menu: Insert > Detail Feature > Face Blend

The **Face Blend** tool is used to create complicated blends tangent to a specified set of faces. A face blend can be created between the faces of the solid or sheet bodies. The wall faces of the blend can be trimmed automatically. To create the face blend feature, choose the **Face Blend** tool from **Menu > Insert > Detail Feature** in the **Top Border Bar**; the **Face Blend** dialog box will be displayed, as shown in Figure 11-31. The options in various rollouts of this dialog box are discussed next.

Type Rollout

The drop-down list in this rollout is used to select the blend type. The face blends are classified into two types based on the number of faces to be selected. The options in this drop-down list are discussed next.

Two-face

This option is selected by default and is used to create a face blend between two intersecting faces.

Three-face

This option is used to create a face blend between three intersecting faces.

Faces Rollout

This rollout is used to select set of faces for blending. There are two or three **Face** buttons available in this rollout depending upon the option selected in the drop-down list of the **Type** rollout. When the **Two-face** option is selected from the drop-down list in the **Type** rollout, two **Face** buttons will be available in the **Faces** rollout. The first **Face** button allows you to select the first set of faces for blending. You can also select edges, instead of faces. After selecting the faces, a vector will be displayed. This vector should point toward the center of the blend. Choose the **Reverse Direction** button to reverse the direction of the vector. The second **Face** button allows you to choose the second set of faces for blending. After selecting the second set of faces, again a vector will be displayed which should point toward the center of the blend.

On selecting the **Three-face** option from the drop-down list in the **Type** rollout, the third **Face** button becomes available in the **Faces** rollout. After selecting the first face and the second face, the third **Face** button will be activated and you will be prompted to select middle faces to blend. Select the middle face and then press the middle mouse button to exit the dialog box. Figure 11-32 shows a preview of the face blend created using the **Three-face** option.

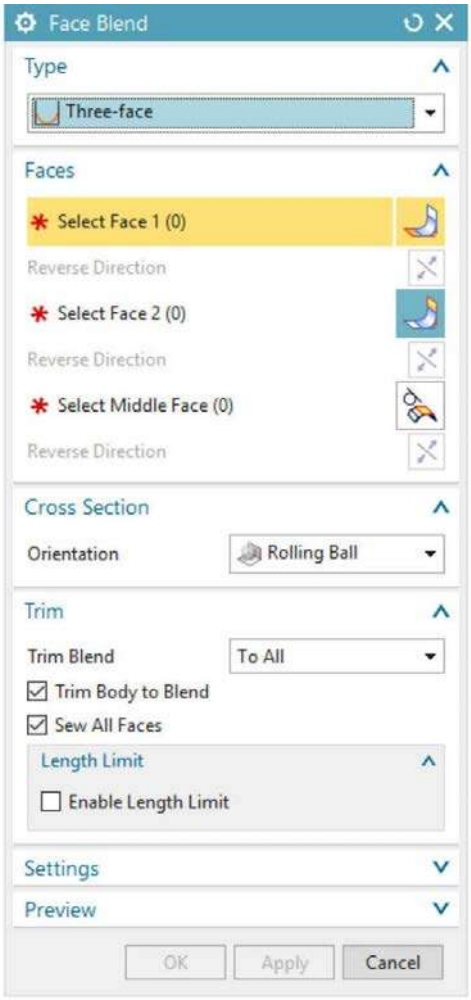


*Figure 11-31 The **Face Blend** dialog box*

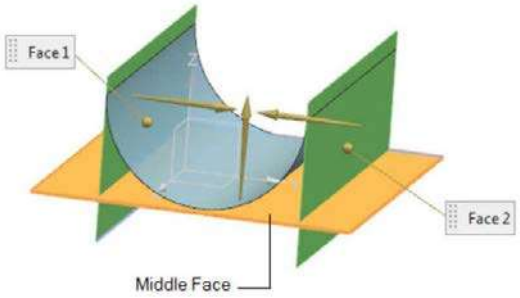


*Figure 11-32 Preview of the face blend created using the **Three-face** option*

Cross Section Rollout

This rollout is used to specify the cross-section radius of bend and its orientation. The options in this rollout are discussed next.

Orientation

The options in this drop-down list are used to define the orientation of the cross-section and are discussed next.

Rolling Ball: Using this option, you can create a face blend as if it was rolled by a ball which is in constant contact with the selected faces.

Swept Disc: This option is used to create a face blend whose surface is controlled by a tangent curve. This tangent curve is swept along a spine

curve.

Width Method

The options in this drop-down list are used to control the width of the face blend.

Shape

On selecting the **Circular** option from this drop-down list, the following options will be available in the **Radius Method** drop-down list to control the cross-section:

Constant: On selecting this option, you can enter the values for the constant radius blends in the **Radius** edit box.

Variable: This option is used to define variable radii at two or more individual points along the spine curve based on the law type selected in the **Law Type** drop-down list.

Limit Curve: This option is used to set the blend radius based on the curves or edges selected for creating the face blend. On selecting this option from the **Radius Method** drop-down list, the **Select Sharp Limit Curve** area will be displayed in the **Width Limits** rollout and you will be prompted to select the sharp limit curve. Note that the curve must lie on one of the faces selected.

On selecting the **Tangent Symmetric** option from the **Shape** drop-down list, the following options will be available to control the cross-section:

Conic Method: By default, the **Boundary and Center** option is selected in the **Conic Method** drop-down list. As a result, you can define the shape of a cone by specifying its boundary and center. Alternatively, you can select the **Boundary and Rho** option or the **Center and Rho** option from this drop-down list to define the conic shape.

Boundary Method: By default, the **Constant** option is selected in the **Boundary Method** drop-down list. As a result, you can define a

constant radius value of the boundary. Alternatively, you can select the **Law Controlled** option from this drop-down list to define the law to specify the boundary.

Boundary Radius: You can specify the radius value of the boundary by using this edit box. Note that this edit box is available only when you select the **Constant** option from the **Boundary Method** drop-down list.

Center Method: By default, the **Constant** option is selected in the **Center Method** drop-down list. As a result, you can define the constant center radius of the symmetric conic. Alternatively, you can select the **Law Controlled** option from this drop-down list to define the law to specify the center radius.

Center Radius: You can specify the center radius value of the symmetric conic by using this edit box.

On selecting the **Tangent Asymmetric** option from the **Shape** drop-down list, the following options will be available to control the cross-section:

Offset 1 Method: By default, the **Constant** option is selected in the **Offset 1 Method** drop-down list. As a result, you can define a constant offset value for the first face selected. Alternatively, you can select the **Law Controlled** option from this drop-down list to define the law.

Offset 1 Distance: This edit box is used to set the distance of the conic offset from the first face.

Offset 2 Method: By default, the **Constant** option is selected in the **Offset 2 Method** drop-down list. As a result, you can define a constant offset value for the second face selected. Alternatively, you can select the **Law Controlled** option from this drop-down list to define the law.

Offset 2 Distance: This edit box is used to set the distance of the conic offset from the second face.

Rho Method: The options available in the **Rho Method** drop-down list

are used to specify the rho method for the conic cross-sections.

On selecting the **Curvature Symmetric** option from the **Shape** drop-down list, the **Curve** button will be activated in the **Cross Section** rollout and the following options will be available to control the cross-section:

Boundary Method: The options available in this drop-down list are same as discussed in the **Tangent Symmetric** option.

Boundary Radius: The options available in this drop-down list are same as discussed in the **Tangent Symmetric** option.

Depth Law Type: The options available in this drop-down list are used to define the depth of blend.

Depth: In this edit box, you can specify the value of depth according to the type of selection made from **Depth Law Type** drop-down list.

On selecting the **Curvature Asymmetric** option from the **Shape** drop-down list, the **Curve** button will be activated in the **Cross Section** rollout and the following additional options will be available to control the cross-section:

Shape Skew Law Type: The options available in this drop-down list are used to specify the skew type of the blend.

Shape Skew: In this edit box, you can specify the value for skew.

The remaining options in the **Cross Section** rollout are same as discussed in previous sections.

Width Limits Rollout

This rollout will be available on selecting the **Two-face** option from the drop-down list in the **Type** rollout. This rollout is used to select the coincident edges and tangency control objects for the blend. The options in this rollout are discussed next.

Select Sharp Limit Curve

The **Curve** button in this area is used to specify the curve with which the blend remains coincident when the blend is large enough to encounter it.

Location

The options available in the **Location** drop-down list are used to specify whether the constraining curve is on the first face chain or the second face chain.

Select Tangent Limit Curve

The **Curve** button in this area is used to control the radius of the sphere, or an offset of the conic by maintaining a tangency between the blend surface and an underlying face set along a specified curve or edge.

Location

The options available in the **Location** drop-down list are used to specify whether the constraining curve is on the first face chain or the second face chain.

Trim Rollout

The options in this rollout are used to specify the trim and sew conditions for the blend. By default, the **To All** option is selected in the **Trim Blend** drop-down list.

The following steps explain the procedure to create a rolling ball face blend:

1. Choose the **Face Blend** tool from **Menu > Insert > Detail Feature** of the **Top Border Bar**; the **Face Blend** dialog box will be displayed. By default, the **Two-face** option is selected in the drop-down list in the **Type** rollout and the **Face** button is chosen from the **Faces** rollout. Therefore, you will be prompted to select the faces or edges to blend.

2. Select the first chain face, refer to Figure 11-33 and make sure that the normal vector points in the upward direction.

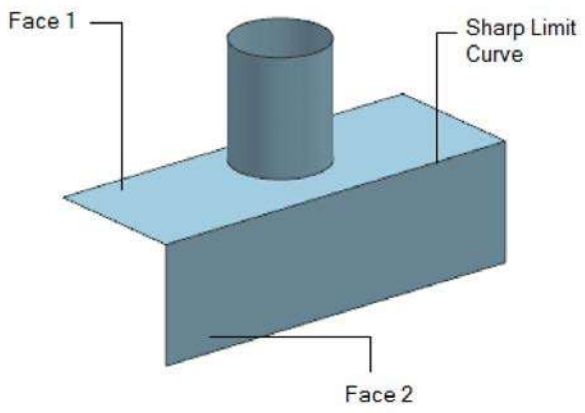


Figure 11-33 *Faces to be selected*

3. Choose the second **Face** button from the **Faces** rollout; you will be prompted to select the faces to blend. Select the second chain face, refer to Figure 11-33.

4. Choose the **Curve** button from the **Width Limits** rollout; you will be prompted to select the sharp limit curve. Select the limit curve, refer to Figure 11-33.

5. Enter the value of the radius in the **Radius** edit box of the **Cross Section** rollout.

6. Select the **To All** option from the **Trim Blend** drop-down list in the **Trim** rollout and choose the **OK** button. The resulting face blend feature is shown in Figure 11-34.

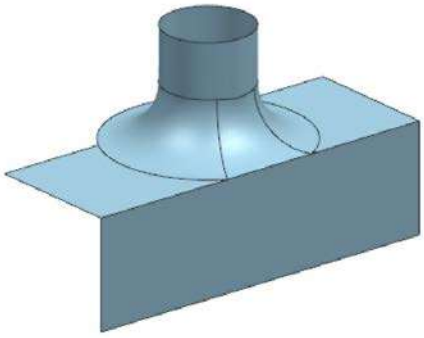


Figure 11-34 *Resulting face blend feature*

The following steps explain the procedure of creating a face blend:

1. Choose the **Face Blend** tool from **Menu > Insert > Detail Feature** in the **Top Border Bar**; the **Face Blend** dialog box will be displayed. Also, you will be prompted to select faces or edges to blend and the **Face** button in the **Select Face 1** area will be activated.

2. Select the first face set, refer to Figure 11-35. Choose the **Reverse Direction** button, if needed.

3. Choose the **Face** button in the **Select Face 2** area and then select the second face set, refer to Figure 11-35. Choose the **Reverse Direction** button, if needed.

4. Select the **Contact Curve** option from the **Width Method** drop-down list in the **Cross Section** rollout; the **Select Contact Curve 1** area will be activated. The **Contact Curve** option helps to maintain tangency between blends to be created and the surface selected as face chain1 and face chain 2.

5. Select the first contact curve, refer to Figure 11-35.

6. Click in the **Select Contact Curve 2** area and select the second contact curve, refer to Figure 11-35.

7. Select the **Curvature Asymmetric** option from the **Shape** drop-down list in the **Cross Section** rollout.

8. Click in the **Select Spine Curve** area and select the Spine string, refer to Figure 11-35.

9. Choose the **OK** button from the **Face Blend** dialog box to create the blend, refer to Figure 11-36.

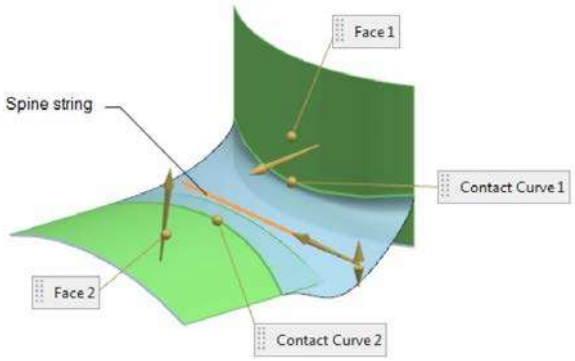


Figure 11-35 Objects to be selected

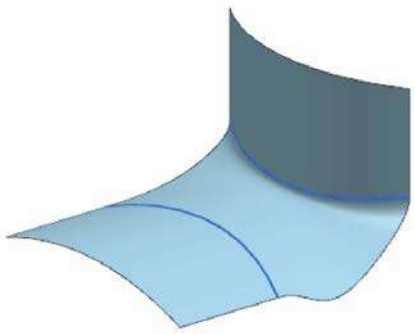


Figure 11-36 Resulting face blend feature

Creating Bridge Features

Ribbon: Home > Create > Blend Gallery > Bridge
Menu: Insert > Detail Feature > Bridge

The **Bridge** tool is used to create the bridge surface that joins the edges of the surfaces. You can specify a tangent or a curvature continuity between the bridge surface and the defining faces.

