



# Chapter 12

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## 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 through curves and through mesh curves.
- 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 a zero thickness. They are used extensively for modeling complex features. The model or 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. Depending on 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 using the surface modeling technique. The tools that are used to create the solid models can also be used to create the 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.

## 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 12-1. 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 the models created henceforth in the **Modeling** application will be sheet models.

## Creating an Extruded Surface

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

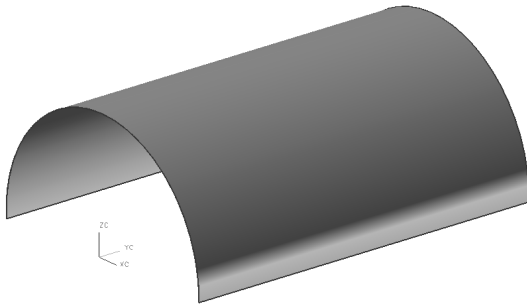


As mentioned earlier, there is no separate tool for creating the surface extrude. After invoking the **Sheet Modeling** environment, you can use the **Extrude** tool to create the extruded sheets. The sketch drawn for creating the extruded surface may be an open or a closed entity. After creating the sketch, choose the **Extrude** button; you will be prompted to select the section geometry to extrude and the **Extrude** dialog box will be displayed. Select the sketch and enter the extrusion value in the **End** edit box. Next, choose the **More Options** button from the **Extrude** dialog box; the **Extrude** dialog box will expand and show more options. Select the **Sheet** radio button and choose the **OK**

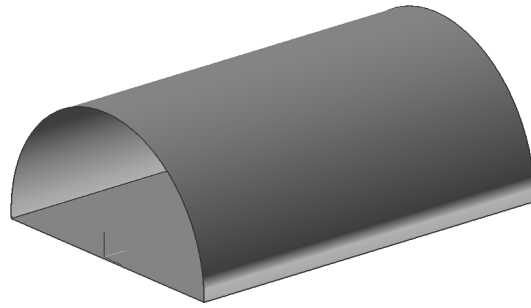


**Figure 12-1** The **Modeling Preferences** dialog box

button; the sheet will be created. The options in the **Extrude** dialog box are the same as discussed in Chapter 3. The surface extrude operations performed on open and closed sketches are displayed in Figures 12-2 and 12-3. In case of sheet bodies, you cannot use the **Until Next** and **Until Selected** options from the **End** drop-down list of the **Extrude** dialog box.



**Figure 12-2** Surface extrude created on an open sketch



**Figure 12-3** Surface extrude created on a closed sketch



#### Note

You can use only the **Create** 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:** Form Feature > Revolve



The **Revolve** tool is used to create the revolved surface. Choose the **Revolve** button from the **Form Feature** toolbar; you will be prompted to select a section string for the revolved body. Also the **Revolve** dialog box will be displayed. Select the sketch and choose the **Inferred Vector** button. Specify the axis of revolution using the same. Specify the start and end angles in the **Start Angle** and the **End Angle** edit boxes, respectively. Next, choose the **More Options** button from the **Revolve** dialog box; the **Revolve** dialog box will expand and show more options. Select the **Sheet** radio button and choose the **OK** button; the revolved sheet will be created. The revolved surface model created using an open sketch and a closed sketch is shown in Figures 12-4 and 12-5, respectively.



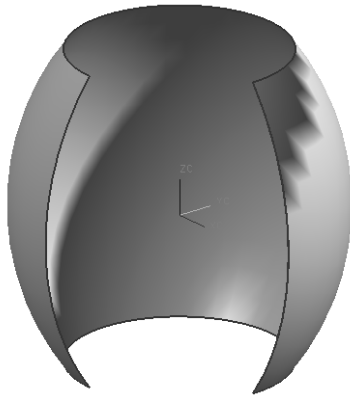
## Creating a Ruled Surface

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

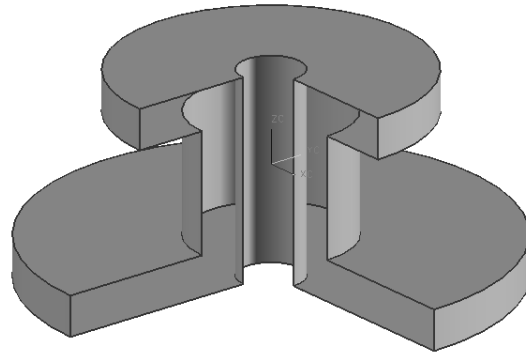


The **Ruled** tool is used to create the ruled surfaces. These surfaces are always created between two similar or dissimilar cross-sections created on different parallel planes. The sketches for this feature may be open or closed. Initially, isoparametric lines are formed to create patches, which are then converted to surfaces. The options for creating isoparametric curves are discussed later in this chapter. For creating the ruled



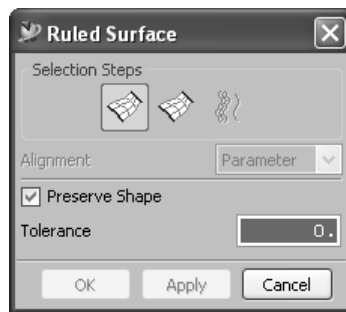


**Figure 12-4** Revolved surface created using an open sketch



**Figure 12-5** Revolved surface created using a closed sketch

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 12-6. 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 cross-section indicating the direction of the surface formation. Next, choose the **Section String 2** button; you will be prompted to select section string 2. Select the second cross-section; an arrow will be displayed from the second cross-section also. The arrows on the first and second cross-sections should point in the same direction.

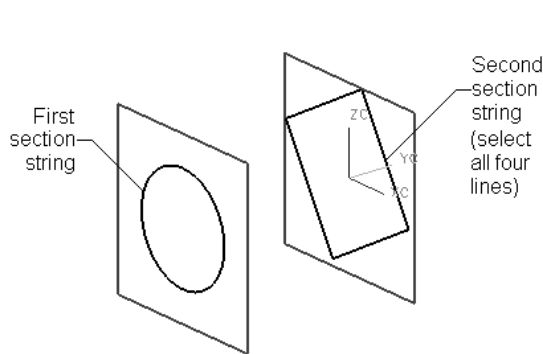


**Figure 12-6** The **Ruled Surface** dialog box

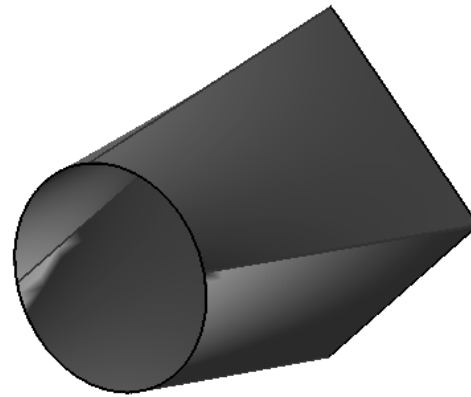
In the **Alignment** drop-down list, you have different methods of distributing the points for creating the isoparametric lines that form the patches. If you select the **Parameter** option, the points will be distributed such that the isoparametric lines are formed at equal parameter intervals. If you select the **Arc length** option, after option the entire curve will be divided into equal segments with respect to the 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, 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 12-7, choose the **OK** button. The resulting ruled surface created is shown in Figure 12-8.



**Figure 12-7** The section strings selected for creating the ruled surface



**Figure 12-8** 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 Surfaces Using the Through Curves Tool

**Menu:** Insert > Mesh Surface > Through Curves  
**Toolbar:** Surface > Through Curves

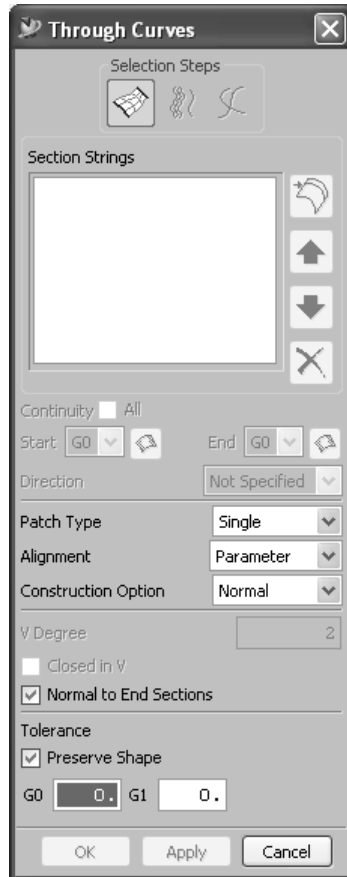


You can create surfaces with the multiple section strings using the **Through Curves** tool. This method of creating the surface allows you to select any number of section strings. For creating surfaces using the **Through Curves** tool, choose the **Through Curves** button from the **Surface** toolbar; the **Through Curves** dialog box will be displayed, as shown in Figure 12-9, and you will be prompted to select the section string. Select the section string and press the middle mouse button; you will again be prompted to select the section string. Likewise, you can select any number of section strings. After selecting the section strings, make sure the arrow point in the same direction.

### Patch Type Drop-down List

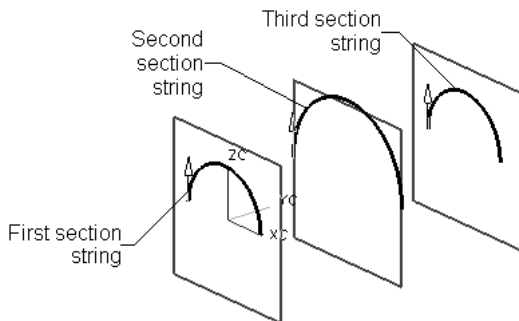
In the **Patch Type** drop-down list, you have three options: **Single**, **Multiple**, and **Match String**. If you select the **Single** option, the surface will be created with a single patch. If you select the **Multiple** option, the surface will be created with multiple patches. The number of patches formed depends on the **Alignment** option selected from the **Alignment** drop-down list. The **Closed in V** check box and the **V 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 **V Degree** edit



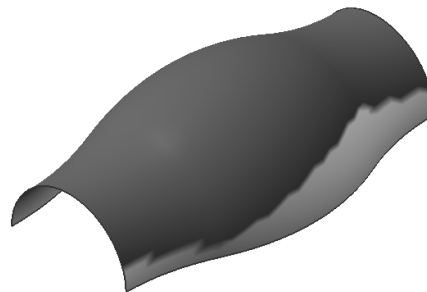


*Figure 12-9 The **Through Curves** dialog box*

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 12-10 shows the section strings selected for creating the surfaces through curves and Figure 12-11 shows the resulting surface.



*Figure 12-10 Section strings selected for creating a **Through Curves** surface*



*Figure 12-11 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 defining the section strings and the guide strings using the **Through Curve Mesh** tool. You can have any number of section strings and guide strings.

If you want to select multiple guide strings, they should be tangentially related to each other. For creating 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 12-12, and you will be prompted to select the primary strings. You need a collection of control strings such as the primary strings and cross strings. Here, the guide strings are called the cross strings. After selecting the first primary string, press middle mouse button to proceed to the next step. Likewise, you can select any number of primary strings. Next, choose the **Cross String** button; you are prompted to select the cross strings. Select the first cross string and press the middle mouse button to proceed to the next step. Likewise, you can select any number of cross strings. Next, choose the **Spine String** button; you will be prompted to select the spine. The selection of the spine curve is optional. The selected spine curve must be normal to all the primary strings. If you want to skip this step, do not choose this button.

### Emphasis Drop-down List

The options in the **Emphasis** drop-down list allow you to define the set of curves that effect the shape of the surface to be created. If the primary strings and cross strings have an equal effect, select the **Both** option from the **Emphasis** drop-down list. The **Intersection Tolerance** edit box defines for the minimum distance available between the two strings. If you select the **Normal** option from the **Construction Options** drop-down list, 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 the curves into temporary curves. The options in the **Rebuild** area will only be enabled by selecting the **Normal** option from the **Construction Options** drop-down list. You can use the options in the **Rebuild** area to join the surface smoothly with the surrounding surfaces. You can rebuild the surface by choosing the **Manual** button and entering the value in the **Degree** spinner. If you choose the **Automatic** button, the **Max Degree** and **Max Segments** spinners will be enabled. You can set

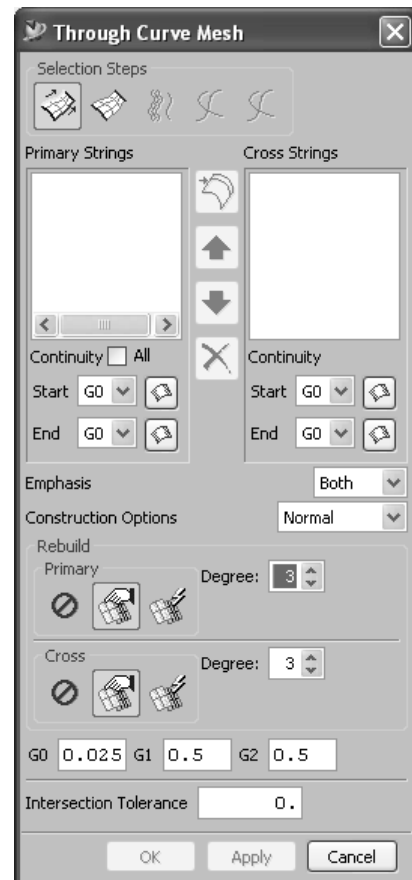
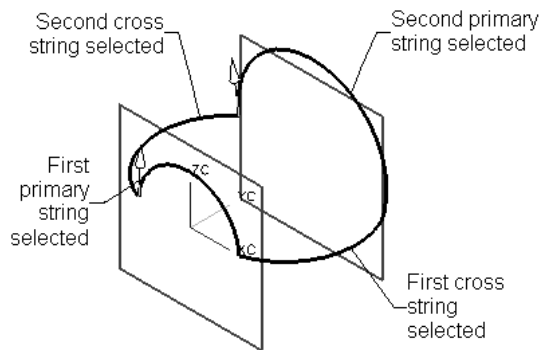
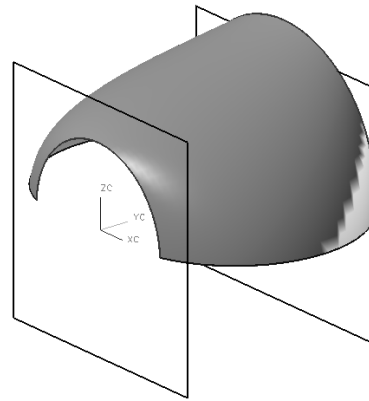


Figure 12-12 The Through Curve Mesh dialog box

the values in the **Max Degree** and **Max Segments** spinners to rebuild the surface automatically. Figure 12-13 shows the control strings selected for creating the through curve mesh surface and Figure 12-14 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 relative tolerance value can be entered in the **G2** edit box.



