Chapter 11

Surface Modeling

Learning Objectives

After completing this chapter, you will be able to:

- *Create extrude, revolve, and sweep surfaces.*
- *Create the ruled surface.*
- *Create surfaces using the Through Curves and Through Curve Mesh tools.*
- *Create a surface by the four points method.*
- *Create the swoop surface.*
- *Create the bounding plane surface.*
- *Create the transition surface.*
- *Create an N-sided surface.*
- *Create silhouette flange surfaces.*
- *Extend and create a surface by using the Law Extension method.*
- *Create uniform and variable surface offsets.*
- *Trim and extend a surface by using the Trim and Extend tool.*
- *Create studio surfaces.*
- *Create styled blend surfaces.*
- *Create styled sweep surfaces.*
- *Sew individual surfaces into a single surface.*
- *Add thickness to the surface.*

INTRODUCTION TO SURFACE MODELING

Surfaces are three dimensional (3D) bodies that possess zero thickness. They are used extensively for modeling complex features. The model or the assembly created using the surface body type possesses a surface area but not the volume or mass properties. In NX, surfaces are created in the form of single or multiple patches. With the increase in the 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 nonuniform surface area are created with the help of the surface modeling technique. 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 **Modeling** application. Before creating the surface model, you need to change the body type to sheet.

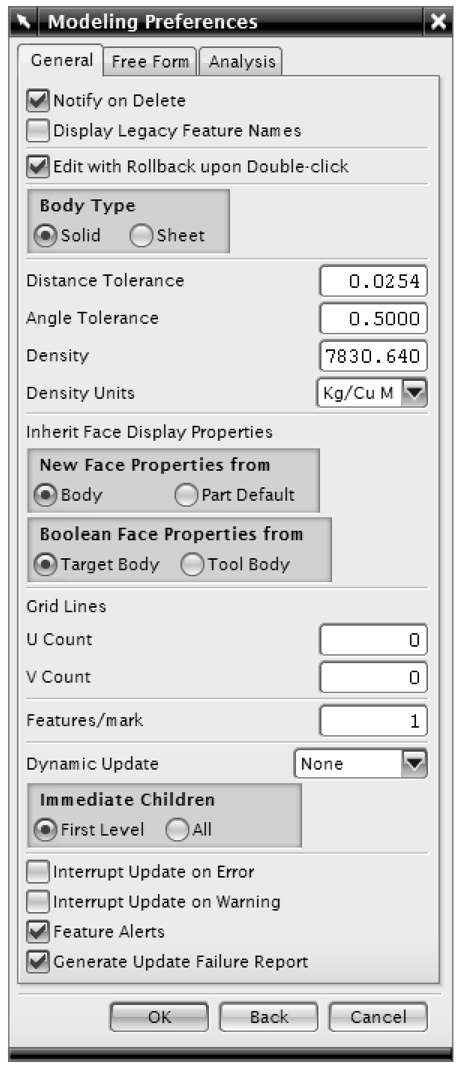


Figure 11-1 *The* ***Modeling Preferences*** *dialog box*

INVOKING THE SHEET MODELING ENVIRONMENT

To invoke the **Sheet Modeling** environment, invoke the **Modeling** environment and then choose **Preferences > Modeling** from the menu bar; the **Modeling Preferences** dialog box will be displayed, as shown in Figure 11-1. Choose the **General** tab and select the **Sheet** radio button from the **Body Type** area. Choose the **OK** button to exit the dialog box. All models created, henceforth, in the **Modeling** application will be sheet models.

Creating an Extruded Surface

Menu: Insert > Design Feature > Extrude
Toolbar: Feature > Extrude

As mentioned earlier, there is no separate tool for creating the surface extrude. After invoking the **Sheet Modeling** environment, you can use the **Extrude** tool to create the extruded sheets. The sketch drawn for creating the extruded surface may be an open or a closed entity. After creating the sketch, choose the **Extrude** button; 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 extrusion values in the **Start** and **End** edit boxes. Choose the **OK** button; the sheet will be created. The options in the **Extrude** dialog box are the same as discussed in Chapter 4. The surface extrude operations performed on open and closed sketches are displayed in Figures 11-2 and 11-3, respectively. In case of sheet bodies, you cannot use the **Until Next** and **Until Selected** options from the **End** drop-down list in the **Extrude** dialog box.

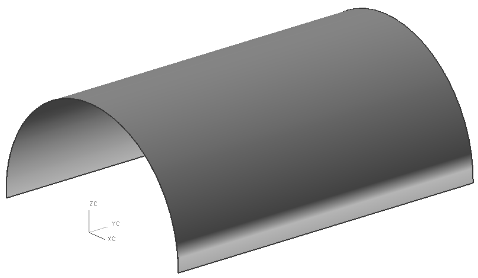


Figure 11-2 *Surface extrude created on an open sketch*

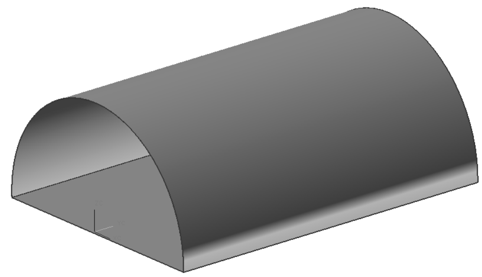


Figure 11-3 *Surface extrude created on a closed sketch*

Note

*You can use only the **None** option from the **Boolean** drop-down list in the **Sheet Modeling** environment. The other options in this drop-down list are not available in this environment.*

Creating a Revolved Surface

Menu:	Insert > Design Feature > Revolve
Toolbar:	Feature > Revolve

The **Revolve** tool is used to create the revolved surface. Choose the **Revolve** button from the **Feature** toolbar; the **Revolve** dialog box will be displayed. Also, you will be prompted to select the section geometry. Select the sketch. Next, choose the **Inferred Vector** button and specify the axis of revolution. Specify the start and end angles in the **Angle** edit boxes. Choose the **OK** button; the revolved sheet will be created. The revolved surface models created using an open sketch and a closed sketch are shown in Figures 11-4 and 11-5, respectively.

Figure 11-4 *Revolved surface created using an open sketch*

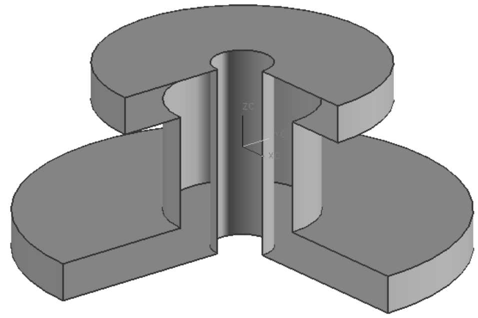


Figure 11-5 *Revolved surface created using a closed sketch*

Creating a Ruled Surface

Menu:	Insert > Mesh Surface > Ruled
Toolbar:	Surface > Ruled *(Customize to add)*

The **Ruled** tool is used to create ruled surfaces. These surfaces are always created between two similar or dissimilar cross-sections created on different parallel planes. The sketches for this feature may be open or closed. Initially, isoparametric lines are formed to create patches, which are then converted into surfaces. The options to create isoparametric curves are discussed later in this chapter. To create the ruled surface, create two cross-sections on two different planes. Choose the **Ruled** button from the **Surface** toolbar; the **Ruled Surface** dialog box will be displayed, as shown in Figure 11-6.

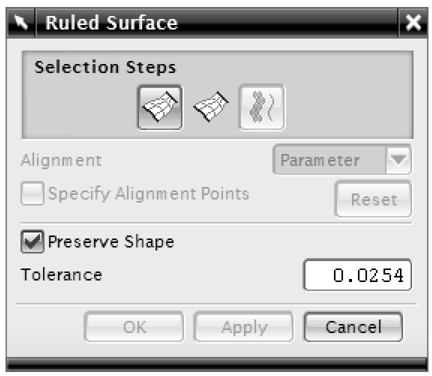


Figure 11-6 *The* ***Ruled Surface*** *dialog box*

By default, the **Section String 1** button will be chosen and you will be prompted to select the first section string. Select the first cross-section; an arrow will be displayed on the first section string indicating the direction of the surface formation. Next, choose the **Section String 2** button; you will be prompted to select the section string 2. Select the second section string; an arrow will be displayed from the second section string also. The arrows on the first and second section strings should point in the same direction.

In the **Alignment** drop-down list, different methods are provided to distribute points for creating the isoparametric lines that form patches. If you select the **Parameter** option, the points will be distributed such that the isoparametric lines are formed at equal parameter intervals. If you select the **Arc length** option, the entire curve will be divided into equal segments with respect to arc length. Also, the isoparametric curve will pass through the dividing points. The **By Points** option will be selected when the cross-sections are of different shapes and have sharp corners. If you select the **Distance** option, the equally spaced isoparametric lines will be created perpendicular to the direction vector selected. If you select the **Angles** option, the isoparametric curves with angles at equal intervals will be created with respect to the common axis line. If you select the **Spine Curve** option, the isoparametric curves will be formed at the intersection points created on the selected curves by the perpendicular planes.

Accept the default tolerance value. After selecting both the section strings, as shown in Figure 11-7, choose the **OK** button. The resulting ruled surface will be created, as shown in Figure 11-8.

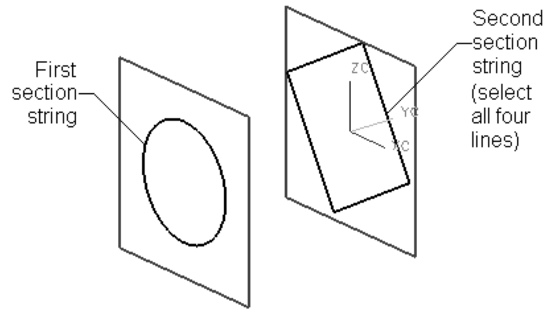


Figure 11-7 The section strings selected for creating the ruled surface

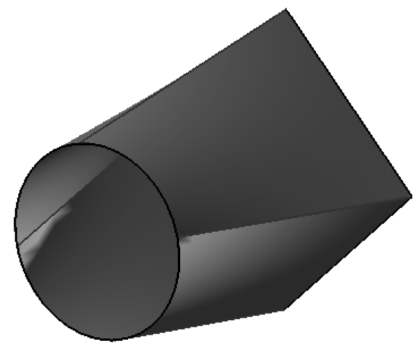


Figure 11-8 The resulting ruled surface created from the selected section strings

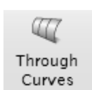
Note

The maximum allowed and the minimum required number of cross-sections for creating the ruled surface is two.

Creating a Surface Using the Through Curves Tool

Enhanced

Menu: Insert > Mesh Surface > Through Curves
Toolbar: 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 create surfaces using the **Through Curves** tool, choose the **Through Curves** button from the **Surface** toolbar; the **Through Curves** dialog box will be displayed, as shown in Figure 11-9, and you will be prompted to select the section. Select the section and press the middle mouse button; you will be again prompted to select the section. Likewise, you can select any number of section strings. After selecting the section strings, make sure that all arrows point in the same direction.

In the **Patch Type** drop-down list of the **Output Surface Options** rollout , you have three options, **Single**, **Multiple**, and **Match String**. If you select the **Single** option, the 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 on the **Alignment** option selected from the **Alignment** rollout.

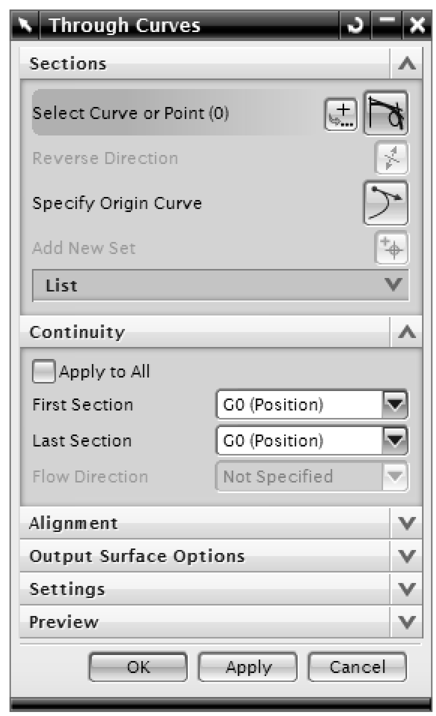


*Figure 11-9 The **Through Curves** dialog box*

The **Closed in V** check box and the **Degree** edit box will be enabled only when you select the **Multiple** option from the **Patch Type** drop-down list. If you select the **Closed in V** check box, the surface body will be closed in the V direction. The value entered in the **Degree** edit box determines the curvature of the isoparametric lines between the selected section strings formed for the creation of the surface. If there are n section strings, the value entered in the **V Degree** edit box should be between 1 to n-1. Figure 11-10 shows the section strings to be selected for creating the surfaces through curves and Figure 11-11 shows the resulting surface.

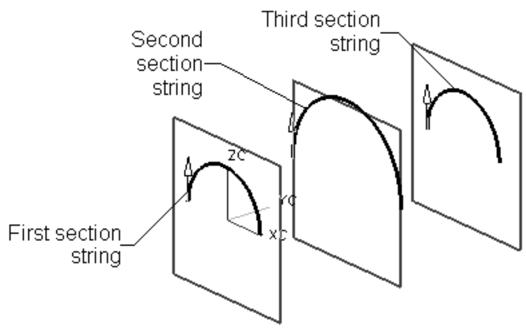


Figure 11-10 *Section strings selected for creating a* ***Through Curves*** *surface*

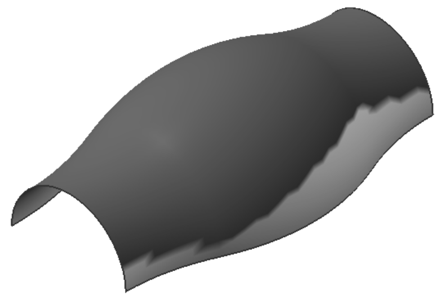
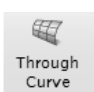


Figure 11-11 *The resulting surface formed from the selected through curves*

Creating a Surface Using the Through Curve Mesh Tool

Menu:	Insert > Mesh Surface > Through Curve Mesh
Toolbar:	Surface > Through Curve Mesh



You can create surfaces by specifying the section strings and the guide strings using the **Through Curve Mesh** tool. You can specify any number of section strings and guide strings. If you want to select multiple guide strings, they should be tangentially related to each other. To create the surface using the **Through Curve Mesh** method, invoke the **Through Curve Mesh** tool from the **Surface** toolbar; the **Through Curve Mesh** dialog box will be displayed, as shown in Figure 11-12, and you will be prompted to select the primary curves. You need to select a collection of control curves such as the primary curves and cross curves for creating surfaces. Select the first primary curve and press the middle mouse button to select the next primary curve. Likewise, 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 the cross curves. Select the first cross curve and press the middle mouse button to select the next cross curve. Likewise, you can select any number of cross curves.