The following steps are required to create the bridge feature:

1. Choose **Menu > Insert > Detail Feature > Bridge** from the **Top Border Bar**; the **Bridge Surface** dialog box will be displayed, as shown in Figure 11-37.

2. By default, the **Select Edge 1** area is activated in the **Edges** rollout and you will be prompted to select a face near an edge or an edge. Select the first edge and the second edge, refer to Figure 11-38.

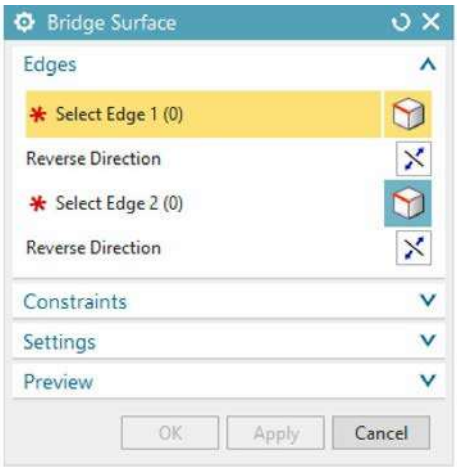


Figure 11-37 The Bridge Surface dialog box

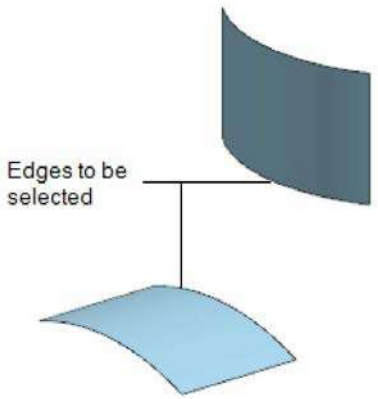


Figure 11-38 Edges to be selected

Note that you may need to use the **Reverse Direction** button, if needed.

3. Next, you need to choose the continuity type from the **Continuity** sub-rollout in the **Constraints** rollout. Choose the continuity type for the first edge and the second edge from the **Edge 1** and **Edge 2** drop-down lists. You can specify **G0 (Position)**, **G1 (Tangent)**, or **G2 (Curvature)** continuity between the selected faces and the bridge surface.

4. Next, you need to specify the tangent value to change the surface as per requirement. You can specify the tangent value using the **Edge 1** and **Edge 2** edit boxes in the **Tangent Magnitude** sub-rollout. Alternatively, use the **Edge 1** and **Edge 2** slider bars to specify the tangent value. You can also use the **Edge 1** and **Edge 2** handles to specify the tangent value, refer to Figure 11-39.

5. Specify the flow direction of the bridge surface using the **Edge 1 and 2** drop-down list in the **Flow Direction** sub-rollout. By default, the **Not Specified** option is selected. You can also select the **Isoparametric** or the **Perpendicular** option to define the flow direction.

You can use the handles displayed on the edges of the bridge surface to change its size and shape, refer to Figure 11-39. Alternatively, you can use the options available in the **Edge Limit** sub-rollout to modify the bridge surface. Choose the **Edge 1** tab from this sub-rollout to modify the Edge 1.

The **%Start** and **%End** edit boxes in this tab are used to modify the start point and the end point of the edge. You can also modify the start point and end point of the edge by using the slider bars available below the respective edit boxes. The **%Offset** edit box is used to specify the offset value of the edge. You can also use the slider bar or the offset handle to offset the edge. Similarly, you can modify the second edge by choosing the **Edge 2** tab.

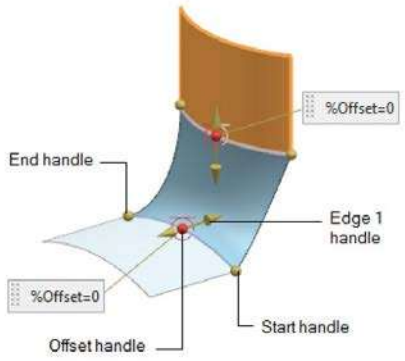


Figure 11-39 *The preview of the bridge surface with the handles*

If you select the **Link Start Handles** check box, the start point of both edges will be linked. Similarly, if you select the **Link End Handles** check box, the end point of both edges will be linked. If you select the **End to End** check box, the start and end points will not be modified.

TUTORIALS

Tutorial 1

In this tutorial, you will create the model shown in Figure 11-40. First, you need to create the surface model using the dimensions and orthographic views shown in Figure 11-41. After creating the surface model, you need to apply thickness of 1 mm in the outward direction. Assume the missing dimensions. Save the model with the name *c11tut1.prt* at the following location: *\NX\c11*. **(Expected time: 45 min)**

Figure 11-40 *Model for Tutorial 1*

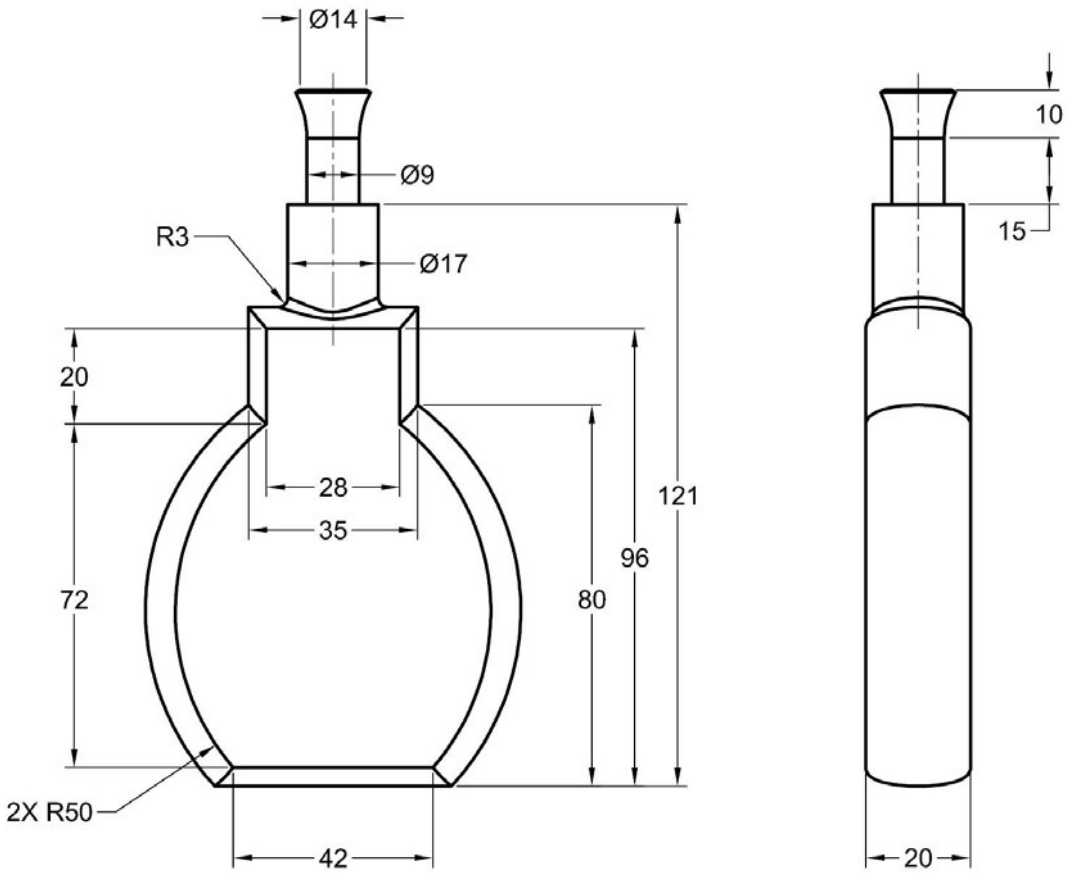


Figure 11-41 *Views and dimensions for Tutorial 1*

The following steps are required to complete this tutorial:

a. Start a new file and set the sheet environment.

b. Create the base feature using the **Bounded Plane** tool, refer to Figures 11-42 and 11-43.

c. Create the second feature using the **Bounded Plane** tool, refer to Figures 11-44 and 11-45.

d. Create the third feature using the **Bridge** tool that will join the two bounded surfaces, refer to Figures 11-46 and 11-47.
e. Mirror the bridge surface about the YC-ZC plane, refer to Figure 11-48.
f. Create the bottom surface of the model using the **Bridge** tool, refer to Figures 11-49 and 11-50.
g. Extrude edges of the model, refer to Figures 11-51 and 11-52.
h. Create the bridge surface over the extruded surface using the **Bridge** tool, refer to Figures 11-53 and 11-54.
i. Create the extrude feature using the **Extrude** tool, refer to Figures 11-55 and 11-56.
j. Create the face blend feature using the **Face Blend** tool, refer to Figures 11-57 and 11-58.
k. Create the surface using the **Bounded Plane** tool, refer to Figure 11-59.
l. Extrude the sketch, refer to Figures 11-60 and 11-61.
m. Create the surface by using the **Through Curve** tool, refer to Figures 11-62 through 11-64.
n. Sew the individual surfaces into a single surface, refer to Figure 11-65.
o. Add thickness to the surface model, refer to Figure 11-66.
p. Save and close the file.

Starting a New File and Setting the Sheet Environment

1. Start a new file with the name *c11tut1.prt* using the **Shape Studio** template, and specify its location as *C:\NX\c11*.

2. Choose **Menu > Preferences > Modeling** from the **Top Border Bar**; the **Modeling Preferences** dialog box is displayed. Select the **Sheet** radio button from the **Body Type** area and choose the **OK** button.

Creating the Base Feature Using the Bounded Plane Tool

The base feature for this tutorial needs to be created using the **Bounded Plane** tool.

1. Draw the fully constrained sketch on the XC-ZC plane, as shown in Figure 11-42. Exit the sketch.

2. Choose **Menu > Insert > Surface > Bounded Plane** from the **Top Border Bar**; the **Bounded Plane** dialog box is displayed and you are prompted to select the bounding string.

3. Select the sketch and choose the **OK** button; the base surface is created using the **Bounded Plane** tool, as shown in Figure 11-43.

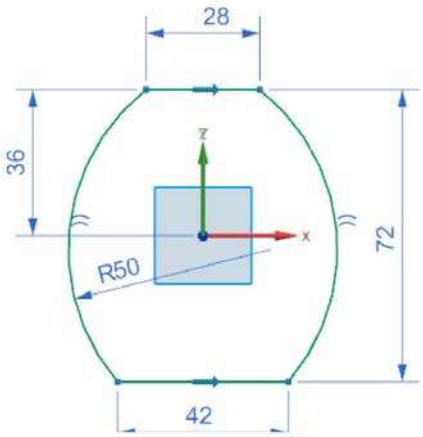


Figure 11-42 *Sketch for the base feature*

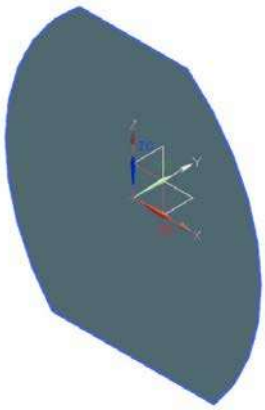


Figure 11-43 *Base feature of the surface model*

Creating the Second Feature Using the Bounded Plane Tool

The second feature for this tutorial will also be created using the **Bounded Plane** tool.

1. Create a plane at an offset of 20 mm from the XC-ZC plane.

2. Select the offset plane as the sketching plane and invoke the Sketch in Task environment.

3. Choose **Menu > Insert > Sketch Curve > Project Curve** from the **Top Border Bar**; the **Project Curve** dialog box is displayed and you are prompted to select the curve to project. Select the base feature and choose the **OK** button.

4. Exit the Sketch in Task environment; the curves are projected, refer to Figure 11-44.

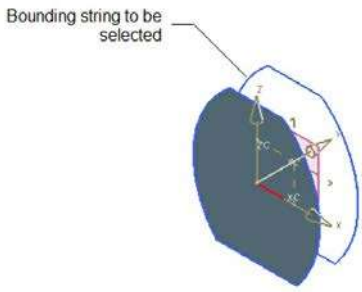


Figure 11-44 Projected curves

5. Choose **Menu > Insert > Surface > Bounded Plane** from the **Top Border Bar**; the **Bounded Plane** dialog box is displayed and you are prompted to select the bounding string.

6. Select the projected curve, as shown in Figure 11-44, and choose the **OK** button; the second feature is created, as shown in Figure 11-45.

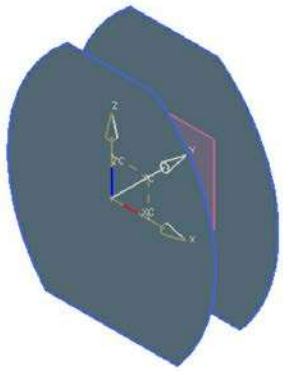


Figure 11-45 Second feature of the surface model

Creating the Bridge Surface Using the Bridge Tool

The third feature for this tutorial is a bridge surface. The procedure to create this feature is discussed next.

1. Choose **Menu > Insert > Detail Feature > Bridge** from the **Top Border Bar**; the **Bridge Surface** dialog box is displayed. By default, the **Select Edge 1** area is activated in the **Edges** rollout and you are prompted to select edges.