**Figure 12-13** The control strings selected for creating the through curve mesh surface



**Figure 12-14** The resulting surface

## Creating a Surface by 4 Points

**Menu:** Insert > Surface > Surface by 4 Points  
**Toolbar:** Free Form Shape > Four Point Surface



The **Surface by 4 Points** tool will be used to create a planar (2D) or nonplanar (3D) surface. For creating the surface by using this method, choose the **Four Point Surface** button from the **Free Form 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 corner. Similarly, you will be prompted to specify the other three corners. Specify the other three corners and choose the **OK** button; the surface will be created. At the time of specifying the point for the corners, 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 12-15 shows the imaginary layout for the surface to be formed after specifying the corner points. Figure 12-16 shows the resulting planar surface formed by enclosing the corner points specified.

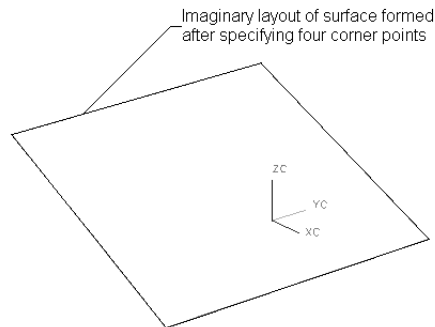
## Creating a Swoop Surface

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

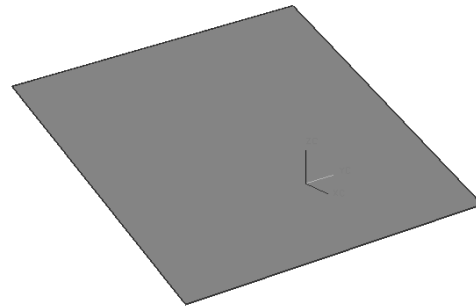


The swoop surfaces will be created as rectangular or square shaped planar (2D) surfaces and later modified to 3D surfaces by using the options in the tool. For creating a swoop surface, choose the **Swoop** button from the **Free Form Shape** toolbar; the **Point Constructor** dialog box will be displayed and you will be prompted





**Figure 12-15** The imaginary layout of the surface formed after specifying the four points for corners

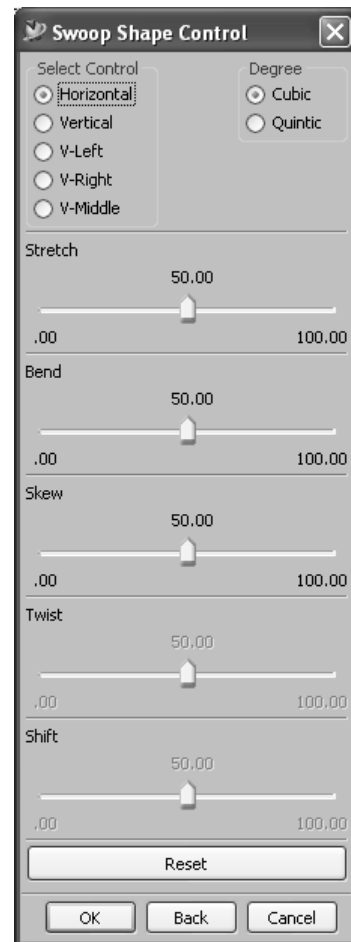


**Figure 12-6** The Resulting Surface

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 and the planar surface will be created. The **Swoop Shape Control** dialog box will be displayed, as shown in Figure 12-17. The vertical and horizontal axes will be displayed in red over the planar surface. The **Swoop Shape Control** dialog box will be used to modify the shape of the default surface formed. In the **Select Control** area, you have all the possible reference positions of the surface. 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 degree 3. Also, it will be comparatively harder. If you select the **Quintic** radio button, the resulting surface will be comparatively smoother.

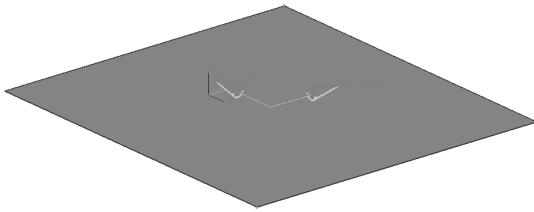
### Sliding Bars

Using the **Stretch** sliding bar, you can 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 the sliders. Using the **Bend** sliding bar, you can bend the surface in a positive or negative direction, along the reference position selected from the **Select Control** area. Using the **Skew** sliding bar, you can create a skewness factor for the surface in the positive or negative direction, along the reference position selected from the **Select Control** area. Using the **Twist** sliding bar, you can provide a twisting effect to the surface in the positive or negative

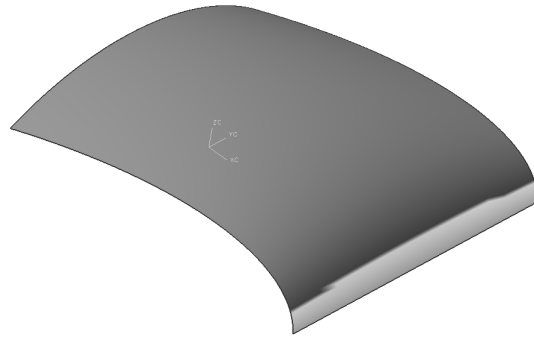


**Figure 12-17** The Swoop Shape Control dialog box

direction along the reference position selected from the **Select Control** area. Using the **Shift** sliding bar, you can shift the other edge of the surface in the positive or negative direction, along the reference position selected from the **Select Control** area. Figure 12-18 shows the planar surface created after specifying both the corners of the rectangle. Figure 12-19 shows the 3D surface modified from the planar surface using the shape modification sliding bars.



**Figure 12-18** The planar surface created after specifying both the corners of the rectangle



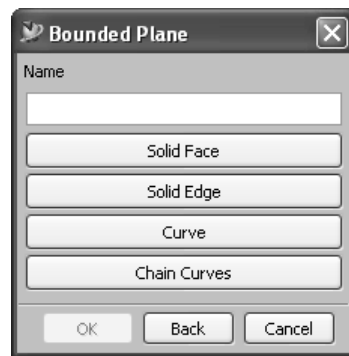
**Figure 12-19** The resulting 3D surface modified from the planar surface created

## Creating the Planar Surfaces from 2D Sketches and Solid Edges

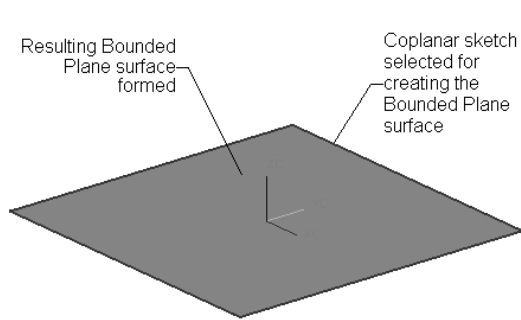
**Menu:** Insert > Surface > Bounded Plane  
**Toolbar:** Form Feature > Bounded Plane (Customize to add)



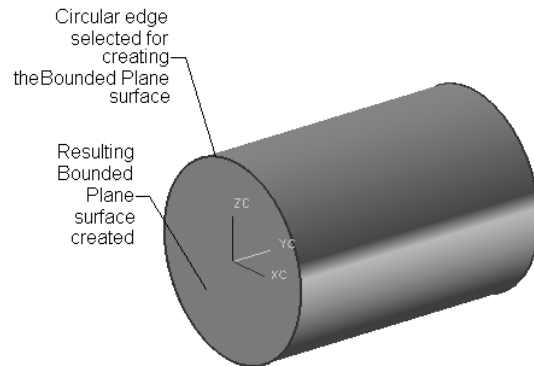
The **Bounded Plane** tool can be used to create a surface from the 2D sketches or closed coplanar edges. If you need to enclose a 2D sketch or a closed coplanar edges with a surface, choose the **Bounded Plane** button from the **Form Feature** toolbar; the **Bounded Plane** dialog box will be displayed, as shown in Figure 12-20, 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 edge. Select the solid edge and choose the **OK** button; the **Bounded Plane** surface will be created. Figure 12-21 shows the bounded plane surface enclosing a 2D sketch and Figure 12-22 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 itself.



**Figure 12-20** The **Bounded Plane** dialog box



**Figure 12-21** Bounded plane surface formed from a 2D sketch



**Figure 12-22** 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 the transition surface involves selecting the required cross-sections and mapping the intersected surface formed between the selected cross-sections automatically. You can also define the shape constraint to the connecting (intersecting) surface. For creating the **Transition** surface, you need to create a minimum of two cross-sections. Generally, three cross-sections are formed. After creating the 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 12-23. The **Section** button will be chosen by default. Select the sections and choose the **OK** button after selecting individual sections.

### Constraint Face, Reverse Normal, and Surface Preview Buttons

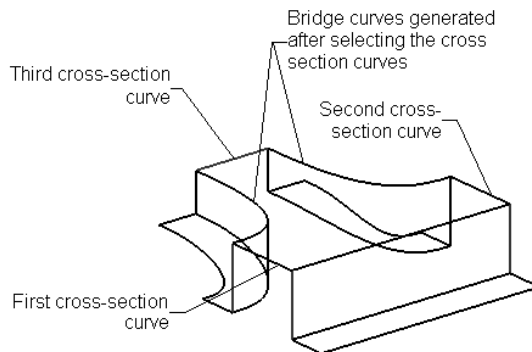
After selecting the first cross-section, the **Constraint Face** button will be enabled. To maintain the shape constraint of the intersecting surface with an existing surface, choose the **Constraint Face** button and select the reference face; the selected cross-sections will be listed in the list box. 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 is maintained. If you select the **G2** option, the curvature continuity is 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 that will 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

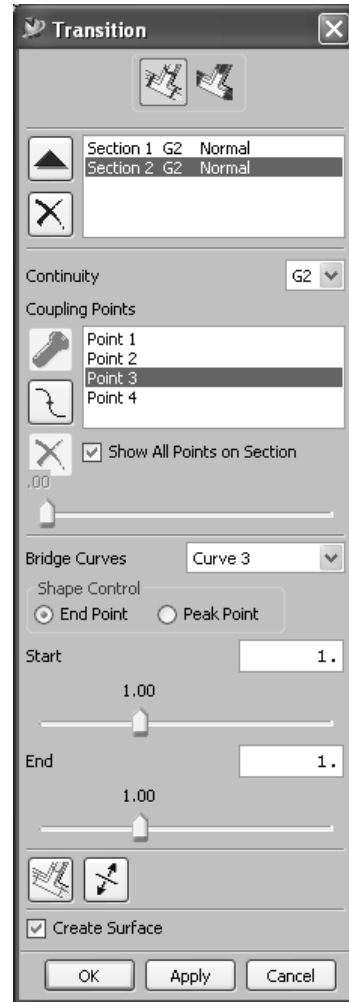
The bridge curves formed after selecting the cross-sections are listed as individual curves and also separate groups in the **Bridge Curves** drop-down list. By selecting the required curve from the **Bridge Curves** drop-down list and adjusting the sliding bar, you can control the shape of the selected 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

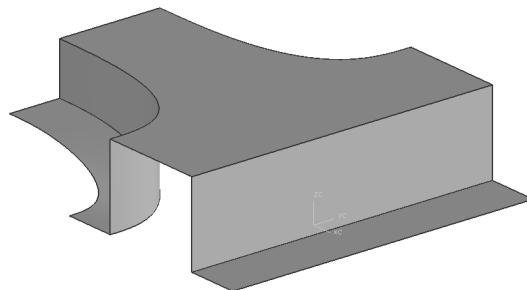
If you select the **Peak Point** radio button, the **Depth** and **Skew** sliding bars will be available for controlling the shape of the selected bridge curve from the **Bridge Curves** drop-down list. The cross-sections for creating the **Transition** surface, along with the bridge curves, is shown in Figure 12-24. The resulting **Transition** surface is formed, as shown in Figure 12-25.



