Chapter 10
Surface Modeling

INTRODUCTION TO SURFACE MODELING

Surfaces are three dimensional (3D) bodies with negligible thickness. They are used extensively for modeling complex features. A model or an assembly created using the surface body type has surface area but no volume or mass properties. In NX, surfaces are created in the form of single or multiple patches. With the increase in patches, the control over the shape of the surface also increases. In NX, surfaces are known as sheets and surface modeling is known as sheet modeling.

Most of the real world models are created using the solid modeling techniques. Only models that are complex in shape and have a non-uniform surface area are created with the help of the surface modeling techniques. The tools that are used to create solid models can also be used to create surface models. It becomes easy for the readers to learn surface modeling if they are familiar with the solid modeling tools. In NX, there is no separate application for surfaces. You need to create the surface model in the Shape Studio environment or Modeling environment. Before creating the surface model, you need to change the body type to sheet.

INVOKING THE SHAPE STUDIO ENVIRONMENT

To invoke the Shape Studio environment, choose the **New** tool from the **Standard** group of the **Home** tab; the **New** dialog box will be displayed. Choose the **Model** tab, if it is not already chosen and select the **Shape Studio** template from the **Templates** rollout of the dialog box. Next, choose the button on the right of the **Name** text box; the **Choose New File Name** dialog box will be displayed. Enter the file name in the **File name** edit box and then, choose the **OK** button to exit the dialog box. Also, to specify the location to save the file, browse to the folder where you need to save the file using the button on the right of the **Folder** edit box. After specifying the location of the file, choose the **OK** button twice; the Shape Studio environment will be displayed.

After invoking the Shape Studio or the Modeling environment, choose **Menu > Preferences > Modeling** from the **Top Border Bar**; the **Modeling Preferences** dialog box will be displayed, as shown in Figure 10-1. Choose the **General** tab, if it is not chosen by default, and then select the **Sheet** radio button from the **Body Type** area. Next, choose the **OK** button to exit this dialog box. All models created, henceforth, in the Modeling environment will be the sheet models.

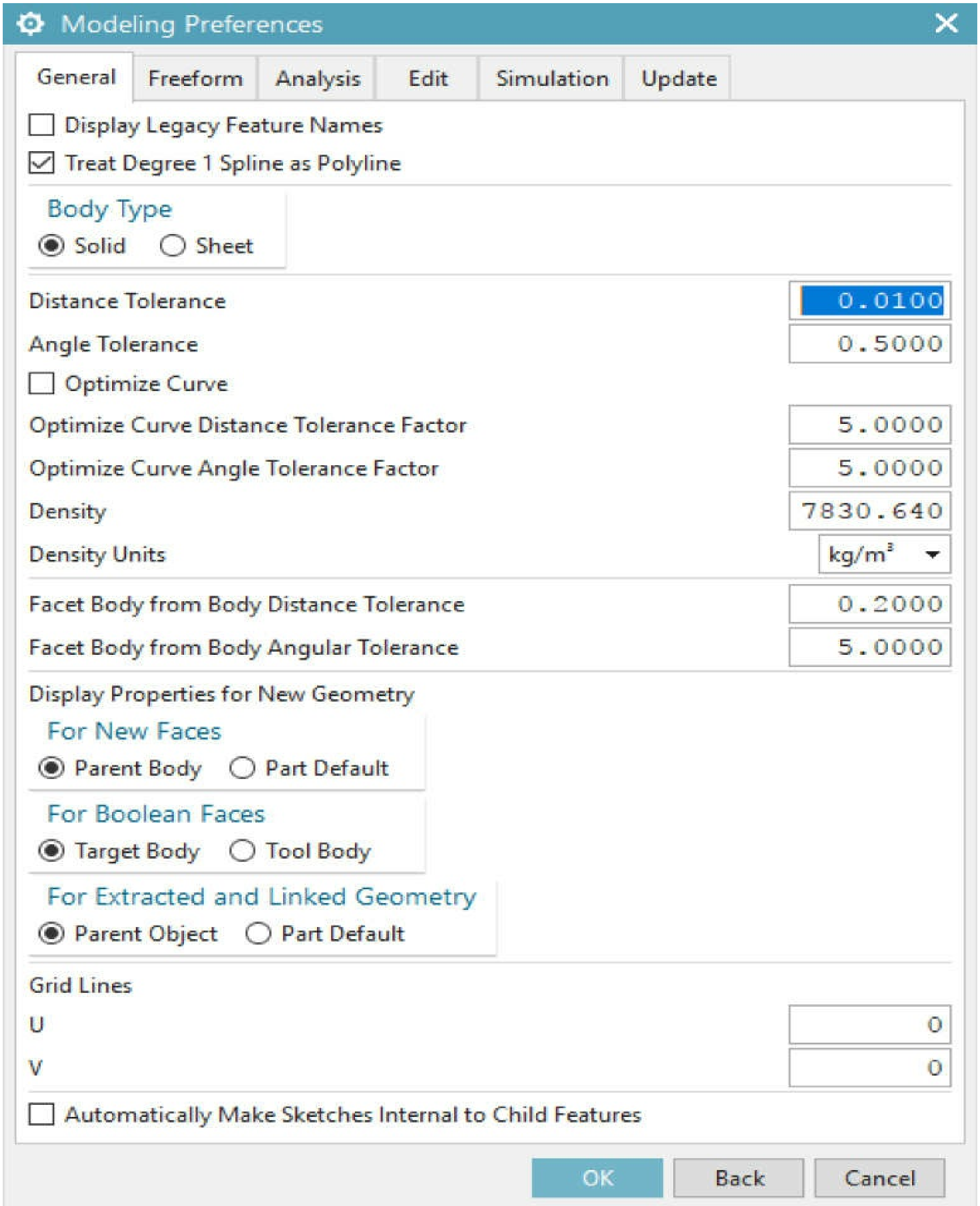


Figure 10-1 *The **Modeling Preferences** dialog box*

Creating an Extruded Surface

Ribbon: Home > More Gallery > Design Feature > Extrude
Menu: Insert > Design Feature > Extrude

As mentioned earlier, there is no separate tool available for creating an extruded surface. After invoking the Shape Studio environment, you can use the **Extrude** tool to create extruded sheets. The sketch drawn for creating an extruded surface may be an open or a closed entity. After creating the sketch, choose the **Extrude** tool from **Menu > Insert > Design Feature** in the **Top Border Bar**; the **Extrude** dialog box will be displayed and you will be prompted to select the section geometry to extrude. Select the sketch and enter the start and end extrusion values in their respective **Distance** edit boxes available below the **Start** and **End** drop-down lists of the **Limits** rollout in the dialog box. Next, choose the **OK** button from the **Extrude** dialog box; a sheet will be created. The options in the **Extrude** dialog box are the same as those discussed in Chapter 4. Figures 10-2 and 10-3 show the extruded surfaces created by using the open and closed sketches, respectively.

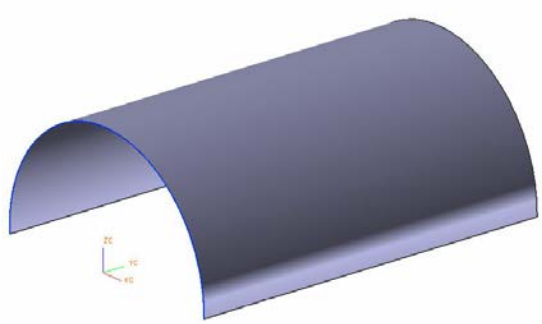


Figure 10-2 *Extruded surface created on an open sketch*

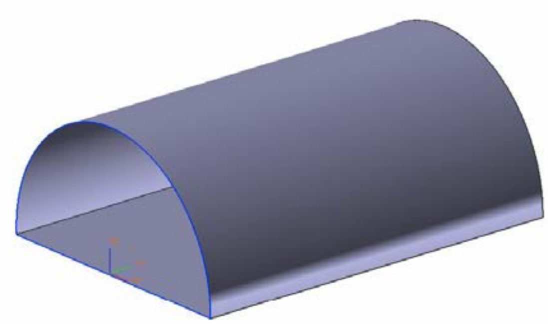


Figure 10-3 *Extruded surface created on a closed sketch*

Note

*You can use only the **None** and **Inferred** options from the **Boolean** drop-down list in the Shape Studio environment. The other options of this drop-down list are not available in this environment.*

Creating a Revolved Surface

Ribbon: Home > More Gallery > Design Feature > Revolve
Menu: Insert > Design Feature > Revolve

The **Revolve** tool is used to create a revolved surface. To create a revolved surface, first create a sketch, and then choose the **Revolve** tool from **Menu > Insert > Design Feature** in the **Top Border Bar**; the **Revolve** dialog box will be displayed. Also, you will be prompted to select the section geometry. Select the sketch drawn for the revolved surface. Next, click on the **Specify Vector** area in the **Axis** rollout of the dialog box and specify the axis of revolution; you will be prompted to select object to infer point. Specify the point where you want to locate the vector and then specify the start and end angles in the **Angle** edit boxes. Next, choose the **OK** button; a revolved surface will be created. The revolved surface models created by using an open sketch and a closed sketch are shown in Figures 10-4 and 10-5, respectively.

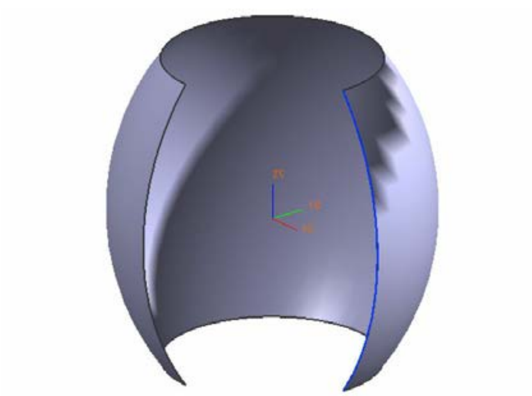


Figure 10-4 *Revolved surface created using an open sketch*

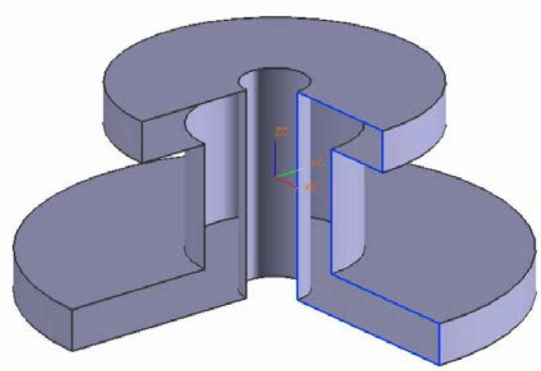


Figure 10-5 *Revolved surface created using a closed sketch*

Creating a Combined Surface

Menu: Insert > Combine > Combine

You can use the **Combine** tool to trim and join regions of multiple sheet bodies. To do so, choose **Menu** > **Insert** > **Combine** from the **Top Border Bar**; the **Combine** dialog box will be displayed, refer to Figure 10-6.

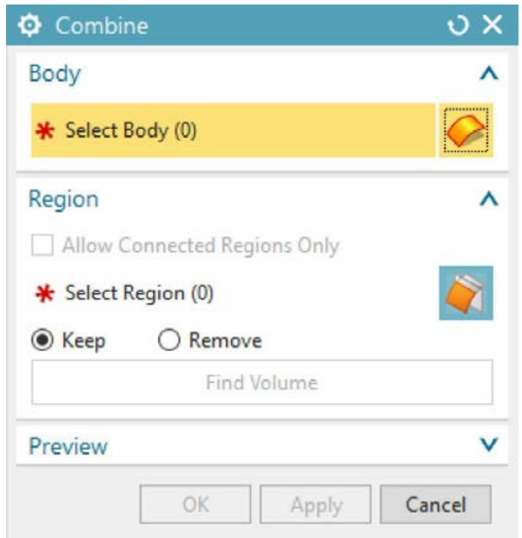


*Figure 10-6 The **Combine** dialog box*

The rollouts available in the **Combine** dialog box are discussed next.

Body Rollout

The **Body** button in this rollout is activated by default. As a result, you are prompted to select the bodies to be combined. Select the bodies to be combined using this rollout.

Region Rollout

The options available in this rollout are discussed next.

Allow Connected Regions Only

If this check box is selected, you can only select the bodies that are edge connected to a previously selected body. On clearing this check box, you can select any body created during the session.

Select Region

This area allows you to select the region of a body to keep or remove. If you select the **Keep** radio button, the selected side of the bodies will be retained and if you select the **Remove** radio button; the selected side of the bodies will be trimmed.

Find Volume

This button allows you to find and select regions that enclose a solid volume, or connected regions that form a volume-like sheet.

Figure 10-7 shows sheet bodies selected to be combined. Figure 10-8(a) shows the resulting body when the **Keep** radio button is selected and Figure 10-8(b) shows the resulting body when the **Remove** radio button is selected.

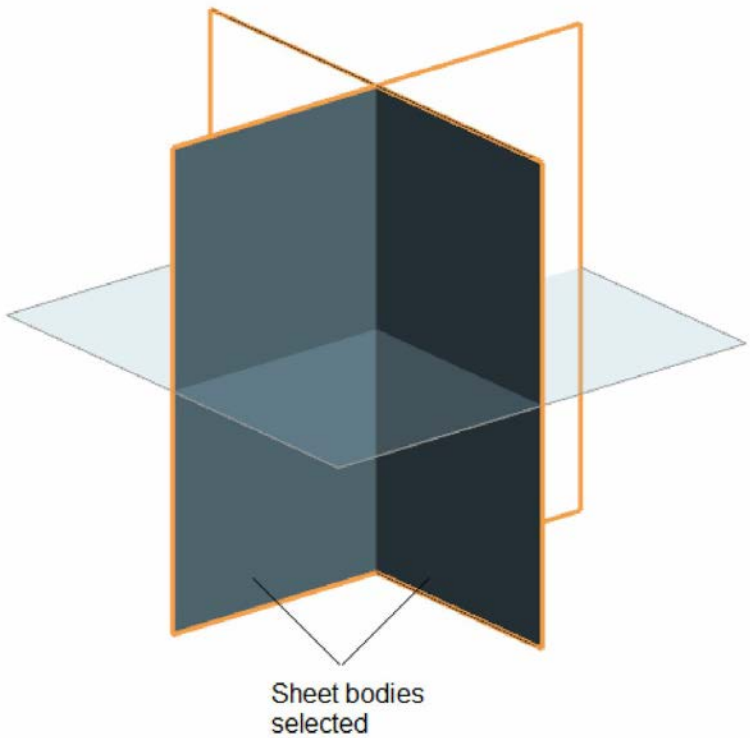


Figure 10-7 *Sheet bodies selected to be combined*

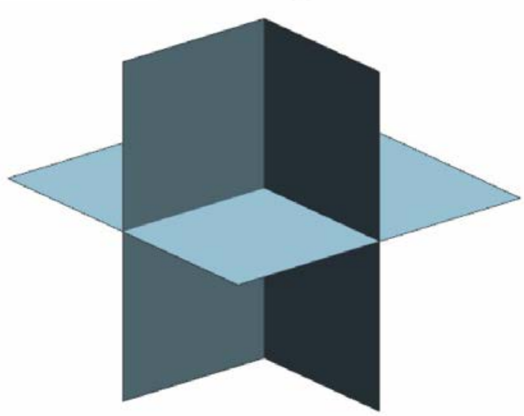


Figure 10-8(a) *Resulting sheet body when the **Keep** radio button is selected*

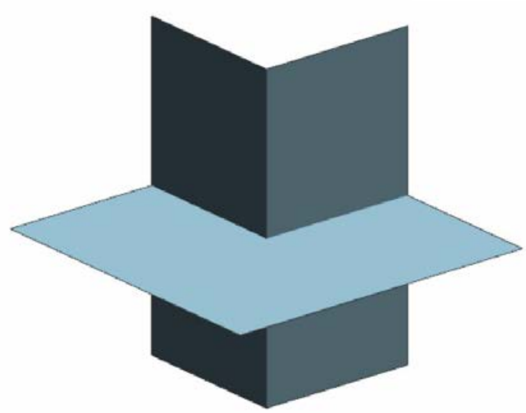


Figure 10-8(b) *Resulting sheet body when the **Remove** radio button is selected*

Creating a Surface Using the Through Curves Tool

Ribbon: Home > More Gallery > Mesh Surface > Through Curves

Menu: Insert > Mesh Surface > Through Curves

You can create surfaces with multiple section strings using the **Through Curves** tool. This method allows you to select any number of section strings. To do so, choose the **Through Curves** tool from the **Mesh Surface** gallery of the **More** gallery, refer to Figure 10-9; the **Through Curves** dialog box will be displayed, as shown in Figure 10-10, and you will be prompted to select the curve or point to section. Select the strings for first section and press the middle mouse button; you will be prompted again to select the curve or point to section. Likewise, you can select any number of section strings. After selecting section strings, make sure that all the arrows on the section strings point in the same direction. All the selected sections will be listed in the **List** sub-rollout of the **Sections** rollout. You can reorder a selected section by using the **Move Up** and **Move Down** buttons available on the right of the **List** sub-rollout. You can also delete a selected section by using the **Remove** button.

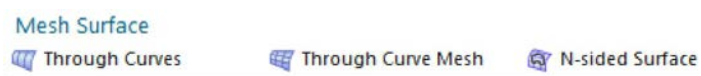


Figure 10-9 *The **Mesh Surface** gallery of the **More** gallery*

In the **Patch Type** drop-down list of the **Output Surface Options** rollout, there are three options, **Single**, **Multiple**, and **Match String**. If you select the **Single** option, a surface will be created with a single patch. If you select the **Multiple** option, the surface will be created with multiple patches. The number of patches formed depends upon the **Alignment** option selected from the **Alignment** rollout.

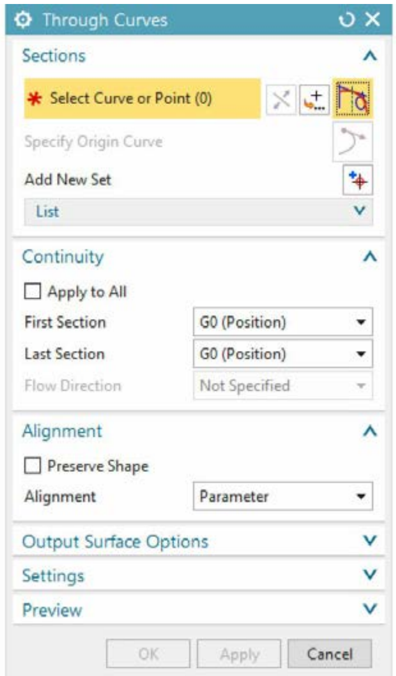


***Figure 10-10** The **Through Curves** dialog box*

When you select the **Multiple** option from the **Patch Type** drop-down list of the **Output Surface Options** rollout, the **Closed in V** and **Normal to End Sections** check boxes will be enabled. If you select the **Closed in V** check box, the surface body will be closed in the V direction and the **Normal to End Sections** check box will be deactivated. If you select the **Normal to End Sections** check box, the resultant surface will be normal to the two end sections and the options in the **Continuity** rollout will be deactivated. Figure 10-11 shows the section strings selected for creating a through curve surface and Figure 10-12 shows the resulting surface.

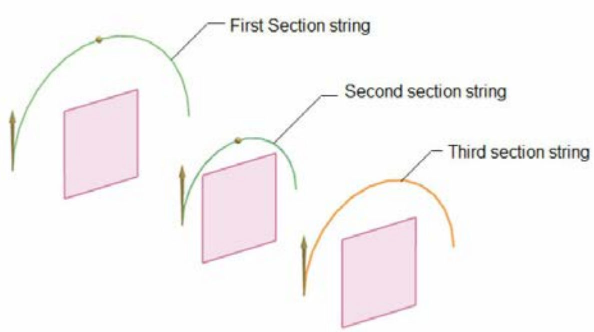


Figure 10-11 Section strings selected for creating the Through Curves surface

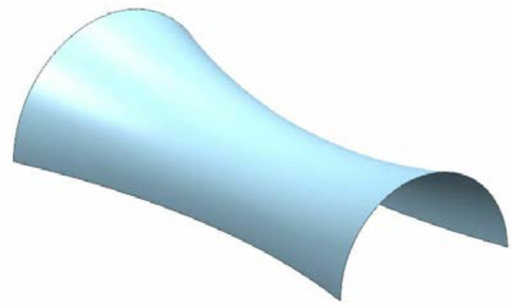


Figure 10-12 The resulting surface

Creating a Surface Using the Through Curve Mesh Tool

Ribbon: Home > More Gallery > Mesh Surface > Through Curve Mesh
Menu: Insert > Mesh Surface > Through Curve Mesh

You can create surfaces by specifying section strings and guide strings using the **Through Curve Mesh** tool. While using the **Through Curve Mesh** tool, any number of section strings and guide strings can be specified. For selecting multiple guide strings, it is required that they are connected end-to-end. To create a surface by using the **Through Curve Mesh** tool, invoke this tool from the **Mesh Surface** gallery of the **More** gallery; the **Through Curve Mesh** dialog box will be displayed, as shown in Figure 10-13, and you will be prompted to select primary curves. You need to select a collection of control curves. Select the first primary curve and press the middle mouse button to select the next curve. Similarly, you can select any number of primary curves. Next, choose the **Cross Curves** button from the **Cross Curves** rollout; you will be prompted to select cross curves. Select the first cross curve and press the middle mouse button to select the next curve. Similarly, you can select any number of cross curves.

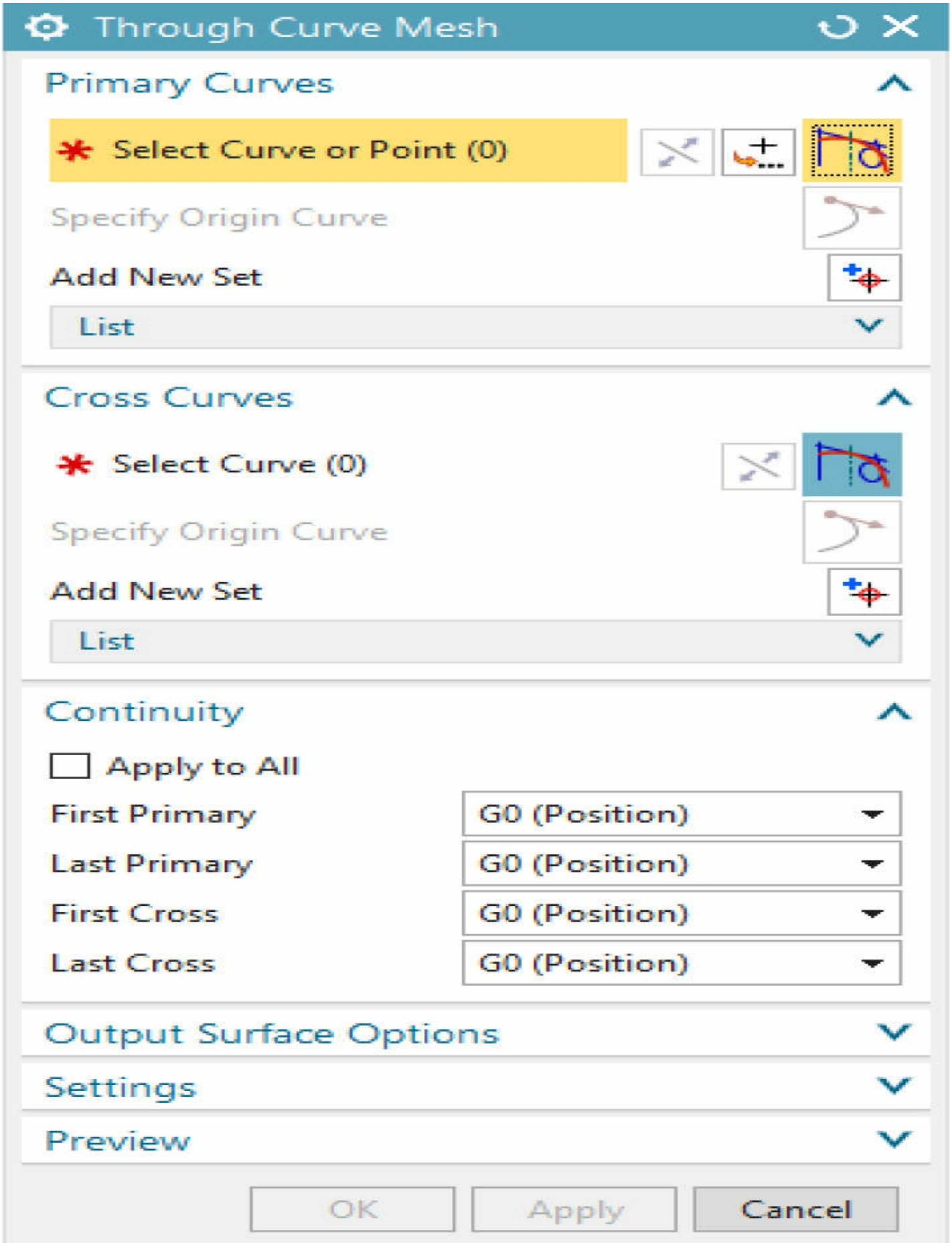


Figure 10-13 *The* ***Through Curve Mesh*** *dialog box*

Note that after selecting two primary curves, the **Spine** rollout will be added to the **Through Curve Mesh** dialog box. The **Spine** button in this rollout allows you to select the spine string. This spine string improves the smoothness of the surface and it must be normal to all primary strings. However, the selection of the spine string is optional. If you want to skip this step, do not choose this button.