2. Select the edges, as shown in Figure 11-46.

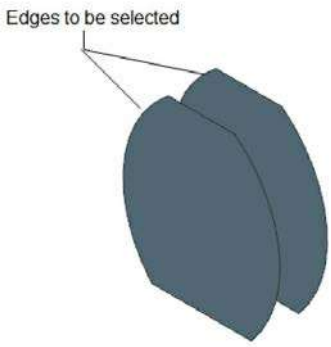


Figure 11-46 Edges to be selected

3. Next, select the **G1(Tangent)** option from the **Edge 1** and **Edge 2** drop-down lists in the **Continuity** sub-rollout of the **Constraints** rollout. Next, you need to choose the **Reverse Direction** button available below the **Edge 1** and **Edge 2** drop-down lists, if required.

4. Enter **1** in the **Edge 1** and **Edge 2** edit boxes available in the **Tangent Magnitude** sub-rollout. Make sure that the **Not Specified** option is selected in the **Edge 1 and 2** drop-down list in the **Flow Direction** sub-rollout.

5. Next, choose the **OK** button from the **Bridge Surface** dialog box. The surface created using the **Bridge** tool is shown in Figure 11-47.

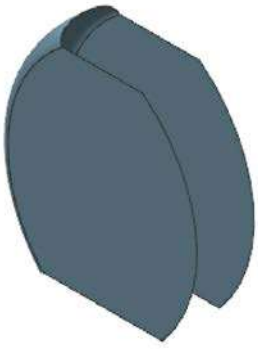


*Figure 11-47 Surface created using the **Bridge** tool*

Mirroring the Bridge Surface

1. After creating the bridge surface, mirror it about the YC-ZC plane. Turn off the display of the plane. The model after creating the mirror feature is shown in Figure 11-48.

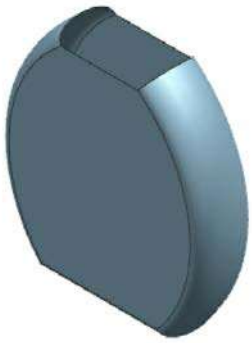


Figure 11-48 The model after mirroring the bridge surface

Creating the Bottom Face of the Model Using the Bridge Tool

The fifth feature for this tutorial is the bottom surface of the model.

1. Choose **Menu > Insert > Detail Feature > Bridge** from the **Top Border Bar**; the **Bridge Surface** dialog box is displayed. Select the first edge and then the second edge, refer to Figure 11-49.

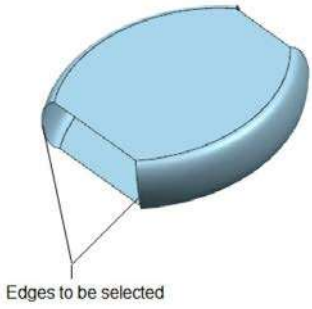


Figure 11-49 Edges to be selected

2. Select the **G0 (Position)** option from the **Edge 1** and **Edge 2** drop-down lists in the **Continuity** sub-rollout available in the **Constraints** rollout.

3. Choose the **OK** button; the surface created using the **Bridge** tool is shown in Figure 11-50.

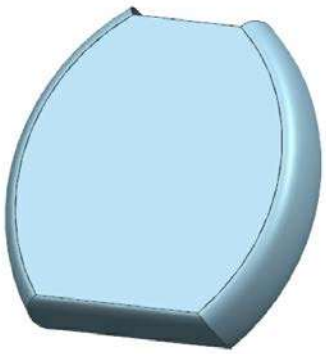


Figure 11-50 *Resulting bridge surface*

Creating the Sixth Feature by Extruding the Edges

In this section, you need to create the sixth feature by extruding the edges of the model.

1. Choose the **Extrude** tool from **Menu > Insert > Design Feature** in the **Top Border Bar** and select the edges of the model, refer to Figure 11-51.

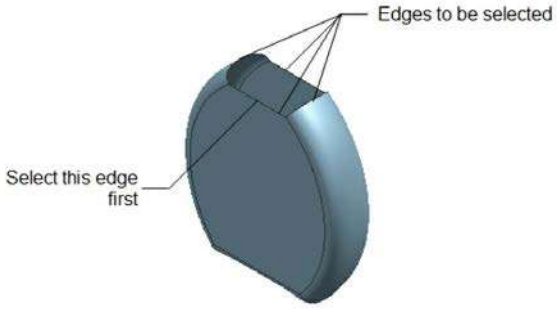


Figure 11-51 *Edges to be selected*

2. Enter the value **20** in the **Distance** edit box available below the **End** drop-down list and choose the **OK** button. The resulting surface model is shown in Figure 11-52.

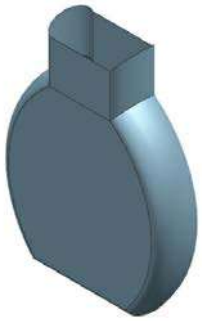


Figure 11-52 The surface model after creating the extrude feature

Creating the Surface Using the Bridge Tool

The seventh feature for this tutorial also needs to be created using the bridge surface.

1. Choose **Menu > Insert > Detail Feature > Bridge** from the **Top Border Bar**; the **Bridge Surface** dialog box is displayed.

2. Select the first edge and then the second edge, refer to Figure 11-53. Choose the **Reverse Direction** button from the **Edges** rollout, if required.

3. Select the **G0 (Position)** option from the **Edge 1** and **Edge 2** drop-down lists in the **Continuity** sub-rollout available in the **Constraints** rollout. Next, choose the **OK** button. The surface created using the **Bridge** tool is shown in Figure 11-54.

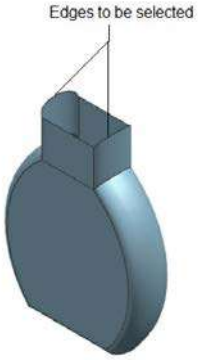


Figure 11-53 Objects to be selected

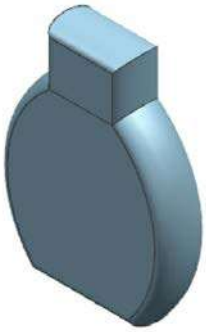


Figure 11-54 Resulting bridge surface

Creating the Extrude Feature

In this section, you need to create the eighth feature by extruding the sketch. You need to create the sketch for this feature on a plane that is at an offset of 81 mm from the XC-YC plane.

1. Create the datum plane at an offset of 81 mm from the XC-YC plane and create the sketch for the eighth feature by selecting the offset plane as the sketching plane, as shown in Figure 11-55.

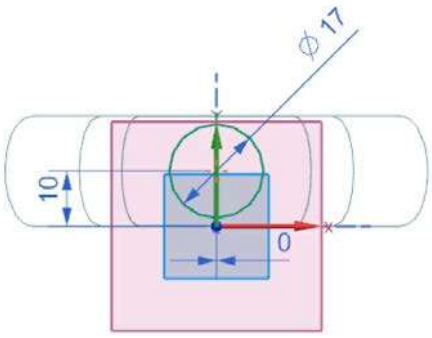


Figure 11-55 *Sketch drawn for the extrude feature*

2. Extrude this sketch through a distance of 25 mm in the downward direction. The resulting surface model is shown in Figure 11-56.

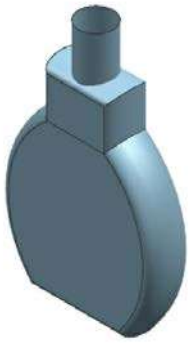


Figure 11-56 *Resulting extruded surface*

Creating the Face Blend Feature

Next, you need to create a face blend feature to blend the extruded surface created in the previous section with the bridge surface. The blending will also ensure that the portion of the extruded surface that extends below the bridge surface is removed.

1. Choose the **Face Blend** tool from **Menu > Insert > Detail Feature** of the **Top Border Bar**; the **Face Blend** dialog box is displayed.

2. By default, the **Two-face** option is selected in the drop-down list displayed in the **Type** rollout and the **Select Face 1** area is activated in the **Faces**

rollout. As a result, you are prompted to select the faces to blend. Select the faces, refer to Figure 11-57; a arrow is displayed. Make sure the arrow points outward. If not, choose the **Reverse Direction** button to flip the direction.

3. Choose the second **Face** button from the **Faces** rollout and select the second face, refer to Figure 11-57. An arrow is displayed on selecting the face. If this arrow does not point upward, reverse the direction.

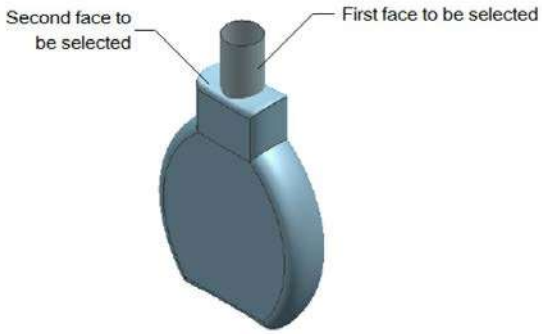


Figure 11-57 *Entities to be selected*

4. Select the **Rolling Ball** option from the **Orientation** drop-down list in the **Cross Section** rollout, if it is not selected by default. Make sure that the **Circular** option is selected in the **Shape** drop-down list.

5. Enter the value **3** in the **Radius** edit box and choose the **OK** button. The resulting face blend feature is shown in Figure 11-58.

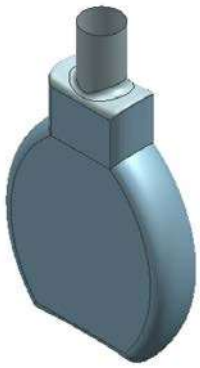


Figure 11-58 *Resulting face blend feature*

Creating the Tenth Feature Using the Bounded Plane Tool

In this section, you need to create the tenth feature using the **Bounded Plane** tool.

1. Choose the **Bounded Plane** tool from **Menu > Insert > Surface** of the **Top Border Bar**; the **Bounded Plane** dialog box is displayed and you are prompted to select the bounding string.

2. Select the circular edge of the extruded feature created earlier.

3. Choose the **OK** button. The surface model after creating the bounded plane feature is shown in Figure 11-59.

Figure 11-59 *The model after creating the bounded plane feature*

Creating the Extrude Feature

In this section, you need to create the eleventh feature by extruding the sketch. You need to create the sketch for this feature on the bounded surface created earlier.

1. Create the sketch for the eleventh feature by selecting the bounded surface created earlier as the sketching plane, refer to Figure 11-60.

2. Choose the **Extrude** tool from **Menu > Insert > Design Feature** in the **Top Border Bar**; the **Extrude** dialog box is displayed and you are prompted to select the section geometry to be extruded.

3. Select the sketch and enter the value **15** in the **Distance** edit box available below the **End** drop-down list. Next, choose the **OK** button from the **Extrude** dialog box. The resulting surface model is shown in Figure 11-61.

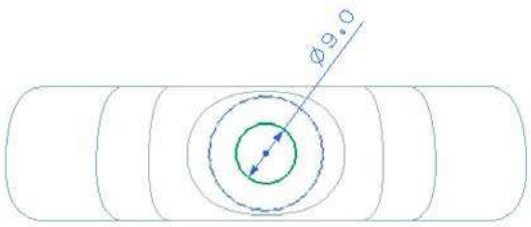


Figure 11-60 Sketch drawn for the eleventh feature

Figure 11-61 The surface model after creating the extrude feature

Trimming the Sheet

Next, you need to trim the unwanted portion of the surface model using the **Trim Sheet** tool.

1. Choose the **Trim Sheet** tool from **Menu > Insert > Trim** in the **Top Border Bar**; the **Trim Sheet** dialog box is displayed.

2. Select the tenth feature created using the **Boundary Plane** tool and then press the middle mouse button.

3. Select the extruded sheet feature created in the previous step and then select the **Keep** radio button from the **Region** rollout of the dialog box. Next, choose the **OK** button. The portion of the boundary plane surface that is enclosed inside the extruded surface is trimmed.

Creating the Through Curves Surface

This feature will be created using the **Through Curves** tool. You need to create the sketch for this feature on a plane at an offset of 106 from the XC-YC plane. The following steps are required to create the twelfth feature:

1. Create a datum plane at an offset of 106 mm from the XC-YC plane in the upward direction.

2. Create the sketch of the twelfth feature by selecting the offset plane as the sketching plane, refer to Figure 11-62.

3. Choose the **Through Curves** tool from **Menu > Insert > Mesh Surface** in the **Top Border Bar**; the **Through Curves** dialog box is displayed and you are prompted to select the curve or point to section.

4. Select the edge as the first section string from the previous extruded feature, refer to Figure 11-63, and press the middle mouse button; you are again prompted to select the section.

5. Select the sketch as the second section string, refer to Figure 11-63 and press the middle mouse button. Make sure the arrows in both sections point in the same direction.

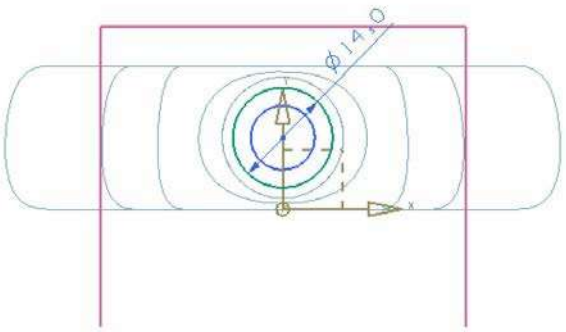


Figure 11-62 Sketch for the through curve surface

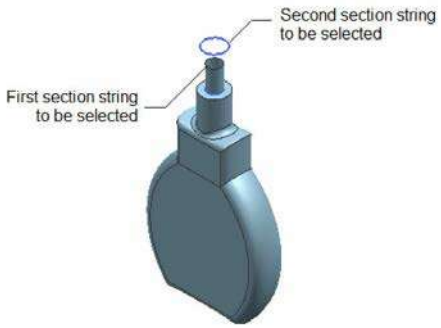


Figure 11-63 Edges to be selected

6. Next, select the **G1 (Tangent)** option from the **First Section** drop-down list in the **Continuity** rollout; the **Select Face** option is activated. As a result, you are prompted to select the continuity constraint face. Select the eleventh feature from the drawing window. Next, select the **Multiple** option from the **Patch Type** drop-down list of the **Output Surface Options** rollout, and then choose the **OK** button from the **Through Curves** dialog box.