**Figure 12-24** The bridge curves generated after selecting the cross-section curves



**Figure 12-23** The **Transition** dialog box



**Figure 12-25** The resulting transition surface created from the cross-sections

## 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 12-26, 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.

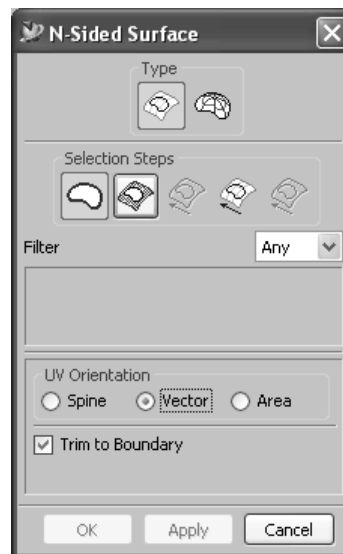


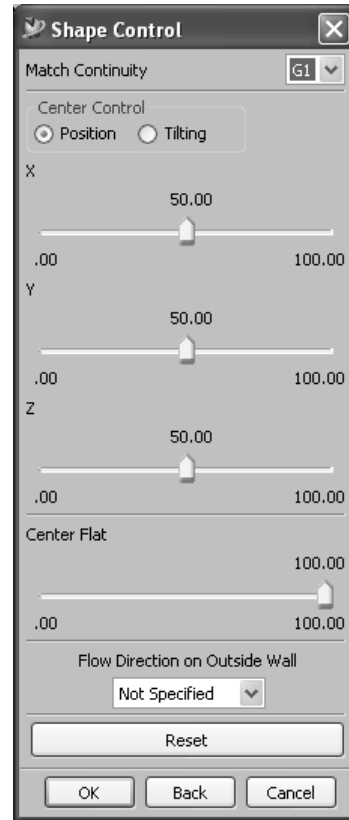
Figure 12-26 The N-Sided Surface dialog box

### Multiple Triangular Patches Button

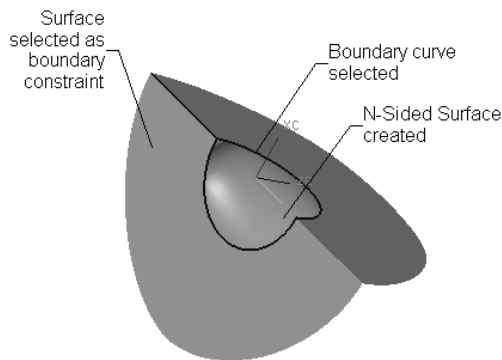
To create a surface with multiple triangular patches, choose the **Multiple Triangular Patches** button from the **Type** area. By default, the **Boundary Curves** button will be chosen from the **Selection Steps** area. After you select a closed boundary of a 2D sketch or a 3D curve, choose the **Boundary Faces** button from the **Selection Steps** area; you will be prompted to select the face for the boundary constraint. After selecting the face or 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.

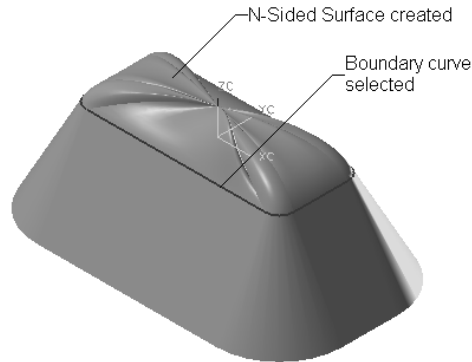
You can also create the surface by choosing the **Multiple Triangular Patches** button from the **Type** area. In the above case, after choosing the **OK** button, the **Shape Control** dialog box will be displayed, as shown in Figure 12-27. If you select the **Position** radio button, then the position of the surface can be varied in 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 the 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 12-28 shows the single patch N-sided surface created for the selected boundary curve and Figure 12-29 shows the multitriangular patch N-sided surface created for the selected boundary curve.



**Figure 12-27** The Shape Control dialog box



**Figure 12-28** Single patch N-sided surface created for the selected boundary curve



**Figure 12-29** 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 will be 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 formed 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



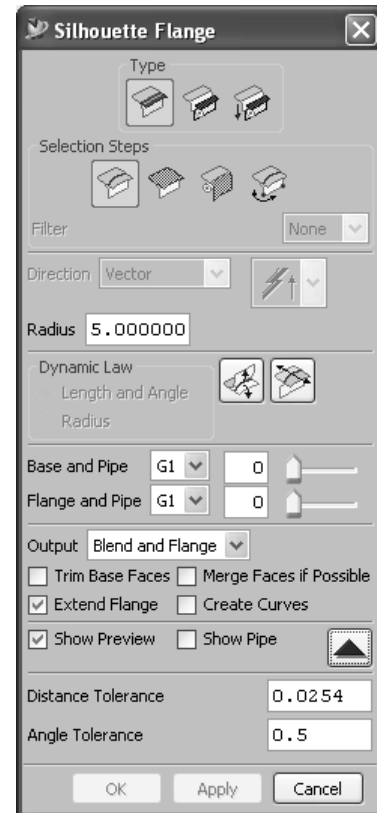
The **Silhouette Flange** tool will be used to create flange surfaces on an edge or a curve by taking any of the adjacent surfaces as the reference.

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 12-30, 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 the 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** area will be enabled.

#### Direction Drop-down List

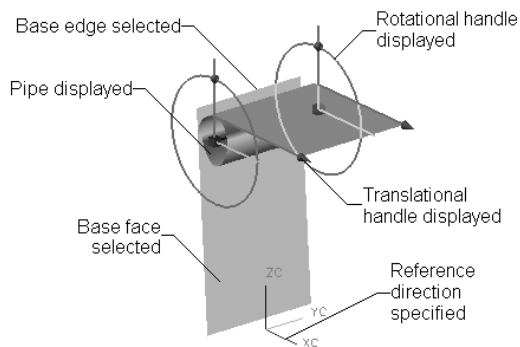
If you select the **Face Normal** option from the **Direction** drop-down list, the face that is normal to the reference face will be selected for specifying the direction. If you select the **Vector** option, the **Vector** drop-down list will be enabled through which you can specify the reference direction. 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.

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

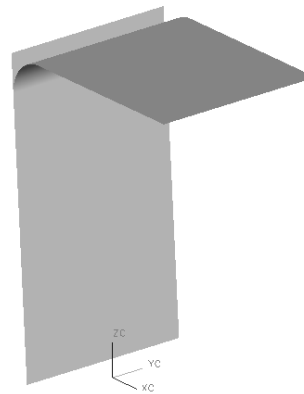


**Figure 12-30** The **Silhouette Flange** dialog box

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 the options for displaying 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 12-31 shows the pipe displayed along with the flange surface created using the **Basic** method and Figure 12-32 shows the resulting flange surface with the fillet.



**Figure 12-31** The silhouette flange surface displayed along with the handles and pipe



**Figure 12-32** The resulting silhouette flange surface created using the **Basic** method

### Creating a Silhouette Flange Surface Using the Absolute Gap Method

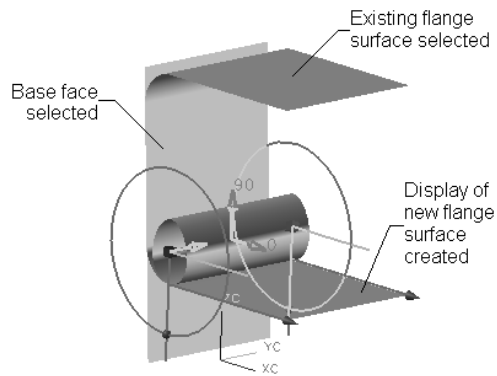


By choosing the **Absolute Gap** button from the **Type** area, you can create a silhouette flange surface relative to the existing silhouette flange surface by maintaining a predefined gap. The minimum gap is calculated by taking the radius of the two pipes and the nearest tangential distance between 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 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 12-33 shows the pipe displayed, along with the preview of the flange surface created using the **Absolute Gap** method and Figure 12-34 shows the resulting flange surface with the fillet.

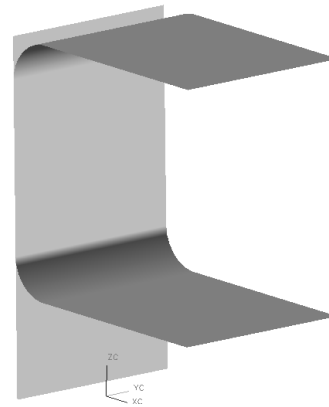
### Creating a Silhouette Flange Surface Using the Visual Gap Method

The **Visual Gap** button from the **Type** area can be chosen to create the flange surface in





**Figure 12-33** The newly created silhouette flange surface displayed along with handles and pipe



**Figure 12-34** The resulting silhouette flange surface created using the **Absolute Gap** method



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.

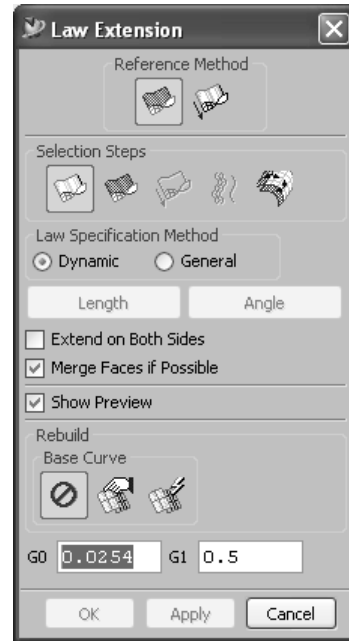
### Extending a Surface Dynamically Using the Dynamic Option

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 12-35, 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 the reference faces. Select the required face as the reference face and choose the **Define Law** button from the **Selection Steps** area. The preview of the surface will be displayed, if you have selected the **Show Preview** check box. 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

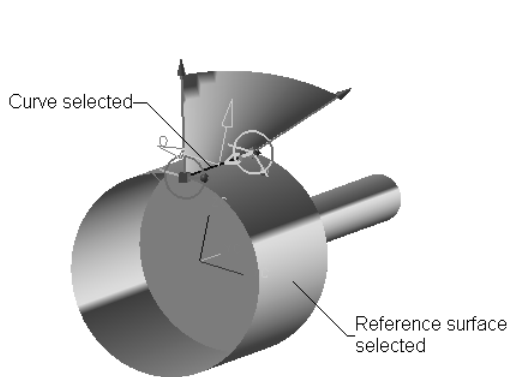
changes. Figure 12-36 shows the preview of the extended surface along with the rotational and translational handles. Figure 12-37 shows the resulting surface created after dynamically modifying it using the handles.

### Extending a Surface Driven by Laws Using the General Option

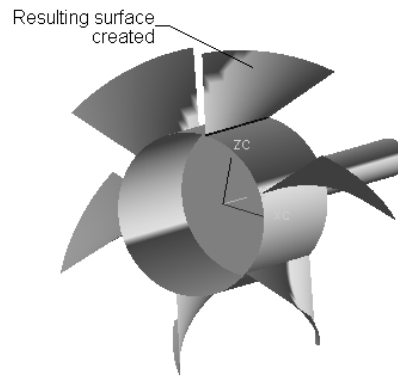
As discussed earlier, you can also extend a surface by defining different types of laws. To do so, choose the **Law Extension** button from the **Surface** toolbar. Select the **General** radio button from the **Law Specification Method** area; the **Length** and **Angle** buttons will be enabled in 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 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 **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 for defining 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 **End Value** edit boxes, respectively, and choose the **OK** button twice; the extended surface will be displayed.



**Figure 12-35** The *Law Extension* dialog box



**Figure 12-36** The preview of the law extension surface created



**Figure 12-37** The resulting law extension surface created and circularly arrayed



#### 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 12-38. By default, the **Faces to offset** button will be chosen from the **Selection Steps** area and you will be prompted to select the faces for the new set. Select the face, as shown in Figure 12-39. Next, enter the offset value in the **Set1D** edit box. If you want create a new set, choose the **Complete set and start the next set** button from the **Selection Steps** area. Choose the **OK** button. The resulting surface offset is shown in Figure 12-39.

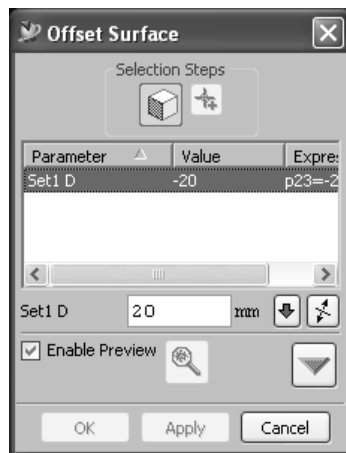


Figure 12-38 The *Offset Surface* dialog box

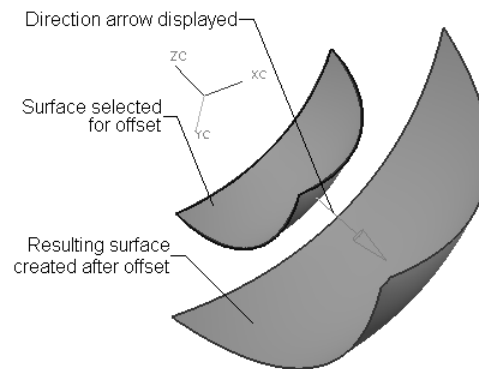


Figure 12-39 The offset surface created

## 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 12-40, and you will be prompted to select the target edges to extend. Select a single edge or multiple edges from the surface that is to be extended. When you select multiple edges for

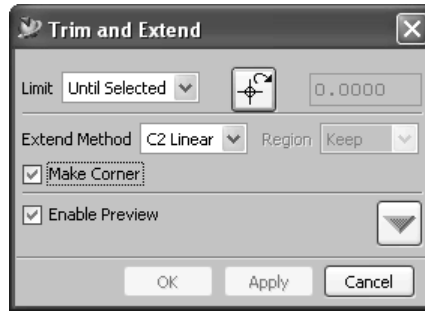


Figure 12-40 The Trim and Extend dialog box

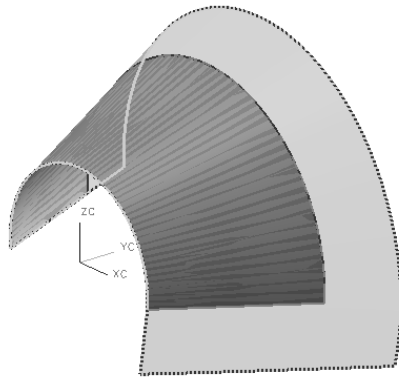
extending, ensure that the selected edges should have a continuity. If you have selected the **Enable Preview** check box, the preview of the extended surface will be displayed. If you select the **Distance** option from the **Limit** drop-down list, you can define the length of the surface extension by specifying a distance value in the edit box provided. If you select the **Percentage** option, the extension length can be specified in terms of the percentage of the original surface. If you select the **Until Selected** option, the surface will be extended up to the reference object selected. This option is also used to trim the selected surface. The options in the **Extend Method** drop-down list will be 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 degree 3 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.