Note that after selecting two primary curves, the **Spine** rollout will be added to the **Through Curve Mesh** dialog box. The **Spline** 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. The options in other rollouts are discussed next.

Output Surface Options Rollout

The options from the **Emphasis** drop-down list in the **Output Surface Options** rollout allow you to define the set of curves that effect 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 have an equal 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.

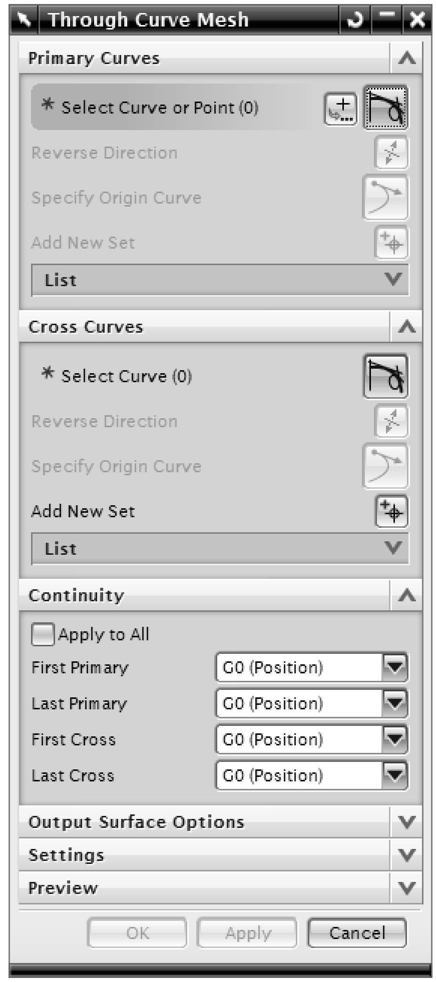


Figure 11-12 *The* ***Through Curve Mesh*** *dialog box*

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 **Manual** option and entering the value in the **Degree** spinner. If you select the **Advanced** 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 11-13 shows the control strings selected for creating the through curve mesh surface and Figure 11-14 shows the resulting surface. You can enter the distance tolerance value between the curves in the **G0** edit box and the angle tolerance value in the **G1** edit box. The curvature tolerance value can be entered in the **G2** edit box.

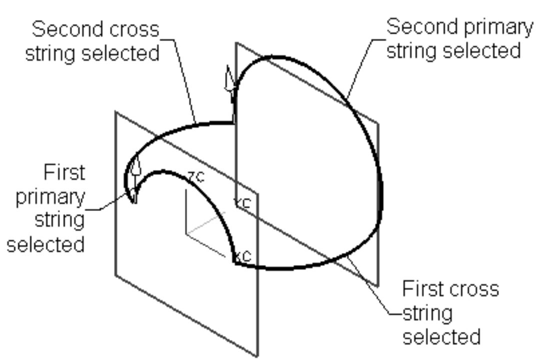


Figure 11-13 *The control strings selected for creating the through curve mesh surface*

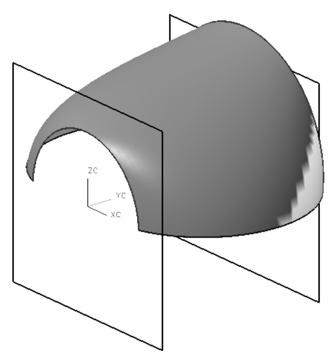


Figure 11-14 *The resulting through curve mesh surface*

Creating a Surface by 4 Points

Menu:	Insert > Surface > Four Point Surface
Toolbar:	Free Form Shape > Four Point Surface

The **Four Point Surface** tool is used to create a planar (2D) or nonplanar (3D) surface. To create a surface by using this method, choose the **Four Point Surface** button from the **Freeform Shape** toolbar; the **Surface by 4 Points Icon Options** will be displayed and you will be prompted to specify the first surface corner. Specify the point for the first surface corner. Similarly, you will be prompted to specify the other three corners. Specify the other three corners and choose the **Apply** button; the surface will be created. You can also reselect the previously selected corner point. To do so, choose the **Delete Last Point** button from the **Surface by 4 Points Icon Options** and specify the point for the corner again. Figure 11-15 shows the imaginary layout of the surface to be formed after specifying the corner points. Figure 11-16 shows the resulting planar surface formed by enclosing the corner points specified.

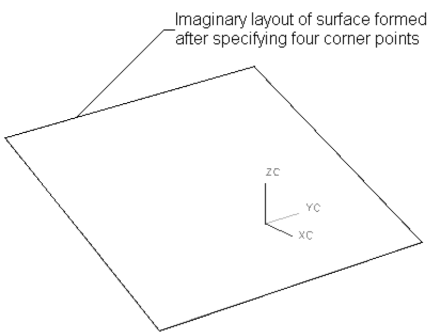


Figure 11-15 *The imaginary layout of the surface formed after specifying the four points for corners*

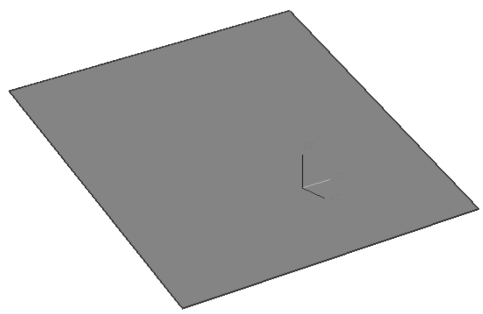


Figure 11-16 *The resulting Surface*

Creating a Swoop Surface

Menu:	Insert > Surface > Swoop
Toolbar:	Free Form Shape > Swoop

The swoop surfaces are created as rectangular or square shaped planar (2D) surfaces, and later modified into 3D surfaces by using the options in the swoop tool. To create a swoop surface, choose the **Swoop** button from the **Freeform Shape** toolbar; the **Point** dialog box will be displayed and you will be prompted to define the first rectangle corner by specifying the inferred point. Specify the point for the first corner of the rectangle; you will be prompted to define the second rectangle corner by specifying the inferred point. Specify the second point; the planar surface will be created. The vertical and horizontal axes will be displayed in red color over the planar surface. Also, the **Swoop Shape Control** dialog box will be displayed, as shown in Figure 11-17. The **Swoop Shape Control** dialog box is used to modify the shape of the default surface formed. In the **Select Control** area, all the possible reference positions of the surface are displayed. At a time, the shape of the surface can be modified only at one reference position. You can select any one option and the shape of the surface will be altered in the selected reference position by using the shape modification sliders. If you select the **Cubic** radio button from the **Degree** area, the final surface formed will be of 3-degrees. Also, it will be comparatively harder. If you select the **Quintic** radio button, the resulting surface will be comparatively smoother.

*Figure 11-17 The **Swoop Shape Control** dialog box*

Sliding Bars

You can use the **Stretch** sliding bar to stretch the surface in a positive or negative direction along the reference position selected from the **Select Control** area. The neutral value is 50 for all sliders. The **Bend** sliding bar, can be used to bend the surface in a positive or negative direction along the reference position selected from the **Select Control** area. The **Skew** sliding bar can be used to create a skewness factor for the surface in the positive or negative direction along the reference position selected from the **Select Control** area. Use the **Twist** sliding bar to provide a twisting effect to the surface in the positive or negative direction along the reference position selected from the **Select Control** area. The **Shift** sliding bar can be used to shift the other edge of the surface in the positive or negative direction along the reference position selected from the **Select Control** area. Figure 11-18 shows the planar surface created after specifying both the corners of the rectangle. Figure 11-19 shows the 3D surface modified from the planar surface using the shape modification sliding bars.

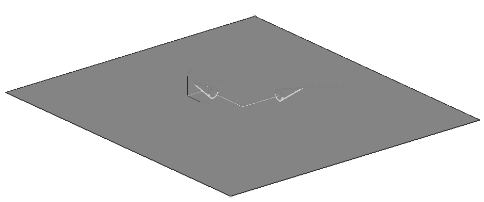


Figure 11-18 The planar surface created after specifying both the corners of the rectangle

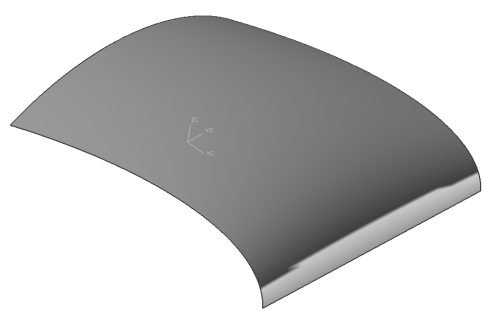


Figure 11-19 The resulting 3D surface modified from the planar surface created

Creating a Planar Surface from 2D Sketches and Edges of Solid

Menu:	Insert > Surface > Bounded Plane
Toolbar:	Feature > Bounded Plane *(Customize to add)*

The **Bounded Plane** tool can be used to create a surface from the 2D sketches or the closed coplanar edges. If you need to enclose a 2D sketch or closed coplanar edges with a surface, choose the **Bounded Plane** button from the **Feature** toolbar; the **Bounded Plane** dialog box will be displayed, as shown in Figure 11-20 and you will be prompted to select the bounding string. Select an entity from the **Bounded Plane** dialog box. For example, on choosing the **Solid Edge** button, the **Bounded Plane** dialog box will be displayed and you will be prompted to select a bounding string or a solid face. Select the solid edge and choose the **OK** button; the **Bounded Plane** surface will be created. Figure 11-21 shows the bounded plane surface enclosing a 2D sketch and Figure 11-22 shows the bounded plane surface created from a circular edge. You can also create the bounded plane surface by selecting a solid face. You can only select 2D faces. The resulting surface, after selecting the solid face, will remain on the same surface.

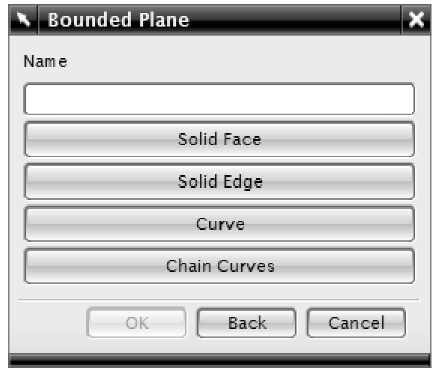


Figure 11-20 *The* ***Bounded Plane*** *dialog box*

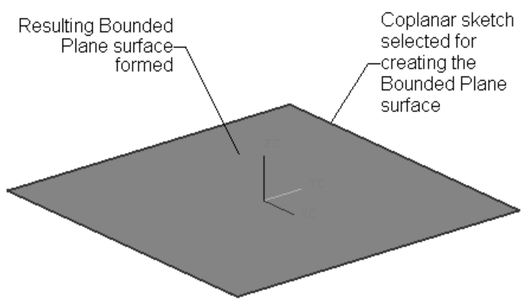


Figure 11-21 *The bounded plane surface formed from a 2D sketch*

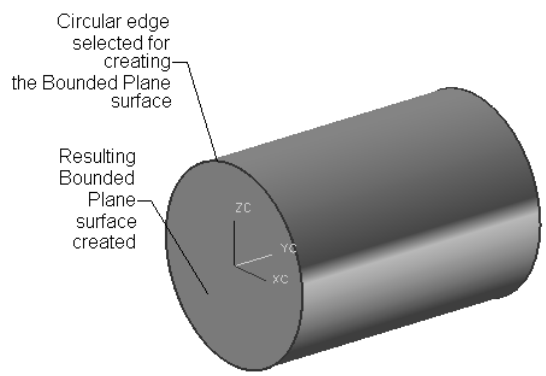
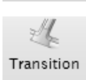


Figure 11-22 *The bounded plane surface formed from a circular edge*

Creating a Transition Surface Using the Transition Tool

Menu:	Insert > Surface > Transition
Toolbar:	Surface > Transition



Generally, the creation of a transition surface involves selection of the required cross-sections and mapping the intersected surface formed between the selected cross-sections automatically. You can also define the shape constraint for the connecting (intersecting) surface. For creating a **Transition** surface, you need to create atleast two cross-sections. Generally, three cross-sections are created. After creating cross-sections, choose the **Transition** button from the **Surface** toolbar; the **Transition** dialog box will be displayed and you will be prompted to select the section. Choose the **More** button to enlarge the dialog box, as shown in Figure 11-23. The **Section** button is chosen by default. Select sections and choose the **OK** button. Note that you need to press the middle mouse button after selecting each section.

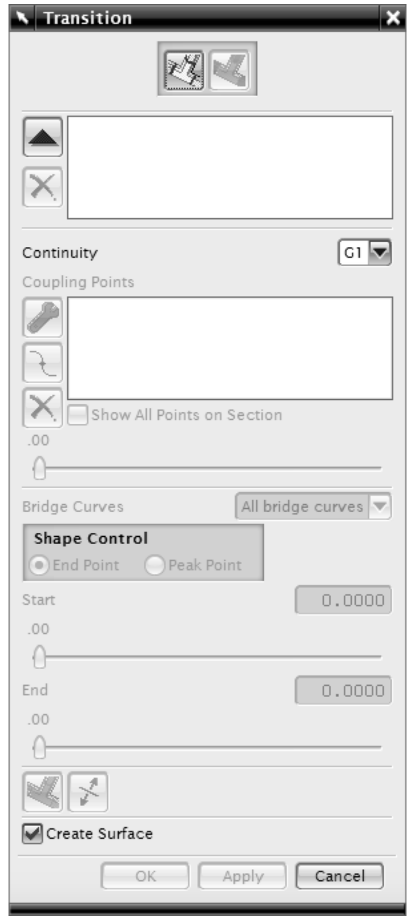


*Figure 11-23 The **Transition** dialog box*

Constrain Face, Reverse Normal, and Surface Preview Buttons

After selecting the first cross-section, the **Constrain Face** button will be enabled. To maintain the shape constraint of the intersecting surface with an existing surface, choose the **Constrain Face** button and select the reference face; the selected cross-sections will be listed in the list box. By default, **G0** is selected from the **Continuity** drop-down list, which implies that there is no shape constraint in the continuity of the intersected surface. If you select the **G1** option, the tangential continuity will be maintained. If you select the **G2** option, the curvature continuity will be maintained. To reverse the direction of the intersected surface, choose the **Reverse Normal** button. The **Surface Preview** button is used to display the preview of the intersected surface to be created. The **Transition** surface will be formed only if the **Create Surface** check box is selected. Else, only the bridge curves will be formed between the selected cross-sections.

Bridge Curves Drop-down List

The bridge curves that are formed after selecting the cross-sections are listed as individual curves and separate groups in the **Bridge Curves** drop-down list. By selecting the required curve from the **Bridge Curves** drop-down list and adjusting the sliding bar, you can control the shape of the selected curve. You can maintain the shape from the endpoint or the peak point. If you select the **End Point** radio button, the **Start** and **End** sliding bars will be available for controlling the shape of the bridge curve selected from the **Bridge Curves** drop-down list.