The resulting surface model is shown in Figure 11-64.

Figure 11-64 The resulting through curves surface feature

Sewing Individual Surfaces into a Single Surface

After creating the surface model, you need to sew individual surfaces into a single surface using the **Sew** tool.

1. Invoke the **Sew** tool by choosing **Menu > Insert > Combine > Sew** from the **Top Border Bar**; the **Sew** dialog box is displayed.

2. Select the **Sheet** option from the drop-down list in the **Type** rollout, if it is not selected.

3. Select the base feature that is created using the **Bounded Plane** tool as the target body and then select all the remaining surfaces as the tool sheet bodies.

4. Choose the **OK** button. The resultant model after sewing individual surfaces into a single surface with a common edge is shown in Figure 11-65.

Adding Thickness to the Surface Model

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

1. Choose **Menu > Insert > Offset/Scale > Thicken** from the **Top Border Bar**; the **Thicken** dialog box is displayed. Also, you are prompted to select the faces to thicken.

2. Select the surface model from the graphics window. Next, enter the thickness value as **0** and **1** in the **Offset 1** and **Offset 2** edit boxes,

respectively.

3. Choose the **OK** button from the dialog box. The final model after applying the thickness is shown in Figure 11-66.

Figure 11-65 *The surface model after sewing the surfaces*

Figure 11-66 *The final model after applying the thickness*

Saving and Closing the File

1. Choose **Menu > File > Close > Save and Close** from the **Top Border Bar** to save and close the file.

Tutorial 2

In this tutorial, you will create the model shown in Figure 11-67. First, you need to create the surface model using the dimensions and orthographic views shown in Figure 11-68. After creating the surface model, you need to apply thickness of 1 mm in the outward direction. Assume the missing dimensions and then save the model with the name *c11tut2.prt* at the following location: *\NX\c11*. **(Expected time: 30 min)**

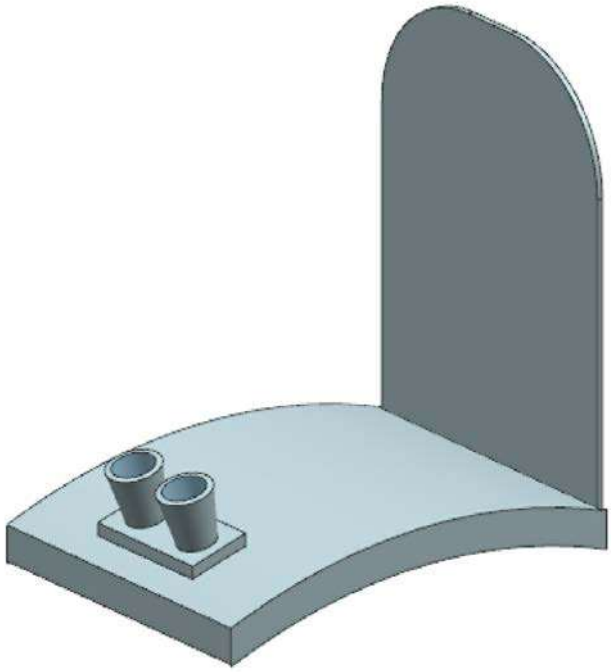
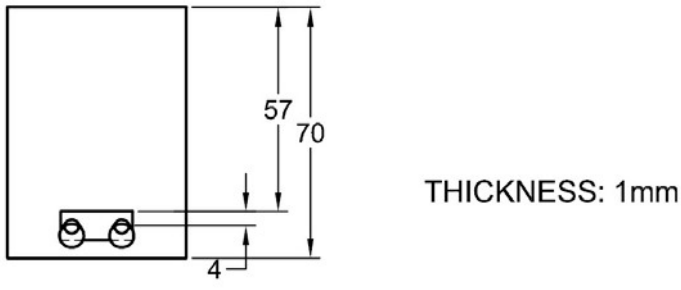


Figure 11-67 *Surface model for Tutorial 2*



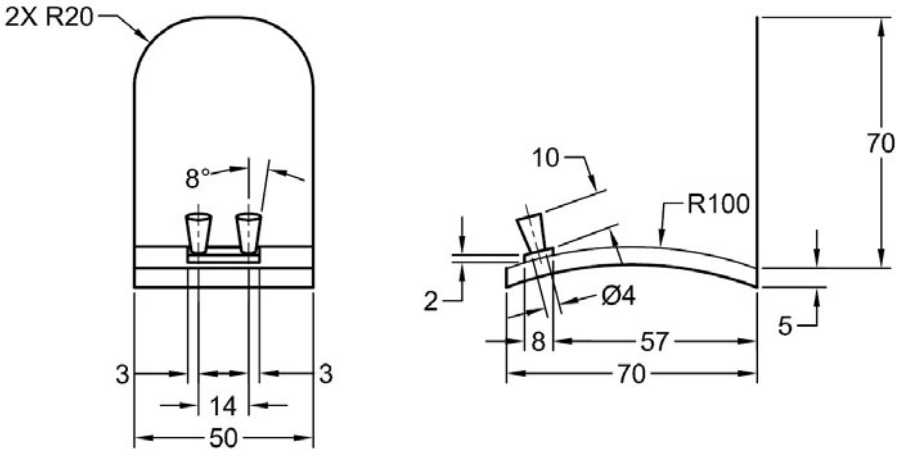


Figure 11-68 *Views and dimensions for Tutorial 2*

The following steps are required to complete this tutorial:

a. Start a new file and set the sheet environment.

b. Create the base feature by extruding the base sketch, refer to Figures 11-69 and 11-70.

c. Create the second feature by extruding the edges of the base feature, refer to Figures 11-71 and 11-72.

d. Create the third feature by extruding the sketch, refer to Figures 11-73 and 11-74.

e. Create the fourth feature by using the **Bounded** tool, refer to Figures 11-75 and 11-76.

f. Trim the fourth feature using the **Trimmed Sheet** tool, refer to Figures 11-77 and 11-78.

g. Create the fifth feature by extruding the sketch and mirror it, refer to Figures 11-79 through 11-82.

h. Create the feature by extruding the sketch and trim it, refer to Figures 11-83 through 11-86.

i. Sew individual surfaces into a single surface.

j. Add thickness to the surface model, refer to Figure 11-87.

k. Save and close the file.

Starting a New File and Setting the Sheet Environment

1. Start a new file with the name *c11tut2.prt* using the **Model** template, and specify its location as *C:\NX\c11*.

2. Choose **Menu > Preferences > Modeling** from the **Top Border Bar** and then select the **Sheet** radio button from the **Body Type** area of the **Modeling Preferences** dialog box. Next, choose the **OK** button.

Creating the Base Feature by Extruding the Sketch

In this section, you need to create the base feature for this tutorial by extruding the sketch.

1. Create the sketch of the base feature on the YC-ZC plane, as shown in Figure 11-69.

2. Extrude the sketch symmetrically on both sides of the sketching plane through a symmetric distance of 25 mm. The base feature of the surface model is shown in Figure 11-70.

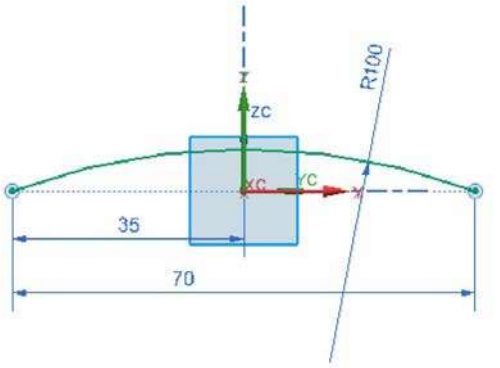


Figure 11-69 Sketch of the base feature

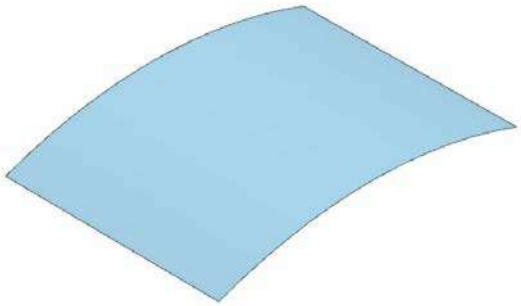


Figure 11-70 Base feature of the surface model

Creating the Second Feature by Extruding the Edges

The second feature will be created by extruding the edges of the base feature. The following steps are required to create the second feature:

1. Invoke the **Extrude** tool and then select the edges of the base surface, refer to Figure 11-71.

2. Define the **-**ZC**-**axis direction as the direction of extrusion using the **Inferred Vector** drop**-**down list. Enter **5** in the **Distance** edit box available below the **End** drop**-**down list.

3. Choose the **OK** button. The resulting surface model is shown in Figure 11**-**72.

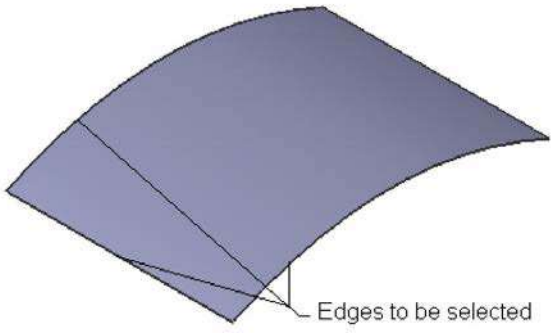


Figure 11-71 Edges to be selected

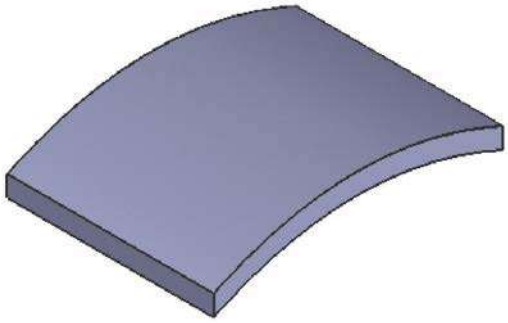


Figure 11-72 The second feature created

Creating the Third Feature by Extruding the Sketch

The third feature will be created by extruding a sketch. You need to create the sketch for the third feature on the YC-ZC plane.

1. Create the sketch for the third feature by selecting the YC-ZC plane as the sketching plane, as shown in Figure 11-73.

2. Extrude the sketch through a symmetric distance of 10 mm. The resulting surface model is shown in Figure 11-74.

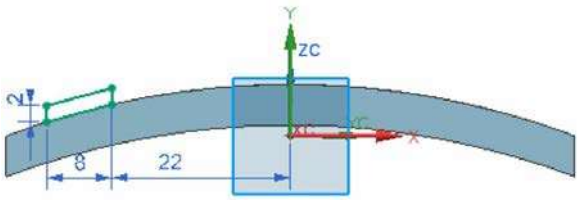


Figure 11-73 Sketch drawn for extrusion

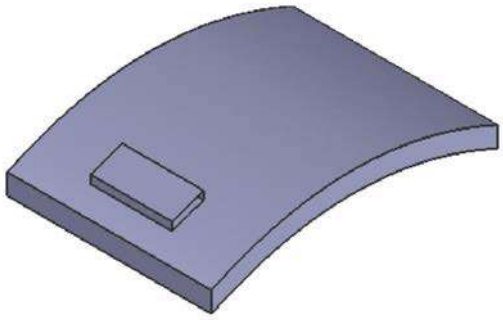


Figure 11-74 Resulting extruded feature

Creating the Bounded Plane Features

Now, you need to create the bounded plane features by using the **Bounded Plane** tool.

1. Choose **Menu > Insert > Surface > Bounded Plane** from the **Top Border Bar**; the **Bounded Plane** dialog box is displayed and you are prompted to select the bounding string.

2. Select the edges of the third feature to create a bounded plane, refer to Figure 11-75.

3. Choose the **OK** button from the dialog box; the bounded plane feature is created, refer to Figure 11-76.

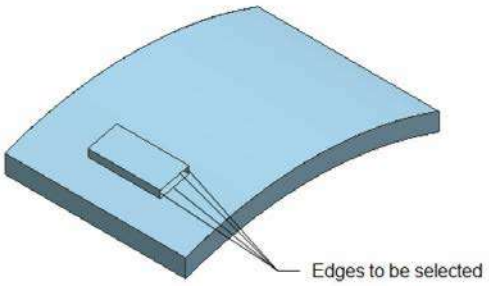


Figure 11-75 Edges to be selected

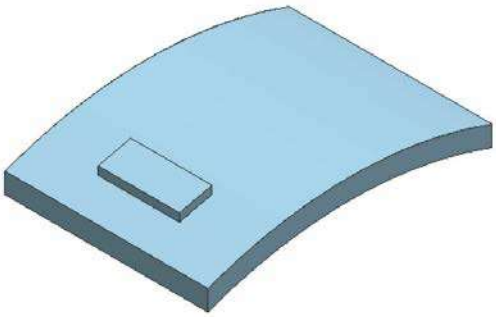


Figure 11-76 Resulting bounded plane feature

4. Similarly, create the bounded plane feature on the other side of the third extruded feature.

Trimming the Surfaces Using the Trim Sheet Tool

Now, the unwanted surfaces need to be trimmed using the **Trim Sheet** tool. The following steps are required to trim the unwanted surfaces.