Output Surface Options Rollout

The options in the **Emphasis** drop-down list of the **Output Surface Options** rollout are used to define the set of curves that affect the shape of the surface to be created. Select the **Both** option from the **Emphasis** drop-down list; the primary curves and cross curves will cast an equal effect. If you select the **Primary** option from the **Emphasis** drop-down list, the primary curves will cast more effect and if you select the **Cross** option from the **Emphasis** drop-down list, the cross curves will cast more effect. If you select the **Normal** option from the **Construction** drop-down list in the **Output Surface Options** rollout, the resulting surfaces will have more number of patches. If you select the **Spline Points** option, the resulting surface will have less number of patches. The surface is formed by reparameterizing curves into temporary curves. If you select the **Simple** option, the resulting surface will be created with or without specifying any constraints.

Settings Rollout

The options in the **Rebuild** drop-down list of the **Settings** rollout will only be enabled if you select the **Normal** option from the **Construction** drop-down list. You can use the options in the **Rebuild** drop-down list to join the mesh surface smoothly with the surrounding surfaces. You can rebuild the mesh surface by selecting the **Degree and Tolerance** option and entering the value in the **Degree** spinner. If you select the **Auto Fit** option, the **Maximum Degree** and **Maximum Segments** spinners will be enabled. You can set the values in these spinners to rebuild the mesh surface automatically.

Figure 10-14 shows the control strings selected for creating the through curve mesh surface and Figure 10-15 shows the resulting surface. You can enter the distance tolerance value between the curves in the **G0 (Position)** edit box and the angle tolerance value in the **G1 (Tangent)** edit box. The curvature tolerance value can be entered in the **G2 (Curvature)** edit box.

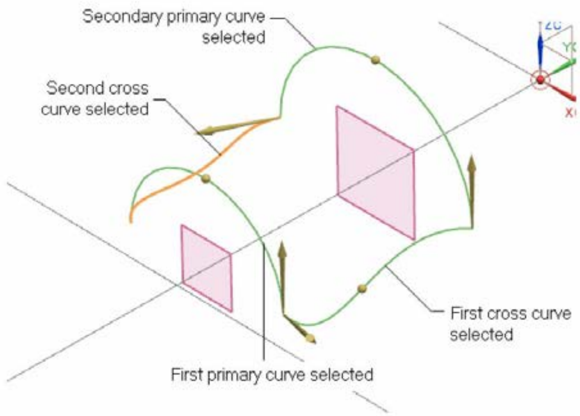


Figure 10-14 *The control strings selected for creating the through curve mesh surface*

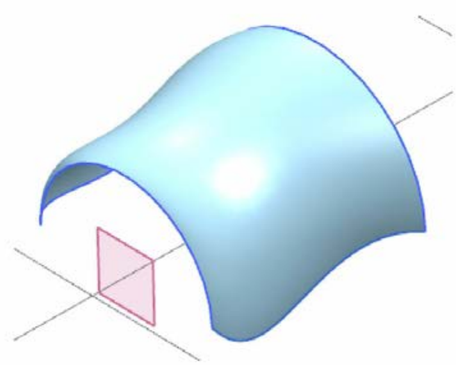


Figure 10-15 *The resulting through curve mesh surface*

Creating a Surface Using the Four Point Surface Tool

Ribbon: Home > Create > Surface Drop-down > Four Point Surface
Menu: Insert > Surface > Four Point Surface

The **Four Point Surface** tool is used to create a planar (2D) or non-planar (3D) surface. To create a surface by using this tool, choose the **Four Point Surface** tool from the **Surface Drop-down** of the **Create** group; the **Four Point Surface** dialog box will be displayed, as shown in Figure 10-16, and you will be prompted to select object to infer point. Specify the point for the first surface corner. Similarly, specify the other three surface corners and choose the **OK** button; the four point surface will be created. You can also redefine the previously selected corner points. To do so, choose the button corresponding to the point that you want to redefine from the **Surface Corners** rollout; the respective point will be highlighted in the drawing window. Again, specify the point for the corner. Figure 10-17 shows the corner points to be selected for creating a surface. Figure 10-18 shows the resulting surface formed by enclosing the specified corner points.

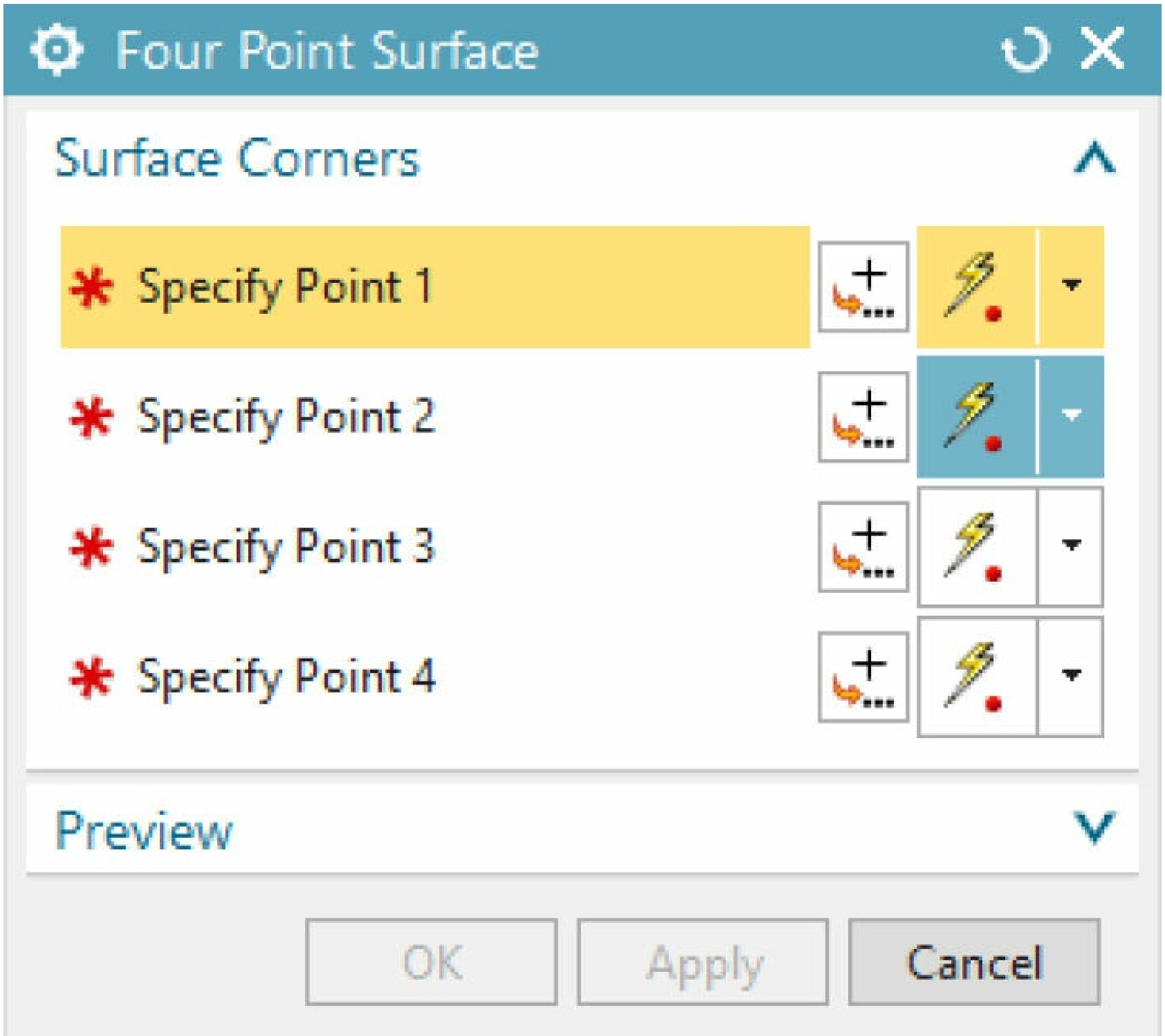


***Figure 10-16** The **Four Point Surface** dialog box*

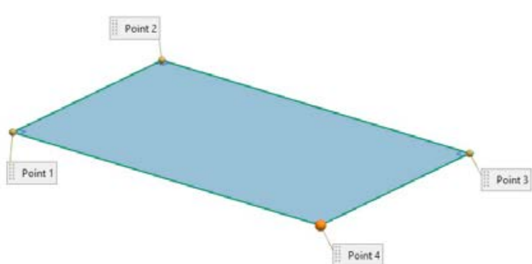


***Figure 10-17** The four corner points to be selected for creating a surface*

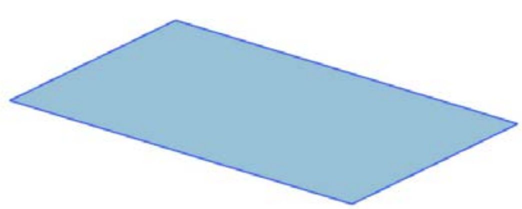


***Figure 10-18** The resulting surface*

Creating Planar Surfaces from 2D Sketches and Edges of Solid or Surface

Ribbon: Home > More Gallery > Surface > Bounded Plane
Menu: Insert > Surface > Bounded Plane

The **Bounded Plane** tool is used to create a surface from 2D sketches or closed coplanar edges. If you need to enclose a 2D sketch or closed coplanar edges with a surface, choose **Menu > Insert > Surface > Bounded Plane** in the **Top Border Bar**; the **Bounded Plane** dialog box will be displayed, as shown in Figure 10-19, and you will be prompted to select curves for the bounded plane. Select the closed coplanar edges of the object or the closed coplanar sketch and then choose the **OK** button; the bounded plane surface will be created. Figure 10-20 shows a bounded plane surface enclosing a 2D sketch and Figure 10-21 shows a bounded plane surface created from a circular edge. You can create a bounded plane surface by selecting the closed coplanar edges of the solid and surface bodies.

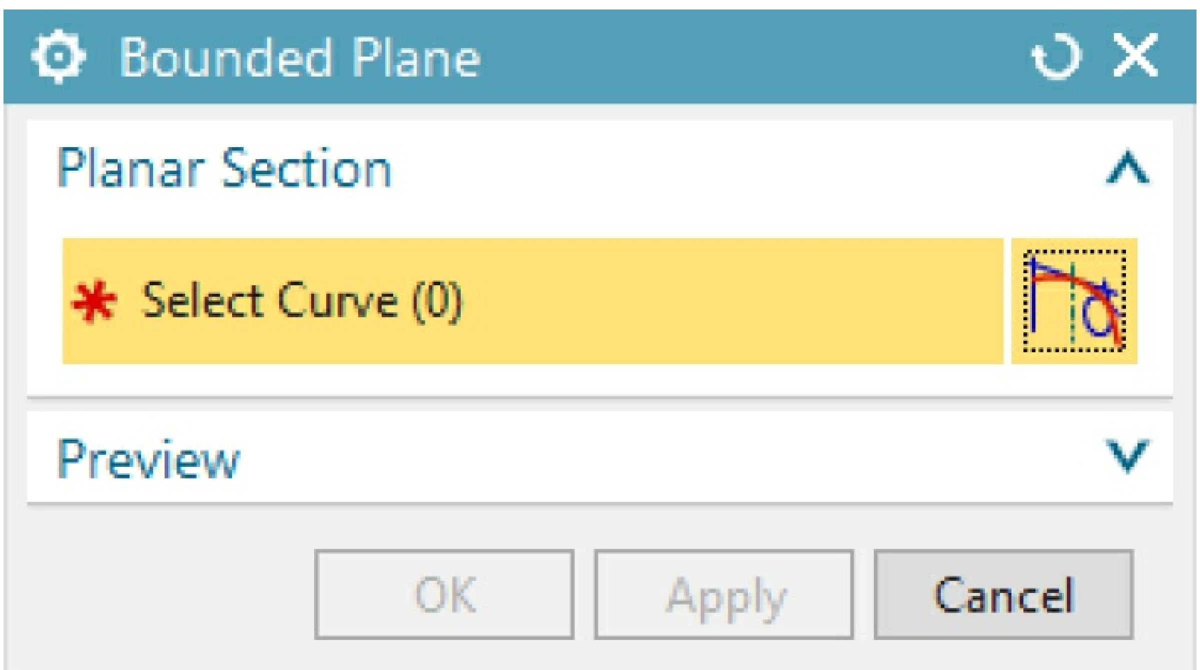


***Figure 10-19** The **Bounded Plane** dialog box*

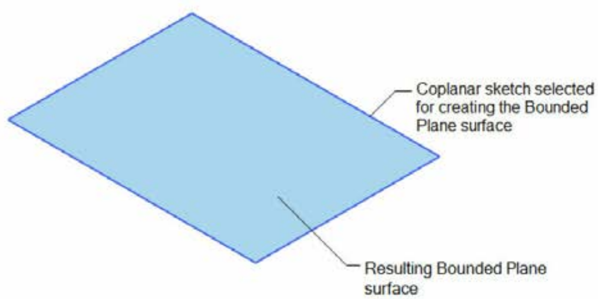


Figure 10-20 The Bounded Plane surface formed from a 2D sketch

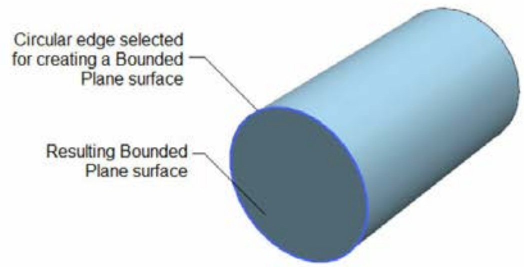


Figure 10-21 The Bounded Plane surface formed from a circular edge

Creating a Transition Surface Using the Transition Tool

Ribbon: Home > More Gallery > Surface > Transition *(Customize to add)*
Menu: Insert > Surface > Transition

Generally, creation of a transition surface involves the selection of required cross-sections and mapping the intersected surface automatically formed between the selected cross-sections. You can define the shape constraint for a connecting (intersecting) surface. To create a transition surface, you need to create two or more than two cross-sections. After creating cross-sections, choose the **Transition** tool from **Menu > Insert > Surface** in the **Top Border Bar**; the **Transition** dialog box will be displayed, as shown in Figure 10-22, and you will be prompted to select curves/edges to section because the **Curve** button in the **Sections** rollout is chosen by default. Select sections and choose the **OK** button. Note that you need to press the middle mouse button after selecting every section. After selecting sections, the wireframe preview of the resultant model will be displayed. Figure 10-23 shows the wireframe view of the resultant model and Figure 10-24 shows the resulting surface.

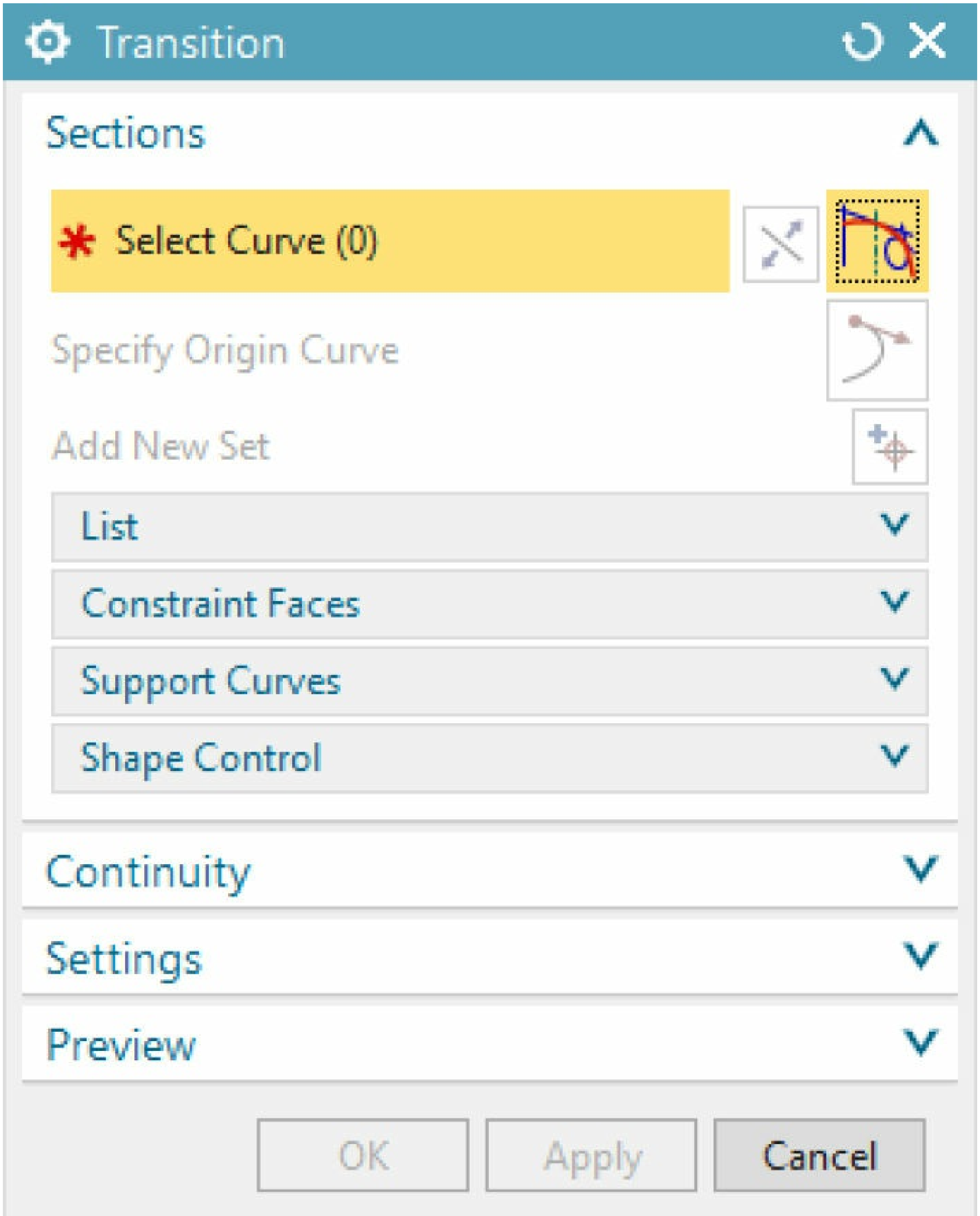


Figure 10-22 *The **Transition** dialog box*

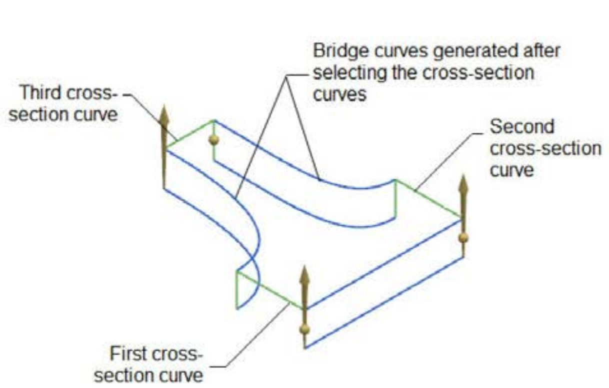


Figure 10-23 The wireframe view of the resulting transition surface created from cross-sections

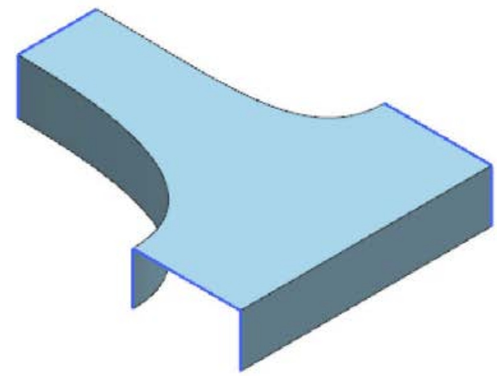


Figure 10-24 The resulting transition surface created from cross-sections

Constraint Faces Sub-rollout and Continuity and Preview Rollouts

To specify constraint surfaces, choose the **Face** button from the **Constraint Faces** sub-rollout; you will be prompted to select the continuity constraint face for the selected section. Select the required face to specify constraint surfaces. By default, the **G1 (Tangent)** option is selected in the **Continuity** drop-down list of the **Continuity** rollout. As a result, there is tangential continuity constraint with the intersected surface. If you select the **G0 (Position)** option from the **Continuity** drop-down list, the positioned continuity will be maintained. If you select the **G2 (Curvature)** option from the **Continuity** drop-down list, the curvature continuity will be maintained. The **Show Result** button in the **Preview** rollout is used to display the preview of the intersected surface to be created. By default, the **Create Surface** check box is selected in the **Settings** rollout. As a result, a transition surface will be created. If you clear this check box, only bridge curve will be formed between cross-sections.

Support Curves

In this sub-rollout, the **Show All Points on Section** check box is cleared by default. If you select this check box, all the section points in the list box of the **Support Curves** sub-rollout will be displayed. Select any point other than **Point 1** in the list box; the **Add** button will be activated. Choose this button; a new section point will be added to the list box as well as to the selected section. You can move this new section point by dragging. To remove the created section point, select it and choose the **Remove** button from this sub-rollout.

Shape Control

The bridge curves formed after selecting the cross-sections of the surfaces are listed as individual curves and separate groups in the **Bridge Curve** drop-down list of this sub-rollout. You can select the required bridge curve from the **Bridge Curve** drop-down list. By selecting the required curve from the **Bridge Curve** drop-down list, you can control the shape of the selected bridge curve in two ways: using the **Tangent Magnitude** and the **Depth And Skew** options available in the **Type** drop-down list. If you select the **Tangent Magnitude** option, you can control the shape of the selected curve from the start point or the end point by sliding the **Start** or **End** slider bars. If you select the **Depth And Skew** option, then the **Depth** and **Skew** slider bars will be available in this sub-rollout to control the depth and the skew angle of the selected bridge curve.