Peak Point Radio Button

If you select the **Peak Point** radio button, the **Depth** and the **Skew** sliding bars will be available for controlling the shape of the selected bridge curve from the **Bridge Curves** drop-down list.

The cross-sections for creating the **Transition** surface along with the bridge curves are shown in Figure 11-24. The resulting **Transition** surface is formed, as shown in Figure 11-25.

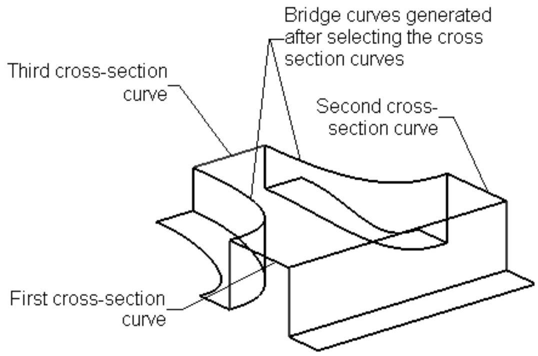


Figure 11-24 The bridge curves generated after selecting the cross-section curves

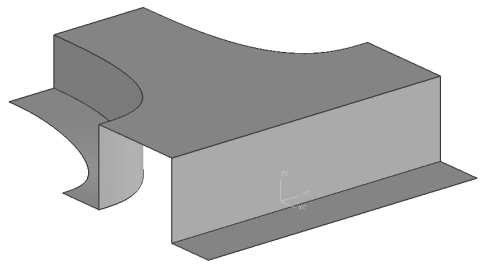


Figure 11-25 The resulting transition surface created from the cross-section curves

Creating an N-sided Surface

Menu:	Insert > Mesh Surface > N-sided Surface
Toolbar:	Surface > N-sided Surface

The **N-sided Surface** tool is used to create a single patch surface or multipatch triangular surfaces that enclose a closed 2D sketch or a closed 3D curve. While doing so, an existing surface can be optionally selected as the reference for maintaining the shape of the surface to be created. For creating the N-sided surface, choose the **N-sided Surface** button from the **Surface** toolbar; the **N-sided Surface** dialog box will be displayed, as shown in Figure 11-26, and you will be prompted to select a closed loop of curves or edges. By default, the **Trimmed Single Sheet** button will be chosen from the **Type** area. This selection will create a surface with a single patch. To create a surface with multiple triangular patches, you need to choose the **Multiple Triangular Patches** button from the **Type** area.

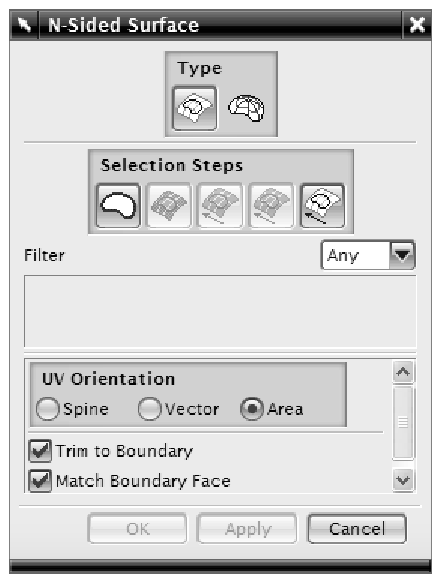


*Figure 11-26 The **N-Sided Surface** dialog box*

To create a single patch surface, invoke the **N-Sided Surface** dialog box. By default, the **Boundary Curves** button is chosen from the **Selection Steps** area. Select the **Spine** radio button from the **UV Orientation** area; you will be prompted to select a closed loop of curves or edges. After selecting a closed boundary of a 2D sketch or a 3D curve, choose the **Boundary Faces** button from the **Selection Steps** area; you will be prompted to select the face for the boundary constraint. After selecting the face or the set of faces, choose the **OK** button; the surface will be formed. If the **Trim to Boundary** check box is selected, the surface formed will be automatically trimmed with respect to the closed loop of the curve or the sketch selected. The orientation of the surface formed can be specified along the **U** and **V** directions by using the options in the **UV Orientation** area. If you select the **Spine** radio button and choose the **UV Orientation - Spine** button from the **Selection Steps** area, you will be prompted to define the V orientation. Select the spine and choose the **OK** button. Likewise, if you select the **Vector** radio button, you will be prompted to select an object to infer a vector. If you select the **Area** radio button and choose the **UV Orientation-Vector** button from the **Selection Steps** area, the **Vector Method** drop-down list will be enabled to define the vector direction.

You can also create the surface by choosing the **Multiple Triangular Patches** button from the **Type** area. In the above case, after choosing the **OK** button, the **Shape Control** dialog box will be displayed, as shown in Figure 11-27. If you select the **Position** radio button, then the

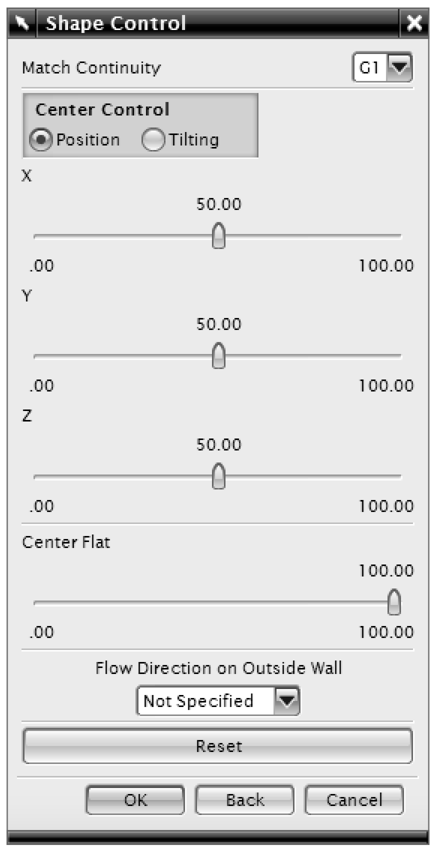


Figure 11-27 *The **Shape Control** dialog box*

position of the surface can be varied in the respective directions by sliding the X, Y, and Z sliding bars. If you select the **Tilting** radio button, you can tilt the surface along the X, Y, and Z directions by using the corresponding sliding bars. Also, you can control the center flatness of the surface by sliding the **Center Flat** sliding bar. The **Flow Direction on Outside Wall** drop-down list contains options for controlling the flow direction of the newly formed surface with the existing reference surface selected. The **Drag** button will be enabled only for the surfaces with multiple triangular patches. By choosing the **Drag** button, you can interact with the **Shape Control** dialog box at any time. Figure 11-28 shows the single patch N-sided surface created for the selected boundary curve and Figure 11-29 shows the multitriangular patch N-sided surface created for the selected boundary curve.

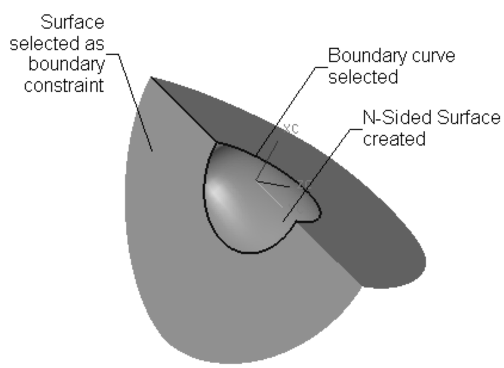


Figure 11-28 *Single patch N-sided surface created for the selected boundary curve*

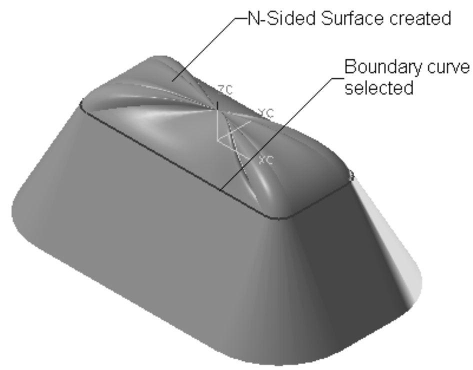


Figure 11-29 *Multitriangular patch N-sided surface created for the selected boundary curve*

Creating a Silhouette Flange Surface

Menu: Insert > Flange Surface > Silhouette Flange
Toolbar: 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 flange surfaces on an edge or a curve by taking any of the adjacent surfaces as the reference. For creating the flange surface, choose the **Silhouette Flange** tool from the **Surface** toolbar; the **Silhouette Flange** dialog box will be displayed, as shown in Figure 11-30, and you will be prompted to select the curves or edges. By default, the **Basic** button is chosen from the **Type** area and the **Base Curves or Edges** button is chosen from the **Selection Steps** area. By selecting the **Basic** button, you can create the flange without the help of other existing flange surfaces. Select the edge or a curve for creating the **Silhouette Flange** surface and choose the **Base Faces** button from the **Selection Steps** area; you will be prompted to select the faces. After selecting the reference face, choose the **Reference Direction** button from the **Selection Steps** area; the reference direction options will be enabled.

Direction Drop-down List

If you select the **Face Normal** option from the **Direction** drop-down list, the face that is normal to the reference face will be selected for specifying the direction. If you select the **Vector** option, the **Vector** drop-down list will be enabled through which you can specify the reference direction. On doing so, you will be prompted to specify the reference direction. Specify the direction for the surface formation by selecting an edge or a curve. You can select only a straight curve or edge for specifying the reference direction.

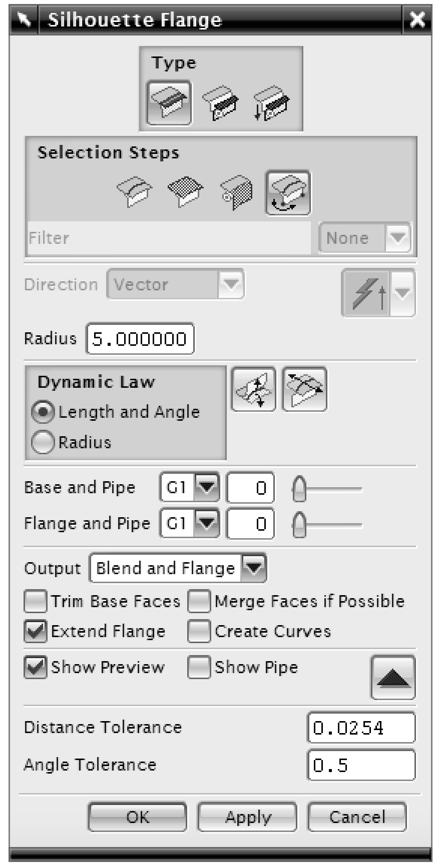


***Figure 11-30** The **Silhouette Flange** dialog box*

After specifying the reference direction, choose the **Shape Silhouette Flange** button from the **Selection Steps** area. If you have selected the **Show Preview** check box, the preview of the flange surface, along with the rotational and linear handles, will be displayed after selecting the **Shape Silhouette Flange** button. If you have selected the **Show Pipe** check box, the circular pipe will be displayed at the point of formation of the flange surface. If you clear the **Show Pipe** check box, only the fillet will be displayed at the point of creation of the flange surface. The radius of the fillet created at the start point of the surface will depend on the value entered in the **Radius** edit box. By choosing the **Reverse Direction** button, you can flip the direction of the flange surface. By choosing the **Reverse Side** button, you can reverse the side of the surface. In the **Output** drop-down list, you have options to display the required output. If you select the **Blend and Flange** option, both the fillet and the flange surfaces will be displayed as the output. If you select the **Pipe Only** option, only the round pipe that controls the fillet will be displayed as the output. If you select the **Flange Only** option, the flange surface will be displayed as the output. Figure 11-31 shows the pipe displayed along with the flange surface created using the **Basic** method and Figure 11-32 shows the resulting flange surface with the fillet.

Creating a Silhouette Flange Surface Using the Absolute Gap Method

By choosing the **Absolute Gap** button from the **Type** area, you can create a silhouette flange surface relative to the existing silhouette flange surface by maintaining a predefined gap. The minimum gap is calculated by taking the radius of the two pipes and the nearest tangential distance between them. You can also maintain a predefined gap between the two silhouette flange surfaces by entering the gap value in the **Gap** edit box. If you choose the **Absolute Gap** button from the **Type** area, the **Gap** edit box will be enabled in the **Silhouette Flange** dialog box. By default, the **Existing Silhouette Flange** button will be chosen from the **Selection Steps** area and you will be prompted to select an existing

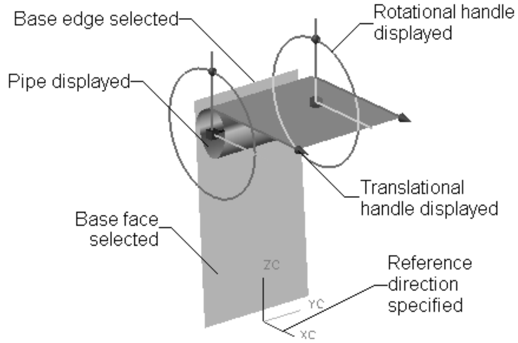


Figure 11-31 The silhouette flange surface displayed along with the handles and pipe

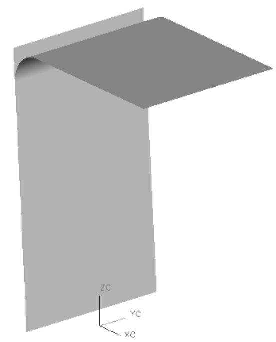


*Figure 11-32 The resulting silhouette flange surface created using the **Basic** method*

silhouette flange feature. Select the existing silhouette flange surface and choose the **Base Faces** button from the **Selection Steps** area. Select the reference face and choose the **Reference Direction** button from the **Selection Steps** area. Specify the reference direction and choose the **Shape Silhouette Flange** button to display the surface created. Figure 11-33 shows the pipe displayed, along with the preview of the flange surface created using the **Absolute Gap** method and Figure 11-34 shows the resulting flange surface with the fillet.

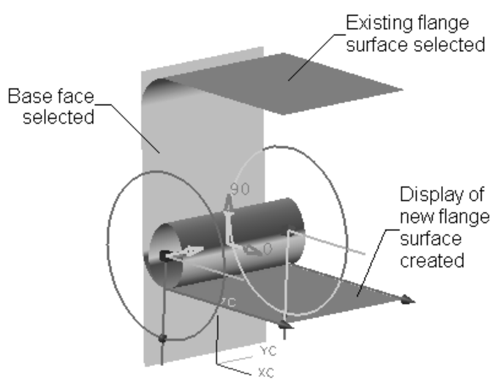


Figure 11-33 The newly created silhouette flange surface displayed along with handles and pipe

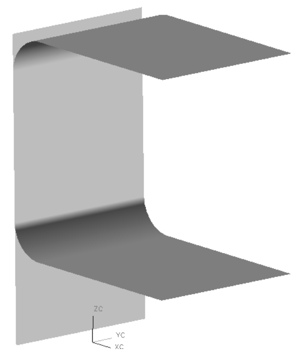


*Figure 11-34 The resulting silhouette flange surface created using the **Absolute Gap** method*

Creating a Silhouette Flange Surface Using the Visual Gap Method

The **Visual Gap** button from the **Type** area can be chosen to create the flange surface in accordance with an existing flange surface by specifying a visual gap attribute between the two flange surfaces. For creating the silhouette flange surface using the visual gap method, choose the **Visual Gap** button from 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 for creating the surface.