1. Choose the **Trim Sheet** tool from **Menu > Insert > Trim** in the **Top Border Bar**; the **Trim Sheet** dialog box is displayed and you are prompted to select the target sheet body.

2. Select the third surface feature created and press the middle mouse button; you are prompted to select the trimming objects.

3. Select the base surface created, and then choose the **Apply** button; the third feature is trimmed. You are again prompted to select the sheet body to be trimmed.

4. Select the base surface created as the sheet body to be trimmed and then press the middle mouse button; you are prompted to select the trimming objects.

5. Select the surfaces as trimming objects, refer to Figure 11-77. Next, choose the **OK** button from the **Trim Sheet** dialog box. The rotated view of the resultant surface model is shown in Figure 11-78.

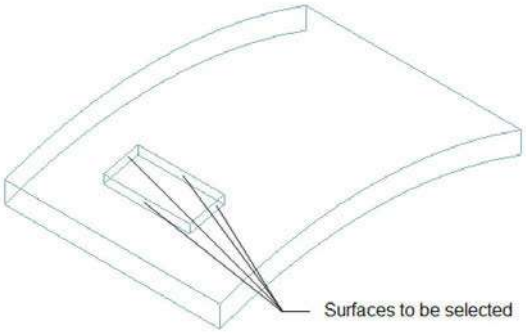


Figure 11-77 Surfaces to be selected

Figure 11-78 Resultant surface model

Creating the Next Feature by Extruding the Sketch

The next feature will be created by extruding the sketch. You need to create the sketch for this feature on the top face of the third feature.

1. Create the sketch for this feature by selecting the top face of the third feature as the sketching plane, as shown in Figure 11-79.

2. Extrude this sketch through a distance of 10 mm with -8 degrees as the draft angle by using the **From Start Limit** option. The resulting surface model is shown in Figure 11-80.

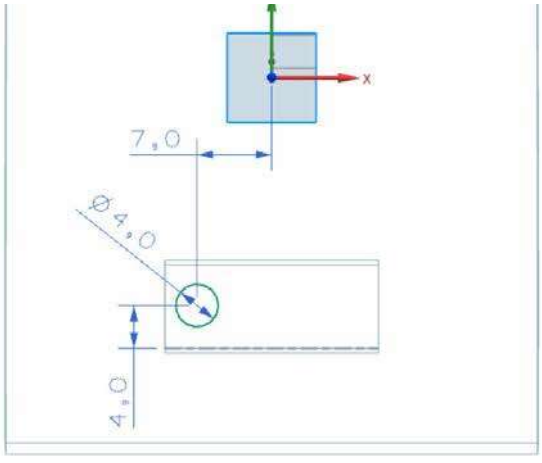


Figure 11-79 Sketch created

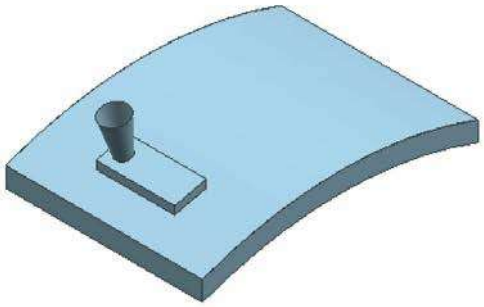


Figure 11-80 Resultant fourth feature

Mirroring the Feature

1. Invoke the **Mirror Feature** dialog box and then select the feature to mirror, refer to Figure 11-81.

2. Choose the **Plane** button from the dialog box and then select the YC-ZC plane as the mirroring plane. Next, choose the **OK** button; the mirror feature is created, refer to Figure 11-82.

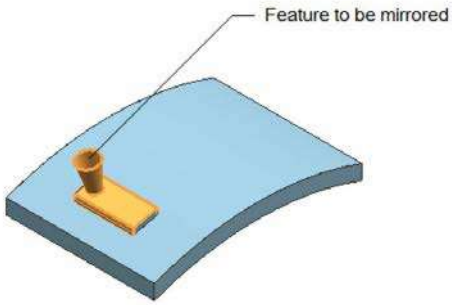


Figure 11-81 Feature to be mirrored

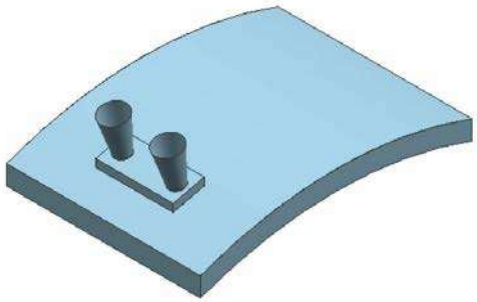


Figure 11-82 Resultant surface model with the mirrored feature

Trimming the Surface Using the Trim Sheet Tool

Now, the surface that is enclosed between the top face of the third feature and the previously created features will be trimmed using the **Trim Sheet** tool.

1. Choose the **Trim Sheet** tool from **Menu > Insert > Trim** in the **Top Border Bar**; the **Trim Sheet** dialog box is displayed and you are prompted to select the target sheet body.

2. Select the third surface feature created and press the middle mouse button; you are prompted to select the trimming objects.

3. Select the previously created extruded surface and its mirror feature and then choose the **OK** button; the surface enclosed between the top face of the third feature and the previously created features is trimmed.

Creating the Next Feature by Extruding the Edge

The next feature will be created by extruding the edge of the base feature.

1. Invoke the **Extrude** dialog box and then select the edge, refer to Figure 11-83. Next, extrude it through a distance of 70 mm in the upward direction. The resulting surface model is shown in Figure 11-84.

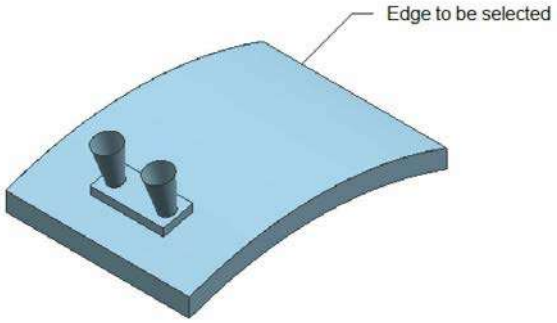


Figure 11-83 The edge to be selected

Figure 11-84 Resultant surface model

Trimming the Previously Created Extruded Surface

Now, the previously created extruded surface will be trimmed using the **Trim Sheet** tool.

1. Select the previously created surface as the sketching plane and draw the sketch, as shown in Figure 11-85.

2. Choose the **Trim Sheet** tool from **Menu > Insert > Trim** in the **Top Border Bar**; the **Trim Sheet** dialog box is displayed and you are prompted to select the target sheet body.

3. Select the previously created extruded surface and press the middle mouse button; you are prompted to select the trimming objects. Select the sketch created and then choose the **OK** button. The resulting surface model is shown in Figure 11-86.

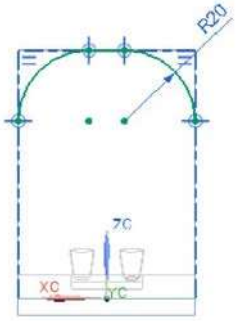


Figure 11-85 Sketch drawn for trimming

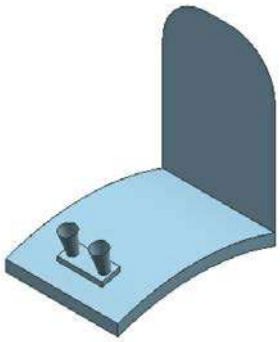


Figure 11-86 The resulting surface model after trimming the fifth feature

Sewing Individual Surfaces into a Single Surface

Now, you need to sew individual surfaces of the surface model into a single

surface by using the **Sew** tool.

1. Invoke the **Sew** tool by choosing **Menu > Insert > Combine > Sew** from the **Top Border Bar**; the **Sew** dialog box is displayed.

2. Select the **Sheet** option from the drop-down list in the **Type** rollout if it is not selected by default.

3. Select the base feature that is created using the **Extrude** tool as the target body and then select all the remaining surfaces as the tool sheet bodies.

4. Choose the **OK** button; all the individual surfaces get sewed into a single surface.

Adding Thickness to the Surface Model

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

1. Choose **Menu > Insert > Offset/Scale > Thicken** from the **Top Border Bar**; the **Thicken** dialog box is displayed. Also, you are prompted to select the faces to thicken.

2. Select the surface model from the graphics window. Next, enter the thickness value as **0** and **1** in the **Offset 1** and **Offset 2** edit boxes, respectively.

3. Choose the **OK** button from the dialog box. The final model after adding the thickness is shown in Figure 11-87.

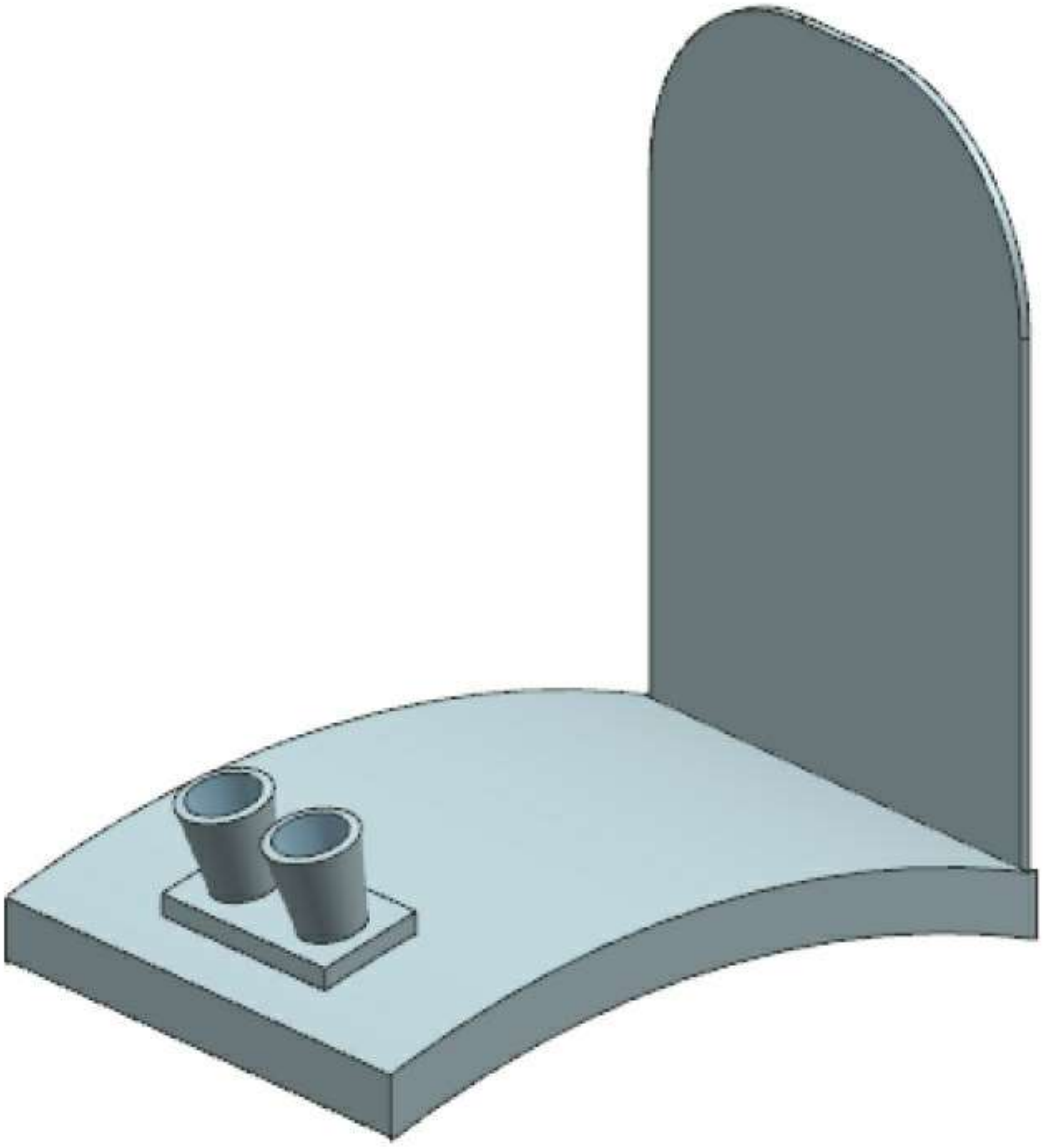


Figure 11-87 *Final surface model after adding the thickness*

Saving and Closing the File

1. Choose **Menu > File > Close > Save and Close** from the **Top Border Bar** to save and close the file.

Tutorial 3

In this tutorial, you will create the cover of a hair dryer, as shown in Figure 11-88. First, you will create the model by using surfaces and then thicken it. The views and dimensions of the model are displayed in Figures 11-89(a) and 11-89(b). **(Expected time: 1.5 hr)**