Creating an N-sided Surface

Ribbon: Home > More Gallery > Mesh Surface > N-sided Surface
Menu: Insert > Mesh Surface > N-sided Surface

The **N-sided Surface** tool is used to create a single patch surface or multi-patch triangular surfaces that enclose a closed 2D sketch or a closed 3D curve. While doing so, an existing surface can optionally be selected as a reference for maintaining the shape of the surface to be created. To create an N-sided surface, choose the **N-sided Surface** tool from **Menu > Insert > Mesh Surface** in the **Top Border Bar**; the **N-sided Surface** dialog box will be displayed, as shown in Figure 10-25, and you will be prompted to select a closed loop of curves or edges. By default, the **Trimmed** option is selected in the drop-down list in the **Type** rollout. As a result, a surface with a single patch will be created. To create a surface with multiple triangular patches, you need to select the **Triangular** option from the drop-down list in the **Type** rollout. Both the options used for creating the N-sided surface are discussed next.

Trimmed

By default, the **Curve** button is chosen in the **Outer Loop** rollout. As a result, you will be prompted to select a closed loop. Select a closed boundary of a 2D sketch, edges, or a 3D curve; the preview of the new surface will be displayed. Next, select the **Trim to Boundary** check box from the **Settings** rollout; the surface created will automatically be trimmed with respect to the closed loop of the curve, as shown in Figure 10-26.

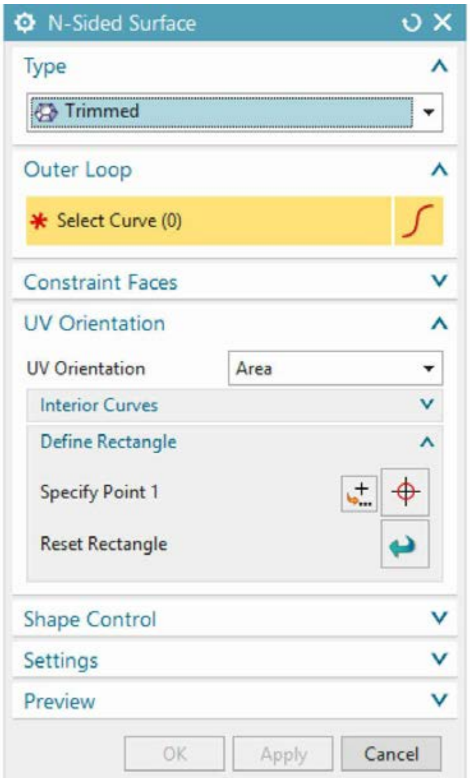


*Figure 10-25 The **N-Sided Surface** dialog box*

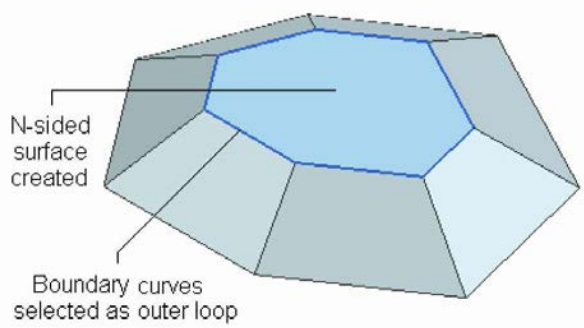


Figure 10-26 Single patch N-sided surface created for the selected boundary curve

By default, the **Area** option is selected in the **UV Orientation** drop-down list of the **UV Orientation** rollout. Click on the **Select Curve** area in the **Interior Curves** sub-rollout to activate it; you will be prompted to select a curve. Select the curve; the surface will be modified such that it passes through the selected interior curves; thereby, deforming the shape of the surface, as shown in Figure 10-27. In this figure, the **Trim to Boundary** check box is cleared for better understanding of the deformation of the surface. You can also define a rectangle by specifying two points as diagonally opposite corners of the rectangle so that the resultant surface is created in the specified rectangle. To do so, click on the **Specify Point 1** area in the **Define Rectangle** sub-rollout; the **Specify Point 1** area will be activated. Next, click in the drawing window; a rectangle will be attached to the cursor. Also, the **Specify Point 2** area will be activated in the **Define Rectangle** sub-rollout. Again, click in the graphics window to specify the second point of the rectangle; a square surface will be created in the graphics window. To reset the rectangle created, choose the **Reset Rectangle** button from the **Define Rectangle** sub-rollout.

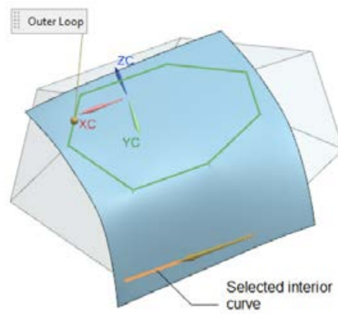


Figure 10-27 *Modified part with interior curve selected*

If you select the **Spine** option from the **UV Orientation** drop-down list in the **UV Orientation** rollout, then the **UV Orientation** rollout will be modified and the options in the **Shape Control** rollout will become active. Click on the **Select Curve** area of the **Spine** sub-rollout; you will be prompted to select a spine curve. Select a curve; the surface will be oriented perpendicular to the selected spine curve. The **Center Flat** slider bar in the **Center Control** sub-rollout of the **Shape Control** rollout is used to modify the shape of the surface created with respect to the selected curve, as shown in Figure 10-28. To reset the options in the **Shape Control** rollout, choose the **Reset Center Control** button from this rollout.

If you select the **Vector** option from the **UV Orientation** drop-down list in the **UV Orientation** rollout, the **UV Orientation** rollout will be modified. Click on the **Specify Vector** area of the **Vector** sub-rollout; you will be prompted to select the object to infer vector. Select a vector; the surface will follow the selected vector direction. The **Center Flat** slider bar in the **Center Control** sub-rollout of the **Shape Control** rollout is used to modify the shape of the surface created with respect to a specified vector, as shown in Figure

10-29. To reset the options in the **Shape Control** rollout, choose the **Reset Center Control** button from the **Shape Control** rollout.

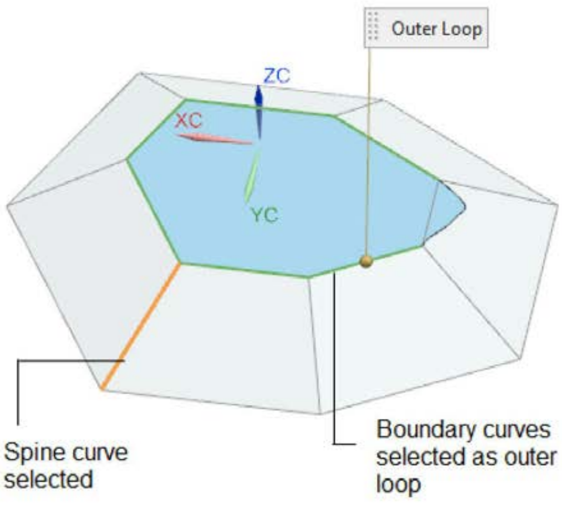


Figure 10-28 Modified part with the spine curve selected

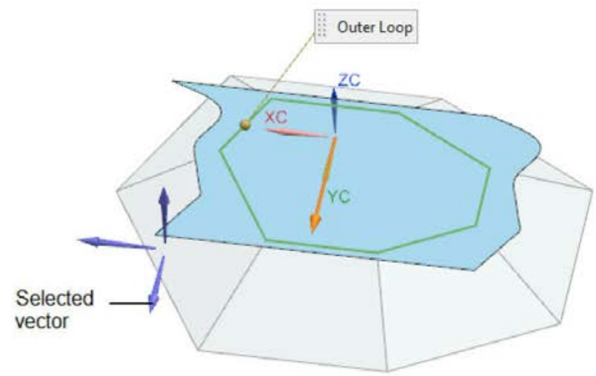


Figure 10-29 Modified part with the selected vector

Triangular

To create a triangular patch surface, select the **Triangular** option from the drop-down list in the **Type** rollout; you will be prompted to select a chain of curves for outer loop. Select the closed entity; the preview of the selected surface will be displayed in the drawing window, as shown in Figure 10-30, and the **Shape Control** and **Settings** rollouts will be modified.

By default, the **Position** option is selected in the **Control** drop-down list of the **Center Control** sub-rollout in the **Shape Control** rollout. You can move the center point of the new surface in the X, Y, and Z directions by using the **X**, **Y**, and **Z** slider bars, respectively, as shown in Figure 10-31. You can specify the flow direction of the new surface as per your requirement by selecting any one of the following options from the **Flow Direction** drop-down list: **Not Specified**, **Perpendicular**, **Iso U/V Line**, or **Adjacent Edges**.

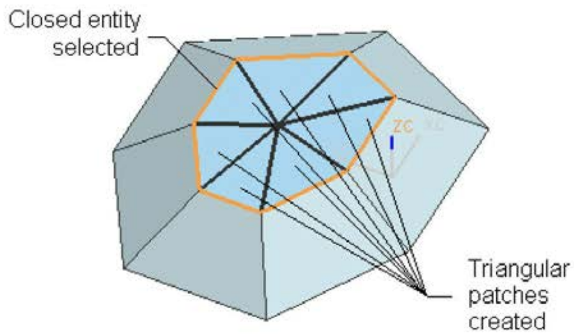


*Figure 10-30 Triangular patches created using the **Triangular** option*

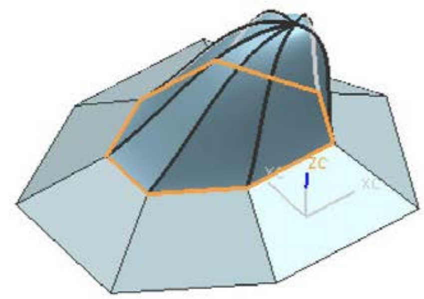


Figure 10-31 Modified surface in the X, Y, and Z directions

Select the **Tilting** option from the **Control** drop-down list and then use the **X** and **Y** slider bars to tilt the created surface in the X and Y directions, respectively, as shown in Figure 10-32. By default, the **Merge Faces if Possible** check box in the **Settings** rollout is clear. As a result, patches are created for each edge of the loop. If you select this check box, the patches of the loop will be removed by treating the tangent continuous edges as a single loop, as shown in Figure 10-33.

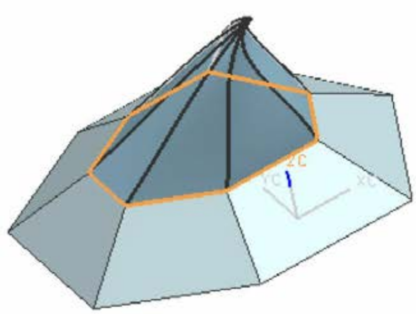


Figure 10-32 The tilted surface in the X and Y directions

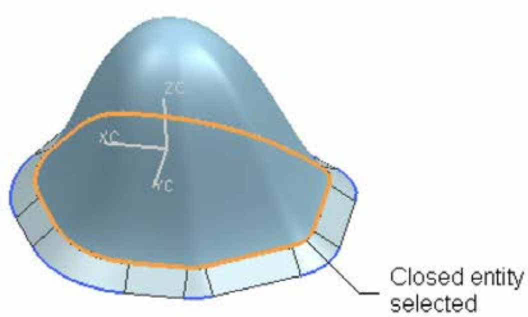


*Figure 10-33 The curves of the loop merged by selecting the **Merge Faces if Possible** check box*

Creating a Silhouette Flange Surface

Ribbon: Home > More Gallery > Flange Surface > Silhouette Flange
Menu: Insert > Flange Surface > Silhouette Flange

The silhouette flange surfaces are created with respect to an existing surface such that the aesthetic shape, quality, and the slope continuity of the existing surface are maintained. The flange surface is created with a full round surface or a fillet at the start point. The flange created can be dynamically modified in shape and size. The silhouette flange surface can be created by using any of the three methods discussed next.

Creating a Silhouette Flange Surface Using the Basic Method

The **Silhouette Flange** tool is used to create silhouette flange surfaces on an edge or on a curve by taking any of the adjacent surfaces as reference. To create a silhouette flange surface, invoke the **Silhouette Flange** tool from the **Flange Surface** gallery of the **More** gallery, refer to Figure 10-34; the **Silhouette Flange** dialog box will be displayed, as shown in Figure 10-35, and you will be prompted to select curve or edges to define base curve. Select the curve or the edge on which you want to create a flange. By default, the **Basic** option is selected in the drop-down list of the **Type** rollout and the **Curve** button is chosen in the **Base Curve** rollout. Selecting the **Basic** option enables you to create a flange without the help of any other existing flange surfaces. Select an edge or a curve for creating the silhouette flange surface and choose the **Face** button from the **Base Face** rollout; you will be prompted to select the face that will act as base face. Select the desired face. The other options in this dialog box are discussed next.

*Figure 10-34 The **Flange Surface** gallery of the **More** gallery*

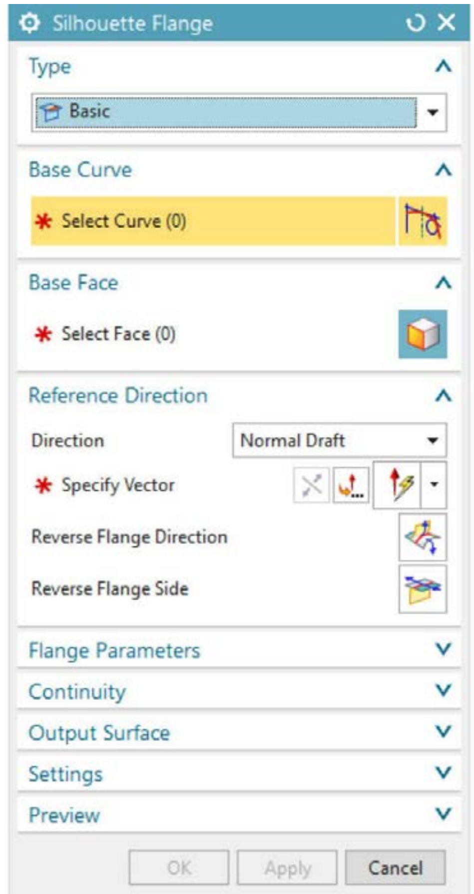


*Figure 10-35 The **Silhouette Flange** dialog box*

Reference Direction Rollout

In this rollout, you can specify the direction of a flange by selecting any one of the options from the **Direction** drop-down list. These options are **Face Normal**, **Vector**, **Normal Draft**, and **Vector Draft**. By default, the **Normal Draft** option is selected in the **Direction** drop-down list. Click on the **Specify Vector** area of this rollout to activate it, and then select the vector; the preview of the flange surface will be displayed, as shown in Figure 10-36. To change the direction of the flange to opposite direction, choose the **Reverse Flange Direction** button from this rollout; the direction of the flange will be reversed. To switch the flange extension to the opposite side of the bend, select the **Reverse Flange Side** button from this rollout. If you do not get the desired result after choosing this button, choose the **Reverse Direction** button from this rollout.

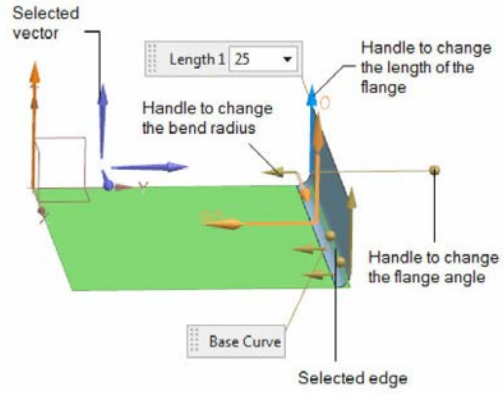


Figure 10-36 *Preview of the flange surface*

Flange Parameters Rollout

You can use the options in this rollout to control the parameters of a flange. Alternatively, you can control the parameters of the flange by using the handle and angular handles in the drawing window. If you select a handle from the drawing window, then the respective dynamic edit box will be displayed. You can enter value in the edit box or drag the handle to modify the respective parameters of the flange.

If you select the **Multi-transition** option from the **Law Type** drop-down list, the **Specify New Location** area is activated in the **Length** sub-rollout. As a result, you will be prompted to select the object to infer point. Select the point on the base curve; a control point and a dynamic edit box will be displayed. Enter the desired value in this edit box to specify the location of the point on curve. To change the radius at this point, drag the handle pointing normal to the flange; the radius at that point will be changed. To change the length of the flange at the selected point, drag the handle pointing parallel to the flange. To change the bend angle of the flange, drag the angular handle; the bend angle of the flange will be changed.

You can change the transition type of the bend radius of the flange to modify the bend radius by selecting the options (**Constant**, **Linear**, **Blend**, and **Minimum/Maximum**) from the **Values along Spine** sub-rollout of the **Flange Parameters** rollout. You can change the transition type of the length of the flange by selecting the options (**Constant**, **Linear**, **Cubic**, **Multi-transition**) from the **Law Type** drop-down list of the **Length** sub-rollout, refer to Figure 10-37.

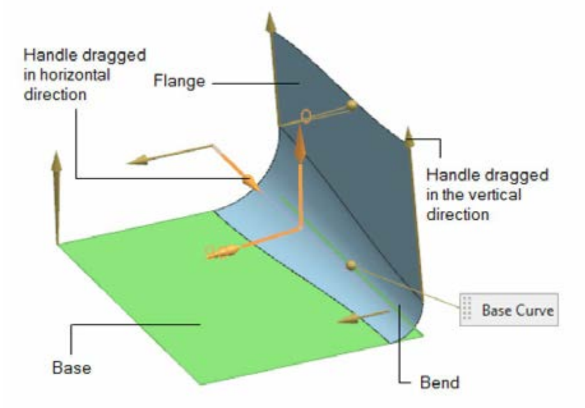


*Figure 10-37 Part modified using the **Flange Parameters** and **Continuity** rollouts*

Continuity Rollout

You can control the continuity between the base and the bent portion using the options in the **Base and Pipe** sub-rollout of the **Continuity** rollout. To do so, select the required **G1 (Tangent)**, **G2 (Curvature)**, and **G3 (Flow)** continuities in the **Continuity** drop-down list of the **Base and Pipe** sub-rollout. To control the amount of edge shift, you can use the **Lead-in** slider bar. Alternatively, you can use the **Lead-in** edit box to control the edge shift. Similarly, you can control the continuity between the flange and the bent portion in the **Flange and Pipe** sub-rollout of this rollout, refer to Figure 10-37.

Output Surface Rollout

In this rollout, the **Blend and Flange** option is selected by default in the **Output Options** drop-down list. If you select the **Pipe Only** option from this drop-down list, then only a pipe will be created. If you select the **Flange Only** option from the **Output Options** drop-down list, then only a flange will be created. By default, the **Trim Base Faces** check box is clear in this rollout. As a result, the portion extended beyond the flange will be retained, as shown in Figure 10-38. If you want to remove the unwanted portion of the flange, select the **Trim Base Faces** check box. The **Extend Flange** check box will be available only when the **Trim Base Faces** check box is clear. If you select this check box, the flange will be extended to cover the entire span of the base surface.

Settings Rollout

By default, the **Create Curves** check box is clear in this rollout. If you select this check box, two curves will be created along the center of the bend radius and at the intersection of the bend and the flange. If you select the **Show Pipe** check box in this rollout, the pipe of the bend radius will be displayed in the preview, as shown in Figure 10-39.

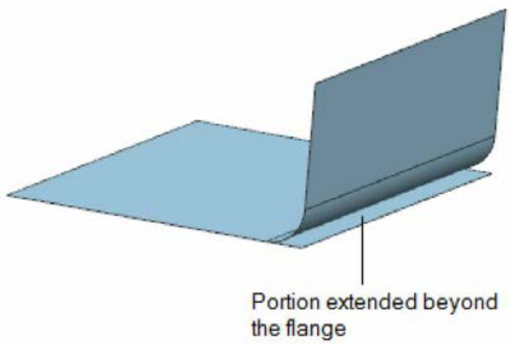


Figure 10-38 Unwanted extended surface

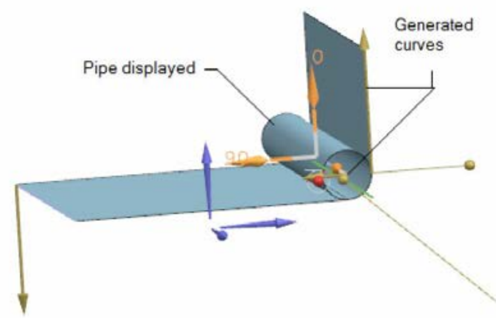


Figure 10-39 Pipe of the bend radius displayed in the preview

Creating a Silhouette Flange Surface Using the Absolute Gap Method

By selecting the **Absolute Gap** option from the drop-down list in the **Type** rollout, you can create a silhouette flange surface relative to an existing silhouette flange surface and also maintain a predefined gap. The minimum gap is calculated by taking the center line of the bend radius of the two pipes and the nearest tangential distance between them. You can also maintain the predefined gap between two silhouette flange surfaces by entering a gap value in the **Gap** edit box.

If you select the **Absolute Gap** option from the drop-down list in the **Type** rollout, the **Base Feature** rollout will be displayed. By default, the **Base Feature** button is chosen in the **Base Feature** rollout. As a result, you will be prompted to select the silhouette flange to define the base flange. Select the existing flange; you will be prompted to select the faces to define the base face. Select the reference face. Next, specify the reference direction in the **Reference Direction** rollout. To do so, choose the **Reverse Flange Side** button in the **Reference Direction** rollout; the preview of the resultant component will be displayed, as shown in Figure 10-40. To change the gap between the created flange and the existing selected flange, you can enter the required value in the **Gap** edit box, which is available at the bottom of the

Flange Parameters rollout.

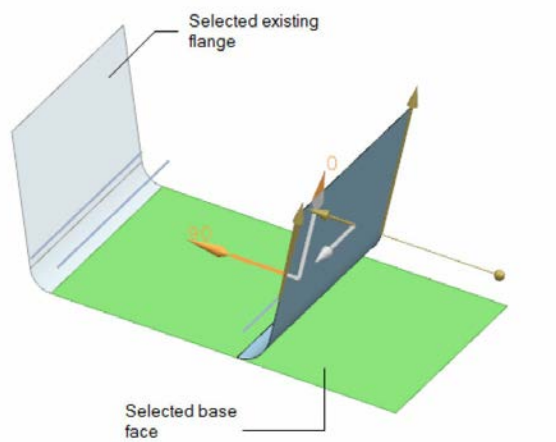


Figure 10-40 *The preview of the silhouette flange surface displayed along with handles and pipe*

Creating a Silhouette Flange Surface Using the Visual Gap Method

The **Visual Gap** option from the drop-down list in the **Type** rollout is used to create a flange surface in accordance with an existing flange surface by specifying a visual gap attribute between the two flange surfaces. To create the silhouette flange surface using the visual gap method, select the **Visual Gap** option from the drop-down list in the **Type** rollout of the **Silhouette Flange** dialog box. The selection procedure for reference objects is the same as discussed in the previous two methods. Enter the gap value in the **Gap** edit box and choose the **OK** button to create the surface. Figure 10-41 shows the silhouette flange created by using the **Visual Gap** method.