Extending a Surface Using the Law Extension Tool

Menu:	Insert > Flange Surface > Law Extension
Toolbar:	Surface > Law Extension

The **Law Extension** tool can be 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 both the methods is discussed next.

Extending a Surface Dynamically Using the Dynamic Option

As discussed earlier, you can also extend a surface dynamically by choosing the **Law Extension** button from the **Surface** toolbar. The **Law Extension** dialog box will be displayed, as shown in Figure 11-35, and you will be prompted to select the base curve string. By default, the **Faces** button is selected from the **Reference Method** area. Using this method, you can extend the surface by taking an existing face as the reference. Select the curve string to be extended from the surface that is to be extended and choose the **Reference Face** button from the **Selection Steps** area; you will be prompted to select reference faces. Select the required face as the reference face and choose the **Define Law** button from the **Selection Steps** area. On selecting the **Show Preview** check box, the preview of the surface will be displayed. If you have selected the **Extend on Both Sides** check box, the surface will be created on both sides of the edge selected. If you have selected the **Merge Faces if Possible** check box, then the newly created surface will merge with the reference face selected, if possible. The preview of the created surface will be displayed along with the rotational and translation handles on both the end points of the edge selected. You can modify the surface by dynamically dragging the rotational and the translational handles. After modifying the surface, choose the **OK** button to reflect the changes. Figure 11-36 shows the preview of the extended surface along with the rotational and translational handles. Figure 11-37 shows the resulting surface created after dynamically modifying it using the handles.

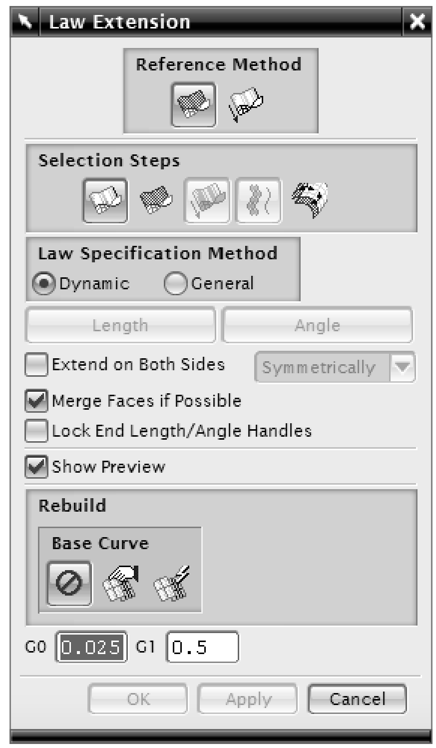


***Figure 11-35** The **Law Extension** dialog box*

Extending a Surface Driven by Laws Using the General Option

As discussed earlier, you can also extend a surface by defining different type of laws. To do so, choose the **Law Extension** button from the **Surface** toolbar. Select the **General** radio button from the **Law Specification Method** area; the **Length** and **Angle** buttons will be enabled in the **Law Extension** dialog box. Select the base curve and the reference face from the surface to be extended after choosing the respective buttons from the **Selection Steps** area. Choose

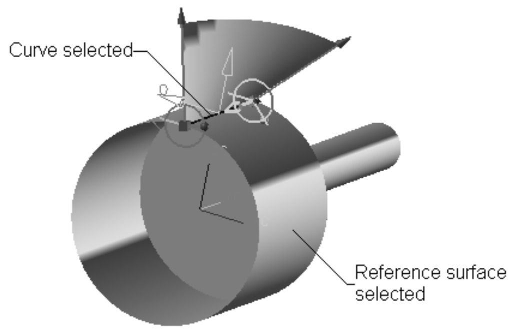


Figure 11-36 The preview of the law extension surface created

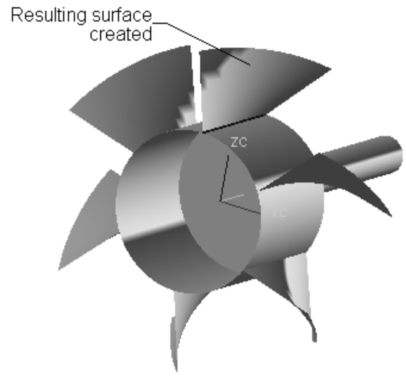


Figure 11-37 The resulting law extension surface created and circularly arrayed

the **Length** button from the **Law Specification Method** area; the **Law Function** dialog box will be displayed. By using the options in the dialog box, you can specify the length law for the surface. For example, choose the **Linear** button from the dialog box; the **Law Controlled** dialog box will be displayed. Enter the start and end values in the **Start Value** and the **End Value** edit boxes, respectively, and choose the **OK** button. Next, choose the **Angle** button from the **Law Specification Method** area; the **Law Function** dialog box will be displayed. Choose any method to define the angle law for the surface. For example, choose the **Linear** button; the **Law Controlled** dialog box will be displayed. Enter the start and end angle values in the **Start Value** and the **End Value** edit boxes, respectively, and choose the **OK** button twice; the extended surface will be displayed.

Note

*The curve selected from the surface for extension should lie on the reference face selected. If you choose the **Vector** button from the **Reference Method** area, the **Vector Method** drop-down list will be available. By using this drop-down list, you can specify the vector direction for extending the surface.*

Creating a Surface Offset Using the Offset Surface Tool

Menu:	Insert > Offset /Scale > Offset Surface
Toolbar:	Surface > Offset Surface

The **Offset Surface** tool can be used to offset a surface in the direction normal to the selected surface. For offsetting a surface, choose the **Offset Surface** button from the **Surface** toolbar; the **Offset Surface** dialog box will be displayed, as shown in Figure 11-38. By default, the **Face** button is chosen from the **Face to Offset** rollout and you will be prompted to select the faces for the new set. Select the face, as shown in Figure 11-39. Next, enter the offset value in the **Offset 1** edit box. If you want create a new set, choose the **Add New Set** button from the **Face to Offset** rollout. Choose the **Reverse Direction** button to flip the offset direction. Choose the **OK** button; the resulting surface offset is shown in Figure 11-39.

Figure 11-38 The ***Offset Surface*** *dialog box*

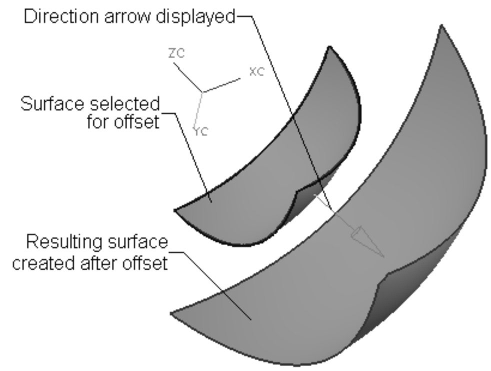


Figure 11-39 The offset surface created

Trimming and Extending a Surface Using the Trim and Extend Tool

Menu:	Insert > Trim > Trim and Extend
Toolbar:	Surface > Trim and Extend

The **Trim and Extend** tool can be used to trim or extend an open or a closed surface. To trim or extend a surface, choose the **Trim and Extend** button from the **Surface** toolbar; the **Trim and Extend** dialog box will be displayed, as shown in Figure 11-40, and you will be prompted to select the target edge to extend. Select a single edge or multiple edges from the surface that is to be extended. When you select multiple edges for extending, ensure that the selected edges are in continuity. If you select the **Preview** check box, the preview of the extended surface will be displayed. The different rollouts in the **Trim and Extend** dialog box are discussed next.

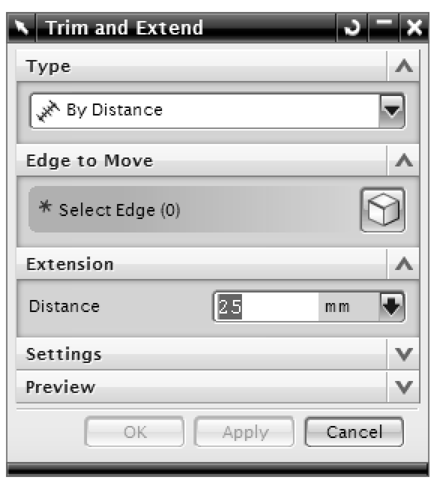


Figure 11-40 The ***Trim and Extend*** *dialog box*

Type Rollout

On selecting the **By Distance** option from the drop-down list in the **Type** rollout, you will be able to define the length of the surface extension by specifying a distance value in the **Distance** edit box. If you select the **Percentage of Measured** option from this rollout, the extension length will be specified in terms of the percentage of the original surface. On selecting the **Until Selected** option, the surface will be extended up to the selected reference object. This option can also be used to trim the 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 **Extention Method** drop-down list of the **Settings** rollout are used to define the 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-degrees 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 **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 from the **Arrow Side** drop-down list in the **Desired Results** rollout are used to retain or discard the selected tool body. If you select the **Retain** option, the selected tool body will be retained after trimming. If you select the **Delete** option, then the material from the tool body will be removed in the direction of the arrow displayed on selecting the tool body. Figure 11-41 shows the preview of the extension surface after selecting the edges. Figure 11-42 shows the surface extended using the **Until Selected** option.

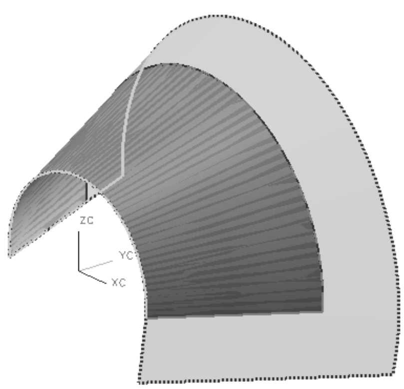


__Figure 11-41__ The preview of the extended surface from the selected edges

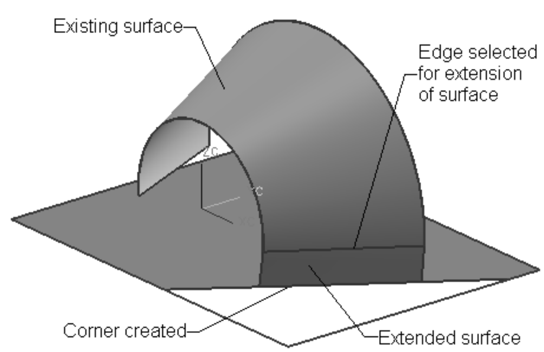


__Figure 11-42__ The resulting surface extended using the __Until Selected__ option

Trimming a Sheet by Using the Trimmed Sheet Tool

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

The **Trimmed Sheet** tool can be 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 intersect fully with the trimming surface. Choose the **Trimmed Sheet** button from the **Surface** toolbar; the **Trimmed Sheet** dialog box will be displayed, as shown in Figure 11-43, and you will be prompted to select a target sheet body. By default, the **Sheet Body** button will be chosen from the **Target** rollout. Select the sheet to be trimmed and press the middle mouse button.

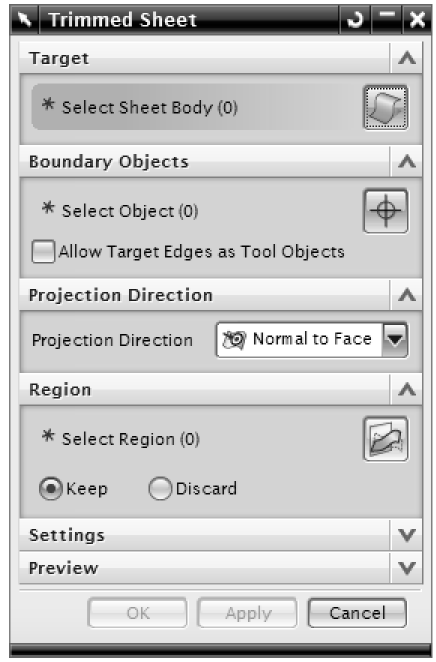


*Figure 11-43 The **Trimmed Sheet** dialog box*

Next, you will be prompted to select the boundary objects. Select the boundary objects. Choose the **Region** button; the surface to be trimmed will be divided into two colors. The dark gray color 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 in gray color has to be kept or discarded. If you select the **Keep** radio button from this rollout, the region in dark gray color will be retained, and the other region will be removed. If you select the **Discard** radio button, the region in dark gray color will be removed (trimmed) and the other region will be retained.

Figure 11-44 shows the entities selected for trimming a surface. Figure 11-45 shows the resulting trimmed surface after selecting the **Discard** radio button from the **Region** rollout.

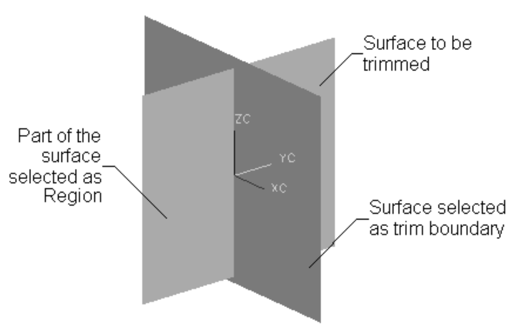


Figure 11-44 The entities selected for trimming a sheet

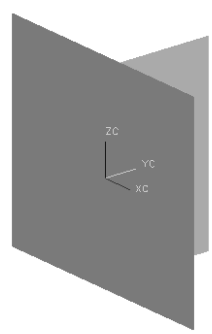


*Figure 11-45 The resulting trimmed sheet after selecting the **Discard** radio button*

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 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 11-46 shows the objects selected when the trim boundary is created by imprinting a curve on the surface to be trimmed. Figure 11-47 shows the resulting trimmed surface after selecting the **Discard** radio button from the **Region** rollout.

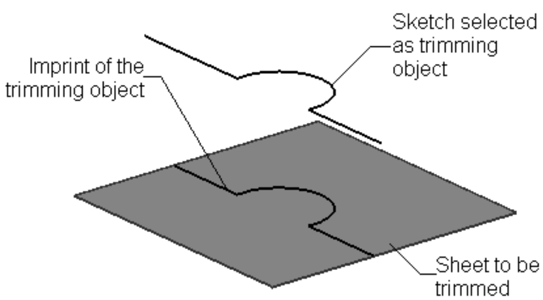


Figure 11-46 *The sketch selected for trimming a sheet*

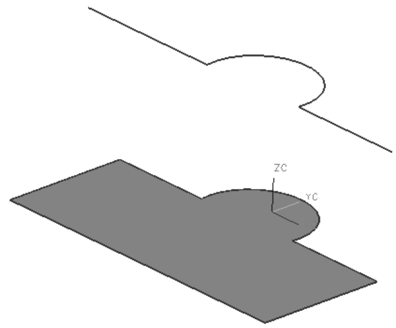


Figure 11-47 *The resulting trimmed sheet after selecting the **Discard** radio button*

Creating a Surface Using the Studio Surface Tool

Enhanced

Menu:	Insert > Mesh Surface > Studio Surface
Toolbar:	Freeform Shape > Studio Surface

The **Studio Surface** tool can be 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 open or closed.