Figure 11-88 *Model for Tutorial 3*

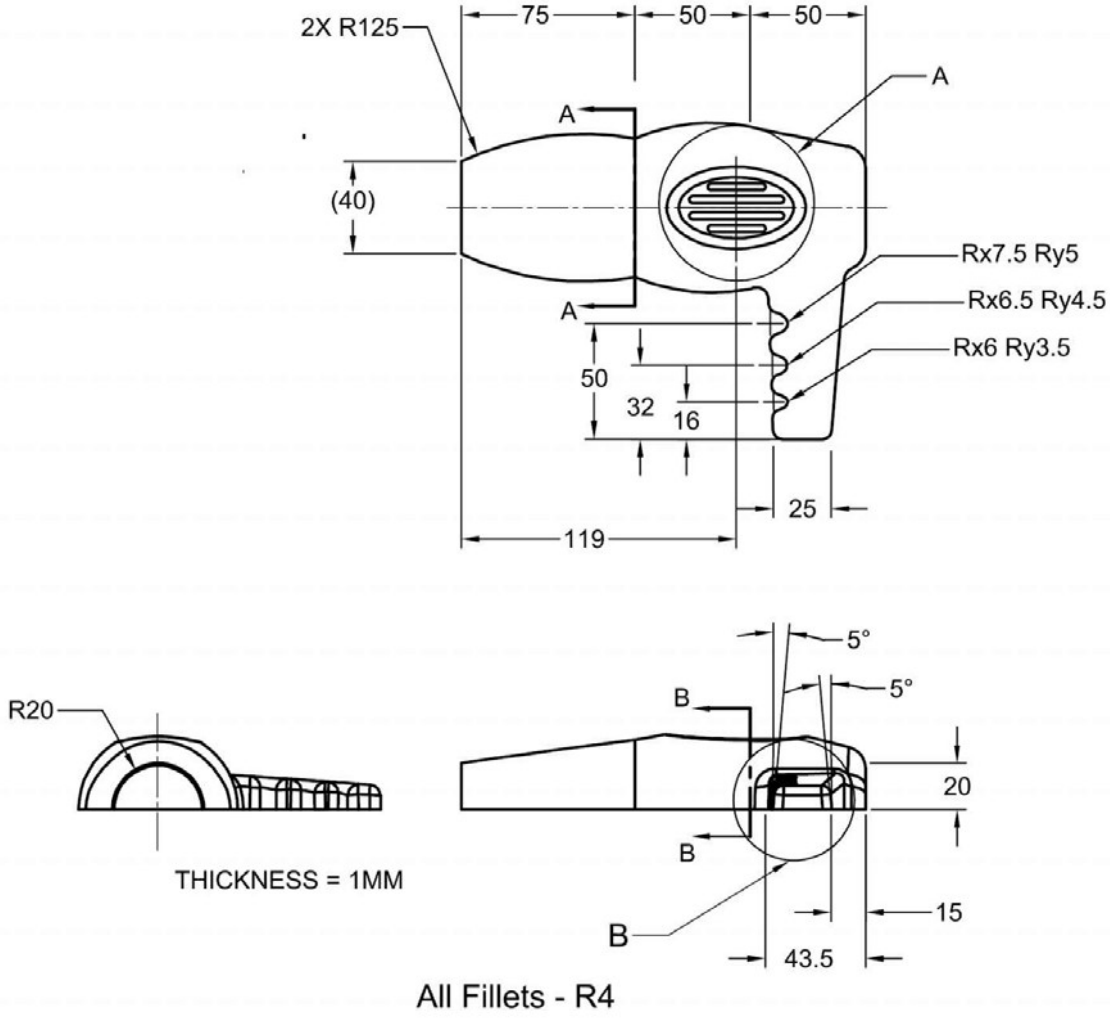


Figure 11-89(a) *Views and dimensions of the model*

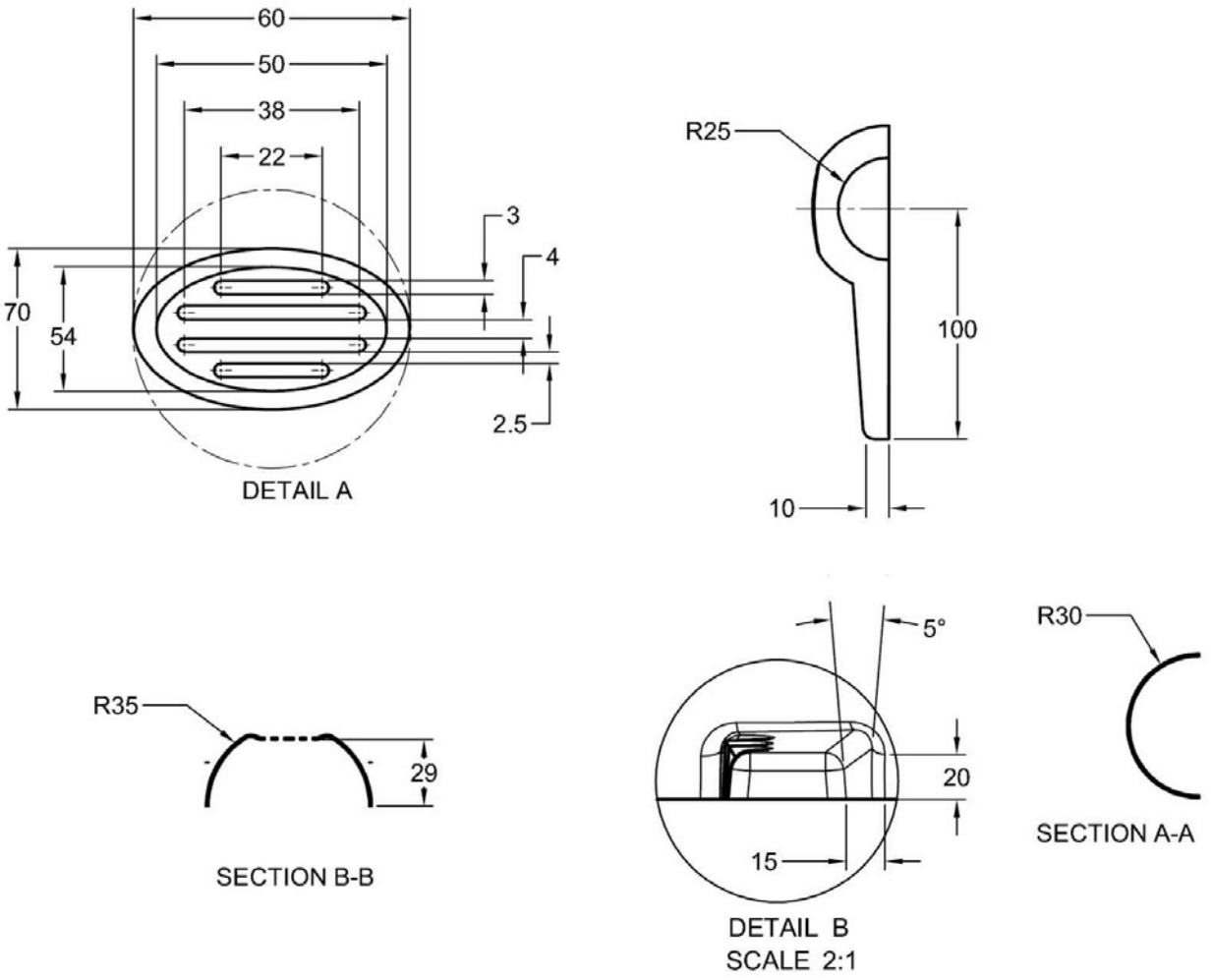


Figure 11-89(b) *Section and Detail views of the model*

The following steps are required to complete this tutorial:

a. First create the base surface. The base surface is created by using the open sections and the guide curves, refer to Figures 11-90 through 11-92.

b. Create a bounded plane surface to close the right face of the base surface, refer to Figures 11-93 and 11-94.

c. Create the basic structure of the handle of the hair dryer cover by creating a through curve surface between two open sections, refer to Figures 11-95 through 11-98.

d. Trim the unwanted portion of the through curve surface that is used to create the handle, refer to Figures 11-99 and 11-100.

e. Create a four point surface to close the front face of the handle, refer to Figures 11-101 and 11-102.

f. Create and extrude the elliptical sketches to create the grips of the handle

and then trim the unwanted surfaces, refer to Figures 11-103 through 11-107.

g. Create a dip on the top surface of the hair dryer, refer to Figures 11-108 through 11-113.

h. Trim the surface to create air vents, refer to Figure 11-114.

i. Sew all surfaces together and add fillets to the model, refer to Figure 11-115.

j. Thicken the surface, refer to Figure 11-116.

k. Save and close the file.

Creating the Base Surface

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

1. Start a new file with the name *c11tut3.prt* using the **Shape Studio** template and specify its location as *C:\NX\c11*.

2. Choose **Menu > Preferences > Modeling** from the **Top Border Bar** and select the **Sheet** radio button from the **Body Type** area of the **Modeling Preferences** dialog box. Next, choose the **OK** button to exit the dialog box.

3. Create three planes at an offset distance from the YC-ZC plane, as shown in Figure 11-90. For the offset distance of planes, refer to Figure 11-89(a).

4. Create sections and guide strings to create a studio surface, as shown in Figure 11-90. For dimensions, refer to Figure 11-89(a).

5. Choose the **Studio Surface** from **Menu > Insert > Mesh Surface** in the **Top Border Bar**; the **Studio Surface** dialog box is displayed and you are prompted to select a section.

6. Select Section 1, refer to Figure 11-91, and then press the middle mouse

button; you are prompted to select a section again.

7. Select Section 2, refer to Figure 11-91, and then choose the **Guide (Cross) Curves** button from the **Guide (Cross) Curves** rollout; you are prompted to select a guide string. Note that you may need to choose the **Reverse Direction** button, if required.

8. Select the Guide Curve 1, refer to Figure 11-91, and then press the middle mouse button; you are prompted to select a guide string again.

9. Select the Guide Curve 2, refer to Figure 11-91. The preview of the studio surface after selecting the sections and the guide curves is shown in Figure 11-102. Next, choose the **Apply** button from the **Studio Surface** dialog box.

10. Select the Section 2, refer to Figure 11-91, and then press the middle mouse button; you are prompted to select a section again.

11. Select the Section 3, refer to Figure 11-91, and then press the middle mouse button; you are prompted to select a section again.

12. Select the Section 4, refer to Figure 11-91, and then choose the **Guide (Cross) Curves** button from the **Guide (Cross) Curves** rollout; you are prompted to select a guide string.

13. Select the Guide Curve 3, refer to Figure 11-91, and then press the middle mouse button; you are prompted to select a guide string again.

14. Select the Guide Curve 4, refer to Figure 11-91. A preview of the studio surface after selecting the sections and the guide curves is shown in Figure 11-92. Next, choose the **OK** button to create the studio surface.

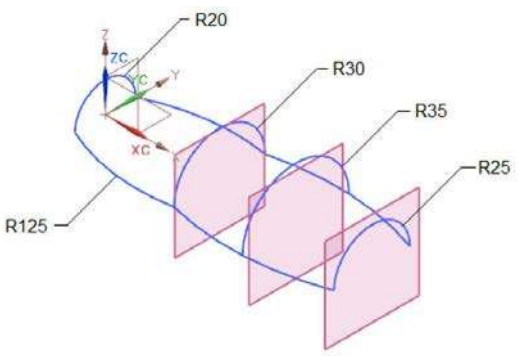


Figure 11-90 *Dimensions of sections and guide curves*

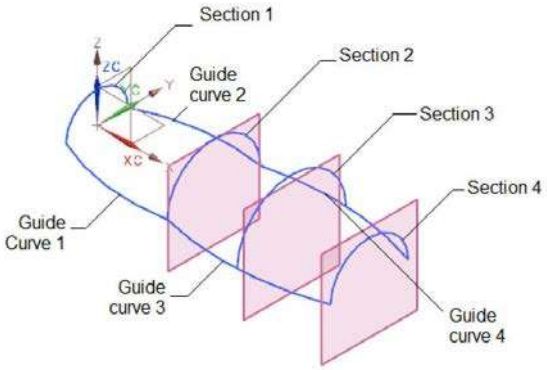


Figure 11-91 *Sections and guide curves to create the studio surface*

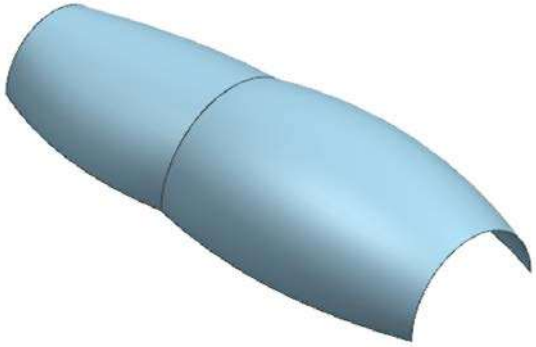


Figure 11-92 *Resultant studio surface*

Creating the Bounded Plane Surface

1. Invoke the Sketch in Task environment by selecting a plane created at an offset of 175mm.

2. Draw a closed sketch to create the bounded plane surface, as shown in Figure 11-93.

3. Invoke the **Bounded Plane** dialog box and then select the closed sketch from the drawing area. Next, choose the **OK** button from the **Bounded Plane** dialog box; the bounded plane surface is created, as shown in Figure 11-94.

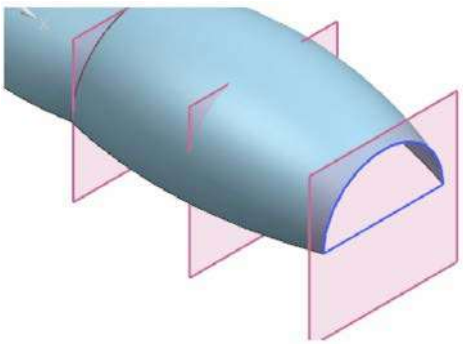


Figure 11-93 *Closed sketch to create a bounded plane surface*

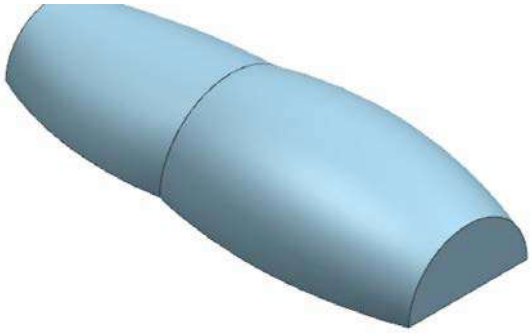


Figure 11-94 *Resultant bounded plane surface*

Creating the Base Surface for the Handle

Next, you need to create the base surface for the handle. The base surface for the handle will be created through two open sections.