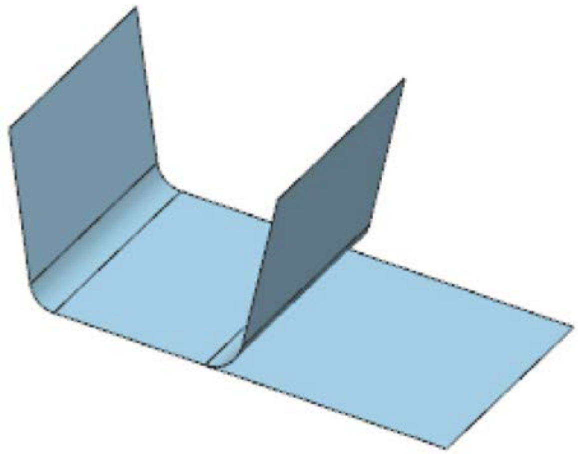


***Figure 10-41** The resultant silhouette flange surface created using the **Visual Gap** method*

Extending a Surface Using the Law Extension Tool

Ribbon: Home > More Gallery > Flange Surface > Law Extension *(Customize to add)* Menu: Insert > Flange Surface > Law Extension

The **Law Extension** tool is used to extend a surface either dynamically or by defining different type of laws for an extension. The extension of the surface can be carried out in both the directions of the edge or the curve selected. The process of extending the surface by using each of these methods is discussed next.

Extending a Surface Dynamically Using the Face Option

As discussed earlier, you can extend a surface dynamically by using the **Law Extension** tool. To do so, choose the **Law Extension** tool from **Menu > Insert > Flange Surface** in the **Top Border Bar**; the **Law Extension** dialog box will be displayed, refer to Figure 10-42, and you will be prompted to select the base curve profile. By default, the **Face** option is selected from the drop-down list in the **Type** rollout. Using this method, you can extend the surface by taking an existing face as reference. Select the curve string that you want to extend from the surface and choose the **Face** button from the **Face** rollout; you will be prompted to select reference faces. Select the required face as the reference face; the preview of the surface will be displayed, as shown in Figure 10-43.

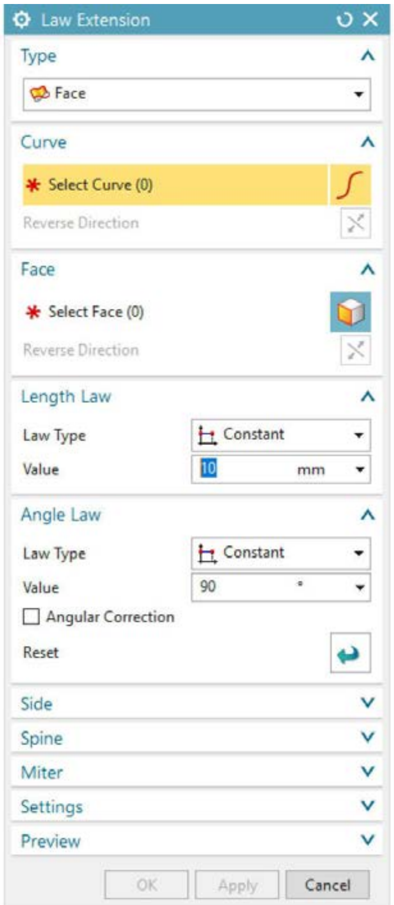


Figure 10-42 *The* ***Law Extension*** *dialog box*

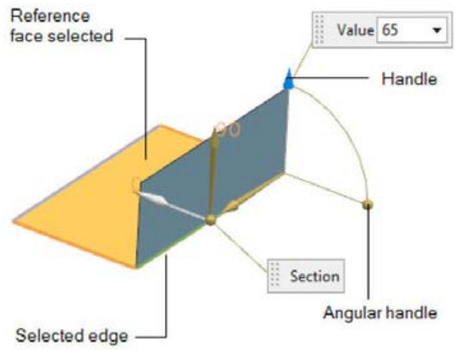


Figure 10-43 *Preview of the component*

Length Law and Angle Law Rollouts

The options in the **Length Law** and **Angle Law** rollouts are the same with the only difference that the length law is applicable for the length of the flange, whereas the angle law is applicable for the angle of the flange. By default, the **Constant** option is selected in the **Law Type** drop-down list in the **Length Law** as well as the **Angle Law** rollouts. As a result, you can specify a constant value of length in the **Value** edit box. On selecting the **Multi-transition** option from the **Law Type** drop-down list, you can change the length or angle of the flange regardless of the other control points. However, you can change the length and angle of the flange by applying other laws such as **Linear**, **Cubic**, **By equation**, and **By Law Curve**. For example, select the **Linear** option from the **Law Type** drop-down list; the **Start** and **End** edit boxes will be displayed in this rollout. Enter the start and end values in the **Start** and **End** edit boxes, respectively, and then choose the **OK** button; the extended surface will be displayed.

Side Rollout

By default, the **One-sided** option is selected in the **Extension Side** drop-down list. If you select the **Symmetric** option from the **Extension Side** drop-down list; a symmetric flange will be created on the opposite side of the created flange. If you select the **Asymmetric** option from the **Extension Side** drop-down list, the **Length Law** sub-rollout will be displayed in this rollout. Also, the new flange will be created on the opposite side of the created flange. You can modify this new flange by using the **Length Law** sub-rollout.

Spine Rollout

This rollout is used to define the spine of the law extension surface. To define a spine, select the **Curve** option from the **Method** drop-down list of the **Spine** rollout. Next, choose the **Curve** button from the **Spine** rollout if it is not already chosen and then select the curve, refer to Figure 10-44. An imaginary plane will be placed perpendicular to the selected curve, with respect to which the angle of the flange will be measured, refer to Figure 10-44. You can also define a spine by selecting a vector. To do so, select the **Vector** option from the **Method** drop-down list and then specify the required vector from the triad displayed in the drawing window.

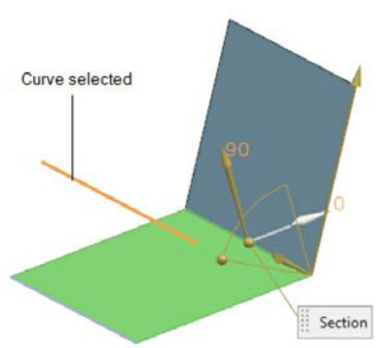


Figure 10-44 Curve selected using the ***Curve*** *button in the* ***Spine*** *rollout*

Miter Rollout

This rollout is used to create miter at the corner where two surfaces meet.

When you select the **Sharp** option from the **Method** drop-down list of the **Miter** rollout, a sharp miter is created at the corner. Select the **Blend** option to create a blend at the corner. Figure 10-45 shows a corner with no miter. Figures 10-46 and 10-47 show corners with sharp and blend miters created, respectively.

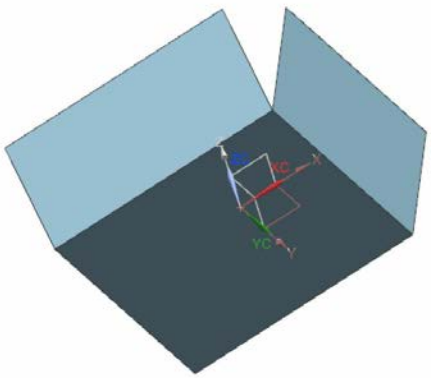


Figure 10-45 *Law extension surfaces with no miter at the corner*

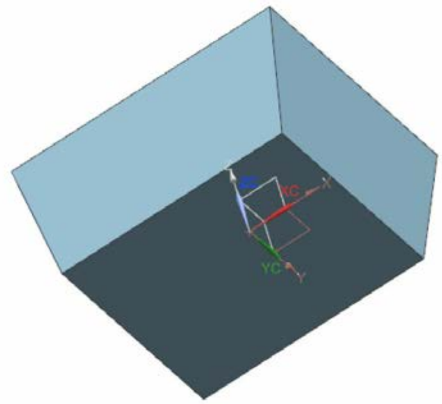


Figure 10-46 *Law extension surfaces with sharp miter at the corner*

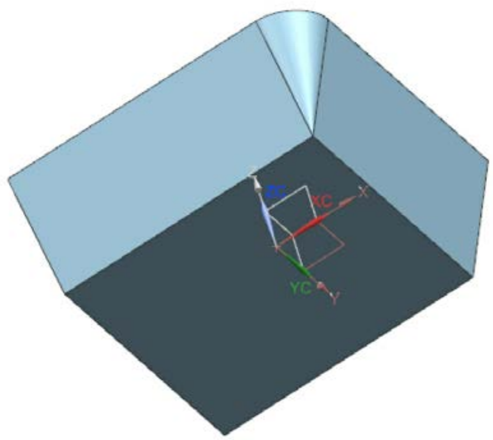


Figure 10-47 *Law extension surfaces with blend miter at the corner*

Settings Rollout

In this rollout, the **Lock End Length/Angle Handles** check box is clear by default. As a result, you can move handles and angular handles irrespective of each other, refer to Figure 10-48. If you select this check box, the end handles of the profile will be locked. As a result, if you drag the handles at the start point, the handle at the end point will be modified simultaneously. Note that this check box is available only when you select the **Multi-transition** option from the **Law Type** drop-down list.

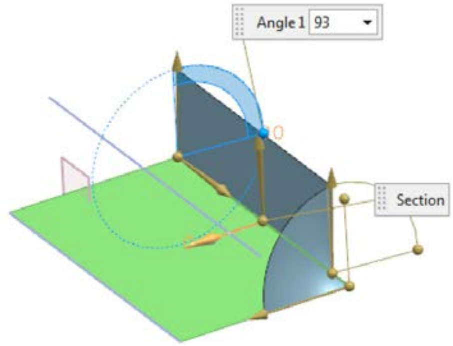


Figure 10-48 End handles dragged irrespective of each other

Extending a Surface Dynamically Using the Vector Option

To extend a surface dynamically by using the **Vector** option, select the **Vector** option from the drop-down list in the **Type** rollout; the **Face** rollout will be replaced by the **Reference Vector** rollout and the rest of the options will remain the same. In this method, instead of selecting reference faces, you can select reference vector so that the extended surface is created along the normal of the selected vector. To do so, select the curve string that you want to extend from the surface and then click on the **Specify Vector** area in the **Reference Vector** rollout; you will be prompted to select a vector. Also, the triad of vector will be displayed. You can select an edge, a line, or an arrow from the triad as a vector. Alternatively, you can specify the vector by selecting the required option from the **Inferred Vector** drop-down list in the **Reference Vector** rollout.

Note

*In the **Face** method, a curve selected from a surface for extension should lie on the selected reference face. In the **Vector** method, a curve selected from a surface for extension need not lie on any face.*

Creating a Surface Offset Using the Offset Surface Tool

Ribbon: Home > Operations > Offset Surface
Menu: Insert > Offset/Scale > Offset Surface

The **Offset Surface** tool is used to offset a surface in the direction normal to a selected surface. To offset a surface, choose the **Offset Surface** tool from **Menu > Insert > Offset/Scale** in the **Top Border Bar**; the **Offset Surface** dialog box will be displayed, as shown in Figure 10-49. By default, the **Face** button is chosen in the **Face** rollout. As a result, you will be prompted to select the faces for the new set. Select the face, refer to Figure 10-50. Next, enter the offset value in the **Offset 1** edit box. If you want to create a new set, choose the **Add New Set** button from the **Face** rollout and select the faces for the second set. To flip the offset direction, choose the **Reverse Direction** button. Next, choose the **OK** button; the resulting offset surface will be created, as shown in Figure 10-50.

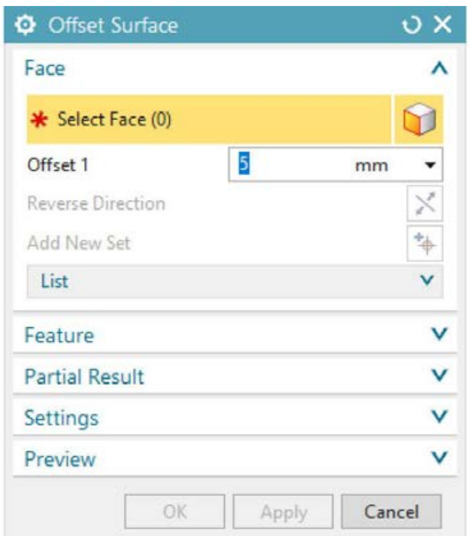


*Figure 10-49 The **Offset Surface** dialog box*

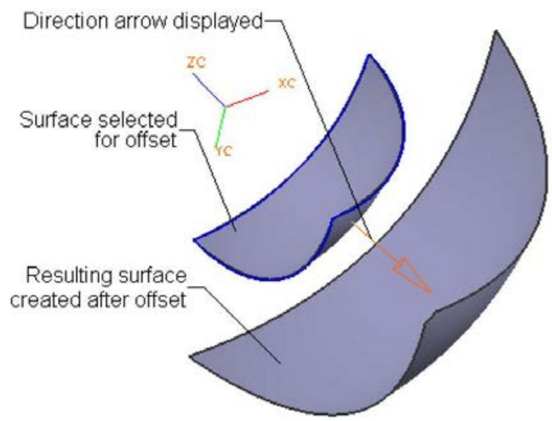


Figure 10-50 Offset surface created

Trimming and Extending a Surface Using the Trim and Extend Tool

Ribbon: Home > Operations > Operations Gallery > Trim and Extend
Menu: Insert > Trim > Trim and Extend

The **Trim and Extend** tool is used to trim or extend an open or a closed surface. To trim or extend a surface, choose the **Trim and Extend** tool from **Menu > Insert > Trim** in the **Top Border Bar**; the **Trim and Extend** dialog box will be displayed, as shown in Figure 10-51, and you will be prompted to select the face or edge to trim or extend. Select a single edge or multiple edges from the surface to be extended. When you select multiple edges for extending them, ensure that the selected edges are in continuity. If the **Preview** check box is selected in the **Preview** rollout, the preview of the extended surface will be displayed. The different rollouts in the **Trim and**

Extend dialog box are discussed next.

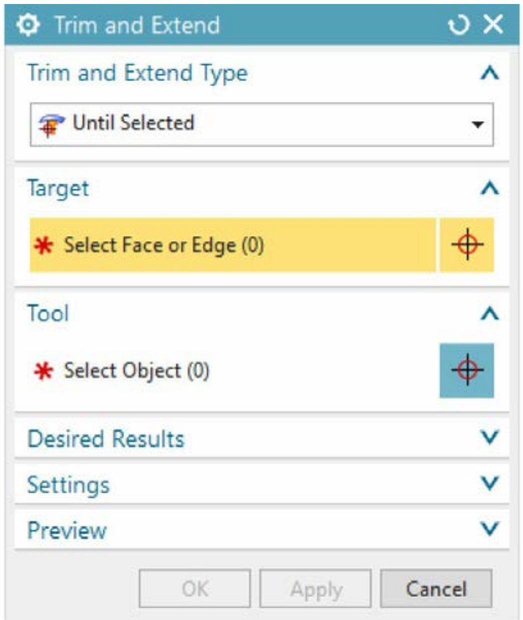


Figure 10-51 The ***Trim and Extend*** *dialog box*

Trim and Extend Type Rollout

On selecting the **Until Selected** option from the drop-down list in the **Trim and Extend Type** rollout, the surface will be extended up to a selected reference object. This option can also be used to trim a selected surface. If you select the **Make Corner** option, a corner will be created at the intersection of the extended surface with the tool body and the tool body will be trimmed.

Settings Rollout

The options in the **Surface Extension Shape** drop-down list of the **Settings** rollout are used to define the type of continuity of the extended surface with the existing surface. If you select the **Natural Curvature** option, the surface will be extended normally to the selected edge. If you select the **Natural Tangent** option, the surface will be extended by maintaining an angular curvature of 3 degree at the start point of the selected edge. If you select the **Mirrored** option, the surface will be extended along the curvature of the existing surface.

Note that if you select the **Until Selected** option from the drop-down list in the **Trim and Extend Type** rollout, you need to select the tool body that serves as the boundary object after selecting the edge for extension. Choose the **Tool** button from the **Tool** rollout and select the boundary object. Next, choose the **OK** button to extend the surface up to the selected boundary object. The options in the **Arrow Side** drop-down list of the **Desired Results** rollout are used to retain or discard a selected tool body. If you select the **Retain** option, the selected tool body will be retained after trimming. If you select the **Delete** option, the material from the tool body will be removed in the direction of the arrow displayed on selecting the tool body. Figure 10-52 shows the preview of the extended surface after selecting the edges. Figure 10-53 shows the surface extended by using the **Make Corner** option.

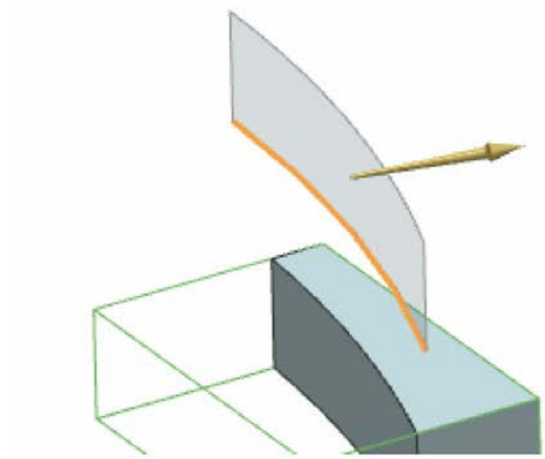


Figure 10-52 *The preview of the extended surface after selecting edges*

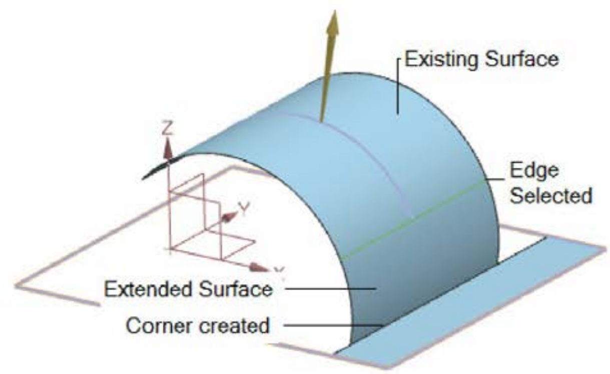


Figure 10-53 *The surface extended using the **Make Corner** option*

Trimming a Sheet by Using the Trimmed Sheet Tool

Ribbon: Home > Operations > Trim Sheet
Menu: Insert > Trim > Trim Sheet

The **Trim Sheet** tool is used to trim a sheet by defining the trim boundary. You can also trim a sheet by projecting a curve and then defining it as trim boundary. If the trim boundary is a surface, then the surface to be trimmed must be intersected completely with the trimming surface. Choose the **Trim Sheet** tool from **Menu > Insert > Trim** of the **Top Border Bar**; the **Trim Sheet** dialog box will be displayed, as shown in Figure 10-54, and you will be prompted to select sheet bodies to trim.

By default, the **Sheet Body** button is chosen in the **Target** rollout. Select the sheet to be trimmed and then press the middle mouse button. Next, you will be prompted to select the boundary objects. Select boundary objects. Next, choose the **Region** button from the **Region** rollout; the surface will be highlighted. The highlighted surface indicates whether this region is to be kept or discarded. You need to select the **Keep** or **Discard** radio button in the **Region** rollout to specify whether the area highlighted has to be kept or discarded. If you select the **Keep** radio button from this rollout, the highlighted region will be retained, and the other region will be removed. If you select the **Discard** radio button, the highlighted region will be removed (trimmed) and the other region will be retained.

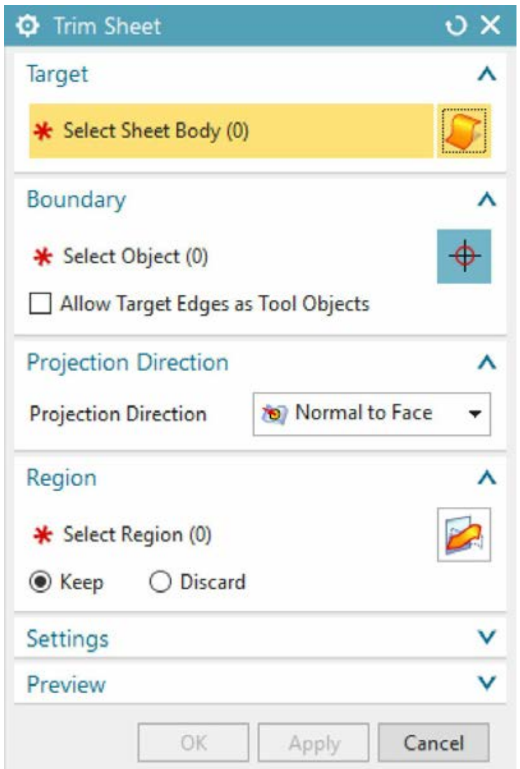


***Figure 10-54** The **Trim Sheet** dialog box*

Figure 10-55 shows the entities selected for trimming a surface. Figure 10-56 shows the resulting trimmed surface created after selecting the **Discard** radio button from the **Region** rollout.

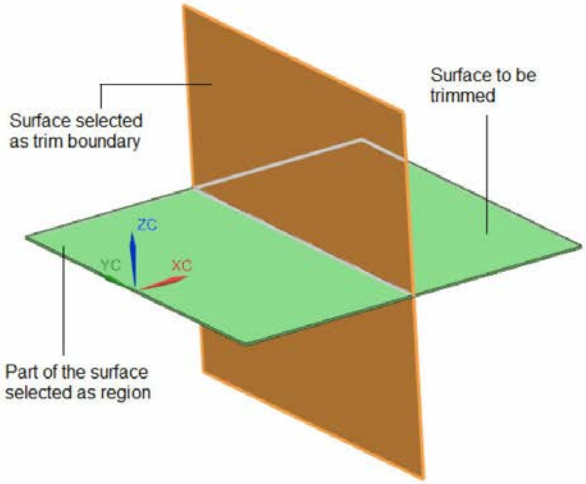


Figure 10-55 Entities selected for trimming a sheet

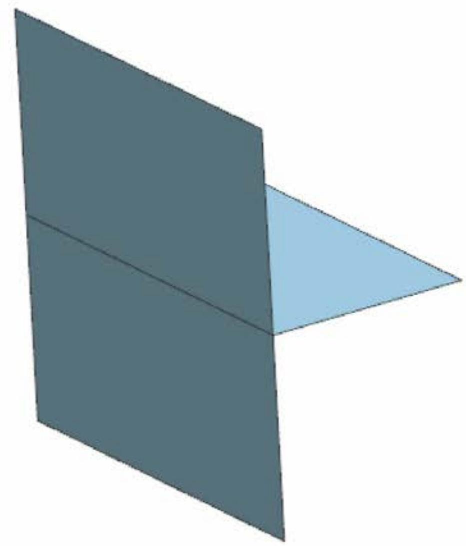


*Figure 10-56 The resulting trimmed sheet created after selecting the **Discard** radio button*

The **Projection Direction** drop-down list contains the options for projecting (imprinting) a curve or a sketch on the surface to be trimmed. The projection curve or sketch can be defined as the trimming boundary. Select the surface to be trimmed and press the middle mouse button. Next, select the curve or the sketch as the trim boundary. The selected curve or the sketch automatically gets imprinted on the surface to be trimmed and forms the trim boundary. The curve projected as the trim boundary should intersect the surface to be trimmed. Figure 10-57 shows the objects selected when the trim boundary is created by imprinting a curve on the surface to be trimmed. Figure 10-58 shows the resulting trimmed surface created after selecting the **Discard** radio button from the **Region** rollout.