If you select the **Until Selected** option from the **Limit** drop-down list, you will have to select the tool body that will serve as the boundary object, after you select the edge for extension. Choose the **Tool Selection** button and select the boundary object. Next, choose the **Apply** button and then the **OK** button to extend the surface up to the boundary object selected. After you select the tool body, the **Region** drop-down list will be enabled. The options in the **Region** drop-down list will be used to retain or discard the selected tool body. If you select the **Keep** option, the selected tool body will be retained after trimming. If you select the **Remove** option, then the material from the tool body is removed in the direction of the arrow displayed while selecting the tool body. If you select the **Make Corner** check box, a corner will be created at the intersection of the extended surface with the tool body and the tool body will be trimmed. If you choose the **More** button, the **Trim and Extend** dialog box will expand and the **Keep Original Edges** check box will be enabled. If you select the **Keep Original Edges** check box, then the edge selected for extension of the surface will be unaffected and will remain in the same state. Figure 12-41 shows the preview of the extension surface after selecting the edges. Figure 12-42 shows the surface extended using the **Until Selected** option from the **Limit** drop-down list.

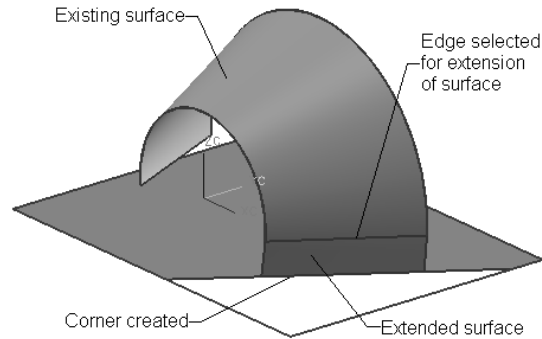
## Trimming a Sheet by Using the Trimmed Sheet Tool

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





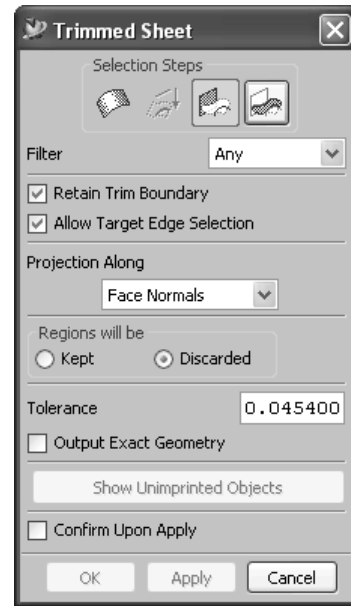
**Figure 12-41** The preview of the extended surface from the selected edges



**Figure 12-42** The resulting surface extended using the **Until Selected** option



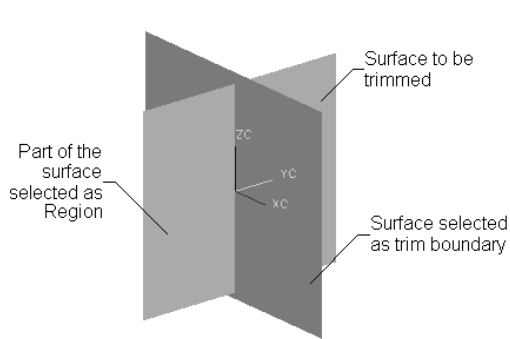
The **Trimmed Sheet** tool can be used to trim a sheet by defining the trim boundary. You can also trim a sheet by projecting a curve and defining it as a trim boundary. If the trim boundary is a surface, then the surface to be trimmed must intersect fully with the trimming surface. Choose the **Trimmed Sheet** tool from the **Surface** toolbar; the **Trimmed Sheet** dialog box will be displayed, as shown in Figure 12-43, and you will be prompted to select a target sheet body. By default, the **Target Sheet Body** button will be chosen from the **Selection Steps** area; select the sheet to be trimmed. Next, you will be prompted to select the trimming objects; select the trimming objects. Choose the **Region** button; you will see a point on the region. The point indicate whether this region is selected to be kept or discarded. The regions selected to kept or discarded is based on the radio button selected from the **Regions will be** area. If you select the **Kept** option from this area, the regions selected will be retained and the unselected regions will be removed. If you select the **Discarded** radio button, the regions selected will be removed (trimmed) and the unselected regions will be retained. Figure 12-44 shows the parameters selected when the trim boundary selected is a surface. Figure 12-45 shows the resulting trimmed surface after selecting the **Discarded** radio button from the **Regions will be** area.



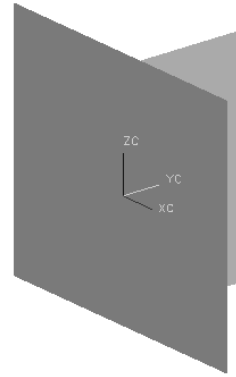
**Figure 12-43** The **Trimmed Sheet** dialog box

The **Projection Along** 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 gets automatically 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 fully. Figure 12-46 shows the parameters selected when the

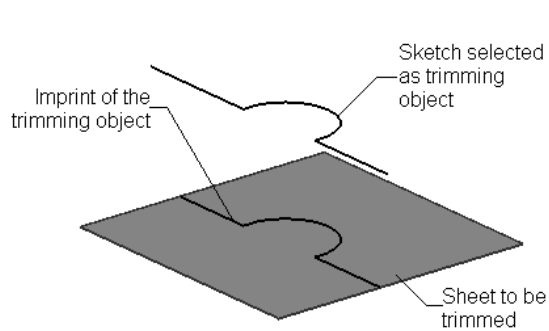
trim boundary is created by imprinting a curve on the surface to be trimmed. Figure 12-47 shows the resulting trimmed surface after selecting the **Discarded** radio button from the **Regions will be area**.



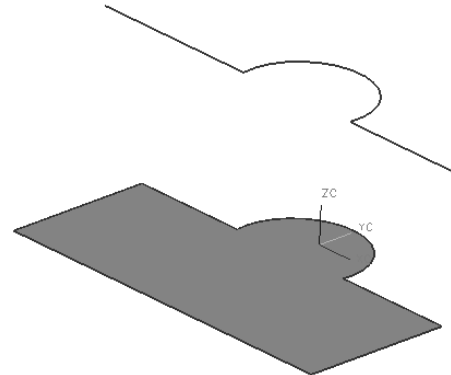
**Figure 12-44** The parameters selected for trimming a sheet



**Figure 12-45** The resulting trimmed sheet after selecting the **Discarded** radio button



**Figure 12-46** The sketch selected for trimming a sheet



**Figure 12-47** The resulting trimmed sheet after selecting the **Discarded** radio button

## Creating a Surface Using the Studio Surface Tools

The studio surface tools can be used to create a surface by sweeping a single section or multiple sections, along single or multiple guide curves. The guide curves and the section curves selected can be open or closed. The studio surface tools for creating various type of surfaces are discussed next.

### Creating a Surface Using the Studio Surface 1x1 Tool

|                 |                                      |
|-----------------|--------------------------------------|
| <b>Menu:</b>    | Insert > Mesh Surface > 1x1          |
| <b>Toolbar:</b> | Free Form Shape > Studio Surface 1x1 |





The name of this tool indicate the number of sections that can be selected and the number of guide curves that can be selected for creating the surface. This means that using the **Studio Surface 1x1** tool, you can create a surface by selecting a single section and a single guide curve. Choose the **Studio Surface 1x1** button from the **Free Form Shape** toolbar; the **Studio Surface 1x1** dialog box will be displayed, as shown in Figure 12-48, and you will be prompted to select a section string.

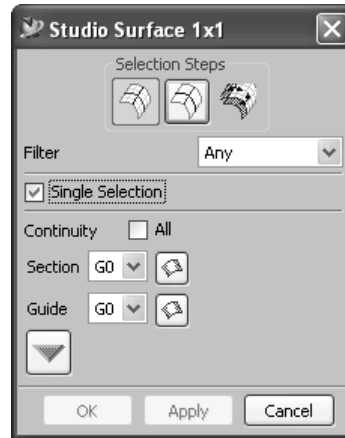


Figure 12-48 The Studio Surface 1x1 dialog box

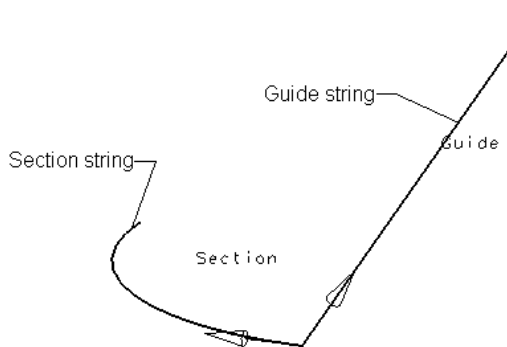
By default, the **Section** button from the **Selection Steps** area will be chosen. Select the section curve and choose the **Guide** button. Select the guide curve and choose the **Face** button; the preview of the surface will be displayed. Choose the **Apply** button and then the **Cancel** button to accept the surface. Note that the arrow directions must be as shown in Figure 12-49. If you choose the **More** button, the dialog box will be expanded. You can enter the intersection value in the respective edit boxes. You can also specify the shape control for the surface to be created using an existing surface as a reference surface by using the **Section** and **Guide** drop-down lists. If you select the **G0** option, the shape control will not be maintained. If you select the **G1** option, the tangential relation will be maintained with the surface selected after choosing the **Constraint Face** button. If you select the **G2** option, a curvature relation will be maintained with the surface selected, after choosing the **Constraint Face** button. The options in the **Rebuild** area have been explained earlier. Figure 12-49 shows the section and guide curve selected. Figure 12-50 shows the preview of the resulting studio surface created when you choose the **Face** button.

### Creating a Surface Using the Studio Surface 1x2 Tool

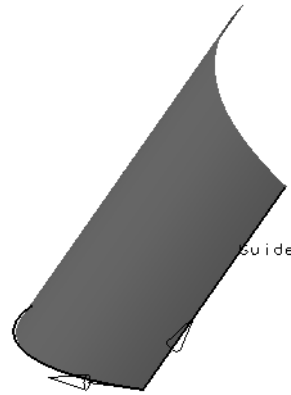
**Menu:** Insert > Mesh Surface > 1x2  
**Toolbar:** Free Form Shape > Studio



The **Studio Surface 1x2** tool can be used to create a surface by selecting a single section and two guide curves. For creating the surface using this tool, choose the **Studio Surface 1x2** button from the **Free Form Shape** toolbar; the **Studio Surface 1x2** dialog box will be displayed, as shown in Figure 12-51, and you will be prompted to select the section string. By default, the **Section** button from the **Selection Steps** area is chosen. Select the section curve and choose the **Guide1** button. Select the first guide curve and

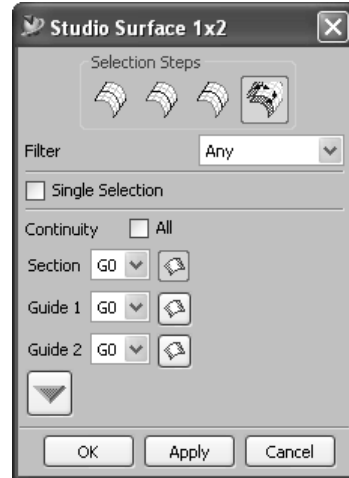


**Figure 12-49** The section curve and the guide curve selected for creating the studio surface



**Figure 12-50** The preview of the studio surface created using the **Studio Surface 1x1** tool

choose the **Guide 2** button. Select the second guide curve and choose the **Face** button; the preview of the resulting surface will be displayed. Choose the **Apply** button and then the **Cancel** button to accept the surface. If you choose the **More** button, the dialog box will expand. You can enter the intersection value in the respective edit boxes. You can also specify the shape control for the surface to be created using an existing surface as a reference surface by using the **Section**, **Guide 1**, and **Guide 2** drop-down lists. If you select the **G0** option, the shape control will not be maintained. If you select the **G1** option, the tangential relation will be maintained with the surface selected after choosing the **Constraint Face** button. If you select the **G2** option, a curvature relation will be maintained with the surface selected, after choosing the **Constraint Face** button. The options in the **Rebuild** area are explained earlier. Figure 12-52 shows the single section and two guide curves selected and Figure 12-53 shows the resulting studio surface.