1. Create a plane at an offset distance of 100 mm from the XC-ZC plane.

2. Invoke the Sketch in Task environment by using the newly created plane as the sketching plane.

3. Create an open sketch, as shown in Figure 11-95, and exit the Sketch in Task environment.

4. Next, invoke the Sketch in Task environment by using the XC-ZC plane as the sketching plane.

5. Again, create another open sketch, as shown in Figure 11-96, and exit the Sketch in Task environment.

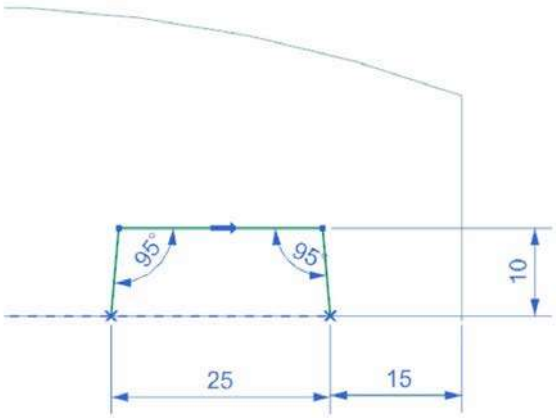


Figure 11-95 Open sketch created

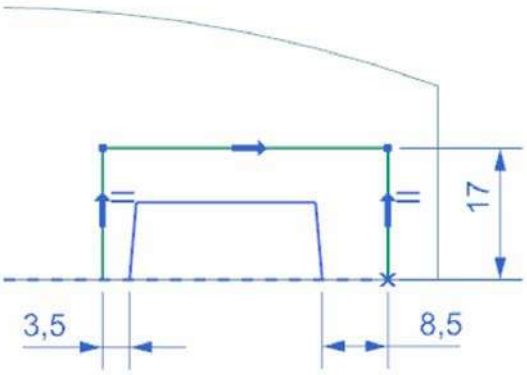


Figure 11-96 Open sketch created

6. Choose the **Through Curves** tool from **Menu > Insert > Mesh Surface** in the **Top Border Bar**; the **Through Curves** dialog box is displayed and you are prompted to select a section.

7. Select all elements from the first section string, refer to Figure 11-97, and then press the middle mouse button; you are prompted to select a section again.

8. Select all elements from the second section string, refer to Figure 11-98. Make sure that the direction arrows are on the same side of the sections.

9. Select the **G0 (Position)** option from the **First Section** and **Last Section** drop-down lists in the **Continuity** rollout.

10. Select the **Preserve Shape** check box from the **Alignment** rollout and make sure that the **Parameter** option is selected in the **Alignment** drop-down list.

11. Choose the **OK** button from the **Through Curves** dialog box; the handle surface is created, as shown in Figure 11-98.

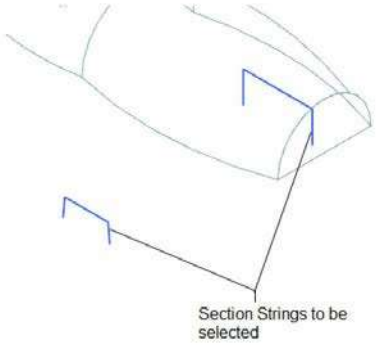


Figure 11-97 Sections to be selected

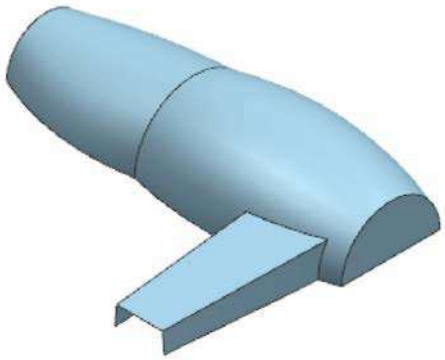


Figure 11-98 Resultant surface for the handle

Trimming the Unwanted Portion from the Surface of the Handle

If you rotate the model after creating the lofted surface for the handle, you will observe that a portion of the lofted surface needs to be trimmed.

1. Choose the **Trim Sheet** tool from **Menu > Insert > Trim** in the **Top Border Bar**; the **Trim Sheet** dialog box is displayed and you are prompted to select the target sheet body.

2. Select the handle surface feature created and press the middle mouse button; you are prompted to select the trimming objects.

3. Select the base surface created, and then select the **Keep** radio button from the **Region** rollout. Next, choose the **Apply** button; the handle feature is trimmed and you are again prompted to select the sheet body to be trimmed.

4. Select the base surface created as the sheet body to be trimmed, and then press the middle mouse button; you are prompted to select the trimming objects.

5. Select the surfaces shown in Figure 11-99 as the trimming objects. Next, choose the **OK** button from the **Trim Sheet** dialog box. The rotated view of the resultant surface model is created, as shown in Figure 11-100.

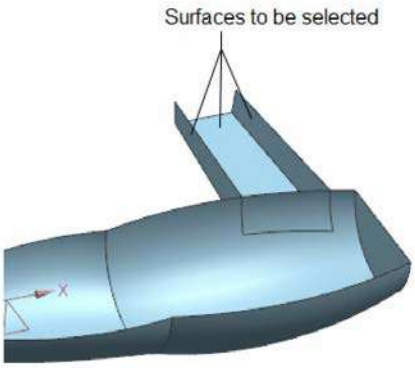


Figure 11-99 Surfaces to be selected

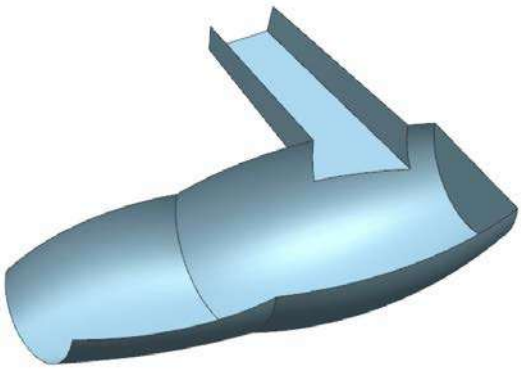


Figure 11-100 Resultant trimmed surface

Creating the Four Point Surface

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

1. Choose the **Four Point Surface** tool from **Menu > Insert > Surface** in the **Top Border Bar**; the **Four Point Surface** dialog box is displayed and you are prompted to select objects to infer points.

2. Select the vertices on the front face of the handle, refer to Figure 11-101, and then choose the **OK** button; the planar surface is created. The model after closing the front face of the handle is shown in Figure 11-102.

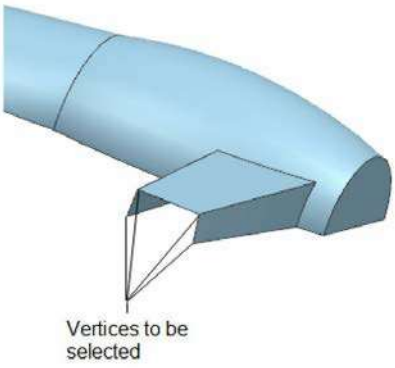


Figure 11-101 Vertices to be selected

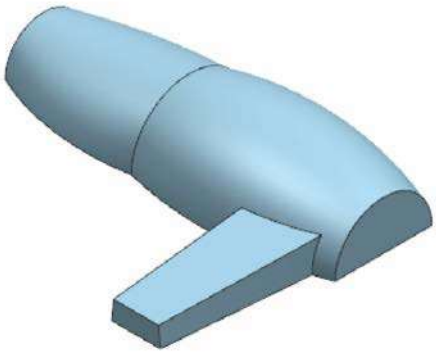


Figure 11-102 Model after closing the front face of the handle

Creating Grips on the Handle

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

1. Invoke the Sketch in Task environment using the XC-YC plane as the

sketching plane.

2. Create a sketch for extruding the surface, as shown in Figure 11-103.

3. Extrude the sketch up to a depth of 25 mm. The extruded surface is displayed, as shown in Figure 11-104.

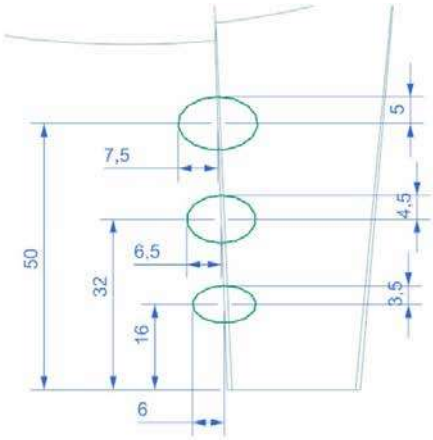


Figure 11-103 *Sketch created for extruding the surface*

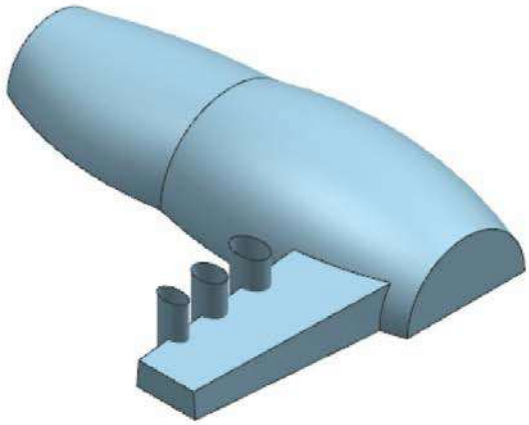


Figure 11-104 *Resultant extruded surface*

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

4. Choose the **Trim Sheet** tool from **Menu > Insert > Trim** in the **Top Border Bar**; the **Trim Sheet** dialog box is displayed and you are prompted to select the target sheet body.

5. Select the extruded surface features and press the middle mouse button; you are prompted to select the trimming objects.

6. Select the faces of the handle, refer to Figure 11-105, and then select the **Discard** radio button from the **Region** rollout.

7. Next, choose the **Apply** button; the extruded features are trimmed and you are again prompted to select the sheet body to be trimmed.

8. Select the handle surface created as the sheet body for trimming, and then

press the middle mouse button; you are prompted to select the trimming objects.

9. Select the surfaces as the trimming objects, as shown in Figure 11-106 and then choose the **Keep** radio button from the **Region** rollout. Next, choose the **OK** button from the **Trim Sheet** dialog box. The rotated view of the resultant surface model is created, as shown in Figure 11-107.

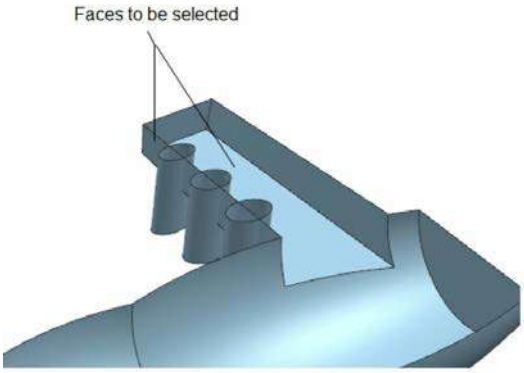


Figure 11-105 *Faces to be selected*

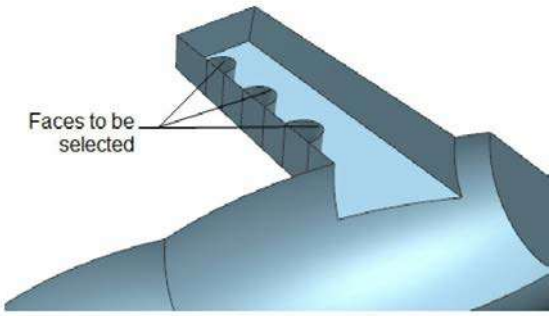


Figure 11-106 *Faces to be selected*

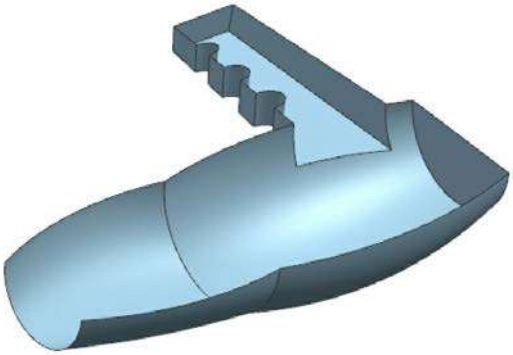


Figure 11-107 *Resultant trimmed surface*

Creating a Dip on the Base Surface

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

1. Create a plane at an offset distance of 35 mm from the XC-YC plane.

2. Invoke the Sketch in Task environment using the newly created plane as the sketching plane.

3. Create a sketch, as shown in Figure 11-108, and then exit the Sketch in Task environment.

4. Next, choose the **Project Curve** tool from the **Curve** gallery of the **Create** group and project the newly created sketch on the base surface; the model after projecting the sketch is displayed, as shown in Figure 11-109.

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

6. Next, invoke the Sketch in Task environment by using the newly created plane as the sketching plane and create a sketch, as shown in Figure 11-110. Next, exit the Sketch in Task environment.

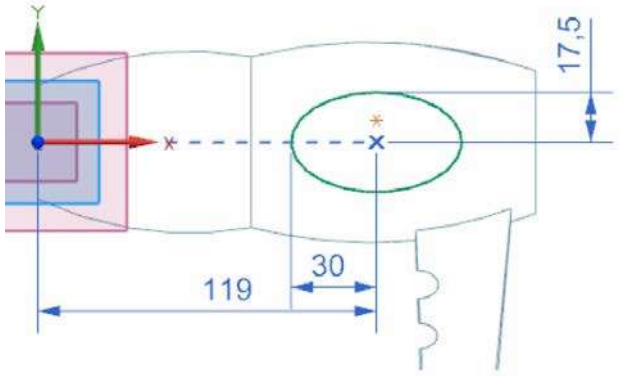


Figure 11-108 *Sketch to be created*

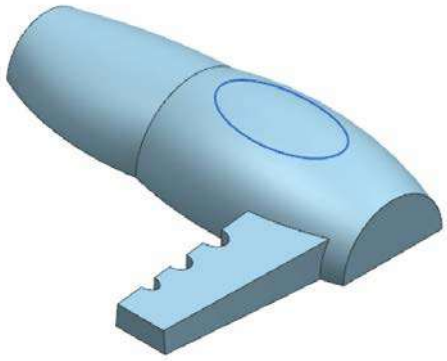


Figure 11-109 *Resultant projected curve*

7. Invoke the **Studio Surface** tool and create a studio surface by using the sketch and the projected curve created earlier. The studio surface after hiding the base surface is shown in Figure 11-111.