Choose the **Studio Surface** button from the **Freeform Shape** toolbar; the **Studio Surface** dialog box will be displayed, as shown in Figure 11-48, and you will be prompted to select a section. By default, the **Section (Primary) Curves** button will be 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 section curves. Note that all the guide curves should also point in one direction.

Other options in the dialog box have been discussed in earlier tools. After selecting all the parameters, choose the **OK** button; the surface will be created. Figure 11-49 shows the section and the guide curve selected for creating studio surface. Figure 11-50 shows the preview of the resulting studio surface.

Figure 11-51 shows a single section and two guide curves selected and Figure 11-52 shows the resulting studio surface.

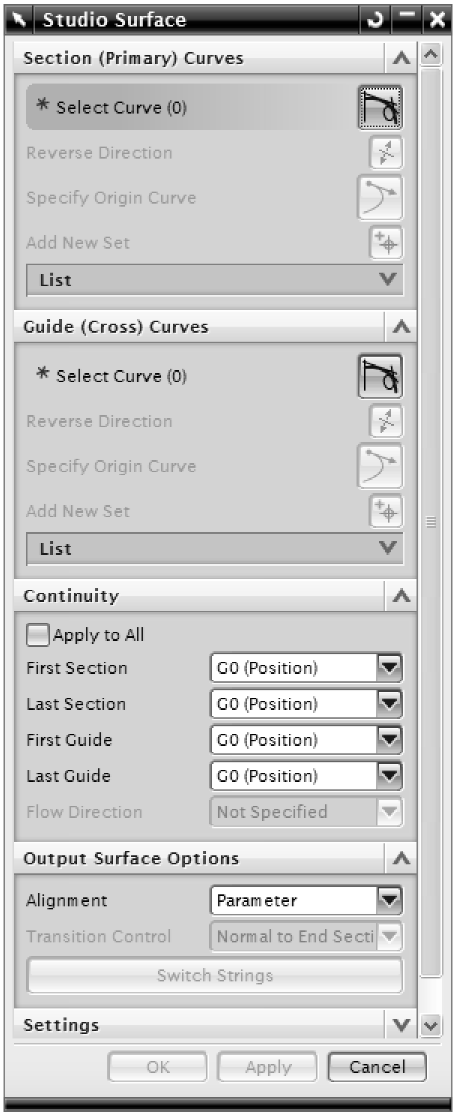


Figure 11-48 *The* ***Studio Surface*** *dialog box*

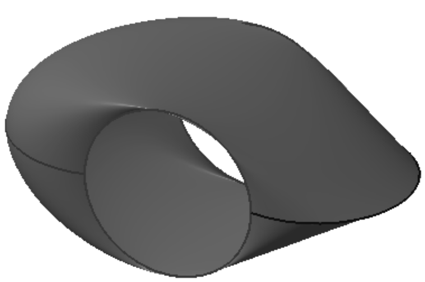


Figure 11-52 *The resulting studio surface*

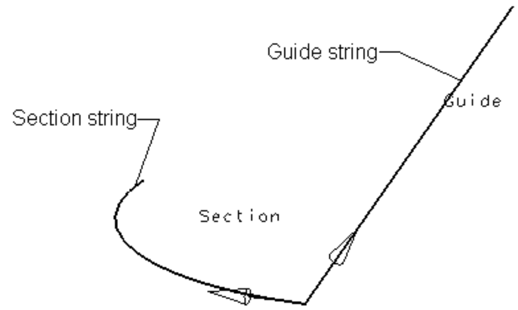


Figure 11-49 *The section curve and the guide curve selected for creating the studio surface*

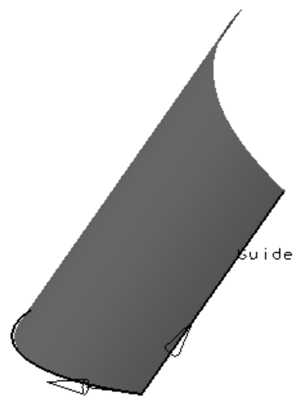


Figure 11-50 *The preview of the studio surface created using the* ***Studio Surface*** *tool*

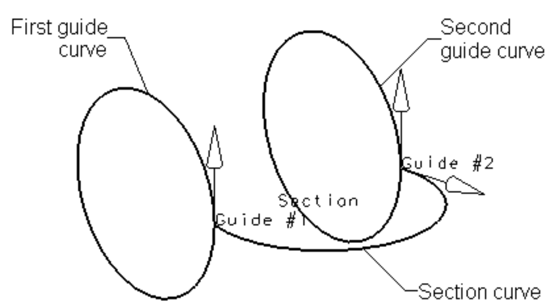


Figure 11-51 *The section curve and the guide curves selected for creating the studio surface*

Figure 11-53 shows the selected start and end section curves and Figure 11-54 shows the resulting studio surface.

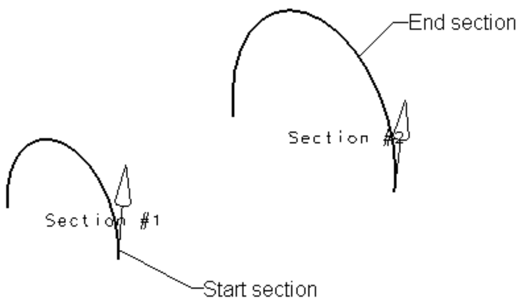


Figure 11-53 The section curves selected for creating a studio surface

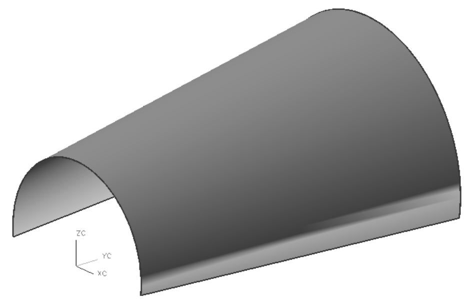


*Figure 11-54 The resulting studio surface created using the **Studio Surface** tool*

Figure 11-55 shows two section curves and two guide curves selected. Figure 11-56 shows the resulting studio surface.

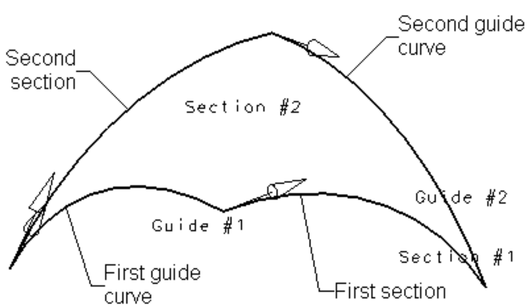


Figure 11-55 The section and guide curves selected for creating the studio surface

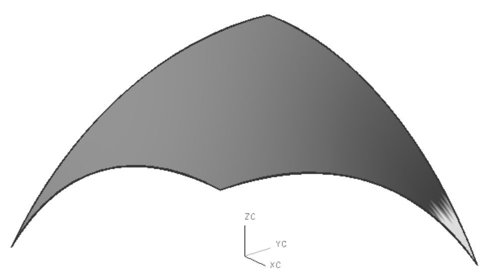


*Figure 11-56 The resulting studio surface created using the **Studio Surface** tool*

Figure 11-57 shows the selected section curves and guide curves. Figure 11-58 shows the resulting studio surface.

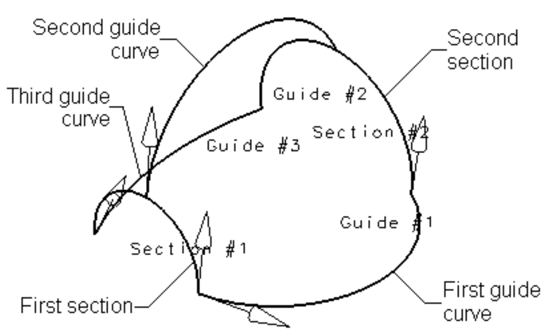


Figure 11-57 The section curves and the guide curves selected to create a studio surface

*Figure 11-58 The resulting surface created using the **Studio Surface** tool*

Creating a Surface Between Two Walls Using the Styled Blend Tool

Menu:	Insert > Detail Feature > Styled Blend
Toolbar:	Freeform Shape > Styled Blend

The **Styled Blend** tool is used to create a fillet surface between two intersecting walls. The tangent holding lines are created first at the intersection point of the surfaces with respect to the pipe radius specified. To create the fillet surface using this tool, choose the **Styled Blend** button from the **Freeform Shape** toolbar; the **Styled Blend** dialog box will be displayed, as shown in Figure 11-59, and you will be prompted to select the faces for wall 1.

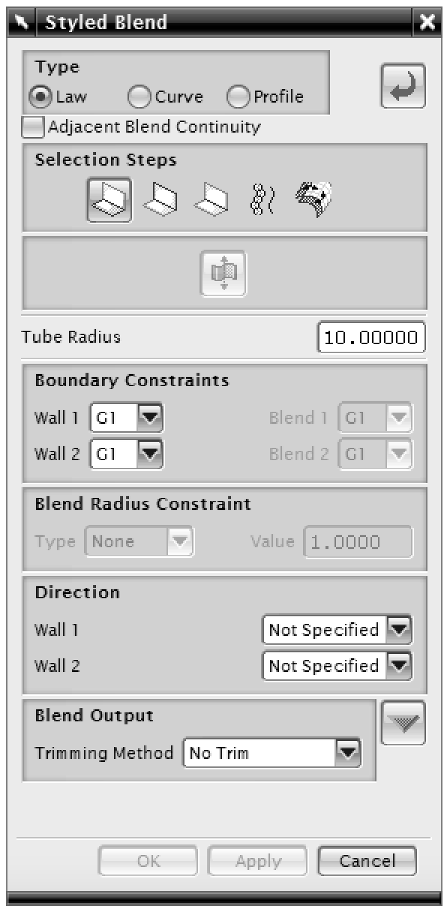


Figure 11-59 *The* ***Styled Blend*** *dialog box*

The option selected from the **Type** area decides the method of formation of the blend surface. If you select the **Law** radio button, the tangent holding lines will automatically be created with respect to the pipe radius specified for the fillet. If you select the **Curve** radio button, you will have to select the tangent holding curves for creating the fillet. If you select the **Profile** radio button, 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** radio button is selected from the **Type** area and the **Wall 1** button is selected from the **Selection Steps** area. Select the first wall and choose the **Wall 2** button from the **Selection Steps** area. While selecting both the walls, you need to ensure that the arrow is displayed from the walls, face inward where the surface is to be created. Select the second wall and choose the **Center curve** button. Select the curve that acts as the hinge for the fillet surface to be created. The center curve is selected to define the center point for the fillet surface to be created. The center curve selected should not be normal to the fillet surface to be created. After selecting the center curve, choose the **Spine** button. Select the spine that is parallel to the center curve selected. The spine curve is selected to define the shape of the fillet. After selecting the spine curve, choose the **Preview** button; the **Labels** check box along with the **Select Feature** and **Reverse Blend Direction** buttons will be available. If the **Labels** check box is selected, the entities that are selected for the blend creation such as wall 1, wall 2, center curve, and spine will be annotated and displayed in the graphics window. The **Select Feature** button is chosen to inherit the properties of an existing blend surface. The **Reverse Blend Direction** button is used to flip the direction of the blend surface created. Figure 11-60 shows the parameters selected for creating the styled blend surface using the **Law** option and Figure 11-61 shows the resulting surface created.

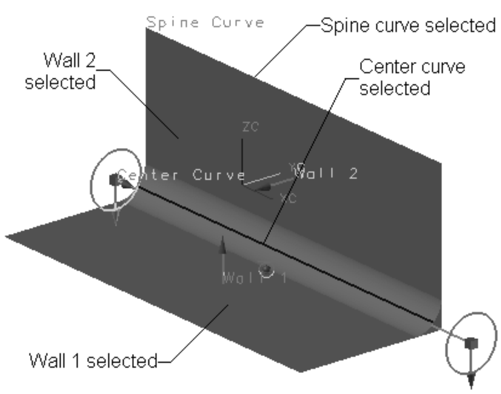


Figure 11-60 *The preview of the styled blend surface displayed for the selected parameters*

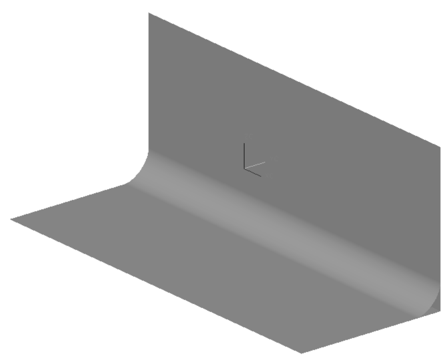


Figure 11-61 *The resulting styled blend surface created using the **Law** option*

Creating a Styled Blend Surface Using the Curve Option

As mentioned earlier, when you select the **Curve** radio button from the **Type** area to create the styled blend surface, you need to select the tangential holding lines. Select the **Curve** radio button from the **Type** area; you will be prompted to select the faces for wall 1. While selecting the walls, ensure that the arrow displayed from the walls after selection should face inward where the surface is to be created. Select the first wall and choose the **Wall 2** button from the **Selection Steps** area. Select the second wall and choose the **Curve Set 1** button. Select the first tangential curve from the first wall selected and choose the **Curve Set 2** button. Select the second tangential curve from the second wall selected. Choose the **Spine** button and select the spine curve; the spine curve will be selected to define the shape of the fillet. After selecting the spine curve, choose the **Face** button. Then, choose the **Preview** button; the **Labels** check box along with the **Select Feature** and **Reverse Blend Direction** buttons will be enabled. If the **Labels** check box is selected, the entities that are selected for the blend creation such as wall 1, wall 2, curve set 1, curve set 2, and spine will be annotated and displayed in the graphics window. Figure 11-62 shows the parameters selected for creating the styled blend surface using the **Curve** option and the resulting surface created.

Creating a Styled Blend Surface Using the Profile Option

As mentioned earlier, when you select the **Profile** option from the **Type** area for creating the **Styled Blend** surface, you need to select the curve or the sketch that is to be imprinted on the surfaces to create the tangential holding lines. Select the **Profile** radio button from the **Type** area; you will be prompted to select the faces for wall 1. While selecting both the walls, ensure that the arrow displayed on the walls after the selection should face inward where the surface is to be created. Select the first wall and choose the **Wall 2** button from the **Selection Steps** area. Select the second wall and choose the **Profile** button. Select the curve or the sketch to be imprinted and choose the **Center Curve** button. Select the center curve and choose the **Spine** button. Next, select the spine curve and choose the **Preview** button; the **Labels** check box along with the **Select Feature** and **Reverse Blend Direction** buttons will be enabled. Figure 11-63 shows the parameters selected for creating the styled blend surface using the **Profile** option and also the resulting surface.