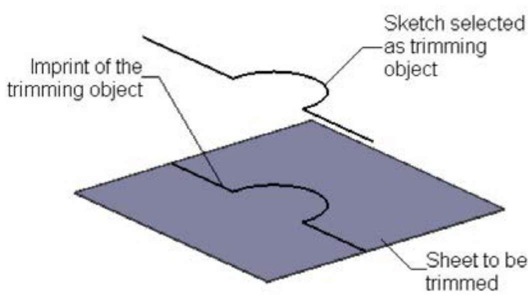


Figure 10-57 Sketch selected for trimming a sheet

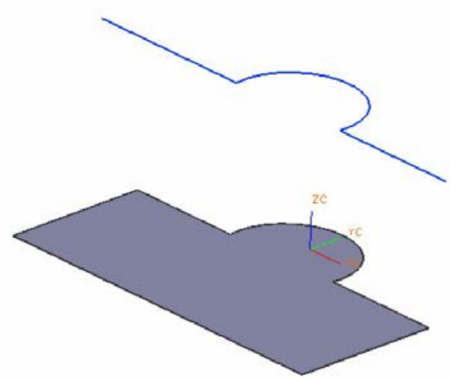


*Figure 10-58 The resulting trimmed sheet created after selecting the **Discard** radio button*

Creating a Surface Using the Studio Surface Tool

Ribbon: Home > Create > Studio Surface
Menu: Insert > Mesh Surface > Studio Surface

The **Studio Surface** tool is used to create a surface by sweeping a single section or multiple sections along single or multiple guide curves. The selected guide and section curves can be opened or closed.

Choose the **Studio Surface** tool from **Menu > Insert > Mesh Surface** in the **Top Border Bar**; the **Studio Surface** dialog box will be displayed, as shown in Figure 10-59, and you will be prompted to select a section. By default, the **Curve** button is chosen from the **Section (Primary) Curves** rollout. Select the section curves one by one. After selecting one section curve, press the middle mouse button to continue selecting other section curves. Note that all section curves should point in one direction. After selecting the section curves, choose the **Guide (Cross) Curves** button from the **Guide (Cross) Curves** rollout; you will be prompted to select a guide curve. Select the guide curves one by one in the same way as you did for the section curves. Make sure that all the guide curves should also point in one direction.

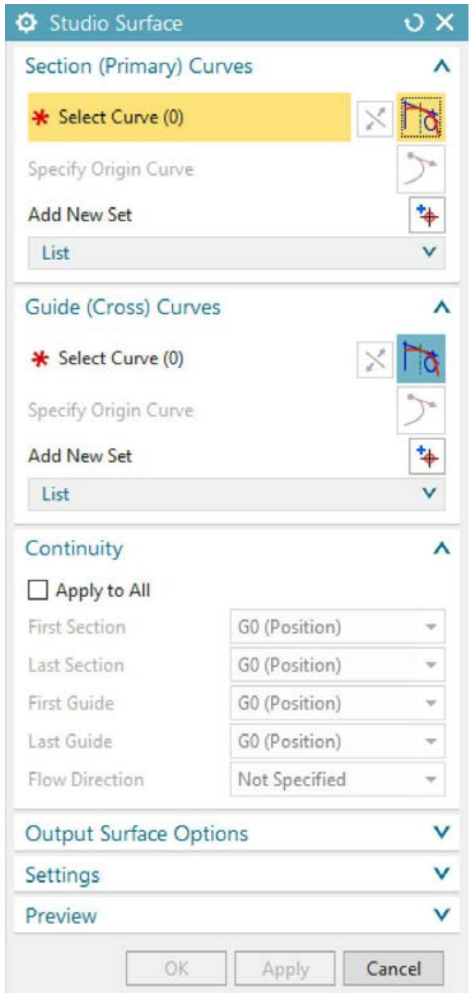


***Figure 10-59** The **Studio Surface** dialog box*

The other options in the dialog box have been discussed in the earlier tools. After selecting all parameters, choose the **OK** button; a surface will be created. Figure 10-60 shows the section and the guide curve selected for creating the studio surface. Figure 10-61 shows the preview of the resulting studio surface.

Figure 10-62 shows a single section and two guide curves selected for creating a studio surface and Figure 10-63 shows the resulting studio surface.

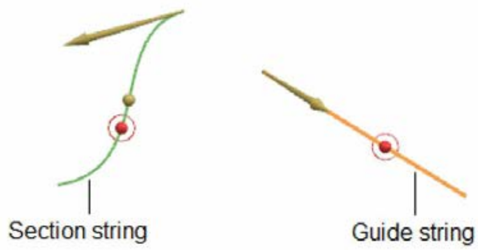


Figure 10-60 The section string and the guide string selected for creating a studio surface

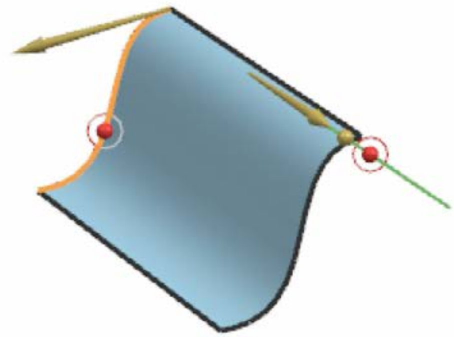


*Figure 10-61 The preview of the studio surface created using the **Studio Surface** tool*

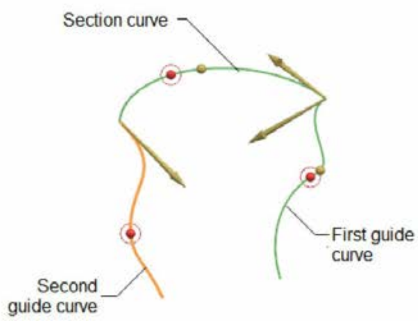


Figure 10-62 The section curve and two guide curves selected for creating the studio surface

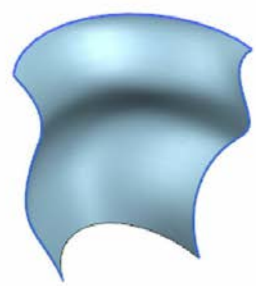


Figure 10-63 The resulting studio surface

Figure 10-64 shows the selected start and end section curves and Figure 10-65 shows the resulting studio surface.

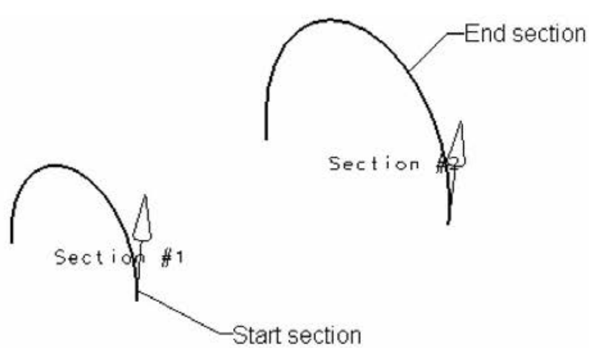


Figure 10-64 The section curves selected for creating a studio surface

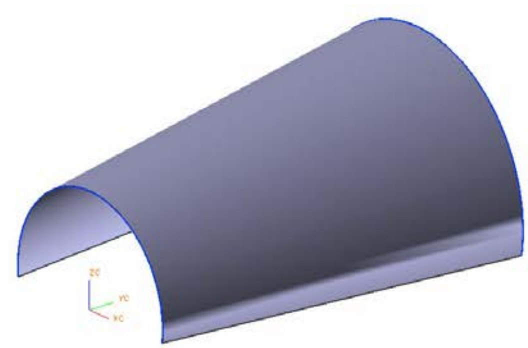


*Figure 10-65 The resulting studio surface created using the **Studio Surface** tool*

Figure 10-66 shows two section curves and two guide curves selected for creating a studio surface. Figure 10-67 shows the resulting studio surface.

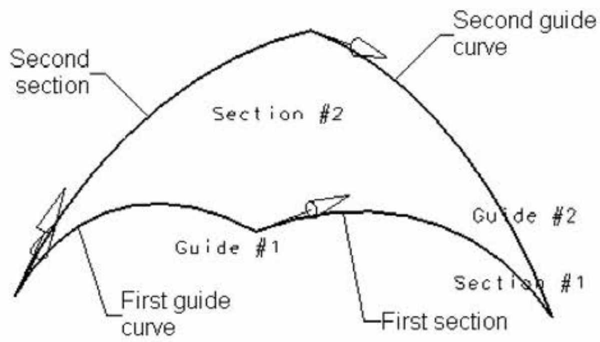


***Figure 10-66** The section and guide curves selected for creating a studio surface*

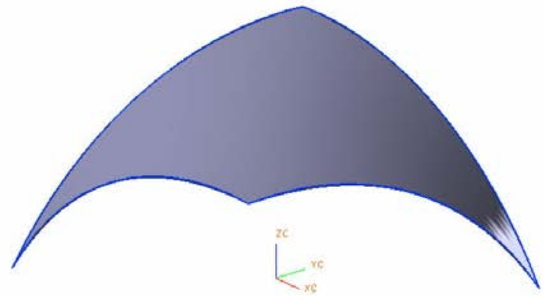


***Figure 10-67** The resulting studio surface created using the **Studio Surface** tool*

Figure 10-68 shows the selected section curves and guide curves. Figure 10-69 shows the resulting studio surface.

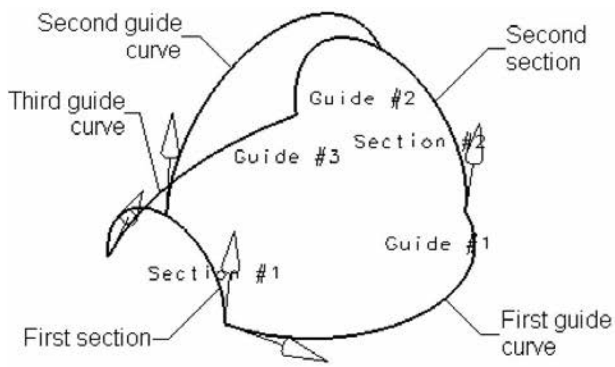


***Figure 10-68** The section curves and the guide curves selected to create a studio surface*

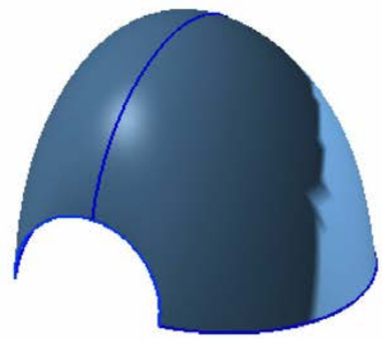


***Figure 10-69** The resulting studio surface created using the **Studio Surface** tool*

Creating a Surface between Two Walls Using the Styled Blend Tool

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

The **Styled Blend** tool is used to create a fillet surfacebetween two intersecting walls. While creating fillet surfaces, you can maintain a tangent or curvature continuity among walls. You can also create variable fillets using this tool. To create a fillet surface, choose the **Styled Blend** tool from **Menu > Insert > Detail Feature** in the **Top Border Bar**; the **Styled Blend** dialog box will be displayed, as shown in Figure 10-70, and you will be prompted to select the faces for wall 1.

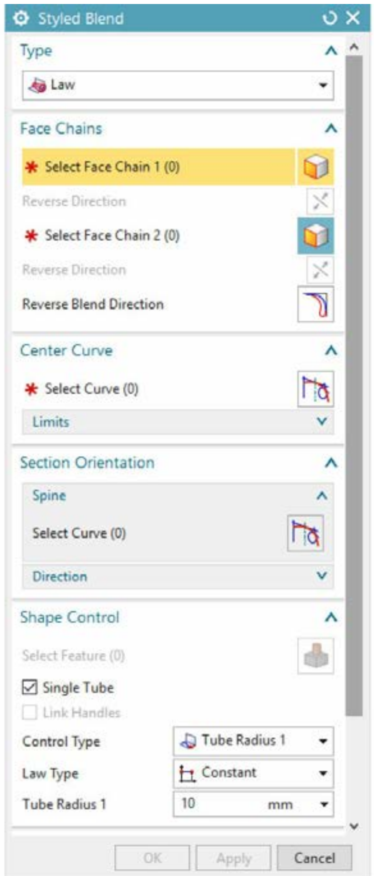


***Figure 10-70** Partial view of the **Styled Blend** dialog box*

The method of formation of the blend surface is defined by the type of option you select from the drop-down list in the **Type** rollout. If you select the **Law** option, the lines holding the tangent will automatically be created with respect to the pipe radius specified for the fillet. If you select the **Curve** option, you need to select the tangent holding curves for creating the fillet. If you select the **Profile** option, the tangent holding lines will be created by imprinting a curve or a sketch on both the surfaces between which the surface is to be created.

Creating a Styled Blend Surface Using the Law Option

By default, the **Law** option is selected in the drop-down list of the **Type** rollout and the **Select Face Chain 1** area is activated in the **Face Chains** rollout. Select the first face and then click on the **Select Face Chain 2** area in the **Face Chains** rollout. While selecting both the faces, you need to ensure that the arrow displayed on the faces are facing inward where the surface is to be created. You can use the **Reverse Direction** button in the **Face Chains** rollout to flip the direction of the arrow. Select the second face and press the middle mouse button; the preview of the fillet along with the handles and dynamic edit box will be displayed, as shown in Figure 10-71. You can view the alternate solutions of the displayed fillet by choosing the **Reverse Blend Direction** button in the **Face Chains** rollout. By default, the **Single Tube** check box is selected in the **Shape Control** rollout. As a result, the parameters of the fillet will change simultaneously for both the walls, as shown in Figure 10-72. If you clear this check box, the parameters of the fillet will change only for the wall 1, as shown in Figure 10-73. Also, the **Tube Radius 2** option will be activated in the **Control Type** drop-down list. By default, the **Tube Radius 1** option is selected in the **Control Type** drop-down list. As a result, you can change the radius of the fillet by using the **Tube Radius** edit box.

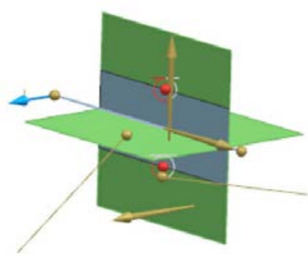


Figure 10-71 *Preview of the fillet created between two walls*

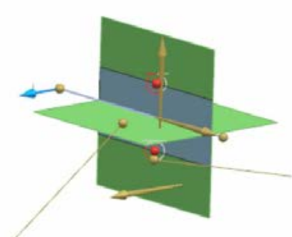


Figure 10-72 *Fillet being created with the* ***Single Tube*** *check box selected*

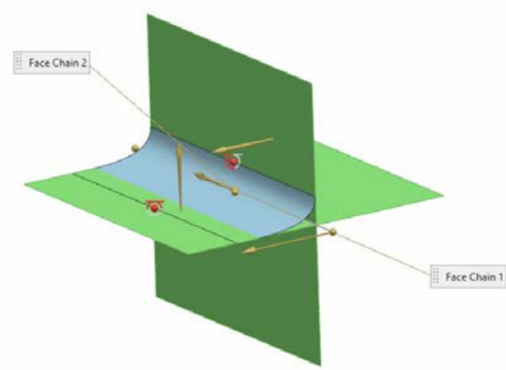


Figure 10-73 *Fillet being created with the* ***Single Tube*** *check box cleared*

If you select the **Depth** option in the **Control Type** drop-down list, you can create the blend by specifying its depth. If you select the **Skew** option, you can create the blend by specifying its skewness. If you select the **Tangent Magnitude** option, you can create the blend by specifying its tangent magnitude.

If you have already created a curve, as shown in Figure 10-74, then you can use that curve as the center of the blend. To do so, choose the **Curve** button from the **Center Curve** rollout and select the curve; the preview of the blend will be displayed by using the selected curve as the center, as shown in Figure 10-75. By default, line curve will be extended by 10% of its original length at the start point and end point. You can edit this value in the **Limits** sub-rollout.

You can restrict the length of the blend by using the **Section Orientation** rollout. To do so, choose the **Curve** button from the **Spine** sub-rollout; you will be prompted to select the curves for spine. Select a curve, as shown in Figure 10-76; the preview of the resultant model will be displayed, as shown in Figure 10-77. By default, the **Use Center Curve as Spine** check box is clear. If you select this check box, the center curve will be used as spine curve.

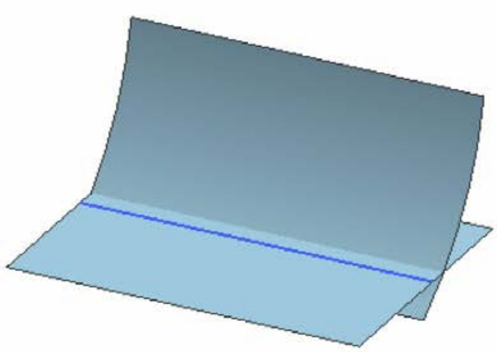


Figure 10-74 *Existing curve to be selected as the center of the bend*

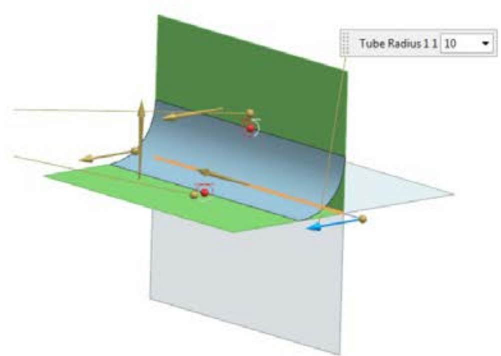
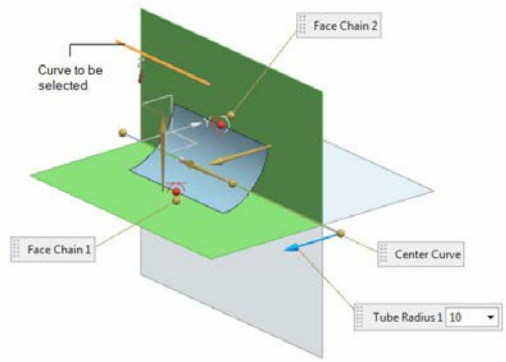


Figure 10-75 *Preview of the resultant model*



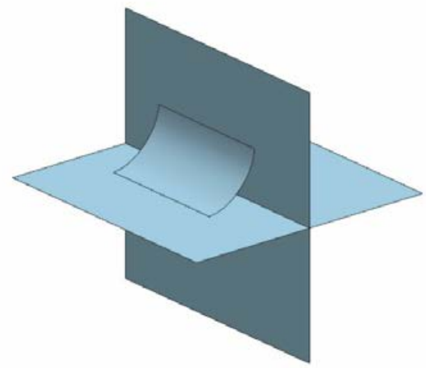


Figure 10-76 *The curve to be selected to restrict fillet*

Figure 10-77 *Preview of the resultant model*

To enable the **Extend Blend** check box in the **Blend Output** rollout, select the **Use Center Curve as Spine** check box from the **Spine** sub-rollout in the **Section Orientation** rollout. If you select the **Extend Blend** check box, the fillet will extend throughout the center line, as shown in Figure 10-78. By default, the **No Trim** option is selected in the **Trimming Method** drop-down list of the **Blend Output** rollout, As a result, it keeps the extended portion after the blend in the resultant model. If you select the **Trim & Attach** option from this drop-down list, the extended portion of the wall after the blend and the extended portion of the blended curve after the boundary edges in the resultant model will be removed, as shown in Figure 10-79. If you select the **Trim Input Face Chains** option, only the extended portion of the wall after the blend will be removed, as shown in Figure 10-80.

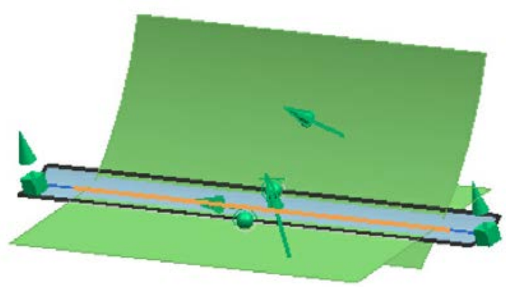


Figure 10-78 *Preview of the fillet with the* ***Extend Blend*** *check box selected*

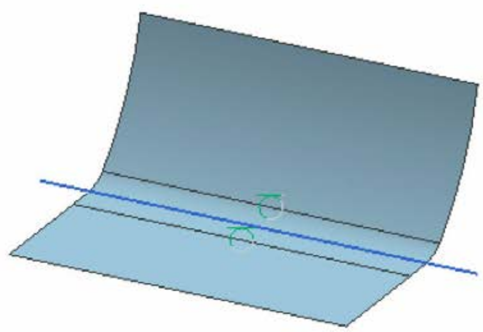


Figure 10-79** The styled blend created using the* ***Trim and Attach *option*

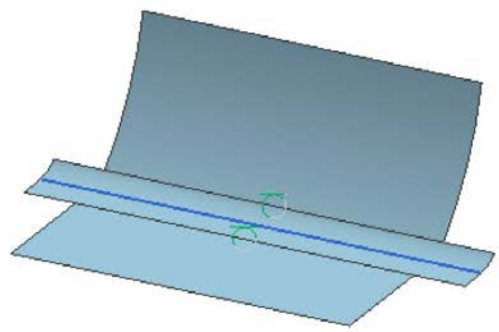


Figure 10-80** The styled blend created using the* ***Trim Input Face Chains *option*

If you select the **Show Tube** check box from the **Show** sub-rollout of the **Settings** rollout, the tube of specified radius will be displayed in the preview. If you select the **Show Depth Curve** check box, the depth of curvature will be displayed in the preview. You can reset the options in the **Settings** rollout by choosing the **Reset All** button in the **Settings** rollout.

The options in the **Constraint Options** rollout will be activated only when two adjacent blends are available in the component. The options in this rollout are used to maintain continuity between two adjacent blends.

Creating a Styled Blend Surface Using the Curve Option

To create a styled blend surface by using the **Curve** option, select the **Curve** option from the drop-down list in the **Type** rollout; you will be prompted to select the faces for face chain 1. Select the first face and then click on the **Select Face Chain 2** area in the **Face Chains** rollout; you will be prompted to select the faces for face chain 2. Select the second face and press the middle mouse button; the **Select Curve Set 1** area in the **Tangent Curves** rollout will be activated and you will be prompted to select the curves for curve set 1. Select the first tangential curve from the first face selected and then click on the **Select Curve Set 2** area; you will be prompted to select the curves for curve set 2. Select the second tangential curve from the selected second wall, refer to Figure 10-81; the preview of the fillet will be displayed, as shown in Figure 10-82. Note that you may need to choose the **Reverse Direction** button to reverse the direction of surface creation.

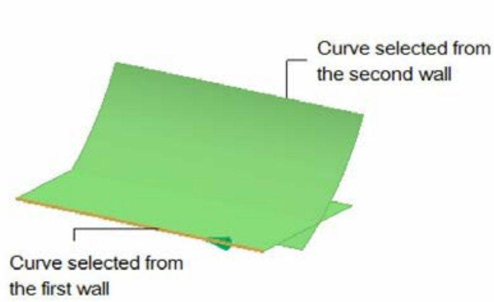


Figure 10-81 Reference selection for the ***Curve*** *option*

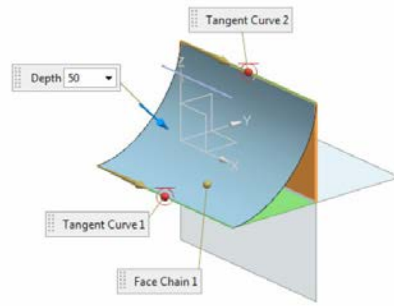


Figure 10-82 Preview of the fillet

Creating a Styled Blend Surface Using the Profile Option

To create a styled blend surface by using the **Profile** option, select the **Profile** option from the drop-down list in the **Type** rollout; you will be prompted to select the faces for face chain 1. Select the first face and press the middle mouse button. Next, select the second face and press the middle mouse button; the **Select Curve** area in the **Profile** rollout will be activated. Also, you will be prompted to select the curves for profile curve. Select the curve, refer to Figure 10-83; the preview of the fillet will be displayed, as shown in Figure 10-84. Remaining options are the same as discussed earlier.