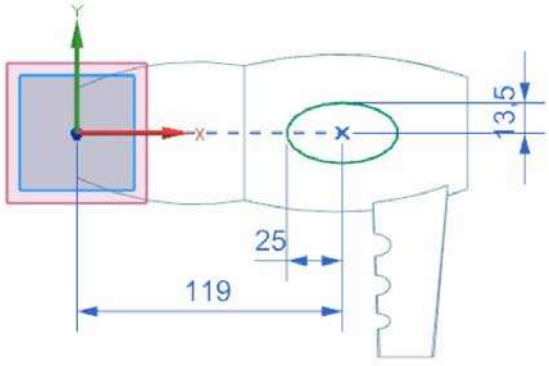


Figure 11-110 *Sketch to be created*

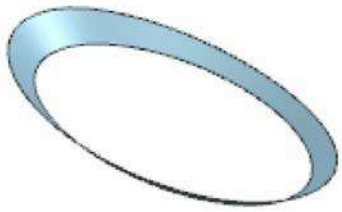


Figure 11-111 *Resulting Studio surface*

8. Invoke the **Class Selection** dialog box and select the base feature. Next,

choose the **OK** button; the selected surface is hidden.

9. Invoke the **Trim Sheet** tool and trim the base surface using the projected curve. The model after trimming the surface is shown in Figure 11-112.

10. Next, create the bounded plane surface by using the **Bounded Plane** tool, as shown in Figure 11-113.

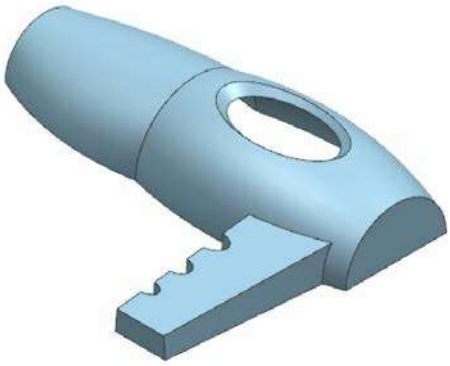


Figure 11-112 *Model after trimming the surface*

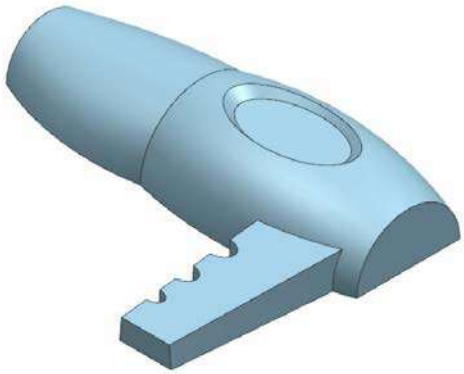


Figure 11-113 *Bounded plane surface*

Creating Air Vents

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

1. Select the newly created bounded plane surface as the sketching plane and then invoke the Sketch in Task environment.

2. Create the sketch of air vents. For dimensions, refer to Figure 11-89(b).

3. Invoke the **Trim Sheet** tool from **Menu > Insert > Trim** in the **Top Border Bar**. Then, select a sketch as the trimming tool to trim the bounded plane surface for creating air vents.

The surface model after creating air vents is displayed in Figure 11-114.

Sewing all Surfaces

After creating all surfaces, you need to sew all surfaces together and then add

fillets to surfaces and thicken the model.

1. Choose the **Sew** tool from **Menu > Insert > Combine** in the **Top Border Bar**; the **Sew** dialog box is displayed.

2. Select the **Sheet** option from drop-down list in the **Type** rollout, if it is not selected.

3. Select the base feature that is created using the **Studio Surface** tool as the target body and then select all the remaining surfaces as the tool sheet bodies.

4. Choose the **OK** button.

5. Add blends at the required places of the surface model. For dimensions, refer to Figure 11-89(a). The model after adding blends is displayed in Figure 11-115.

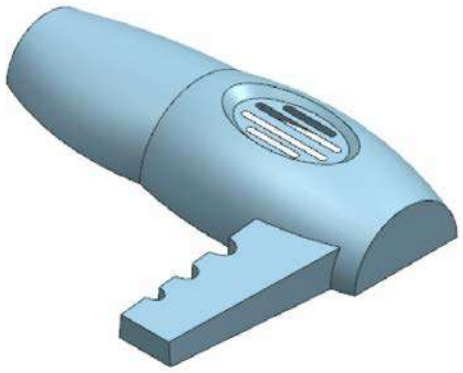


Figure 11-114 *Surface model after trimming the bounded plane surface*

Figure 11-115 *Surface model after adding blends*

Adding Thickness to the Surface Model

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

1. Choose the **Thicken** tool from **Menu > Insert > Offset/Scale** in the **Top Border Bar**; the **Thicken** dialog box is invoked and you are prompted to select the surface to thicken.

2. Select the surface model from the graphics window. Next, enter the

thickness value as **0** and **1** in the **Offset 1** and **Offset 2** edit boxes, respectively.

3. Choose the **OK** button from the dialog box. The final model after applying the thickness is shown in Figure 11-116.

Figure 11-116 *Final model*

Saving and Closing the File

1. Choose **Menu > File > Close > Save and Close** from the **Top Border Bar** to save and close the file.

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

1. The _____ tool is used to create intersection curves between two sets of objects.

2. The _____ option creates a face blend as if it was subtended by a ball rolling in constant contact with two sets of input faces.

3. The _____ option is used to specify the planes spaced at equal angles which are further used to section the selected bodies.

4. The maximum number of primary faces that can be used to create the bridge surface is _____.

5. The _____ tool is used to create the section curves between specified planes and solid bodies, surfaces, and curves.

6. Select the _____ check box to bridge the gap between two segments of a projected curve.

7. The _____ tool is used to project a closed or an open curve on one or more than one planar or curved faces.

8. The _____ tool is used to create complicated blends tangent to the specified sets of faces.

9. The **Emboss Body** tool is used to emboss the shape of one solid body onto another solid or sheet body. (T/F)

10. You can project a curve onto a plane. (T/F)

Answer the following questions:

1. Which of the following tools is used to stitch the surfaces into a single surface?

(a) **Face Blend** (b) **Emboss Body**
(b) **Bridge** (d) **Sew**

2. Which of the following tools is used to create intersection curves between two sets of objects?

(a) **Curve Intersection** (b) **Intersection Curve**
(b) **Project Curve** (d) **Section Curve**

3. Which of the following tools is used to create the sheet body that joins the two edges of faces?

(a) **Bridge** (b) **Fillet**

(c) **Face Blend** (d) None of these

4. The _____ option is used to create a face blend between three faces.

5. The _____ option is used to create section curves along the planes perpendicular to the selected curve.

6. The isoparametric blend is used for turbine blades. (T/F)

7. You can create a surface body from a closed sketch. (T/F)

8. Using the **Three-face** option in the **Face Blend** dialog box, you can create a face blend between three faces. (T/F)

9. The **Emboss Body** tool is used to emboss the shape of one solid body onto another solid or sheet body. (T/F)

10. The **Project Curve** tool is used to project curves on planar or curved faces. (T/F)

EXERCISES

Exercise 1

Create the surface model shown in Figure 11-117. The drawing views and dimensions of the surface model are shown in Figure 11-118. Assume the missing dimensions. After creating the surface model, save it with the name *c11exr1.prt* at the location *\NX\c11*. **(Expected time: 45 min)**

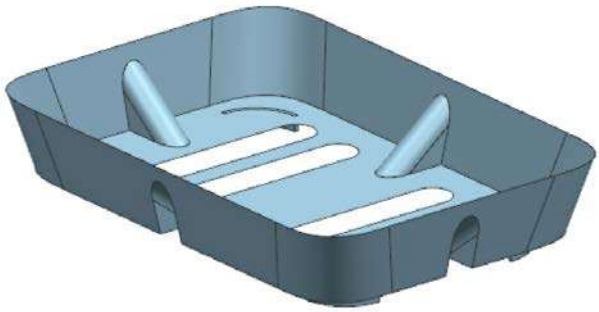


Figure 11-117 *Surface model for Exercise 1*

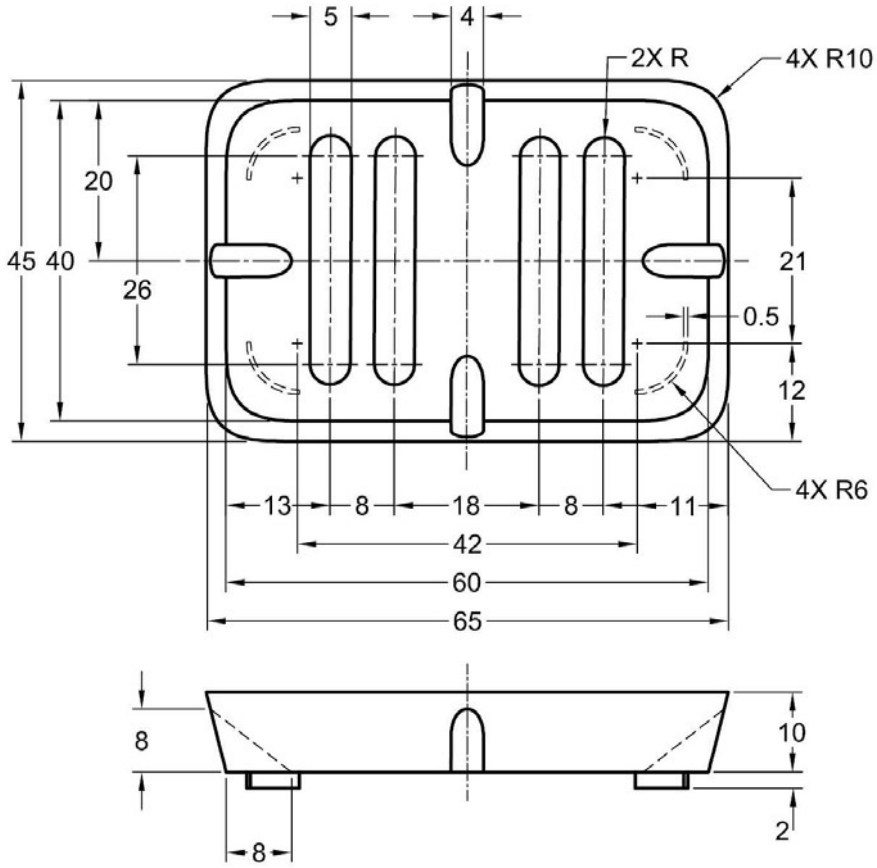


Figure 11-118 *Views and dimensions for Exercise 1*

Exercise 2

Create the surface model shown in Figure 11-119. The drawing views and dimensions of the surface model are shown in Figure 11-120. Assume the missing dimensions. After creating the surface model, save it with the name *c11exr2.prt* at the location *\NX\c11*. **(Expected time: 45 min)**

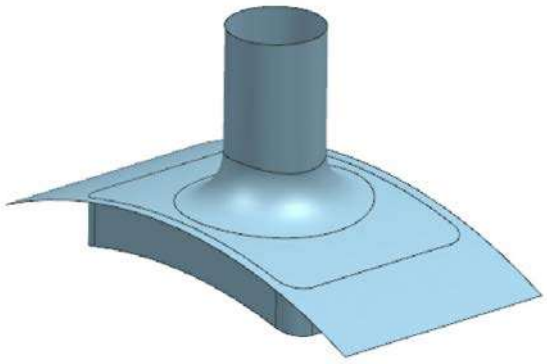


Figure 11-119 *Surface model for Exercise 2*

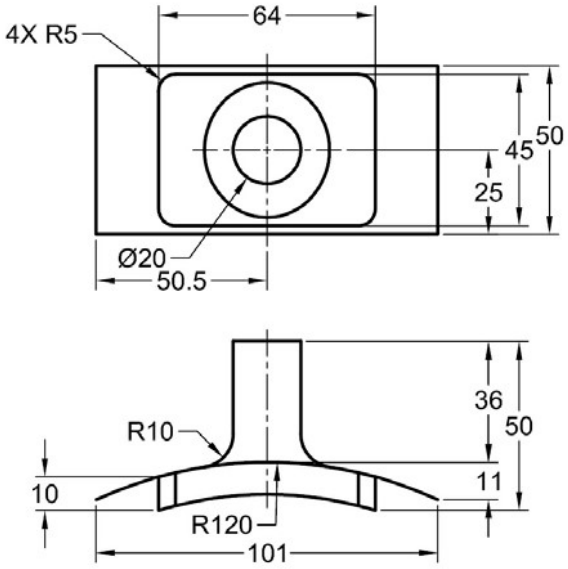


Figure 11-120 *Views and dimensions for Exercise 2*

Answers to Self-Evaluation Test

1. Intersection Curve, **2. Rolling Ball**, **3. Radial Planes**, **4.** two, **5. Section Curve**, **6. Create Curves to Bridge Gaps**, **7. Project Curve**, **8. Face Blend**, **9.** T, **10.** T