**Figure 12-51** The **Studio Surface 1x2** dialog box

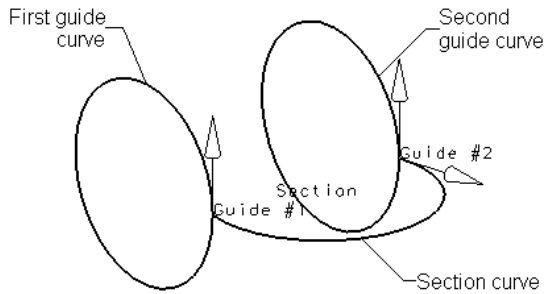
## Creating a Surface Using the Studio Surface 2x0 Tool

**Menu:** Insert > Mesh Surface > 2x0  
**Toolbar:** Free Form Shape > Studio Surface 2x0

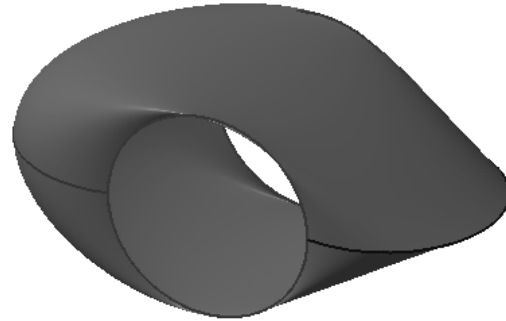


The **Studio Surface 2x0** tool can be used to create a surface by selecting two section curves. You cannot select a guide curve. For creating the surface using this tool, choose the **Studio Surface 2x0** button from the **Free Form Shape** toolbar; the **Studio Surface 2x0** dialog box will be displayed, as shown in Figure 12-54, and you will be prompted to select the first section string. By default, the **Start Section** button from the **Selection Steps** area will be chosen. Select the first section curve and choose the **End Section** button. Select the second section curve and choose the **Face** button; the preview of the surface created will be displayed. Choose the **Apply** button and then the **Cancel** button to accept the



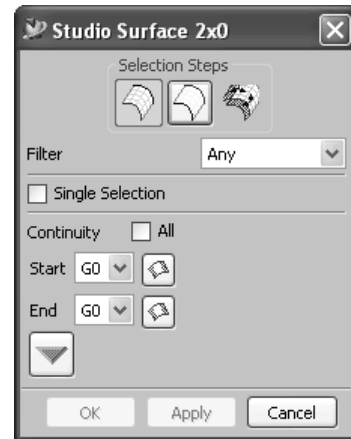


**Figure 12-52** The section curve and the guide curves selected for creating the studio surface

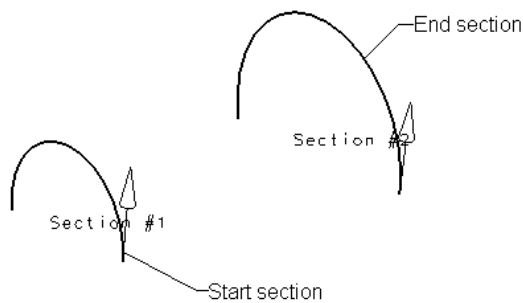


**Figure 12-53** The resulting studio surface

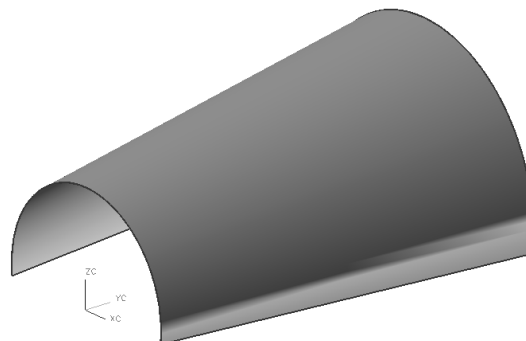
surface. Figure 12-55 shows the start and end section curves selected and Figure 12-56 shows the resulting studio surface created.



**Figure 12-54** The Studio Surface 2x0 dialog box



**Figure 12-55** The section curves selected for creating a studio surface



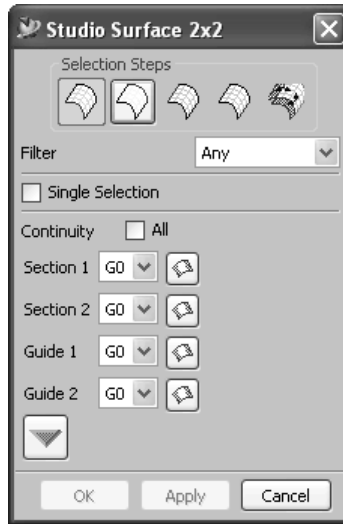
**Figure 12-56** The resulting studio surface created using the Studio Surface 2x0 tool

## Creating a Surface Using the Studio Surface 2x2 Tool

**Menu:** Insert > Mesh Surface > 2x2  
**Toolbar:** Free Form Shape > Studio

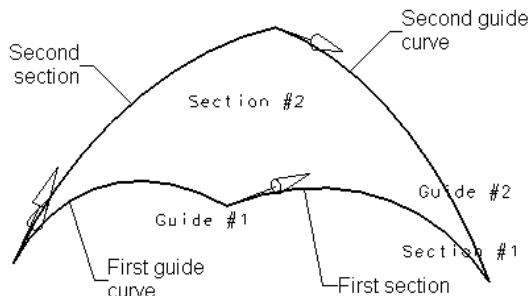


The **Studio Surface 2x2** tool can be used to create a surface by selecting two section curves and two guide curves. For creating the surface using this tool, choose the **Studio Surface 2x2** button from the **Free Form Shape** toolbar; the **Studio Surface 2x2** dialog box will be displayed, as shown in Figure 12-57, and you will be prompted to select the first section string.

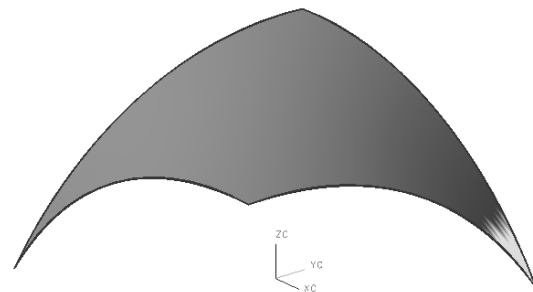


*Figure 12-57 The Studio Surface 2x2 dialog box*

By default, the **Section 1** button from the **Selection Steps** area is chosen. Select the first section curve and choose the **Section 2** button. Select the second section curve and choose the **Guide 1** button. Select the first guide curve and choose the **Guide 2** button. After selecting the two guide curves, choose the **Face** button; the preview of the surface created will be displayed. Choose **Apply** and then **Cancel** to accept the surface. Figure 12-58 shows the two section and two guide curves selected. Figure 12-59 shows the resulting studio surface created.



*Figure 12-58 The section curves and the guide curves selected for creating the studio surface*



*Figure 12-59 The resulting studio surface created using the Studio Surface 2x2 tool*

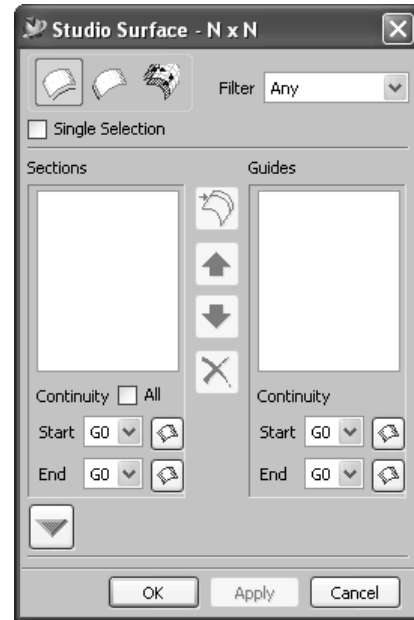
## Creating a Surface Using the Studio Surface nxn Tool

**Menu:** Insert > Mesh Surface > nxn  
**Toolbar:** Free Form Shape > Studio

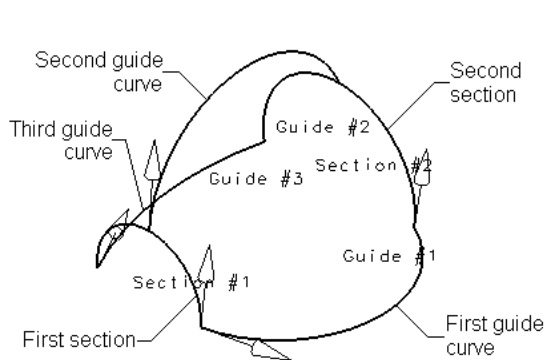


The **Studio Surface nxn** tool can be used to create surfaces by selecting any number of section curves and any number of guide curves.

For creating the surface using this tool, choose the **Studio Surface nxn** button from the **Free Form Shape** toolbar; the **Studio Surface- NxN** dialog box will be displayed, as shown in Figure 12-60, and you will be prompted to select a curve, edge, or a curve feature for a section string. By default, the **Add to Sections** button will be chosen from the **Selection Steps** area. Select the section curve and choose the **OK** button. After selecting each section curve, choose the **OK** button. You can select any number of section curves. After you select the section curves, choose the **Add to Guides** button from the **Selection Steps** area. Select the guide curves and choose the **OK** button; the preview of the created surface will be displayed, if you have selected the **Preview Body** check box from the expanded area of the dialog box. Choose the **Apply** button and then the **Cancel** button to accept the surface. You can also control the shape of the surface created by selecting the option from the **Start** and **End** drop-down lists of the **Continuity** area. Before selecting the reference surface to maintain the shape control, choose the **Face** button. Next, select the reference surface. Figure 12-61 shows the section curves and guide curves selected. Figure 12-62 shows the resulting studio surface created.



**Figure 12-60** The Studio Surface - NxN dialog box



**Figure 12-61** The section curves and the guide curves selected to create a studio surface



**Figure 12-62** The resulting surface created by using the Studio Surface nxn tool

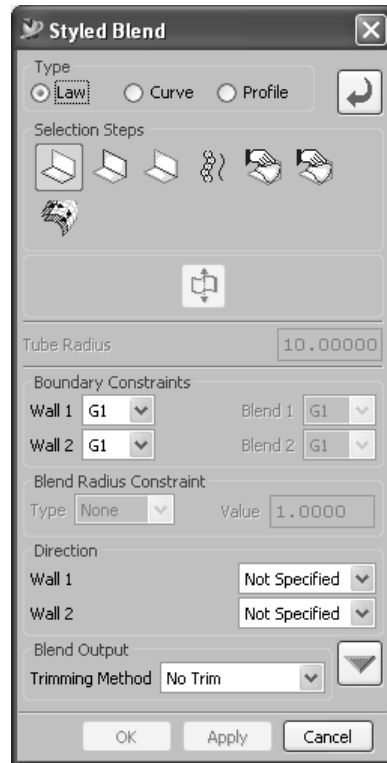
## Creating a Surface Between Two Walls Using the Styled Blend Tool



**Menu:** Insert > Detail Feature > Styled Blend  
**Toolbar:** Free Form Shape > Styled Blend



The **Styled Blend** tool can be 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. For creating the fillet surface using this tool, choose the **Styled Blend** button from the **Free Form Shape** toolbar; the **Styled Blend** dialog box will be displayed, as shown in Figure 12-63, and you will be prompted to select the faces for wall 1.

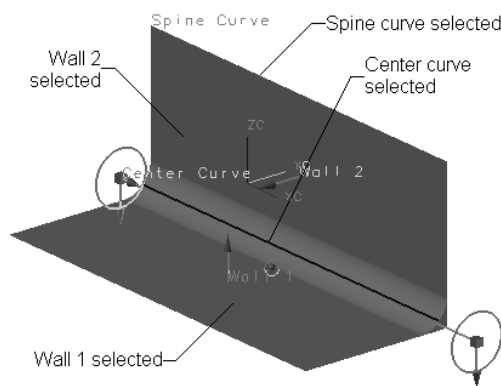


**Figure 12-63** The *Styled Blend* dialog box

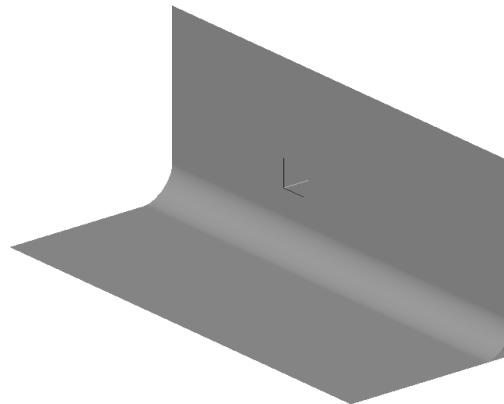
The option selected from the **Type** area decides the method of formation of the blend surface. If you select the **Law** radio button, the tangent holding lines will be automatically created with respect to the pipe radius specified for the fillet. If you select the **Curve** radio button, you will 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 the both 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 in 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, ensure that the arrow displayed from the walls after selection, 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 will be used to flip the direction of the blend surface created. Figure 12-64 shows the parameters selected for creating the styled blend surface using the **Law** option and Figure 12-65 shows the resulting surface created.