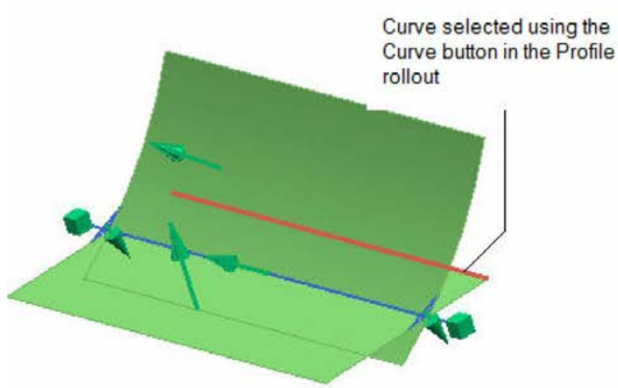


Figure 10-83 The curve selected for creating the Styled Blend surface using the ***Profile*** *option*

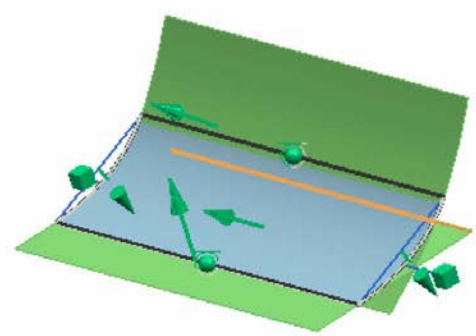


Figure 10-84 The preview of the Styled Blend surface using the ***Profile*** *option*

Creating Surfaces Using the Styled Sweep Tool

<u>Ribbon: Home > Create > Styled Sweep</u>
<u>Menu: Insert > Sweep > Styled Sweep</u>

The **Styled Sweep** tool is used to create surfaces by sweeping cross-sections along one or two guide curves. A surface created by using this tool can be modified dynamically by dragging the pivot point displayed along with the surface. To create a styled sweep surface, choose the **Styled Sweep** tool from **Menu > Insert > Sweep** in the **Top Border Bar**; the **Styled Sweep** dialog box will be displayed, as shown in Figure 10-85.

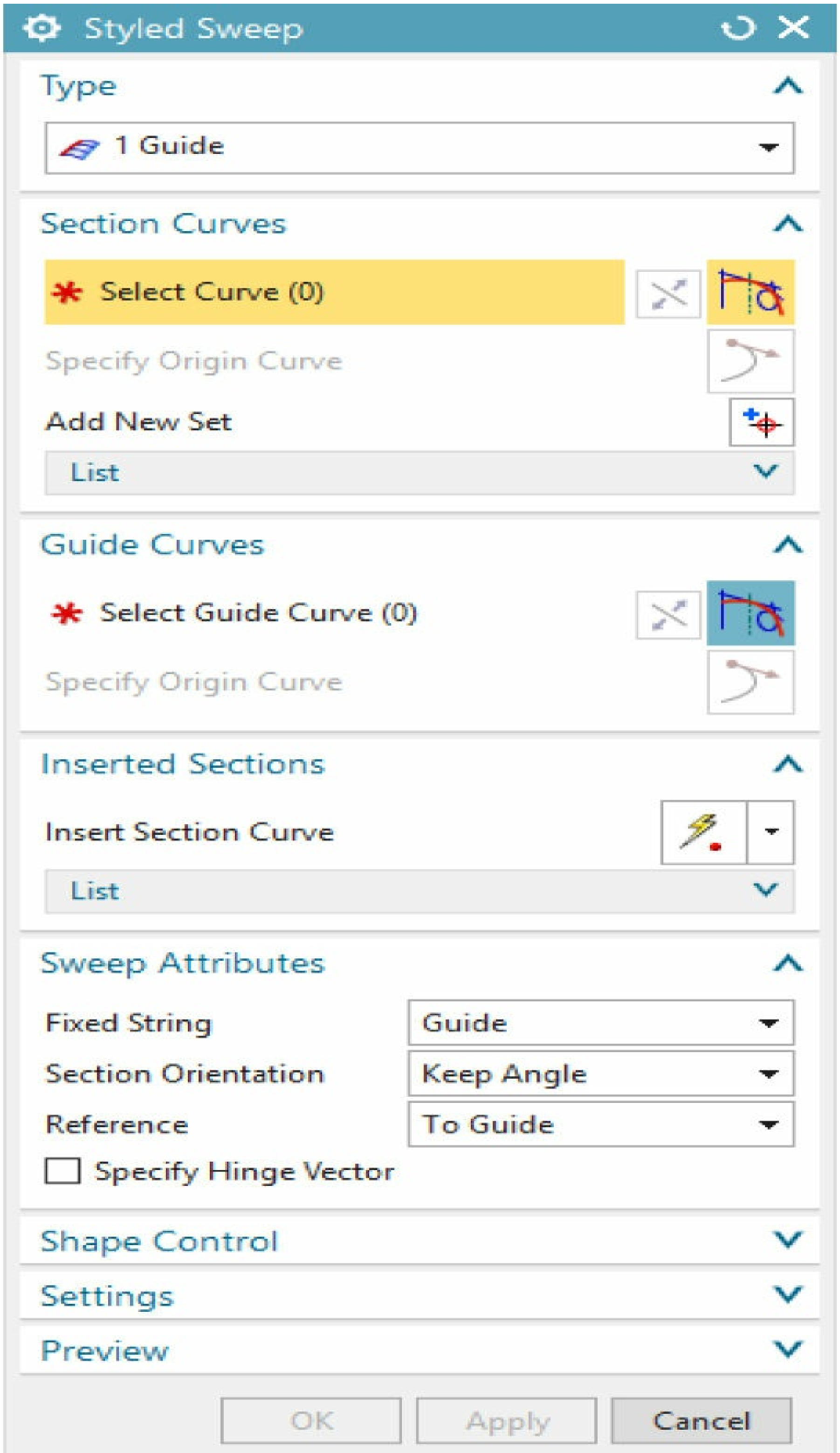


Figure 10-85 *The* ***Styled Sweep*** *dialog box*

The options in the drop-down list of the **Type** rollout are used to specify the number of guide, touch, and orientation strings. These options are discussed next.

1 Guide

This option allows you to select only one guide string. However, you can select up to 150 section strings.

1 Guide, 1 Touch

This option allows you to select one guide string, one touch string, and one section string.

1 Guide, 1 Orientation

This option allows you to select one guide string and one orientation string.

2 Guides

This option allows you to select two guide strings only. However, you can select up to 150 section strings.

Select the required option from the drop-down list in the **Type** rollout. Next, select the section and guide curves by using the **Section Curves** and **Guide Curves** rollouts, respectively. Note that you need to press the middle mouse button to add each section and guide string. Figure 10-86 shows the entities to be selected for creating the styled sweep surface by using the **1 Guide, 1 Touch** option. After selecting the section and guide strings, the preview of the surface along with the pivot point will be displayed, as shown in Figure 10-87.

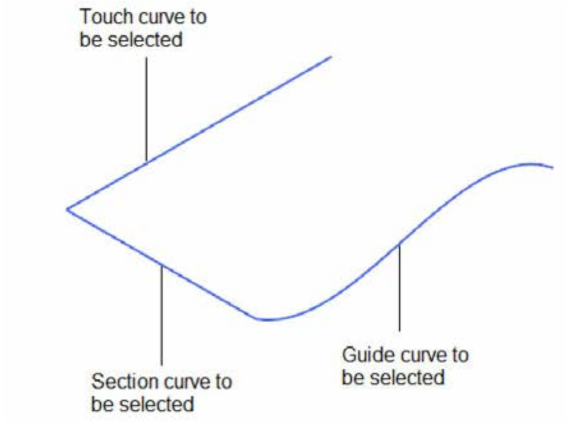


Figure 10-86 Entities to be selected for creating the styled sweep surface

Figure 10-87 Preview of the resulting styled sweep surface

You can modify the shape and size of the surface by dragging the pivot point. Alternatively, you can use the slider bar in the **Shape Control** rollout to modify the shape and size of the surface. The options in the **Shape Control** rollout are used to display different types of handles for the surface created, and the handles can be used to modify the surface dynamically.

Sewing Individual Surfaces into a Single Surface

Ribbon: Home > Operations > Sew
Menu: Insert > Combine > Sew

The **Sew** tool is used to stitch individual surfaces into a single surface with a common edge. When a selected individual surface encloses a volume, it creates a solid body. A sheet to which all other individual sheets are to be stitched is known as target sheet, and the individual sheets to be stitched are known as the tool sheets. You cannot stitch the tool sheets that intersect a target sheet and extend beyond it.

To stitch individual surfaces to a single surface, choose **Menu > Insert > Combine > Sew** from the **Top Border Bar**; the **Sew** dialog box will be displayed, as shown in Figure 10-88. By default, the **Sheet** option is selected from the drop-down list in the **Type** rollout and you are prompted to select the target sheet body. Select the target sheet body; you will be prompted to select the tool sheet bodies to sew. The border of the selected tool sheets should lie within the target sheet boundary. Otherwise, the selected tool sheet will not get stitched to the target sheet. Select the tool sheet bodies and choose the **OK** button; the sheet bodies will be stitched.

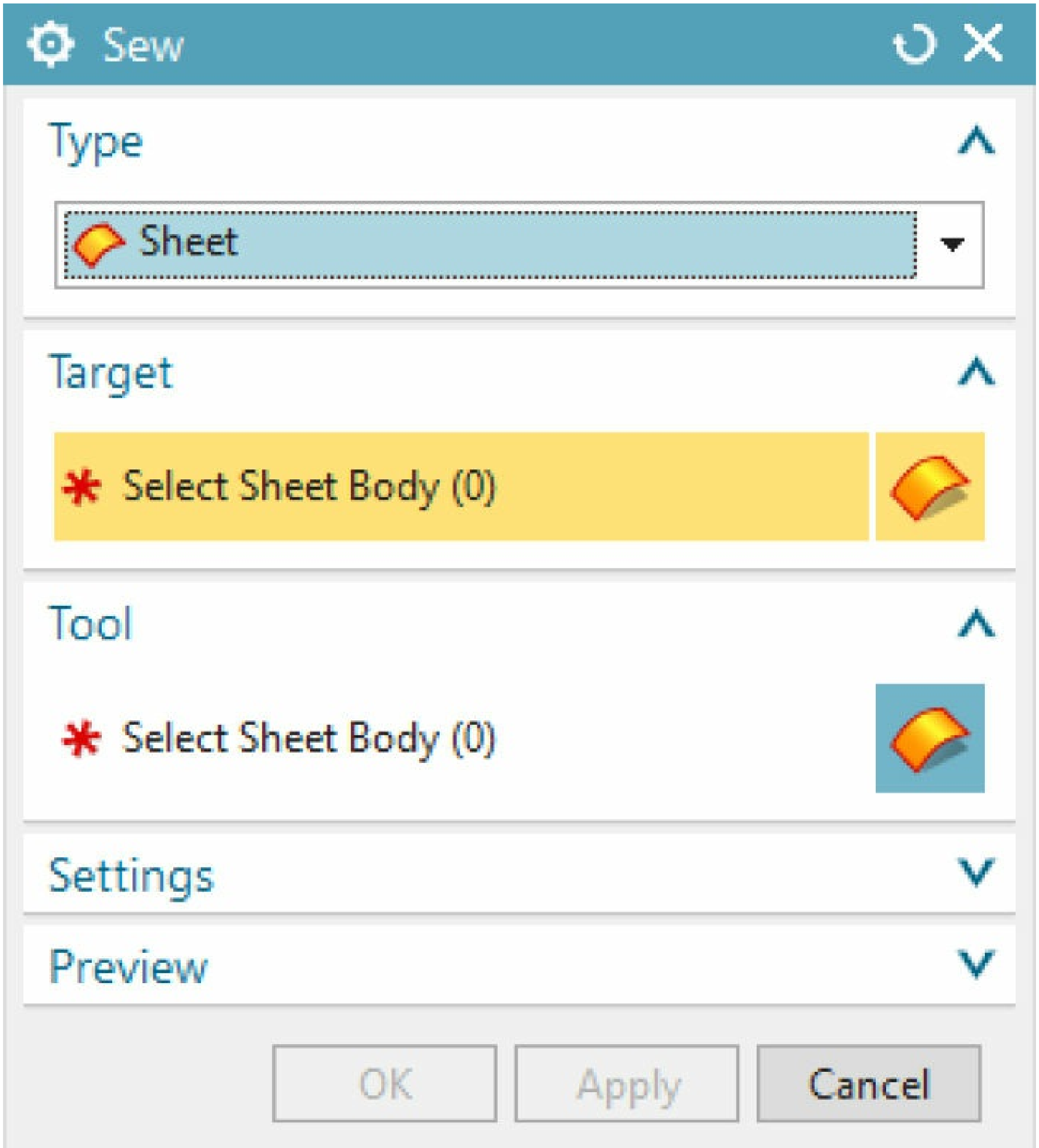


***Figure 10-88** The **Sew** dialog box*

Similarly, to combine solid bodies, select the **Solid** option from the drop-down list in the **Type** rollout. Note that you can sew two solid bodies together only if they share one or more common (coincident) faces. You can also enter sew tolerance value in the **Tolerance** edit box of the **Settings** rollout.

Adding Thickness to a Surface

Ribbon: Home > More Gallery > Offset/Scale > Thicken (Customize to add)
Menu: Insert > Offset/Scale > Thicken

The **Thicken** tool is used to add thickness to a sheet. Once you add thickness to a sheet, it is converted into a solid. To add thickness to a sheet, choose **Menu > Insert > Offset/Scale > Thicken** from the **Top Border Bar**; the **Thicken** dialog box will be displayed, as shown in Figure 10-89, and you will be prompted to select faces to thicken. Select the sheet bodies to which the material is to be added.

Enter thickness value in the **Offset 2** edit box to add thickness along the direction of handle. Enter a negative thickness value in the **Offset 1** edit box to assign thickness on the other side of the surface.

The **Boolean** drop-down list in the **Boolean** rollout provides options to perform the boolean operations on an existing solid body. Select the required option from the **Boolean** drop-down list and choose the **Body** button from the **Boolean** rollout. Next, select the target solid body; the boolean operation will be performed.

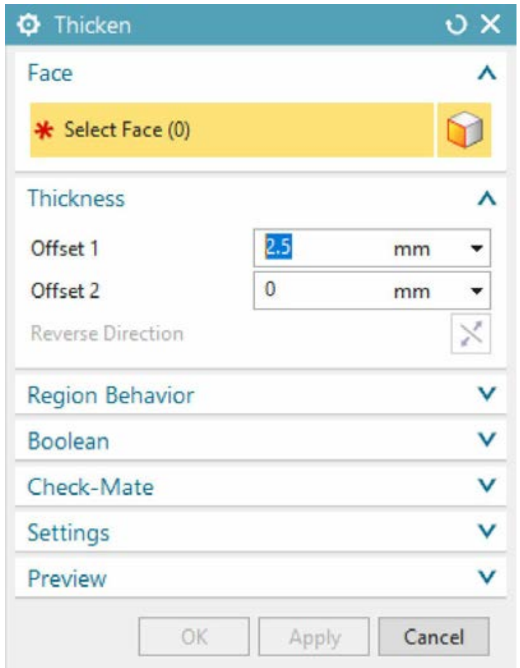


*Figure 10-89 The **Thicken** dialog box*

Figure 10-90 shows the sheet selected for adding thickness and Figure 10-91 shows the resulting solid body.

Figure 10-90 The sheet selected for adding material

Figure 10-91 The sheet after adding thickness

Adding a Draft

Ribbon: Feature > Feature > Draft
Menu: Insert > Detail Feature > Draft

In earlier chapters, you learned to add a draft using some of the options in the **Draft** dialog box. In this section, you will learn to add the split draft using the **Parting Face** option in the **Draft Method** drop-down list. Note that this option is available when the **Face** option is selected from the drop-down list in the **Type** rollout. The procedure of adding the split draft and the step draft angle using the **Parting Face** option is discussed next.

Parting Face

The **Parting Face** option enables you to select a parting geometry that acts as a pivot for the draft to be added to the faces. To add a draft by using this option, invoke the **Draft** dialog box and then select the **Face** option from the drop-down list in the **Type** rollout of the **Draft** dialog box. Next, select the **Parting Face** option from the **Draft Method** drop-down list in the **Draft References** rollout; you will be prompted to select the parting surface. Select the construction surface to be used as parting surface, as shown in Figure 10-92. Next, choose the **Face** button from the **Faces to Draft** rollout; you will be prompted to select the faces to be added to the draft. Select the faces. You need to enter the draft angle value in the **Angle 1** edit box; the preview of the draft will be displayed, as shown in Figure 10-93.

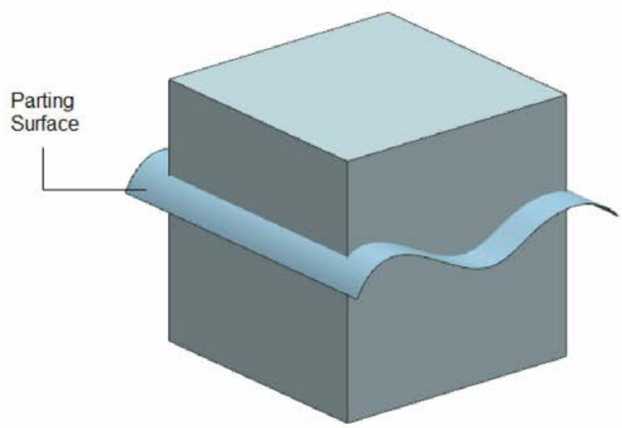


Figure 10-92 *Construction surface to be used as the parting surface*

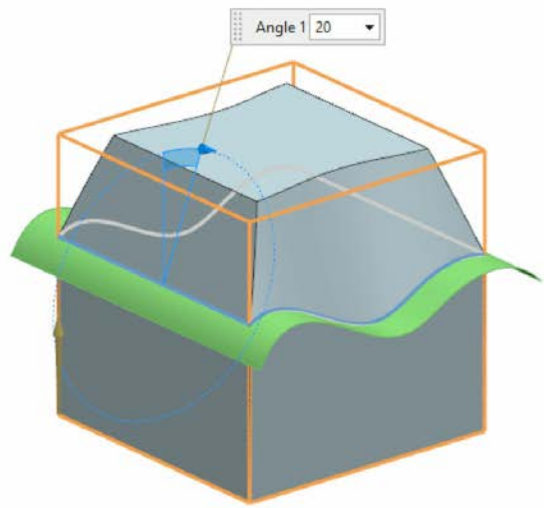


Figure 10-93 *Preview of the draft added to the faces*

To create two different drafts such that the face of a model is split using a surface, select the **Draft Both Sides** check box in the **Draft References** rollout. The parting surface acts as a hinge about which the draft angles will be added. Figure 10-92 shows the construction surface that is used as a parting surface. The parting surface splits the selected faces. As a result, two different draft angles are added to the faces, refer to Figure 10-94.

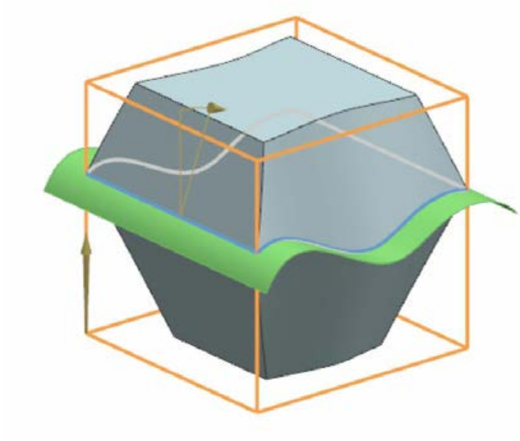


Figure 10-94 Two different drafts added to the faces

TUTORIALS

Tutorial 1

In this tutorial, you will create the surface model shown in Figure 10-95. The dimensions and orthographic views of the model are shown in Figure 10-96. After creating the surface, save it with the name *c10tut1.prt* at the location *\NX\c10*. **(Expected time: 30 min)**

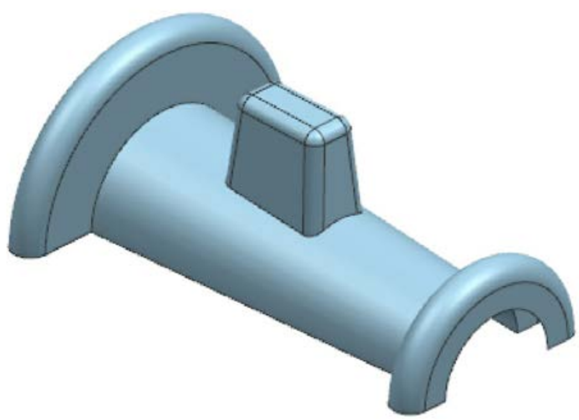


Figure 10-95 The isometric view of the surface model

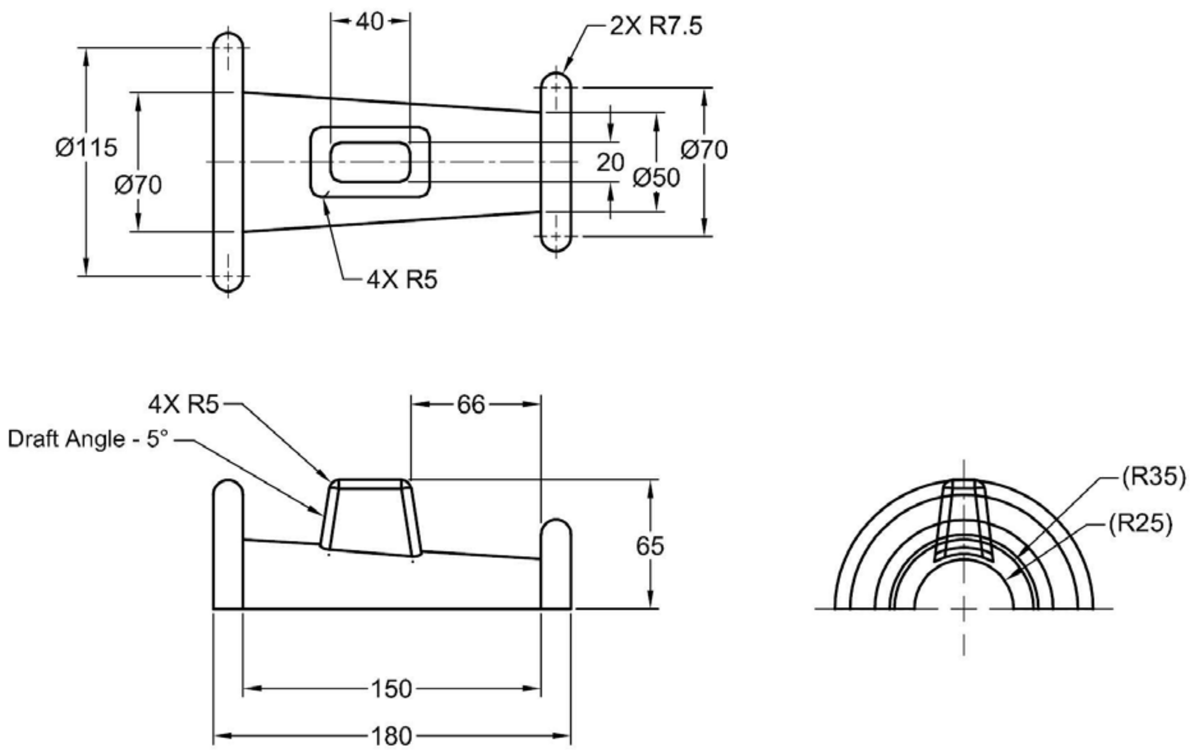


Figure 10-96 Dimensions and drawing views of the surface model

The following steps are required to complete this tutorial:

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

b. Create the sketch for the base surface and then revolve it, refer to Figures 10-97 and 10-98.

c. Create the second feature, which is an extruded surface, refer to Figures 10-99 and 10-100.

d. Trim the base surface, refer to Figures 10-101 and 10-102.

e. Trim the extended part of the third feature, refer to Figures 10-103 and 10-104.

f. Create the top surface of the extruded feature, refer to Figures 10-105 and 10-106.

g. Stitch the bounded plane and extruded surfaces with the revolved surface.

h. Fillet the stitched surface, refer to Figures 10-107.

i. Save and close the file.