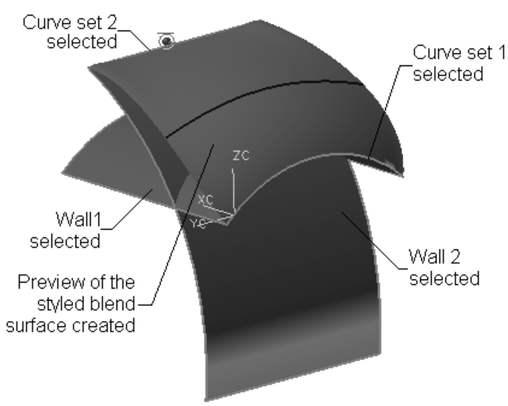


Figure 11-62 The preview of the styled blend surface displayed after selecting the parameters for the ***Curve*** *option*

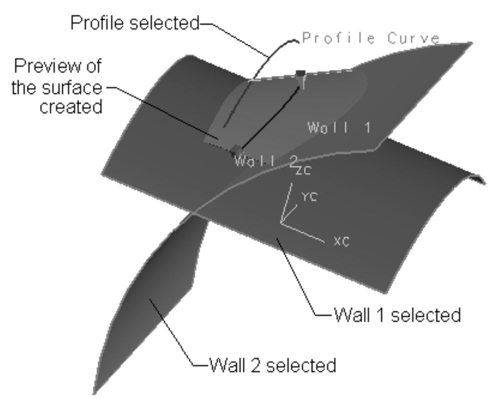


Figure 11-63 The preview of the styled blend surface displayed after selecting the parameters for the ***Profile*** *option*

Creating a Surface Using the Styled Sweep Tool

Menu:	Insert > Sweep > Styled Sweep
Toolbar:	Freeform Shape > Styled Sweep

The **Styled Sweep** tool is used to create surfaces by sweeping the cross-sections across the guide curves. The surface created using this tool can be modified dynamically by dragging the handles that are displayed along with the surface. To create the styled sweep surface, invoke the **Styled Sweep** tool from the **Freeform Shape** toolbar; the **Styled Sweep** dialog box will be displayed, as shown in Figure 11-64.

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

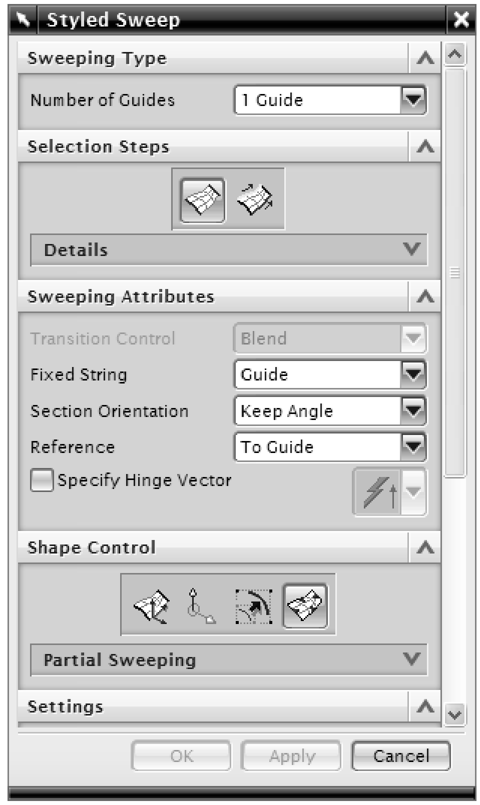


Figure 11-64 The ***Styled Sweep*** *dialog box*

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 and 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.

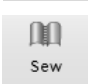
Select the required option from the **Number of Guides** drop-down list of the **Sweeping Type** rollout. Next, select the section and guide strings, using the **Section String** and **Guide String** buttons, respectively from the **Selection Steps** rollout. Note that you need to press the middle mouse button to add each section and guide string. After selecting the section and guide strings, the preview of the surface will be displayed along with the handles. You can modify the shape and size of the surface by dragging the handles. The options in the **Shape Control** rollout are used to display different types of handles for the surface created, which can be used to modify the surface dynamically.

Figure 11-65 shows the entities selected for creating the styled sweep surface using the **2 Guides** option and Figure 11-66 shows the resulting styled sweep surface.

Sewing Individual Surfaces into a Single Surface

Enhanced

Menu:	Insert > Combine Bodies > Sew
Toolbar:	Feature Operation > Sew *(Customize to add)*



The **Sew** tool from the **Feature Operation** toolbar is used to stitch individual surfaces into a single surface with a common edge. This tool allows you to sew two solid bodies together if they share one or more common faces. When the selected individual surface encloses a volume, a solid body is created. The sheet to which all the other individual sheets are to be stitched is known as the target sheet, and the individual

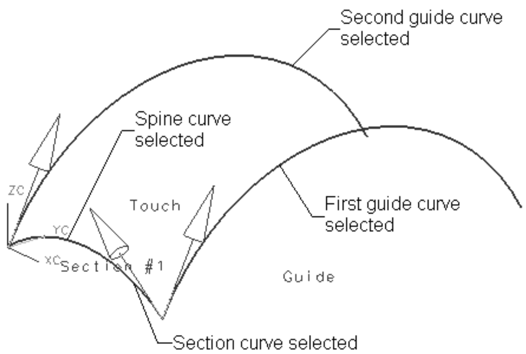


__Figure 11-65__ The entities selected for creating the styled sweep surface

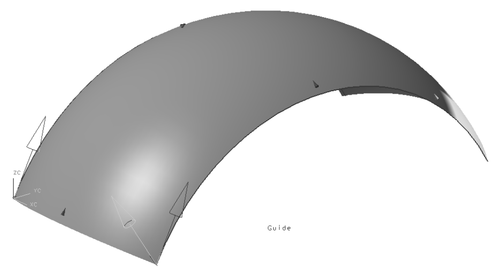


__Figure 11-66__ The resulting styled sweep surface

sheets that are to be stitched are known as the tool sheets. You cannot stitch the tool sheets that intersect a target sheet and extend from it.

To stitch the individual surfaces to a single surface, choose the **Sew** button from the **Feature Operation** toolbar; the **Sew** dialog box will be displayed, as shown in Figure 11-67. By default, the **Sheet** option will be selected from the drop-down list in the **Type** rollout and you will be prompted to select the target sheet body. Select the target sheet body; the border of the target sheet will be displayed in cyan color and you will be prompted to select the tool sheet bodies to sew. The border of the selected tool sheets should lie within the cyan 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.

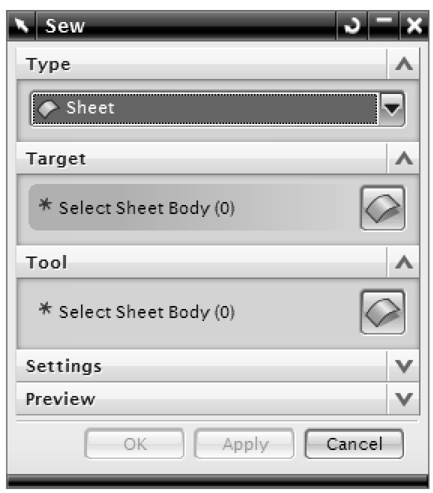
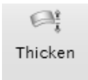


__Figure 11-67__ The __Sew__ dialog box

Similarly, to combine the solid bodies, select the **Solid** option from the drop-down list in the **Type** rollout. You can also enter the sew tolerance in the **Tolerance** edit box of the **Settings** rollout. If the selected objects are an instance of an array and the **Sew All Instances** check box is selected, all the instances of the array will be stitched together. Note that this check box will be enabled only after selecting the **Solid** option from the drop-down list in the **Type** rollout.

Adding Thickness to a Surface

Menu:	Insert > Offset/Scale > Thicken
Toolbar:	Feature > Thicken *(Customize to add)*



This tool is used to add thickness to the sheet. Once you add thickness to the sheet, it is converted into a solid. To add thickness to the sheet, invoke the **Thicken** tool from the **Feature** toolbar; the **Thicken** dialog box will be displayed, as shown in Figure 11-68 and you will be prompted to select faces to thicken. Select the sheet bodies to which the material is to be added.

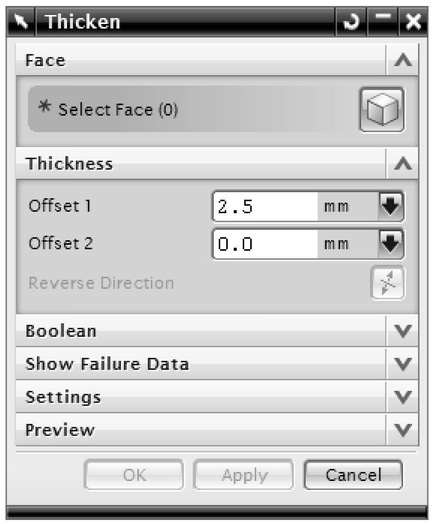


***Figure 11-68** The **Thicken** dialog box*

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

The **Boolean** drop-down list in the **Boolean** rollout provides the options for performing 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 11-69 shows the sheet to add thickness and Figure 11-70 shows the resulting solid body.

***Figure 11-69** The sheet selected for adding the material*

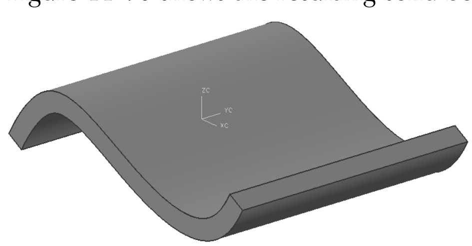


***Figure 11-70** The resulting solid body after adding thickness to the sheet*

TUTORIALS

Tutorial 1

In this tutorial, you will create the surface model shown in Figure 11-71. The dimensions and orthographic views are shown in Figure 11-72. After creating the surface, save it in the name given below:

\NX 5\c11\c11tut1.prt **(Expected time: 30 min)**

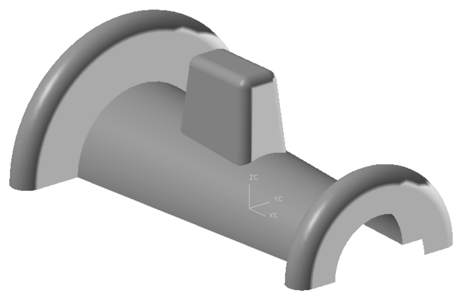


Figure 11-71 *The isometric view of the surface model*

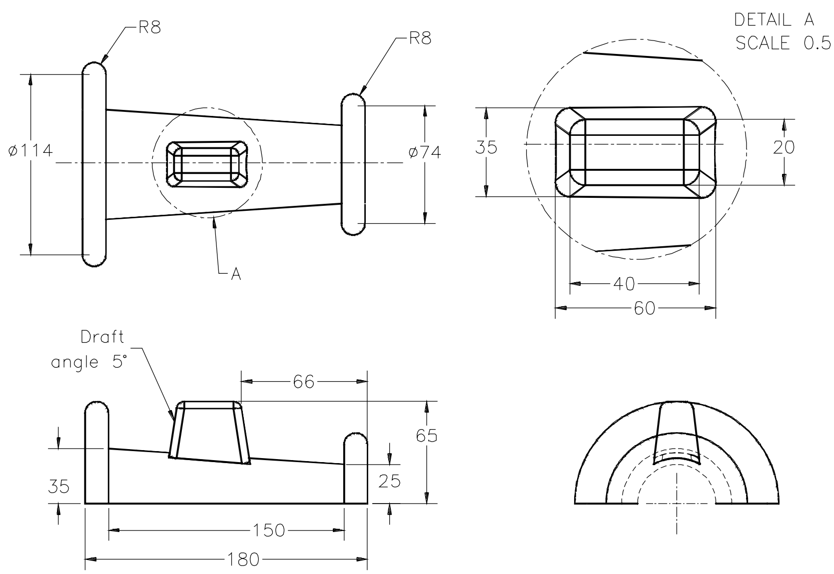


Figure 11-72 *The dimensions and orthographic views of the surface model*

The following steps are required to complete this tutorial:

a. Start a new file and then set the sheet environment.
b. Create the sketch for the base surface and then revolve it.
c. Create the second feature, which is an extruded surface.
d. Trim the base surface.

e. Trim the extended part of the third feature.
f. Create the bounded plane surface.
g. Stitch the bounded plane and extruded surfaces with the revolved surface.
h. Fillet the stitched surface.
i. Save and close the file.

Starting a New File and Setting the Sheet Environment

1. Start a new file using the Model template and save the file in the *C:\NX 5\c11 folder* with the name *c11tut1.prt*. Create three fixed datum planes.

2. Choose **Preferences > Modeling** from the menu bar; the **Modeling Preferences** dialog box is displayed.

3. Select the **Sheet** radio button from the **Body Type** area 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 11-73.

2. Revolve the sketch through an angle of 180-degrees. The resulting revolved base feature created using the **Sheet** option is shown in Figure 11-74.

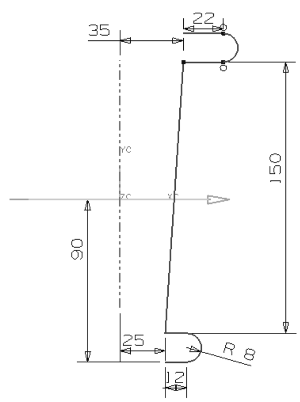


Figure 11-73 *The sketch created for the base feature*

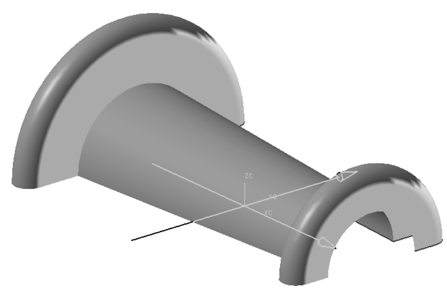


Figure 11-74 *The base feature created by revolving the sketch*

Creating the Second Feature by Extruding the Sketch

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

2. Create the sketch for the second feature by selecting the offset plane as the sketching plane, as shown in Figure 11-75.

3. Extrude the sketch through a distance of 50 in the downward direction and the draft angle of -5. The resulting extruded surface model is shown in Figure 11-76.