**Figure 12-64** The preview of the styled blend surface displayed for the selected parameters



**Figure 12-65** The resulting styled blend surface created using the **Law** option

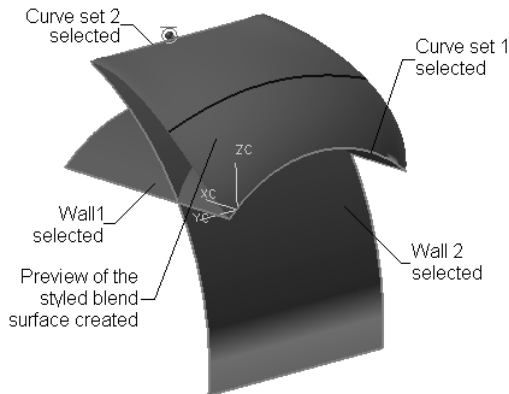
### Creating a Styled Blend Surface Using the Curve Option

As mentioned earlier, when you select the **Curve** option from the **Type** area for creating the styled blend surface, you need 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. After choosing the **Preview** button, the **Labels** check box along with the **Select Feature** and **Reverse Blend Direction** buttons will be enabled. If the **Labels** check

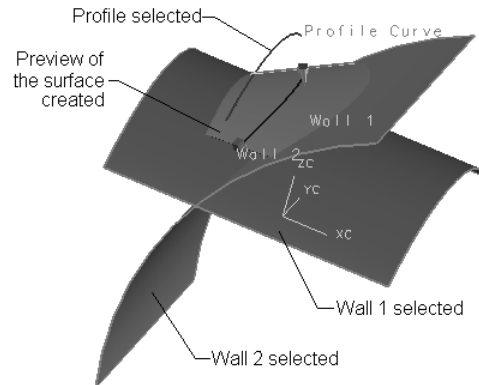
box is selected, the entities that are selected for the blend creation such as wall 1, wall 2, curve set 1, curve set 2, and spine will be annotated and displayed in the graphics window. Figure 12-66 shows the parameters selected for creating the styled blend surface using the **Curve** option and the resulting surface created.

### Creating a Styled Blend Surface Using the Profile Option

As mentioned earlier, when you adopt the **Profile** option from the **Type** area for creating the **Styled Blend** surface, you need select the curve or 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 faces for wall 1. While selecting both 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 **Profile** button. Select the curve or 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 12-67 shows the parameters selected for creating the styled blend surface using the **Profile** option and also the resulting surface.



**Figure 12-66** The preview of the styled blend surface displayed after selecting the parameters for the **Curve** option



**Figure 12-67** The preview of the styled blend surface displayed after selecting the parameters for the **Profile** option

### Creating Surfaces Using the Styled Sweep Tool

**Menu:** Insert > Sweep > Styled Sweep  
**Toolbar:** Free Form Shape > Styled Sweep



The **Styled Sweep** tool can be used to create surfaces that need sweeping of the cross-sections across the guide curves. The surface created across the cross-sections by following the guide curve can be modified dynamically by dragging the handles displayed along with the surface created. For creating the surface, choose the **Styled Sweep** tool from the **Free Form Shape** toolbar. The **Styled Sweep** dialog box will be displayed, as shown in Figure 12-68, and you will be prompted to select the guide string. By default, the **Guide String** button is chosen from the **Selection Steps** area. Select the first guide string and choose the **Guide String** button again to add the guide curve selected. The first guide string

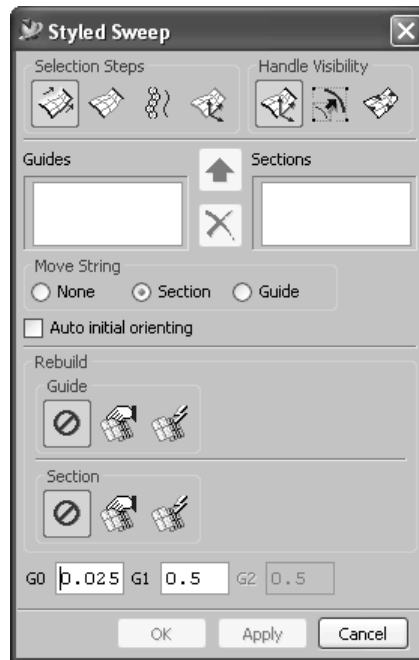


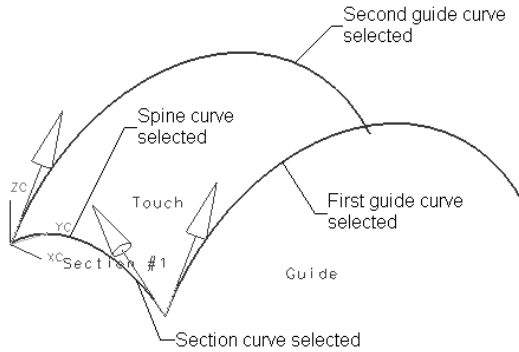
Figure 12-68 The *Styled Sweep* dialog box

selected is known as the guide and the shape of the surface is guided along the shape of the guide string. After choosing the **Guide String** button for the second time, the selected guide string will be listed in the **Guides** list box. The second guide string is known as the touch curve. The surface will be created by touching the overall length of this curve. At the maximum, you can only select two guide curves. Next, choose the **Section String** button and select the section string. After selecting the section string, choose the **Section string** button to add the selected section in the **Sections** drop-down list. Next, choose the **Spine String** button from the **Selection Steps** area. Select the curve to be followed as the spine. Otherwise, after selecting the cross-sections, choose the **Shape Control** button from the **Selection Steps** area. The preview of the surface will be displayed, along with the handles. By dragging the handles, you can modify the shape and size of the surface. The options in the **Handle Visibility** area are used to display the different types of handles for the surface created. You can dynamically modify the surface using the handles displayed.

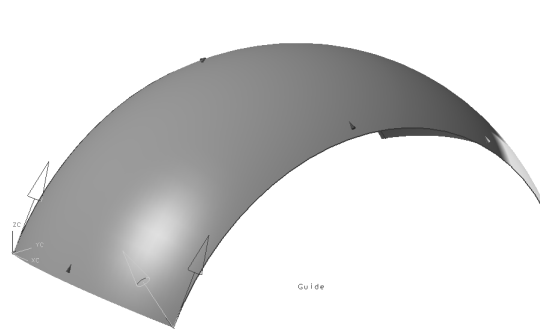
### Handle Visibility Area

After you select the **Section** radio button from the **Move String** area, the **Handle Visibility** area will be enabled. You can also rebuild the guide strings and the section strings by using the options from the **Rebuild** area. You can rebuild the strings automatically or manually, by selecting the **Automatic** or **Manual** button, respectively. If you need to rebuild the curves automatically, choose the **Automatic** button from the **Rebuild** area; the **Max Segments** and **Max Degrees** spinners will be enabled. By adjusting the values in the spinner, you can rebuild the curves automatically. If you need to rebuild the curves manually, choose the **Manual** button from the **Rebuild** area; the **Max Degrees** spinners will be enabled. By adjusting the values in the spinner, you can rebuild the curves automatically. Figure 12-69 shows the parameters selected for

creating the styled sweep surface and Figure 12-70 shows the resulting styled sweep surface.



**Figure 12-69** The parameters selected for creating the styled sweep surface



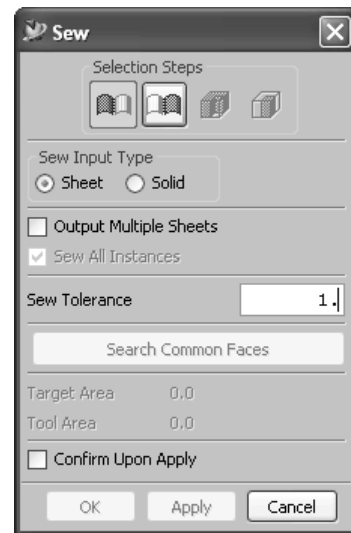
**Figure 12-70** The resulting styled sweep surface

## Sewing Individual Surfaces into a Single Surface

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



The **Sew** tool from the **Feature Operation** toolbar is used to stitch the 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 surfaces enclose a volume, a solid body will be created. The sheet to which all the other individual sheets are to be stitched is known as the target sheet, and the individual sheets that are to be stitched are known as tool sheets. You cannot stitch tool sheets that intersect a target sheet and extend from it. For stitching the individual surfaces in to a single surface, choose the **Sew** button from the **Feature Operation** toolbar; the **Sew** dialog box will be displayed, as shown in Figure 12-71, and you will be prompted to select the target sheet to sew. Select the target sheet to sew; the border of the target sheet will be displayed in yellow and you will be prompted to select the tool sheets to sew. The border of the selected tool sheets should lie in the yellow boundary displayed. Otherwise, the selected tool sheet will not be stitched with the target sheet. To stitch surfaces, select the **Sheet** radio button from the **Sew Input Type** area; the **Target Sheet** and **Tool Sheets** buttons will be enabled in the **Selection Steps** area. To combine the solid bodies, select the **Solid** radio button. After selecting this radio button, the **Target Faces** and **Tool Faces** buttons will be enabled in the **Selection Steps** area. You can also enter the sew tolerance in the **Sew Tolerance** edit box. If the selected objects are an instance of an array and the **Sew All Instances** check box is selected, all



**Figure 12-71** The **Sew** dialog box



the instances of the array are stitched together. This check box will be enabled only after selecting the **Solid** radio button.

## Adding Thickness to a Surface

**Menu:** Insert > Offset / Scale > Thicken Sheet  
**Toolbar:** Form Feature > Thicken Sheet (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. For adding the thickness, choose the **Thicken Sheet** tool from the **Form Feature** toolbar; the **Thicken Sheet** dialog box will be displayed, as shown in Figure 12-72. You will be prompted to select a sheet body to thicken. Select the sheet to which the material is to be added. Enter the thickness value in the **Second Offset**



Figure 12-72 The **Thicken Sheet** dialog box

edit box. You can enter the offset distance in the **First Offset** edit box. The **Action** drop-down list provides the options for performing the boolean operation with an existing solid body. To perform any type of boolean operation with an existing solid body, select the respective option from the **Action** drop-down list. Next, choose the **Target Solid Body** button from the **Selection Steps** area and select the target solid body. Figure 12-73 shows the sheet displayed to add the thickness and Figure 12-74 shows the resulting solid body.



Figure 12-73 The sheet selected for adding the material

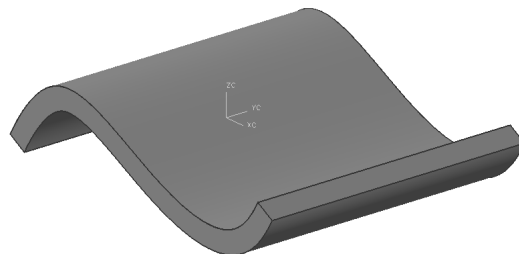


Figure 12-74 The resulting solid body after adding thickness to the sheet

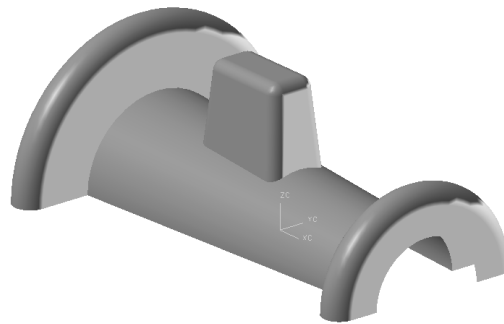
## TUTORIALS

### Tutorial 1

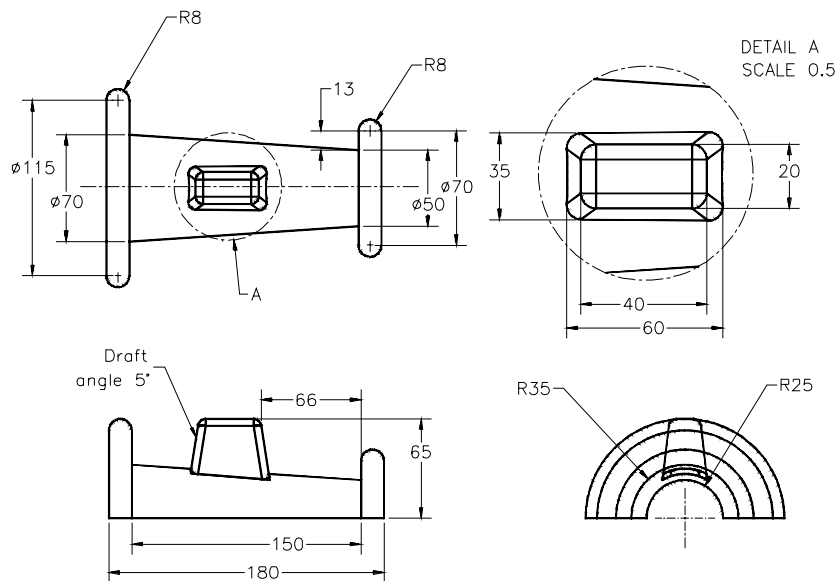
In this tutorial, you will create the surface model shown in Figure 12-75. The dimensions and orthographic views are shown in Figure 12-76. After creating the surface, save it in the name mentioned below.

\\NX 4\\c12\\c12tut1.prt

(Expected Time: 30 min)



**Figure 12-75** The isometric view of the surface model



**Figure 12-76** The dimensions and drawing views of the surface model

The following steps are required to complete this tutorial:

- Start a new file and then set the sheet environment.
- Create the sketch for the base surface and then revolve it, refer to Figures 12-77 and 12-78.
- Create the second feature, which is an extruded surface, refer to Figures 12-79 and 12-80.
- Trim the base surface, refer to Figures 12-81 and 12-82.
- Trim the extended part of the third feature, refer to Figures 12-83 and 12-84.
- Create the bounded plane surface, refer to Figure 12-85.
- Stitch the bounded plane and extruded surfaces with the revolved surface.
- Fillet the stitched surface, refer to Figure 12-86.
- Save and close the file.