Starting a New File and Setting the Sheet Environment

1. Start a new file with the name *c10tut1.prt* using the **Shape Studio** template

and specify its location at *C:\NX\c10*.

2. Choose **Menu > Preferences > Modeling** from the **Top Border Bar**; the **Modeling Preferences** dialog box is displayed.

3. In this dialog box, select the **Sheet** radio button from the **Body Type** rollout and choose the **OK** button.

Creating the Base Feature by Revolving the Sketch

1. Create the sketch for the base surface on the XC-YC plane, as shown in Figure 10-97.

2. Revolve the sketch about an angle of 180 degrees. The resulting revolved base feature created by using the **Sheet** option is shown in Figure 10-98.

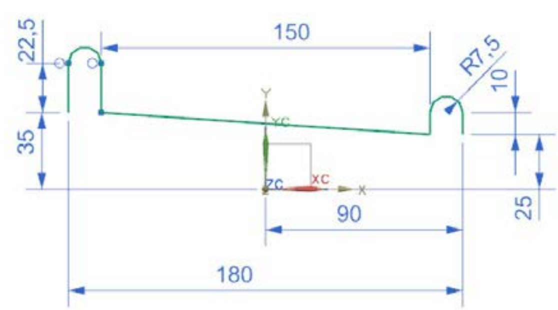


Figure 10-97 *The sketch created for the base feature*

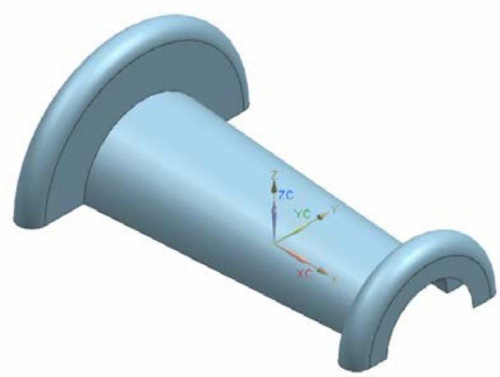


Figure 10-98 *The base feature created by revolving the sketch*

Creating the Second Feature by Extruding the Sketch

1. Create a datum plane in the upward direction at an offset of 65 mm from the XC-YC plane.

2. Create the sketch for the second feature by selecting the offset plane as the sketching plane, as shown in Figure 10-99.

3. Extrude the sketch through a distance of 50 mm in the downward direction and at the draft angle of -5 degrees by selecting the **From Start Limit**

option from the **Draft** drop-down list. The resulting extruded surface model is shown in Figure 10-100.

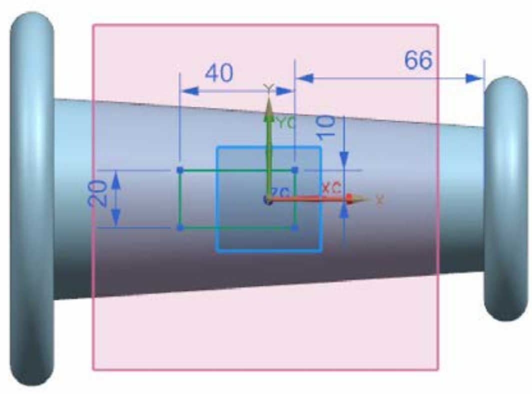


Figure 10-99 *The sketch drawn for creating the extruded feature*

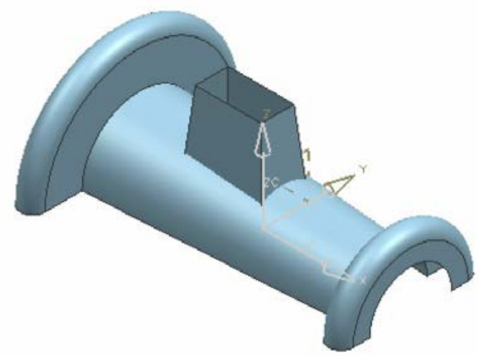


Figure 10-100 *The resulting extruded surface*

Trimming the Base Surface with Respect to the Second Surface

Next, you need to trim the base surface with respect to the second feature.

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 sheet bodies to trim.

2. Select the sheet to be trimmed, as shown in Figure 10-101. Make sure that you select the sheet by using the selection points shown in this figure. After selecting the target sheet body, press the middle mouse button; you are prompted to select the boundary objects.

3. Select the trimming surfaces, as shown in Figure 10-101. Make sure that you select the surfaces by using the points shown in this figure.

4. Select the **Keep** radio button from the **Region** rollout and then choose the **OK** button. The trimmed surface model is shown in Figure 10-102.

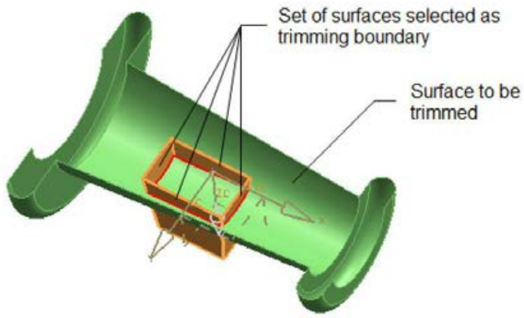


Figure 10-101 *The surface to be trimmed and the trimming surfaces*

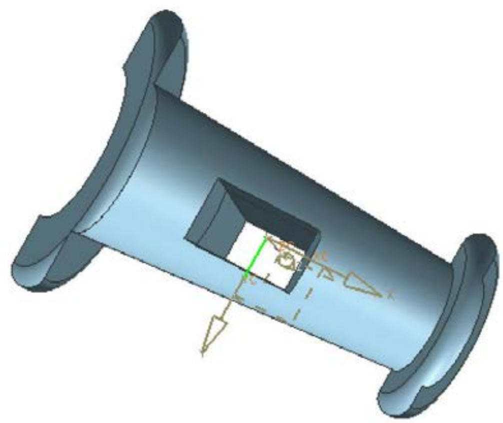


Figure 10-102 *The surface model after trimming the base surface with respect to the extruded surface*

Trimming the Second Surface with the Base Surface

After trimming the base surface, you need to trim the unwanted portions of the second feature.

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

2. Select the sheet to be trimmed, as shown in Figure 10-103, and then press the middle mouse button; you are prompted to select the boundary objects.

3. Select the trimming surface, as shown in Figure 10-103. Select the **Keep** radio button from the **Region** rollout and then choose the **OK** button. The resulting surface after trimming the extended portion of the extruded surface is shown in Figure 10-104.

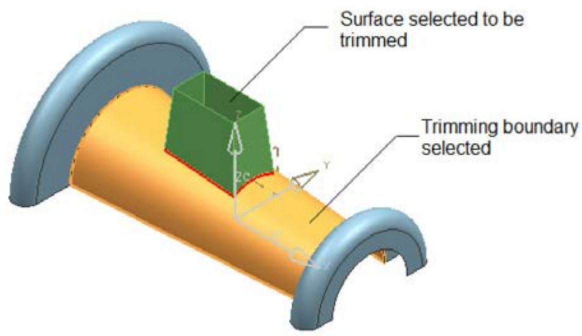


Figure 10-103 The surface to be trimmed and the trimming surface

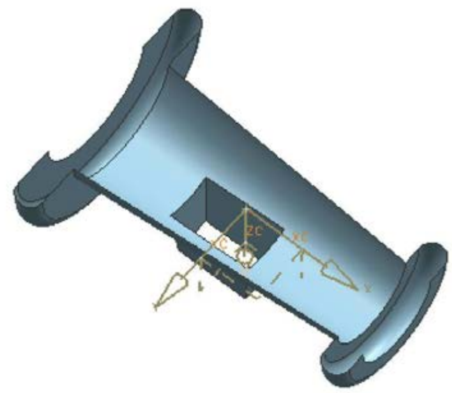


Figure 10-104 The resulting surface after trimming the extended portion of the extruded surface

Creating the Top Surface on the Extruded Feature

Next, you need to create the top surface of the extruded feature.

1. Choose the **Four Point Surface** tool from the **Surface Drop-down** in the **Create** group; the **Four Point Surface** dialog box is displayed and you are prompted to select object to infer point.

2. Select the vertices at the top of the extruded feature, refer to Figure 10-105, and then choose the **OK** button. The resulting surface is shown in Figure 10-106.

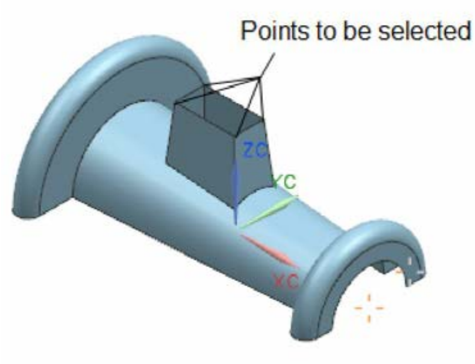


Figure 10-105 Vertices to be selected

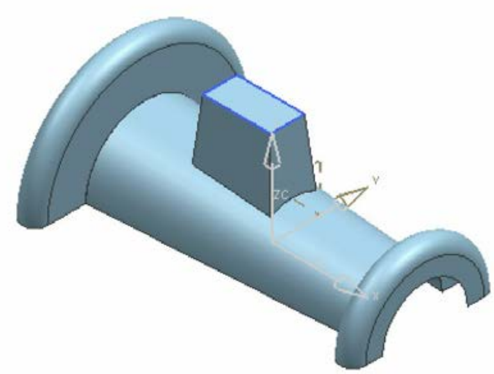


Figure 10-106 The resulting surface

Stitching the Extruded Surface and its Top Surface with the Revolved Surface

After creating all the surfaces, you need to stitch them together by using the

Sew tool.

1. Choose the **Sew** tool from the **Menu > Insert > Combine** of the **Top Border Bar**; the **Sew** dialog box is displayed and you are prompted to select the target sheet to sew.

2. Select the revolved surface; you are prompted to select the tool sheets to sew.

3. Select the extruded surface and its top surface.

4. Choose the **OK** button; all the surfaces are stitched together.

Note
After stitching individual surfaces into a single surface, you can hide the sketch created for revolving the base feature, the sketch created for extruding the second feature, and the datum plane. To hide these entities, press CTRL+B; the ***Class Selection*** *dialog box is displayed. Select the entities and choose the* ***OK*** *button.*

Creating Fillets on Edges Using the Edge Blend Tool

Next, you need to fillet edges.

1. Choose the **Edge Blend** tool from **Menu > Insert > Detail Feature** of the **Top Border Bar**; the **Edge Blend** dialog box is displayed and you are prompted to select the edges for a new set.

2. Select the edges of the surface, refer to Figure 10-107. Enter **5** as the fillet radius value in the **Radius 1** edit box.

3. Choose the **OK** button from the **Edge Blend** dialog box. The completed surface model after adding fillets to edges is shown in Figure 10-108.

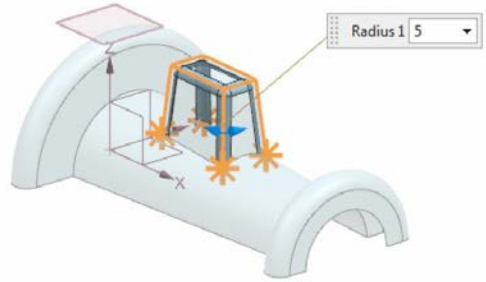


Figure 10-107 The preview of the fillets displayed after selecting the edges

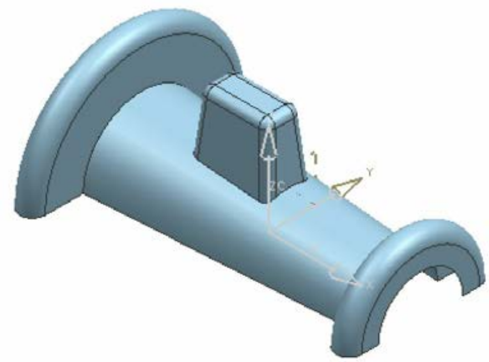


Figure 10-108 The final surface model

Saving and Closing the File

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

Tutorial 2

In this tutorial, you will create the surface model shown in Figure 10-109. The drawing views and dimensions of the surface model are shown in Figure 10-110. After creating the model, save it with the name *c10tut2.prt* at the location *\NX\c10.* **(Expected time: 45 min)**

Figure 10-109 The isometric view of the surface model

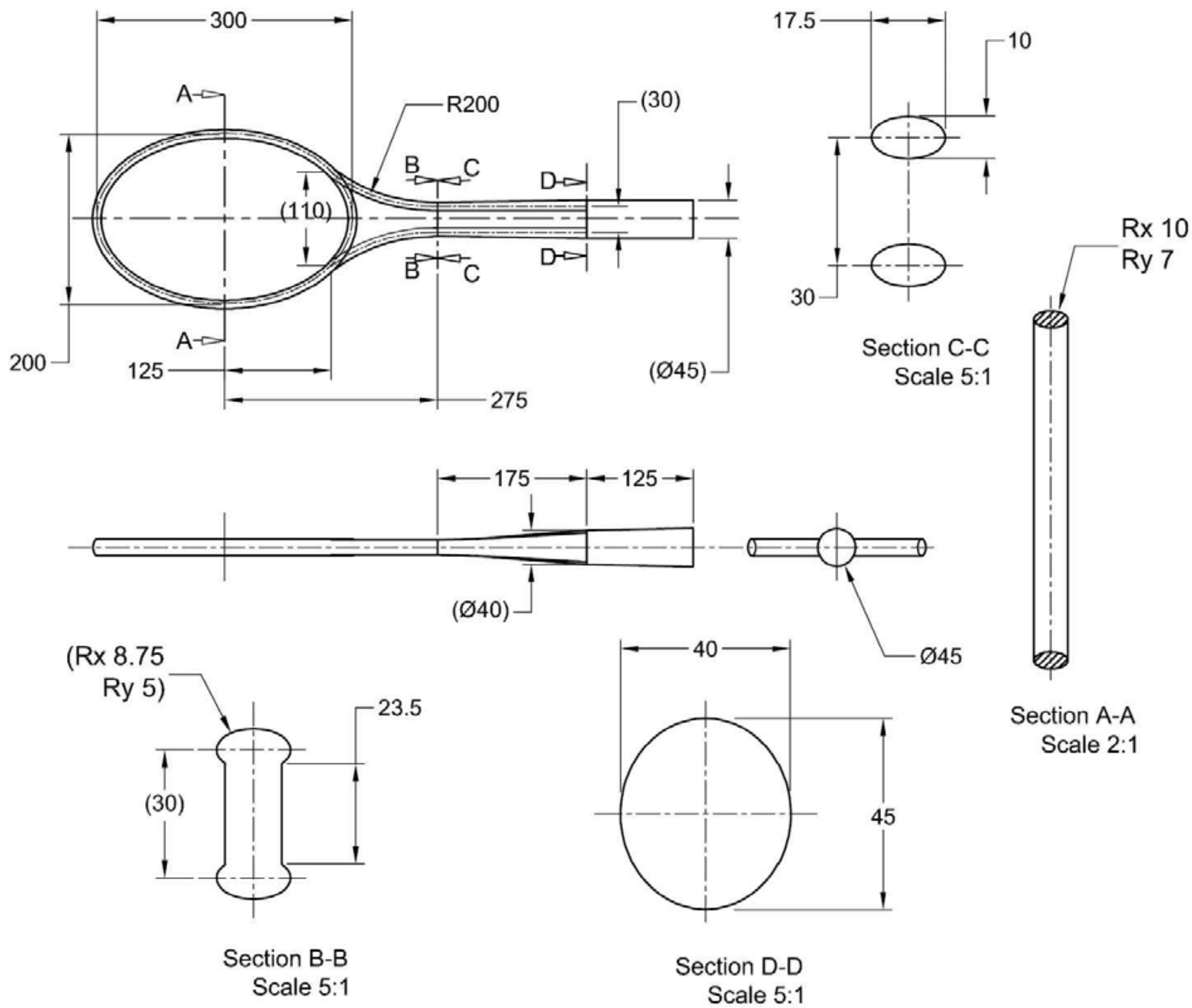


Figure 10-110 *The drawing views and dimensions of the surface model*

The following steps are required to complete this tutorial:

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

b. Create the styled sweep surface as the base feature, refer to Figures 10-111 through 10-113.

c. Create the sweep surface as the second feature and mirror the surface, refer to Figures 10-114 through 10-117.

d. Create the studio surface as the third feature, refer to Figures 10-118 through 10-122.

e. Create the through curves surface as the fourth feature, refer to Figures 10-124 and 10-125.

f. Create the bounded plane surface as the fifth feature, refer to Figure 10-126.

g. Create another bounded plane surface as the sixth feature, refer to Figure 10-127.

h. Save and close the file.

Starting the New File and Setting the Sheet Environment

1. Start a new file with the name *c10tut2.prt* using the **Model** template and specify its location at *C:\NX\c10*.

2. Choose **Menu** > **Preferences** > **Modeling** from the **Top Border Bar**; the **Modeling Preferences** dialog box is displayed.

3. Select the **Sheet** radio button from the **Body Type** rollout and choose the **OK** button.

Creating the Styled Sweep Surface as the Base Feature

As mentioned earlier, the styled sweep surface will be the base feature.

1. Draw an ellipse on the XC-YC plane, as shown in Figure 10-111, and exit the sketching environment.

2. Draw another ellipse on the XC-ZC plane, as shown in Figure 10-112, and exit the sketching environment.

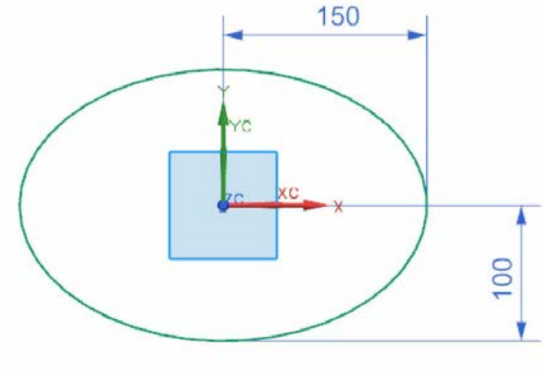


Figure 10-111 *The sketch for the guide string*

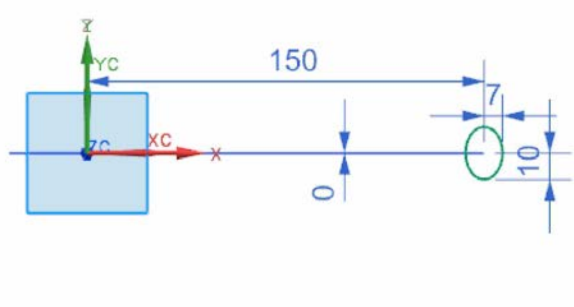


Figure 10-112 *The sketch for the section string*

3. Choose the **Styled Sweep** tool from **Menu** > **Insert** > **Sweep** of the **Top Border Bar**; the **Styled Sweep** dialog box is displayed and you are prompted to select the section curves.

4. Select the **1 Guide** option from the drop-down list of the **Type** rollout in the dialog box, if it is not selected by default.

5. Select the section curve drawn on the XC-ZC plane and press the middle mouse button.

6. Choose the **Guide** button from the **Guide Curves** rollout and then select the guide curve drawn on the XC-YC plane; the preview of the styled sweep surface is displayed in the drawing window. Next, press the middle mouse button.

7. Select the **Guide and Section** option from the **Fixed String** drop-down list of the **Sweep Attributes** rollout in the dialog box.

8. Choose the **OK** button to create the styled sweep surface. Hide the sketches that are used to create the styled surface. The resulting styled sweep surface is shown in Figure 10-113.

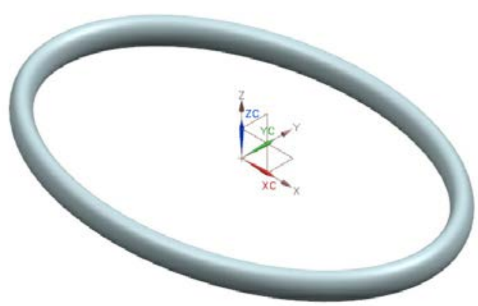


Figure 10-113 *The base feature of the surface model*

Creating the Sweep Surface

The second feature is a sweep surface.

1. Select the XC-YC plane as the sketching plane and draw the guide curve, as shown in Figure 10-114. Next, exit the sketching environment.

2. Create a new datum plane perpendicular to the guide curve by entering **0** as the arc length value.

Note

In this tutorial, the datum plane perpendicular to the guide curve is created by selecting the entity of the guide curve that measures 30 mm.

3. Draw the sketch by selecting the newly created plane as the sketching plane. Note that you may need to specify horizontal or vertical reference for the sketch in the **Create Sketch** dialog box to invoke the sketching environment. Next, draw the ellipse (section curve), refer to Figure 10-115, and then exit the sketching environment.

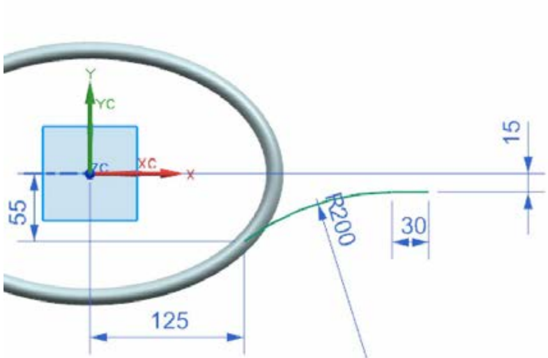


Figure 10-114 The guide curve for creating the sweep surface

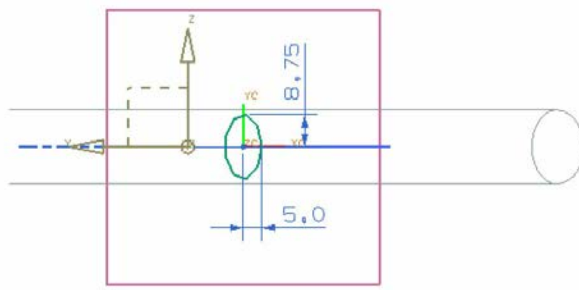


Figure 10-115 The section curve drawn for creating the sweep surface

4. Choose **Menu > Insert > Sweep > Sweep along Guide** from the **Top Border Bar**; the **Sweep Along Guide** dialog box is displayed and you are prompted to select the chain of curves for the section.

5. Select the section curve and then press the middle mouse button; you are prompted to select the guide curve.

6. Select the guide curve and then choose the **OK** button; the sweep surface is created. Hide the sketches that are used to create the sweep surface. The resulting surface model is shown in Figure 10-116.

Mirroring the Sweep Surface

1. Mirror the last created surface by using the XC-ZC plane as the mirror plane. The surface model after mirroring the sweep surface is shown in Figure 10-117.