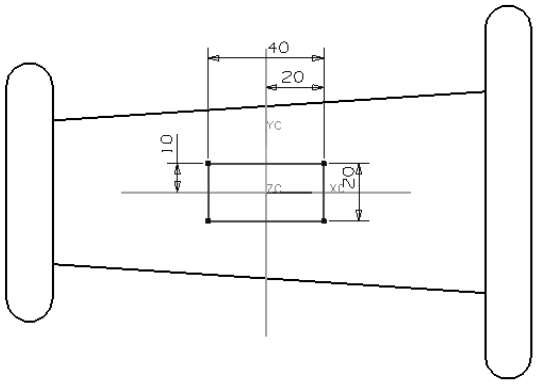


Figure 11-75 *The sketch drawn for creating the extruded feature*

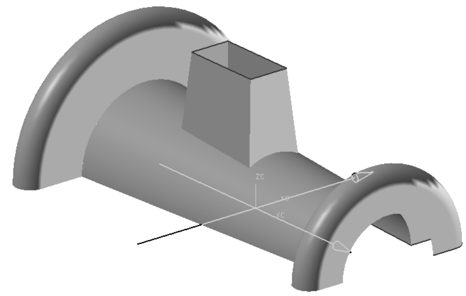


Figure 11-76 *The resulting extruded surface*

Trimming the Base Surface with Respect to the Second Surface

Next, you need to trim the base surface with respect to the second feature. To trim the base surface, follow the steps given below:

1. Choose the **Trimmed Sheet** button from the **Surface** toolbar; the **Trimmed Sheet** dialog box is displayed and you are prompted to select the target sheet body.

2. Select the sheet to be trimmed, as shown in Figure 11-77. Make sure that you select the sheet using the selection points, shown in the figure. After selecting the target sheet body, press the middle mouse button; you are prompted to select the trimming objects.

3. Select the trimming surfaces, as shown in Figure 11-77. Make sure that you select the surfaces using the points shown in the figure.

4. Select the **Discard** radio button from the **Region** rollout and then choose the **OK** button. The resulting surface model after trimming it, is shown in Figure 11-78.

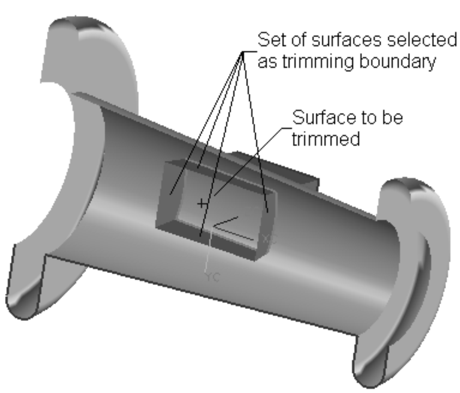


Figure 11-77 *The surface to be trimmed and the trimming surfaces selected from the model*

Figure 11-78 *The surface model after trimming the base surface with respect to the extruded surface*

Trimming the Second Surface Created with the Base Surface

After trimming the base surface, you need to trim the unwanted portions of the second feature. To trim the extended portion, follow the procedure that is discussed next.

1. Choose the **Trimmed Sheet** button from the **Surface** toolbar; the **Trimmed Sheet** dialog box is displayed and you are prompted to select a target sheet body.

2. Select the sheet to be trimmed, as shown in Figure 11-79; you are prompted to select the trimming objects.

3. Select the trimming surface, as shown in Figure 11-79. Select the **Discard** 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 11-80.

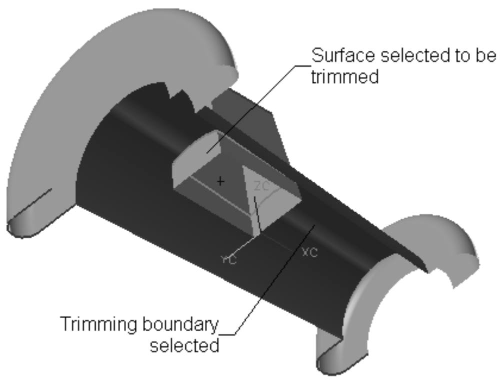


Figure 11-79 *The surface to be trimmed and the trimming surface selected from the model*

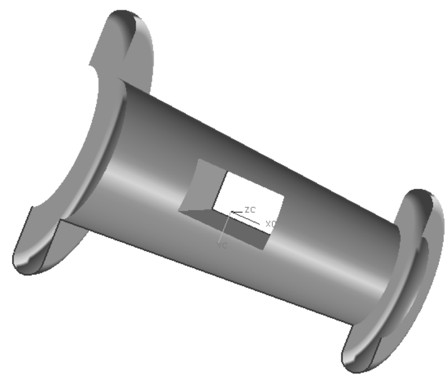
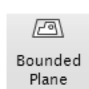


Figure 11-80 *The resulting surface after trimming the extended portion of the extruded surface*

Creating the Bounded Plane Surface

Next, you need to create the bounded surface to close the top face of the second feature. For creating the bounded plane surface, follow the procedure that is discussed next.

1. Choose the **Bounded Plane** button from the **Feature** toolbar; the **Bounded Plane** dialog box is displayed and you are prompted to select bounding string.



2. Select the sketch that is used to create the second surface and choose the **OK** button. The resulting bounded plane surface is shown in Figure 11-81.

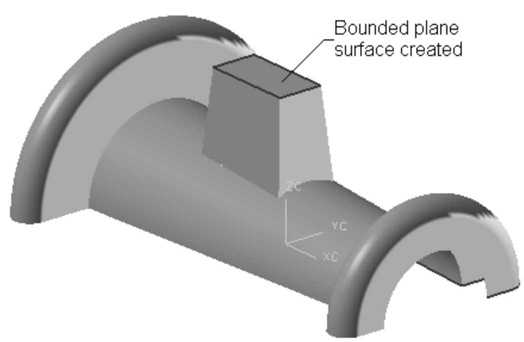
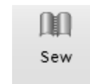


Figure 11-81 *The resulting bounded plane surface*

Stitching the Bounded Plane Surface and Extruded Surface with the Revolved Surface

After creating all surfaces, you need to stitch them together. This is done using the **Sew** tool.



1. Choose the **Sew** button from the **Feature Operation** toolbar; the **Sew** dialog box is displayed and you are prompted to select a target sheet to sew.

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

3. Select the bounded plane and the extruded surface.

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

Note
After stitching the individual surfaces together into a single surface, you can hide the sketch created for revolving the base feature, the sketch created for extruding the second feature, the unwanted datum planes, and the datum axis. 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 the Edges Using the Edge Blend Tool

Next, you need to fillet the edges.

1. Choose the **Edge Blend** button from the **Feature Operation** toolbar; 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, as shown in Figure 11-82. Enter the fillet radius value as **5** in the **Radius 1** edit box.

3. Choose the **OK** button from the **Edge Blend** dialog box. The completed surface model, after adding the fillets to the edges, is shown in Figure 11-83.

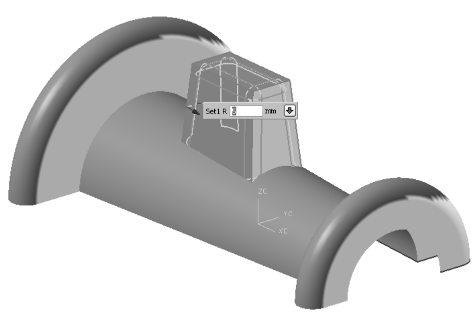


Figure 11-82 *The preview of the fillets displayed after selecting the edges*

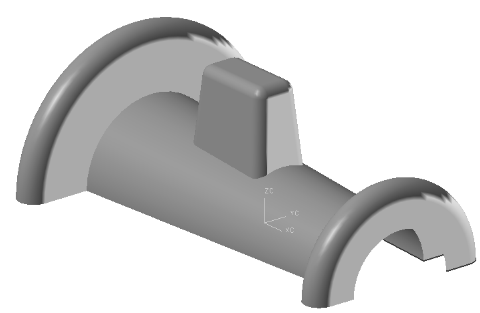


Figure 11-83 *The completed surface model after adding the fillets to the edges*

Saving and Closing the File

1. Choose **File > Close > Save and Close** from the menu bar to save and close the file.

Tutorial 2

In this tutorial, you will create the surface model shown in Figure 11-84. The drawing views and the dimensions of the surface model are shown in Figure 11-85. After creating the model, save it with the name *\NX 5\c11\c11tut2.prt.* **(Expected time: 45 min)**

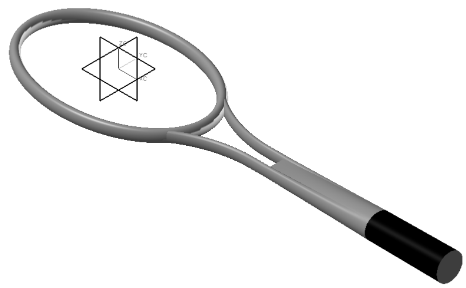


Figure 11-84 *The isometric view of the surface model*

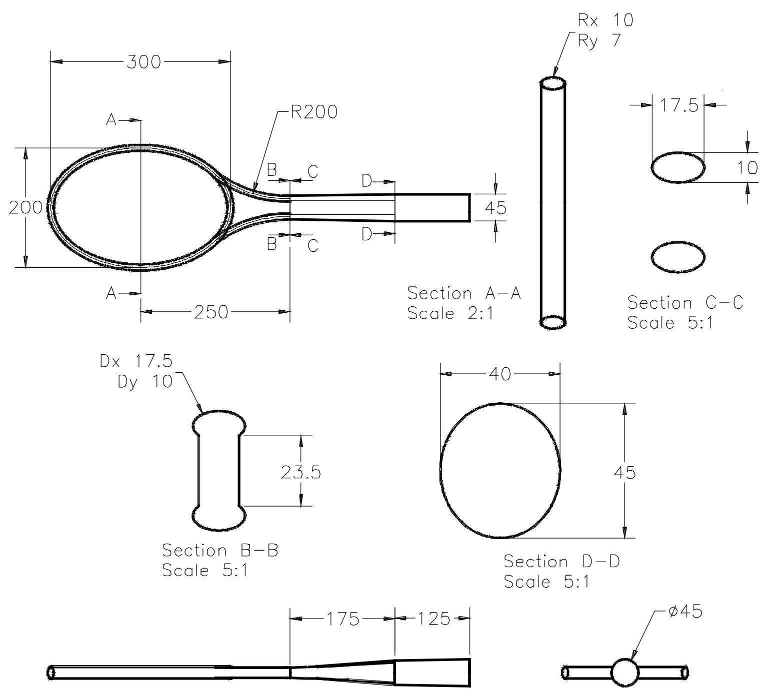


Figure 11-85 *The drawing views and dimensions of the surface model*

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.
c. Create the sweep surface as the second feature and mirror the surface.
d Create the studio surface as the third feature.

e. Create the ruled surface as the fourth feature.
f. Create the bounded plane surface as the fifth feature.
g. Create another bounded plane surface as the sixth feature.
h. Save and close the file.

Starting a New File and Setting the Sheet Environment

1. Start a new file using the Model template and save the file in the *C:\NX 5\c11 folder* with the name *c11tut2.prt*. Create three fixed datum planes.

2. Choose **Preferences > Modeling** from the menu bar; the **Modeling Preferences** dialog box is displayed.

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

4. Create three fixed datum planes and turn on the display of WCS.

Creating a Styled Sweep Surface as the Base Feature

As mentioned earlier, the styled sweep surface will be the base feature. To create the base feature, follow the procedure that is discussed next.

1. Draw an ellipse on the XC-YC plane as per the dimensions shown in Figure 11-86 and exit the **Sketcher** environment.

2. Draw another ellipse on the XC-ZC plane as per the dimensions shown in Figure 11-87 and exit the **Sketcher** environment.

3. Choose the **Styled Sweep** button from the **Freeform Shape** toolbar; the **Styled Sweep** dialog box is displayed and you are prompted to select the section string.

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

5. Choose the **Guide String** button and select the guide curve drawn on the XC-YC plane. Press the middle mouse button; the preview of the styled sweep surface will be displayed.

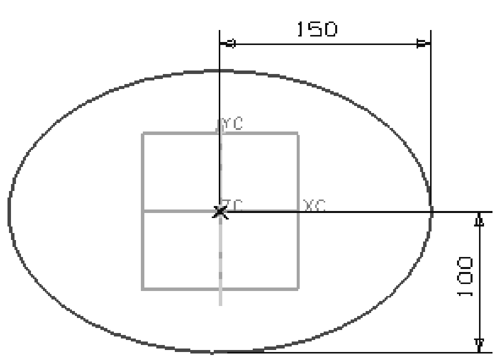


Figure 11-86 The sketch for the guide string

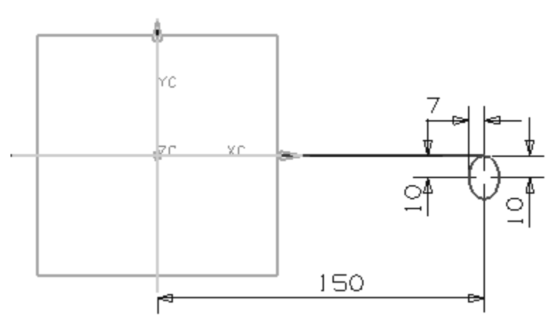


Figure 11-87 The sketch for the section string

6. Choose the **OK** button to create the styled sweep surface. The resulting styled sweep surface is shown in Figure 11-88.

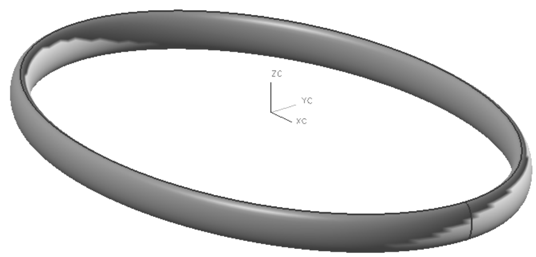


Figure 11-88 The resulting base feature of the surface model

Creating the Sweep Surface as the Second Feature

The second feature is the sweep surface. Before creating the sketch for the second feature, you need to create a datum plane parallel to the XC-YC plane and at a distance of -10.

1. Create a datum plane at an offset of -10 from the XC-YC plane.

2. Select the newly created plane as the sketching plane and draw the guide curve, as shown in Figure 11-89.

3. Create a new datum plane perpendicular to the guide curve by entering the arclength value as 0.

4. Select the newly created plane and draw the ellipse (section curve), as shown in Figure 11-90. Exit the **Sketcher** environment.

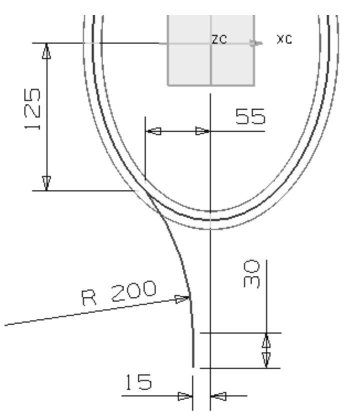


Figure 11-89 The dimensions for the guide curve of the sweep surface

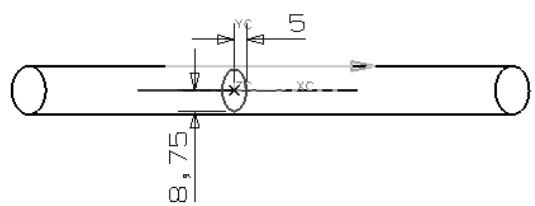


Figure 11-90 The section curve drawn for creating the sweep surface

5. Choose the **Sweep along Guide** button from the **Feature** toolbar; the **Sweep along Guide** dialog box is displayed and you are prompted to select the section string.