### Starting a New File and Setting the Sheet Environment

- Start a new file using the **Master Model Template** and save it with the specified name.
- Choose **Preferences > Modeling** from the menu bar; the **Modeling Preferences** dialog box is displayed.
- Select the **Sheet** radio button from the **Body Type** area and choose the **OK** button.

### Creating the Base Feature by Revolving the Sketch

- Create the sketch for the base surface on the XC-YC Plane, as shown in Figure 12-77.

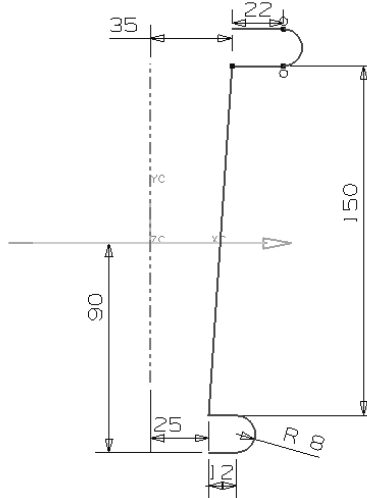
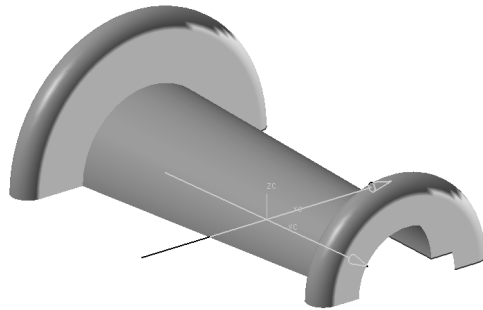


Figure 12-77 The sketch created for the base feature

- Revolve the sketch through an angle of 180-degrees. The resulting revolved base feature created using the **Sheet** option is shown in Figure 12-78.

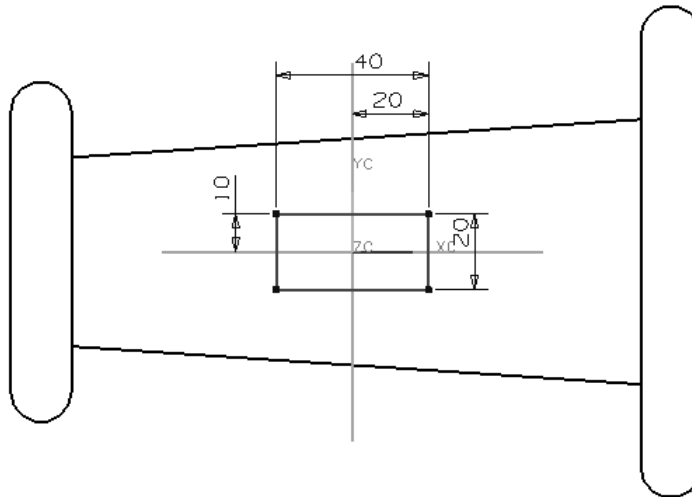
### Creating the Second Feature by Extruding the Sketch

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



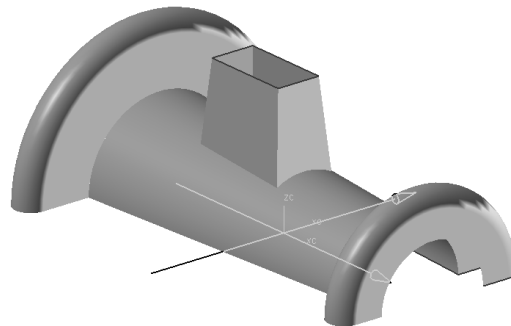
**Figure 12-78** The base feature created by revolving the sketch

2. Create the sketch for the second feature by selecting the offset plane as the sketching plane, as shown in Figure 12-79.



**Figure 12-79** The sketch drawn for creating the extruded feature

3. Extrude the sketch through a distance of 50 in the downward direction and the draft angle of -5. Select the **Sheet** radio button from the **Body Type** area of the **Extrude** dialog box. The resulting extruded surface model is shown in Figure 12-80.

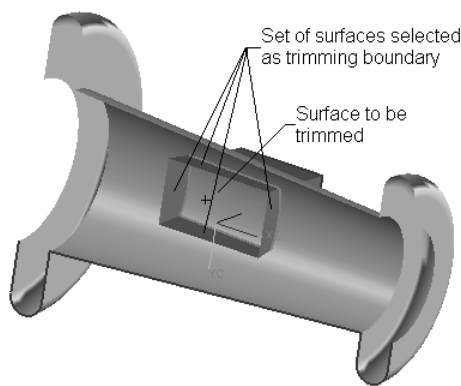


**Figure 12-80** The resulting extruded surface

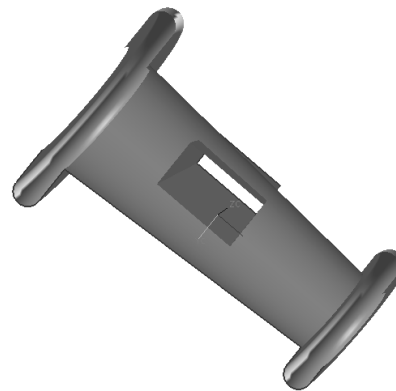
### Trimming the Base Surface with the Second Surface

Next, you need to trim the base surface with respect to the second feature. For trimming the base surface, follow the procedure 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 the target sheet body.
2. Select the sheet to be trimmed, as shown in Figure 12-81. Make sure you select the sheet using the selection point shown in the figure. After you select the target sheet body, you are prompted to select the trimming objects.
3. Select the trimming surfaces, as shown in Figure 12-81. Make sure you select the surfaces using the points shown in the figure.
4. Select the **Discarded** radio button from the **Regions will be** area and then choose the **OK** button. The resulting surfaces, after they are trimmed, is shown in Figure 12-82.



**Figure 12-81** The surface to be trimmed and the trimming surfaces selected from the model

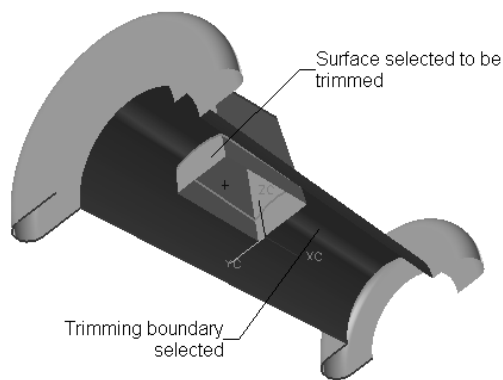


**Figure 12-82** The surface model after trimming the base surface with the extruded surface

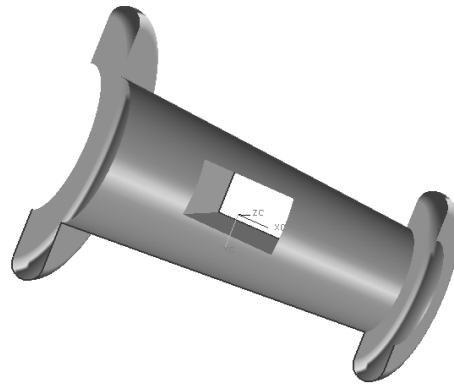
### Trimming the Second Surface Created with the Base Surface

After trimming the base surface, you need to trim the unwanted portions of the second feature. For trimming the extended portion, follow the procedure 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 12-83; you are prompted to select the trimming objects.
3. Select the trimming surface, as shown in Figure 12-83. Select the **Discarded** option from the **Regions will be** area and then choose the **OK** button. The resulting surfaces after they are trimmed, are as shown in Figure 12-84.



**Figure 12-83** The surface to be trimmed and the trimming surface selected from the model

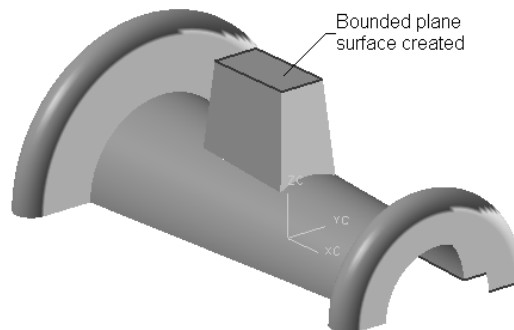


**Figure 12-84** The resulting surface after trimming the extended portion of the extruded surface

### 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 discussed next.

1. Choose the **Bounded Plane** button from the **Form Feature** toolbar; the **Bounded Plane** dialog box is displayed and you are prompted to select a bounding string.
2. Select the sketch used to create the second surface and choose the **OK** button. The resulting bounded plane surface is shown in Figure 12-85.



**Figure 12-85** The resulting bounded plane surface created

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

After creating all the 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 **Apply** button and then the **Cancel** button; all surfaces are stitched together.

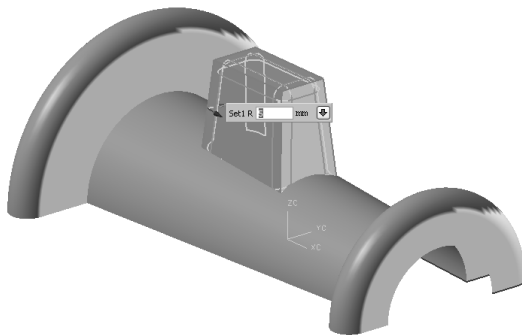
**Note**

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. For hiding these entities, press **Ctrl+B**; The **Blank Icon Options** are displayed. Select the entities and choose the **OK** button.

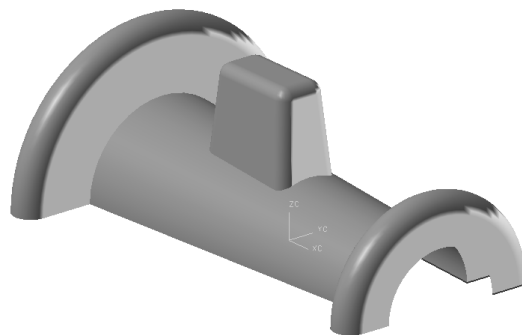
### Creating Fillets on the Edges Using the Edge Blend Tool

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 12-86. Enter the fillet radius value as **5** in the **Set 1 R** edit box.
3. Choose the **OK** button from the **Edge Blend** dialog box. The completed surface model, after you add the fillets to the edges, is as shown in Figure 12-87.



**Figure 12-86** The preview of the fillets displayed after selecting the edges



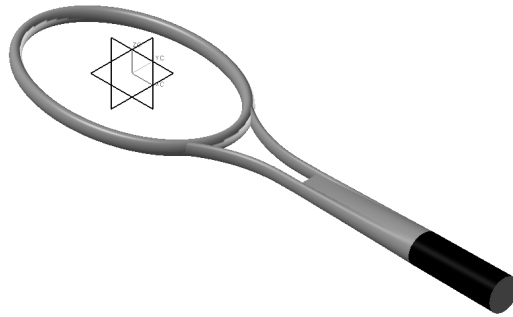
**Figure 12-87** The completed surface model after adding the fillets to the edges

### Saving the Surface Model

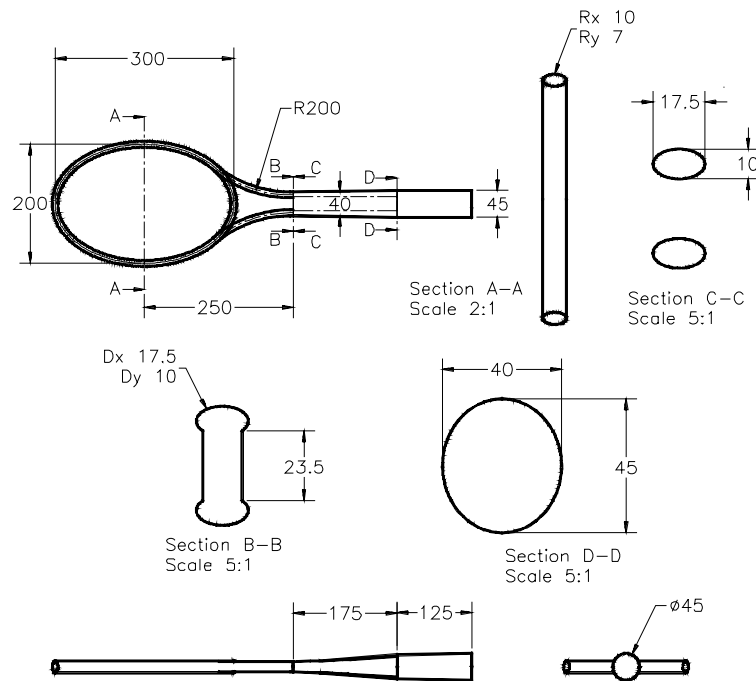
1. Choose **File > Save** from the menu bar.

## Tutorial 2

In this tutorial, you will create the surface model shown in Figure 12-88. The drawing views and the dimensions of the surface model are shown in Figure 12-89. After creating the model, save it with the name `|NX 4|c12\c12-tut2.prt` **(Expected Time: 45 min)**



**Figure 12-88** The isometric view of the surface model



**Figure 12-89** The drawing views and dimensions of the surface model



The following steps are required to complete this tutorial:

- Start a new file and set the sheet environment.
- Create the styled sweep surface as the base feature, refer to Figure 12-92.
- Create the sweep surface as the second feature and mirror the surface, refer to Figures 12-95 and 12-96
- Create the surface through curve mesh as the third feature, refer to Figures 12-97 through 12-99.
- Create the ruled surface as the fourth feature, refer to Figure 12-101 and 12-102.
- Create the bounded plane surface as the fifth feature, refer to Figure 12-103.
- Create another bounded plane surface as the sixth feature, refer to Figure 12-104.
- Save the surface model.