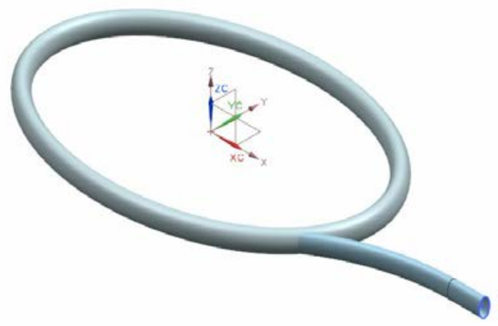


Figure 10-116 *The surface model after creating the sweep surface*

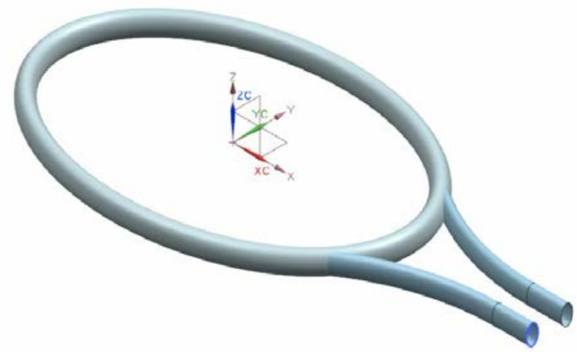


Figure 10-117 *The surface model after mirroring the sweep surface*

Creating the Studio Surface

The third feature is a surface and it will be created by using the **Studio Surface** tool.

1. Choose the **Sketch in Task Environment** tool to draw the section of the sweep surface by using the last datum plane.

2. Choose **Menu > Insert > Recipe Curve > Project Curve** from the **Top Border Bar**; the **Project Curve** dialog box is displayed and you are prompted to select the geometry to be projected. Make sure the **Original** option is selected in the **Output Curve Type** drop-down list of the **Settings** rollout.

3. Select the edges of the sweep surfaces and create a profile, as shown in Figure 10-118. Exit the Sketch in Task environment.

4. Create a datum plane parallel to the YC-ZC plane at a distance of 450 mm.

5. Select the newly created datum plane as the sketching plane and create the second section curve, as shown in Figure 10-119. For dimensions, refer to Figure 10-110.

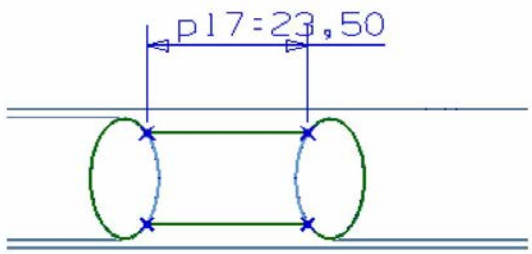


Figure 10-118 The first section curve drawn to create the through curve mesh surface

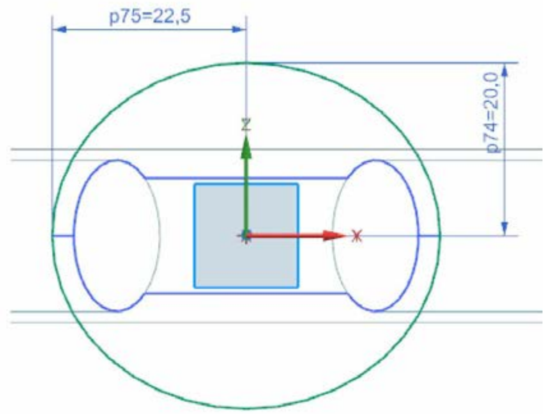


Figure 10-119 The second section curve

6. Select the XC-YC plane as the sketching plane and create the guide strings, as shown in Figure 10-120.

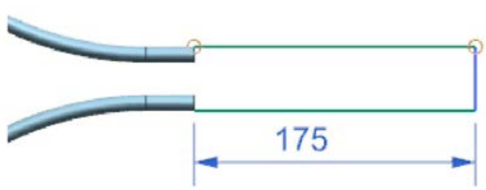


Figure 10-120 Sketch for the guide strings

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

8. Select all the elements from first section string, refer to Figure 10-121, and then press the middle mouse button; you are prompted to select the section again.

9. Select the second section string, refer to Figure 10-121, and then choose the **Curve** button from the **Guide (Cross) Curves** rollout; you are prompted to select the guide string.

10. Select the first guide curve, refer to Figure 10-121, and then press the middle mouse button; you are prompted to select the guide string again.

11. Select the second guide curve, refer to Figure 10-121. The preview of the

studio surface after selecting the sections and the guide curves is shown in Figure 10-122. Next, choose the **Apply** button and then the **Cancel** button to create the studio surface.

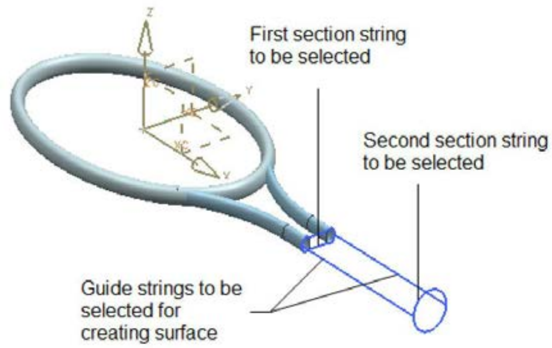


***Figure 10-121** The second section curve and the guide strings to create a surface*

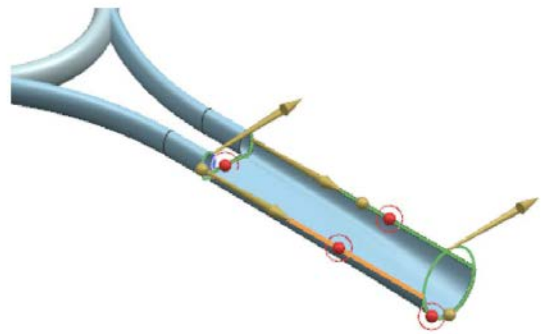


***Figure 10-122** Entities to be selected for creating the studio surface*

Mirroring the Studio Surface

1. Mirror the studio surface using the XC-YC plane as the mirror plane. The surface model after mirroring the studio surface is shown in Figure 10-123.

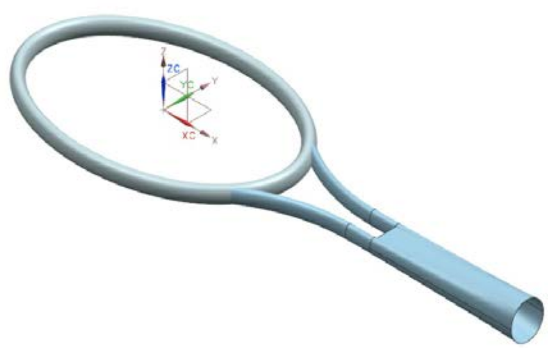


***Figure 10-123** The surface model after mirroring the studio surface*

Creating the Through Curves Surface

Next, you need to create the ruled surface.

1. Create a plane parallel to the YC-ZC plane at a distance of **575** mm.

2. Select the newly created plane as the sketching plane and draw the sketch, as shown in Figure 10-124.

3. Exit the Sketch in Task environment and choose **Menu > Insert > Mesh Surface > Through Curves** from 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 first section string, refer to Figure 10-125. Next, press the middle mouse button; you are again prompted to select the curve or point to section.

5. Select the second section string, which is the edge of the surface created earlier, refer to Figure 10-125. Note that the arrows should point in the same direction. Next, choose the **OK** button; the surface is created.

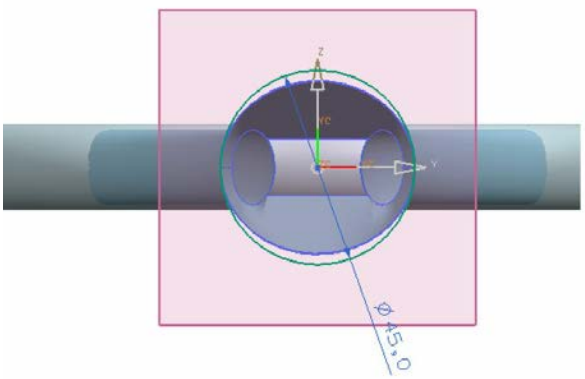


Figure 10-124 The first section curve drawn to create the through curves surface

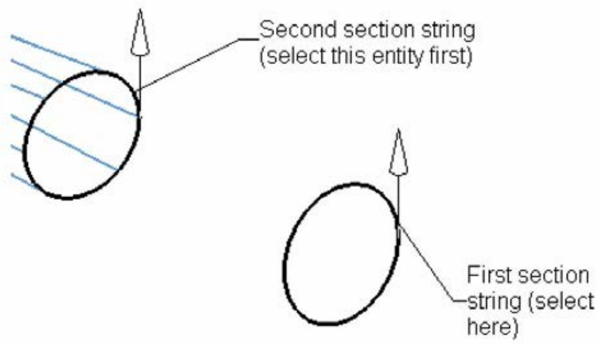


Figure 10-125 The through curves surface created after selecting both cross-sections

6. Change the color of the through surface to black.

Creating the N-sided Surface

1. Choose the **N-Sided Surface** tool from **Menu > Insert > Mesh Surface** of the **Top Border Bar**; the **N-Sided Surface** dialog box is displayed and you are prompted to select chain of curves for outer loop.

2. Select the **Trimmed** option from the drop-down list in the **Type** rollout and then select the circular edge, refer to Figure 10-126. The preview of the bounded plane surface is displayed. Next, select the **Trim to Boundary** check box from the **Settings** rollout and choose the **OK** button from the dialog box; the fill surface is created.

3. Again, invoke the **N-Sided Surface** dialog box and select bounding string, refer to Figure 10-127. Make sure that the **Trim to Boundary** check box is

selected. Next, choose the **OK** button from this dialog box. The bounded plane surface is created. The final surface model after hiding all the sketches and datum planes is shown in Figure 10-128.

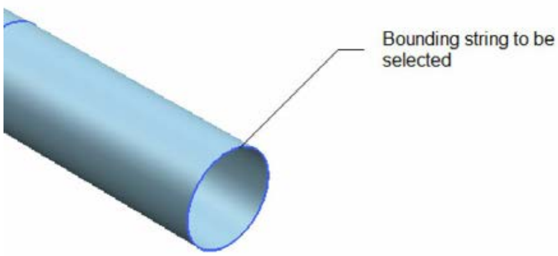


Figure 10-126 The bounding string to be selected

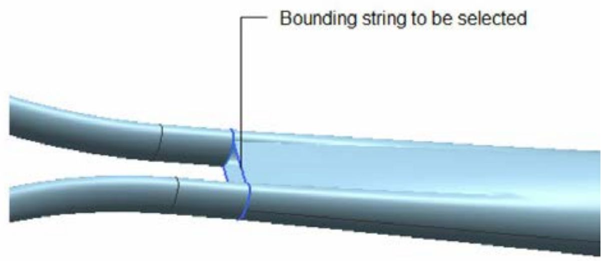


Figure 10-127 The bounding string to be selected

Figure 10-128 The final surface 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 a revolved surface.

2. The _____________ tool is used to create a sheet from *n* number of guide

curves and *n* number of section curves.

3. The _____________ tool is used to stitch individual surfaces into a single surface.

4. The _____________ tool is used to trim and extend a surface.

5. The _____________ tool is used to create a planar surface.

6. The _____________ tool is used to extend a surface dynamically or by defining the laws for extension.

7. The _____________ tool is used to create an offset surface.

8. In NX, surfaces are termed as sheets. (T/F)

9. The **Trim and Extend** tool is used to trim or extend an open or a closed surface. (T/F)

10. You can use the **Until Selected** and **Until Next** options from the **End** drop-down list of the **Extrude** dialog box to create a base sheet. (T/F)

11. You can create surfaces by specifying section strings and guide strings using the **Through Curves Mesh** tool. (T/F)

Answer the following questions:

1. How many points are required to create a surface while using the **Four Point Surface** tool?

(a) Three (b) Four
(c) Five (d) None of these

2. Which of the following tools is used to create a single patch surface or a

multi-patch triangular surface that encloses a closed 2D sketch or a closed 3D curve?
(a) **N-Sided Surface** (b) **Silhouette Flange**
(b) **Law Extension** (d) None of these

3. Which of the following options is available in the **Type** rollout of the **Law Extension** rollout?
(a) **Faces, Vector** (b) **Visual Gap, Absolute Gap**
(c) **Basic, Absolute Gap** (d) None of these

4. Before adding a fillet at the intersection of two surfaces, the surfaces have to be:
(a) Stitched using the **Sew** tool (b) Merged
(c) Trimmed (d) None of these

5. The _____________ tool is used to create a surface by sweeping a single section or multiple sections along single or multiple guide curves.

6. You can invoke the **Bounded Plane** tool by choosing **Menu > Insert > Surface > Bounded Plane** from the **Top Border Bar** in the Shape Studio environment. (T/F)

7. You can select an open sketch to create a bounded plane surface. (T/F)

8. Surface models do not have mass properties. (T/F)

9. You can create a surface from a closed or an open sketch. (T/F)

10. Once thickness has been added to a sheet, it is converted into a solid. (T/F)

11. You can create a hole feature on a planar surface by using the **Hole** tool. (T/F)

EXERCISES

Exercise 1

Create the surface model shown in Figure 10-129. The drawing views and dimensions of the surface model are shown in Figure 10-130. Save the model with the name *c10exr1.prt* at the location *\NX\c10*. **(Expected time: 30 min)**

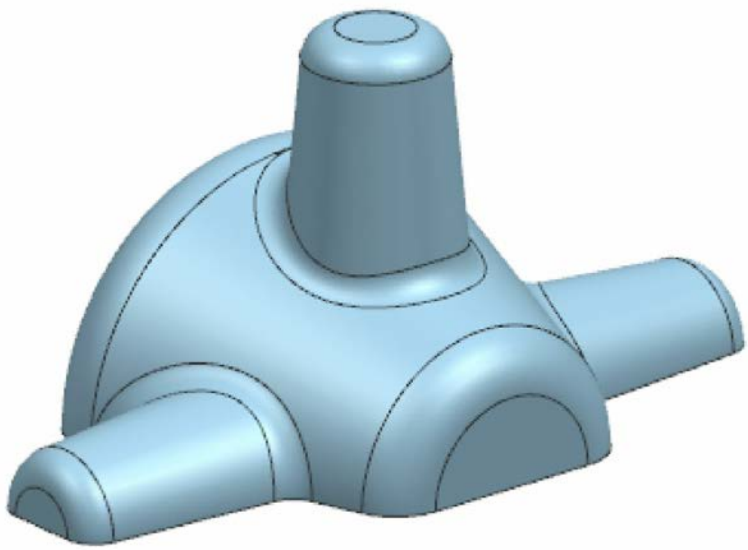


Figure 10-129 *Surface model for Exercise 1*

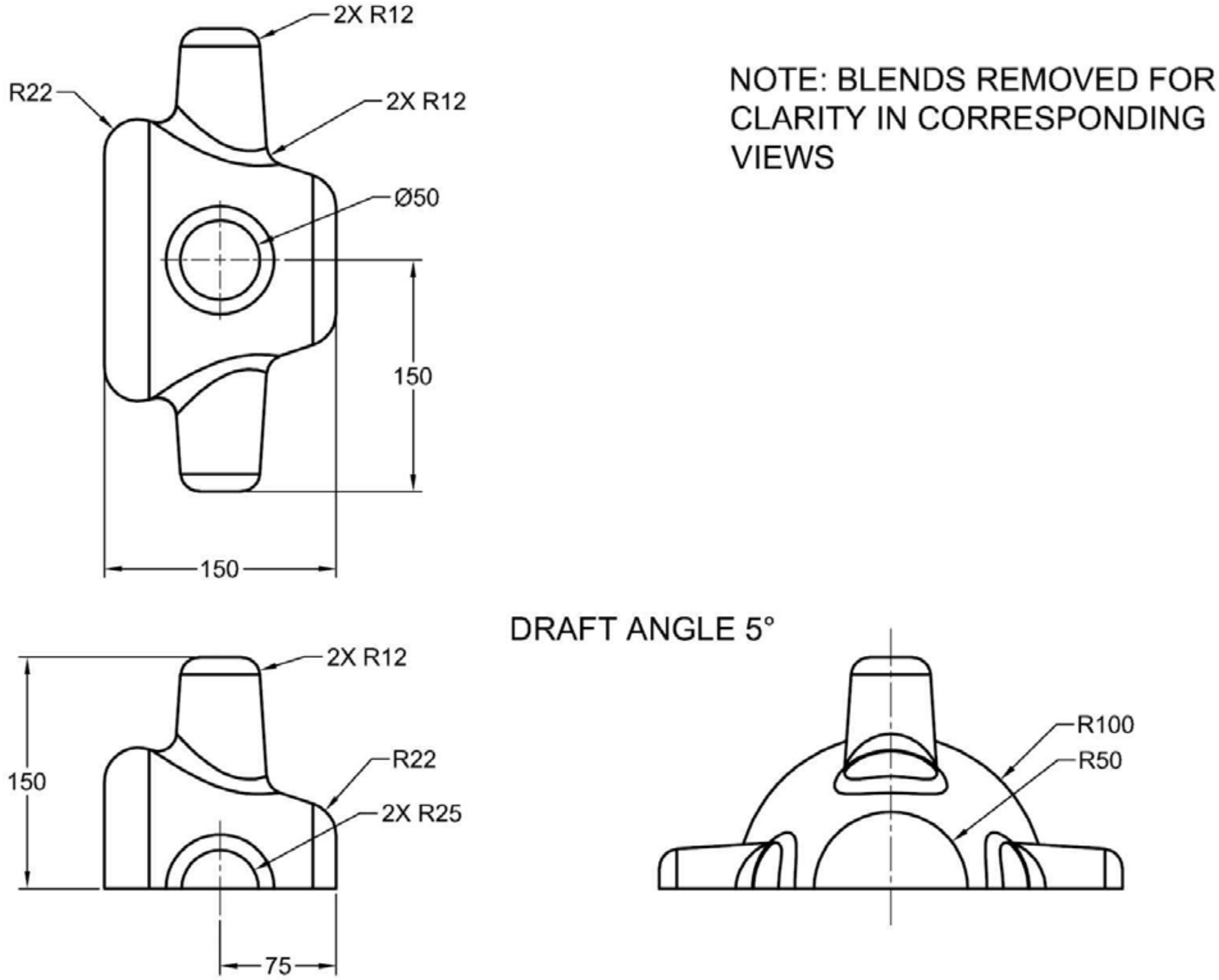


Figure 10-130 Drawing views and dimensions of the surface model for Exercise 1

Exercise 2

Create the surface model shown in Figure 10-131. The drawing views and dimensions of the surface model are shown in Figure 10-132. Save the model with the name *c10exr2.prt* at the location *\NX\c10.* **(Expected time: 30 min)**

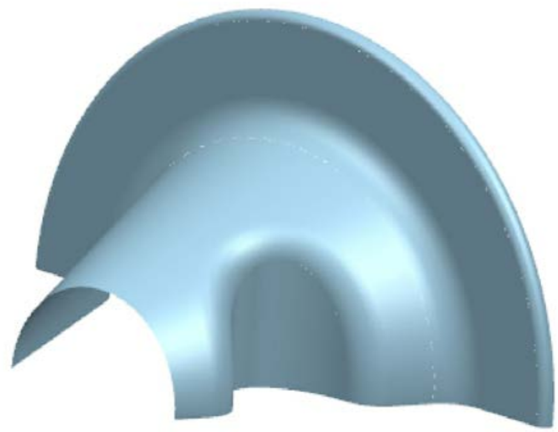


Figure 10-131 The isometric view of the surface model for Exercise 2

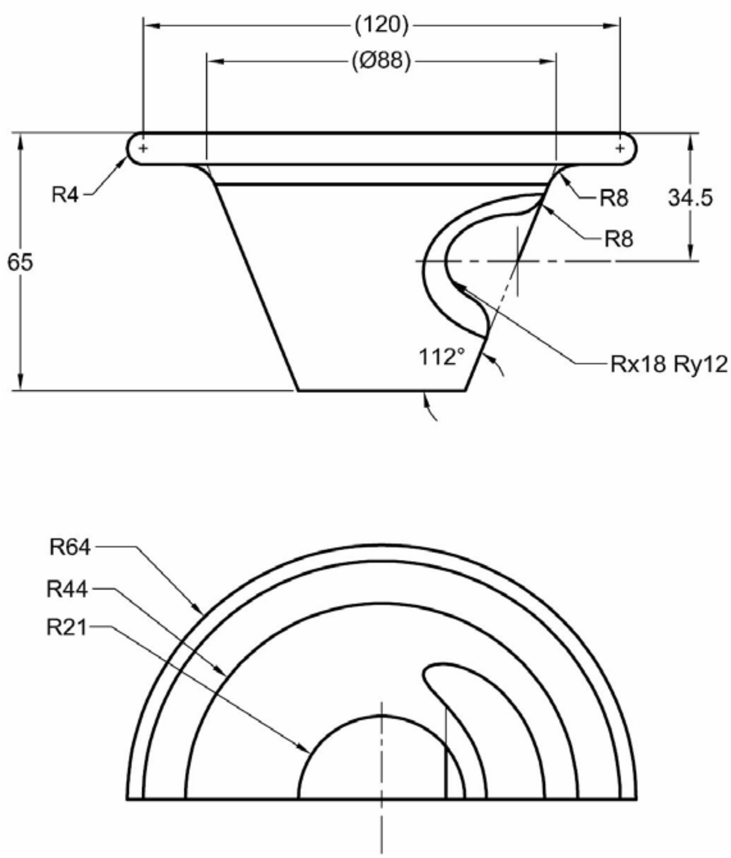


Figure 10-132 The dimensions and drawing views of the surface model

Answers to Self-Evaluation Test

1. Revolve, **2. Studio Surface**, **3. Sew**, **4. Trim and Extend**, **5. Bounded Plane**, **6. Law Extension**, **7. Offset Surface**, **8.** T, **9.** T, **10.** F, **11.** T

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.

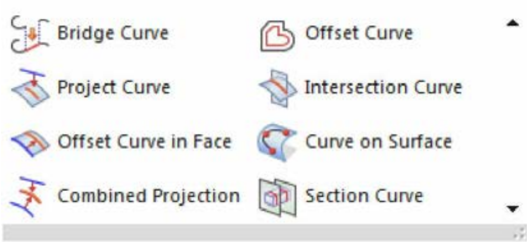


***Figure 11-1** Tools in the **Curve** gallery*

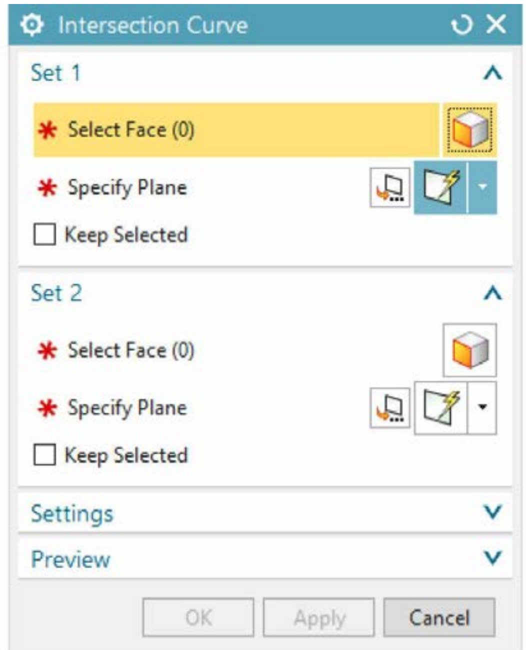


***Figure 11-2** The **Intersection Curve** dialog box*

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