6. Select the section curve and choose the **OK** button; you are prompted to select the guide curve.

7. Select the guide curve and choose the **OK** button; the **Sweep along Guide** dialog box is displayed.

8. Choose the **OK** button twice; the resulting sweep surface will be created, as shown in Figure 11-91.

Mirroring the Sweep Surface

1. Mirror the last surface using the XC-ZC plane as the mirror plane. The resulting surface model, after mirroring the sweep surface, is shown in Figure 11-92.

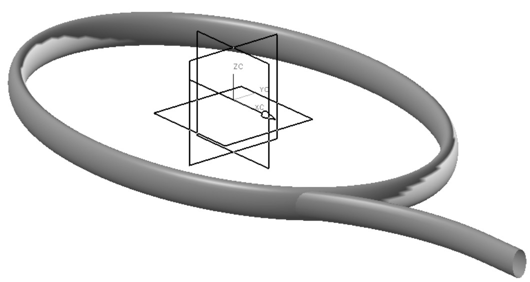


Figure 11-91 The sweep surface created

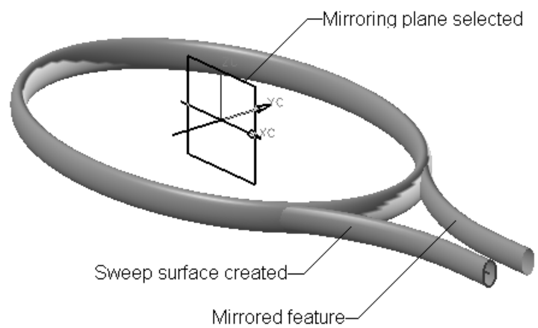


Figure 11-92 The resulting surface model after mirroring the sweep feature

Creating the Studio Surface

The third feature is the surface that is created using the **Studio Surface** tool.

1. Invoke the **Sketcher** environment using the last datum plane created to draw the section of the sweep surface.

2. Choose **Insert > Project Curve** from the menu bar; the **Project Curve** dialog box is displayed and you are prompted to select geometry to project.

3. Select edges and create the profile, as shown in Figure 11-93. Exit the **Sketcher** environment.

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

5. Select the datum plane created as the sketching plane and create the second primary string, as shown in Figure 11-94. The dimensions can be obtained from Figure 11-85.

6. Select the offset datum plane and create the guide strings, as shown in Figure 11-94.

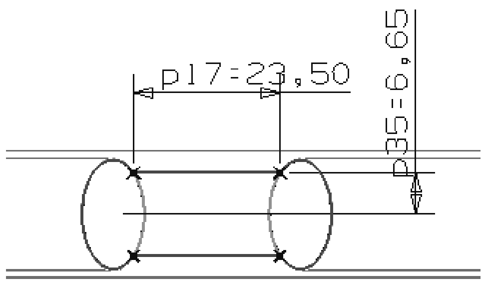


Figure 11-93 *The first section curve drawn to create the through curve mesh surface*

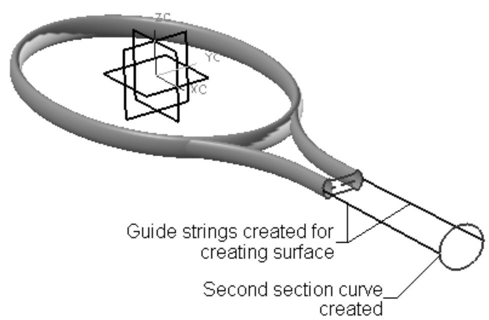


Figure 11-94 *The second section curve and the guide strings created to create the surface*

7. Choose **Insert > Mesh Surface > Studio Surface** from the menu bar; the **Studio Surface** dialog box is displayed and you are prompted to select the section.

8. Select the first section string, as shown in Figure 11-95. Press the middle mouse button; you are prompted to select the section again.

9. Select all elements from the second section string. Make sure that the arrows in both the section strings point in the same direction, as shown in Figure 11-95. Choose the **Guide (Cross) Curves** button; you are prompted to select the guide string.

10. Select the first guide curve, as shown in Figure 11-95, and press the middle mouse button; you are prompted to select the guide string again.

11. Select the second guide curve, as shown in Figure 11-95. Choose the **Apply** button and then the **Cancel** button to create the surface.

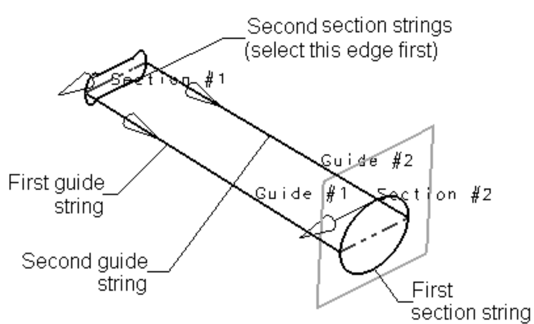


Figure 11-95 *The entities to be selected to create the studio surface*

Mirroring the Studio Surface

1. Mirror the studio surface using the XC-YC plane as the mirror plane. The resulting surface model is shown in Figure 11-96.

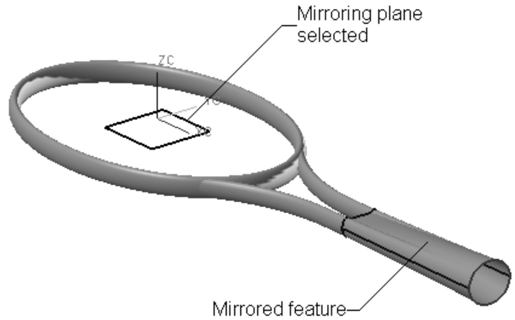


Figure 11-96 *The resulting surface model after mirroring the surface*

Creating the Ruled Surface

Next, you need to create the ruled surface.

1. Create a plane parallel to the YC-ZC plane at a distance of 575.
2. Select the created plane as the sketching plane and draw the sketch, as shown in Figure 11-97.
3. Exit the **Sketcher** environment and choose the **Ruled** button from the **Surface** toolbar; the **Ruled Surface** dialog box is displayed and you are prompted to select the first section string.

4. Select the first section string, as shown in Figure 11-98. Next, press the middle mouse button; you are prompted to select the second section string.
5. Select the second section string, which is the edge of the surface created earlier, as shown in Figure 11-98. Note that the arrows should point in the same direction. Choose the **OK** button; the ruled surface is created.
6. Change the color of the ruled surface to black.

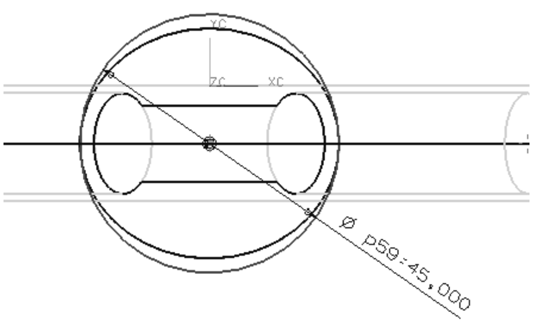


Figure 11-97 *The first section curve drawn to create the ruled surface*

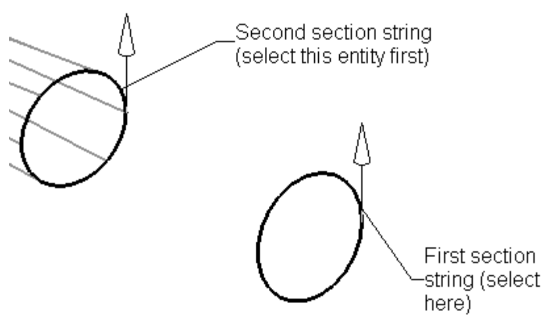
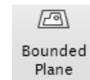


Figure 11-98 *The resulting ruled surface created after selecting both cross-sections*

Creating the Bounded Plane Surface



1. Choose the **Bounded Plane** button from the **Feature** toolbar; the **Bounded Plane** dialog box is displayed and you are prompted to select the bounding string.

2. Select the bounding string for creating the bounded plane surface, as shown in Figure 11-99. The resulting bounded plane surface is displayed, as shown in Figure 11-99.

3. Again, select the sketch shown in Figure 11-100 to create the bounding surface. The resulting bounded plane surface is displayed, as shown in Figure 11-100.

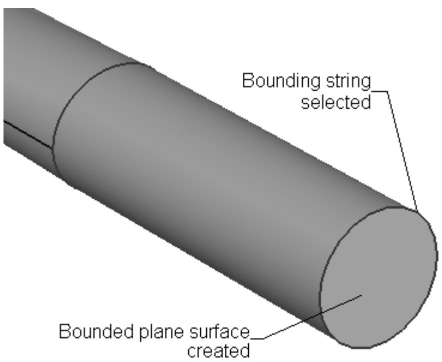


Figure 11-99 *The bounding string selected and the resulting bounded plane surface created*

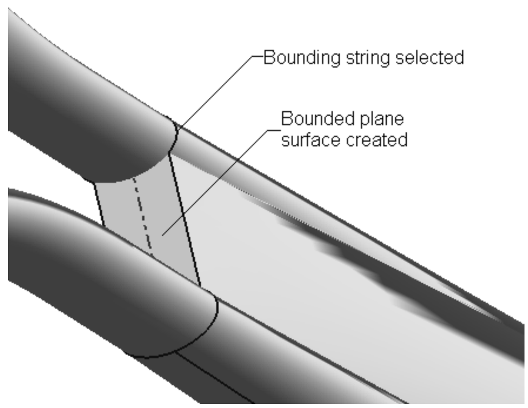


Figure 11-100 *The bounding string selected and the resulting bounded plane surface created*

Saving and Closing the File

1. Choose **File > Close > Save and Close** from the menu bar to save and close the file.

Self-Evaluation Test

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

1. The surfaces are termed as sheets in NX. (T/F)

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

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

4. The default tolerance value for the creation of the sheet is 0.0254. (T/F)

5. The maximum number of sections that can be used to create a sheet using the **Ruled** tool from the **Surface** toolbar is ___________.

6. The ____________ tool is used to create a sheet from n number of guide curves and n number of section curves.

7. The ____________ tool is used to stitch the individual surfaces into a single surface.

8. The ____________ tool is used to trim and extend a surface.

9. The ____________ tool is used to create a planar surface.

10. The ____________ tool is used to create a surface offset.

Review Questions

Answer the following questions:

1. How many tools are available to create a studio surface?

   (a) Three (b) Four
   (c) Five (d) None of the above

2. Which tool is used to create the flange surface?

   (a) **Silhouette Flange** (b) **N-Sided Surface**
   (b) **Law Extension** (d) None of the above

3. Which tool is used to create a surface by using multiple sections only?

   (a) **Through Curve Mesh** (b) **Ruled**
   (c) **Studio Surface** (d) None of the above

4. Before adding a fillet at the intersection of two surfaces, the surface has to be

   (a) Stitched using the **Sew** tool (b) Merged
   (c) Trimmed (d) None of the above

5. The **Bounded Plane** tool is used to close the ends of the closed surface. (T/F)

6. You can select an open sketch to create the bounded plane surface.

7. The surface models do not have mass properties. (T/F)

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

9. Once you have added thickness to the sheet, it is converted into a solid. (T/F)

10. 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 11-101. The drawing views and the dimensions of the surface model are shown in same figure. Save the model with the name given below:

\NX 5\c11\c11exr1.prt **(Expected time: 30 min)**

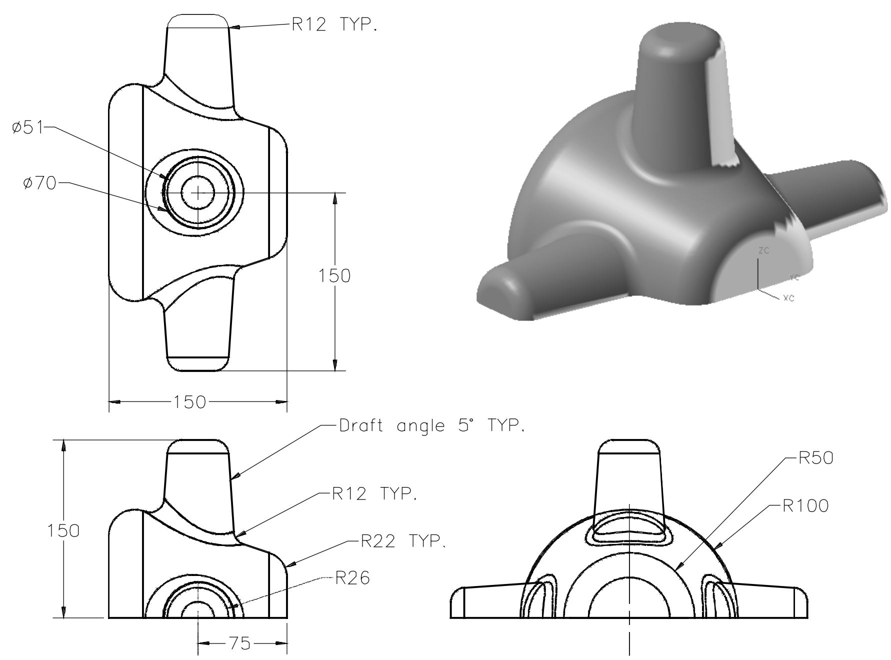


Figure 11-101 *Drawing views and dimensions of the surface model for Exercise 1*

Exercise 2

Create the surface model shown in Figure 11-102. The drawing views and the dimensions of the surface model are shown in Figure 11-103. Save the model with the name given below:

\NX 5\c11\c11exr2.prt **(Expected time: 30 min)**

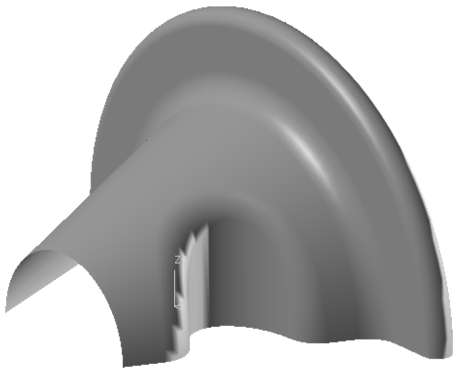


Figure 11-102 The isometric view of the surface model for Exercise 2

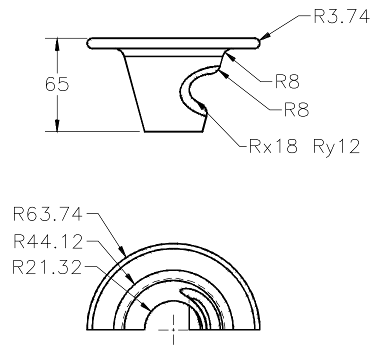


Figure 11-103 The dimensions and drawing views of the surface model

Answers to Self-Evaluation Test

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