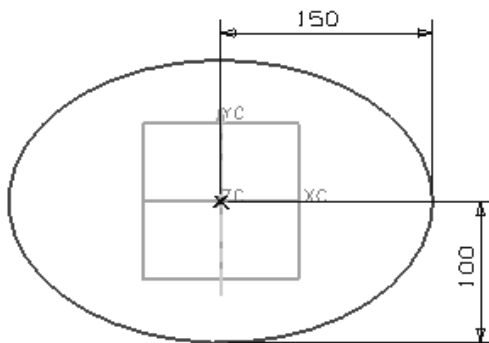
### Starting a New File and Setting the Sheet Environment

- Start a new file using the **Master Model Template** and save it with the specified name.
- Choose **Preferences > Modeling** from the menu bar; the **Modeling Preferences** dialog box is displayed.
- Select the **Sheet** radio button from the **Body Type** area and choose the **OK** button.

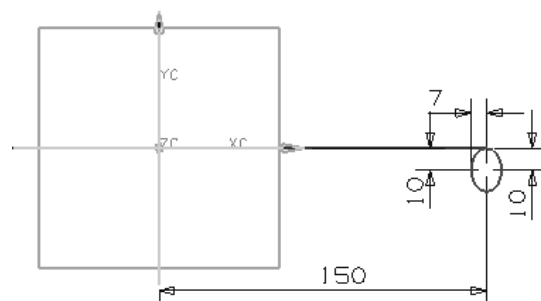
### Creating a Styled Sweep Surface as the Base Feature

As mentioned earlier, the base feature will be the styled sweep surface. For creating the base feature, follow the procedure discussed next.

- Draw an ellipse on the XC-YC Plane as per the dimensions shown in Figure 12-90 and exit the sketching environment.
- Draw another ellipse on the XC-ZC Plane as per the dimensions shown in Figure 12-91 and exit the sketching environment.



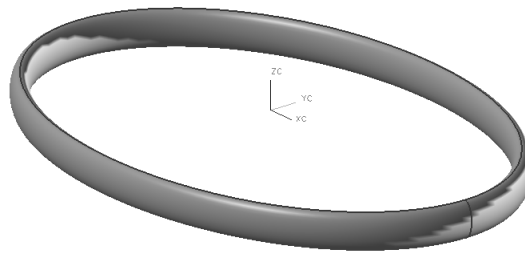
**Figure 12-90** The sketch for the guide string



**Figure 12-91** The sketch for the section string

- Choose the **Styled Sweep** button from the **Free Form Shape** toolbar; the **Styled Sweep** dialog box is displayed and you are prompted to select the guide string.

4. Select the guide curve and choose the **Guide String** button again; you are prompted to select the tough string.
5. Choose the **Section String** button and select the section curve. Again, choose the **Section String** button.
6. Choose the **Shape Control** button to display the preview of the surface. Choose the **Apply** button and then the **Cancel** button to create the styled sweep surface. The resulting styled sweep surface created is shown in Figure 12-92.

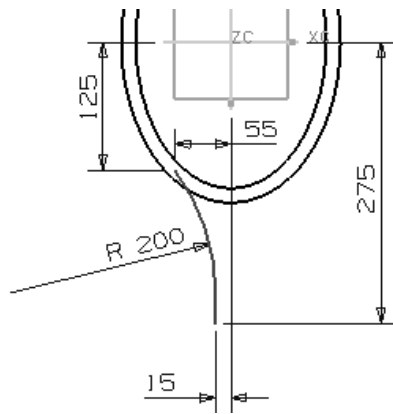


*Figure 12-92 The resulting base feature of the surface model*

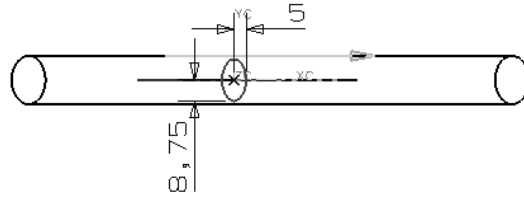
### **Creating the Sweep Surface as the Second Feature**

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 12-93.
3. Create a new datum plane perpendicular to the guide curve, by entering the location value as 0.
4. Select the newly created plane and draw the ellipse (section curve), as shown in Figure 12-94.
5. Choose the **Sweep along Guide** button from the **Form 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. The resulting sweep surface will be created, as shown in Figure 12-95.



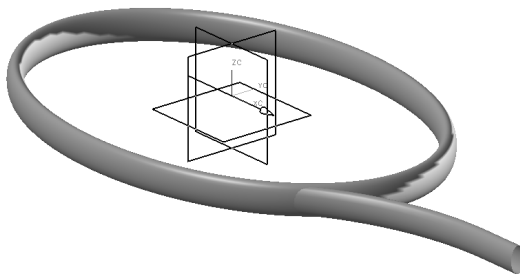
**Figure 12-93** The dimensions for the guide curve of the sweep surface



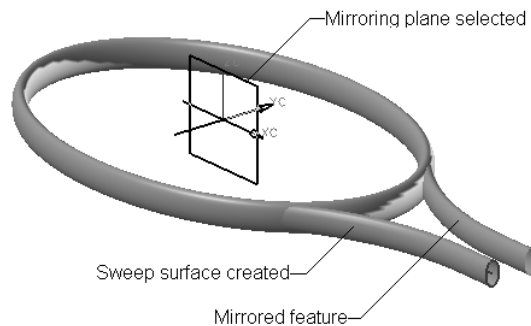
**Figure 12-94** The section curve drawn for creating the sweep surface

### Mirroring the Sweep Surface

1. Mirror the last surface using the ZC-XC Plane as the mirror plane. The resulting surface model, after mirroring the sweep surface, is shown in Figure 12-96.



**Figure 12-95** The resulting sweep surface created after selecting the guide curve and the section curve



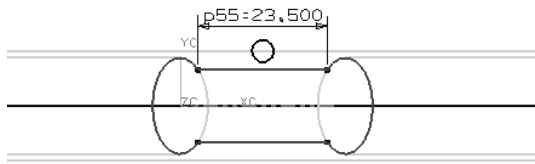
**Figure 12-96** The resulting surface model after mirroring the sweep feature

### Creating the Through Curve Mesh Surface

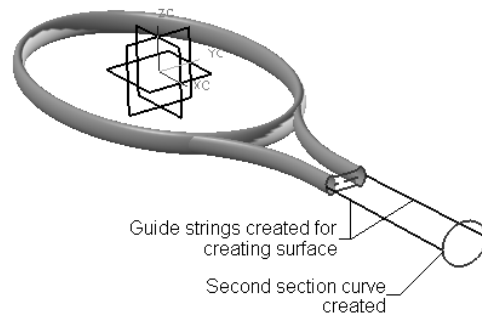
The third feature is the surface that is created using the through curve mesh tool.

1. Invoke the sketching environment using the datum plane created to draw the section of the sweep surface.
2. Choose **Insert > Project** from the menu bar; the **Project Icon Options** are displayed and you are prompted to select the geometry to project into the sketch.

3. Select the edges and create the profile, as shown in Figure 12-97. Exit the sketching environment.
4. Create a datum plane parallel to the YC-ZC Plane and 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 12-98. The dimensions can be obtained from Figure 12-89.

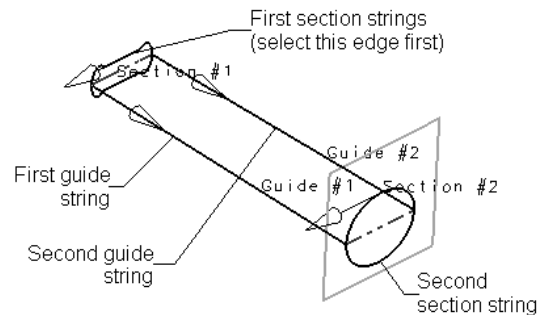


**Figure 12-97** The first section curve drawn to create the through curve mesh surface



**Figure 12-98** The second section curve and the guide strings created to create the surface

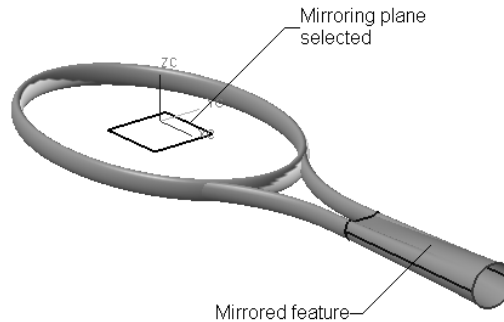
6. Select the offset datum plane and create the guide strings, as shown in Figure 12-98.
7. Choose **Insert > Mesh Surface > 2x2** from the menu bar; the **Studio Surface 2x2** dialog box is displayed and you are prompted to select the first section string.
8. Select all the elements in the first section string, as shown in Figure 12-99. Choose the **Section 2** button; you are prompted to select the second section string.
9. Select the second section string. Make sure the arrows in both section strings point in the same direction, as shown in Figure 12-99. Choose the **Guide 1** button; you are prompted to select the first guide string.
10. Select the first guide curve, as shown in Figure 12-99, and choose the **Guide 2** button; you are prompted to select the second guide string.
11. Select the second guide curve, as shown in Figure 12-99. Choose the **Apply** button and then the **Cancel** button to create the surface.



**Figure 12-99** The entities to be selected to create the 2x2 surface mesh

### Mirroring the Through Curve Mesh Surface

1. Mirror the 2x2 surface using the XC-YC Plane as the mirror plane. The resulting surface model is shown in Figure 12-100.



*Figure 12-100 The resulting surface model after mirroring the surface*

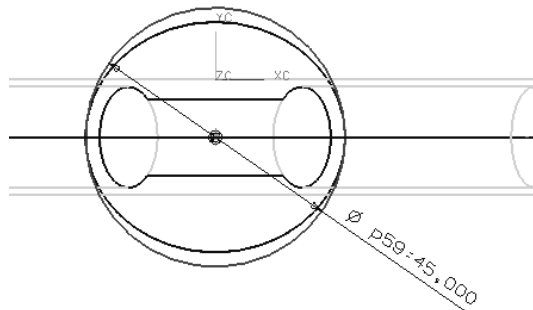
### Creating the Ruled Surface

Next, you need to create the ruled surface.

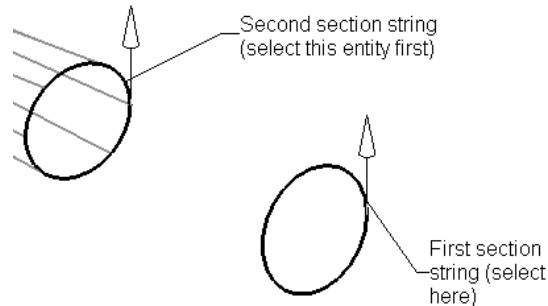
1. Create a plane parallel to the YC-ZC Plane and at a distance of 575.
2. Select the created plane as the sketching plane and draw the sketch, as shown in Figure 12-101.
3. Exit the sketching environment and choose the **Ruled** button from the **Surface** toolbar; the **Ruled** dialog box is displayed and you are prompted to select the first section string.
4. Select the first section string, as shown in Figure 12-102. After selecting the first section string, choose the **OK** 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 102. Note that the arrows should point in the same direction. Choose the **OK** button; the **Ruled** surface is created.

### Creating the Bounded Plane Surface

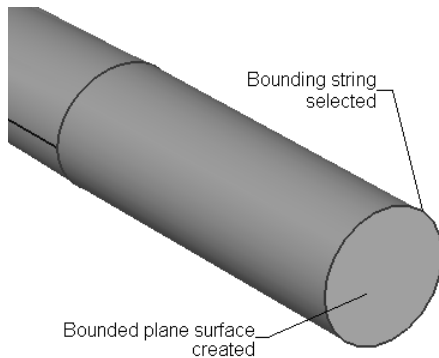
1. Choose the **Bounded Plane** button from the **Form Feature** toolbar; the **Bounded Plane** dialog box is displayed and you are prompted to select the bounding string.
2. Select the bounding string for creating the bounded plane surface, as shown in Figure 12-103. The resulting bounded plane surface is displayed, as shown in Figure 12-103.
3. Again, select the sketch shown in Figure 12-104 to create the bounding surface. The resulting bounded plane surface is displayed, as shown in Figure 12-104.



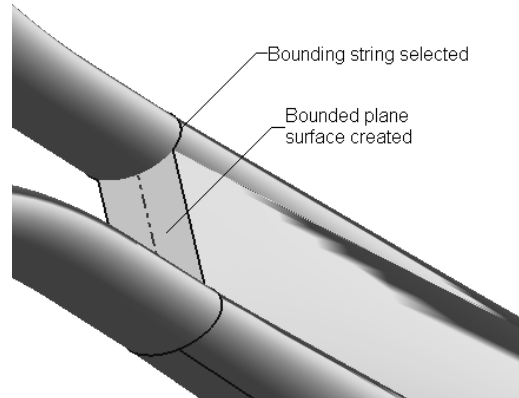
**Figure 12-101** The first section curve created to create the through curve mesh surface



**Figure 12-102** The resulting ruled surface created after selecting both the cross-sections



**Figure 12-103** The Bounding string selected and the resulting bounded plane surface created



**Figure 12-104** The Bounding string selected and the resulting bounded plane surface created

### **Saving the Surface Model**

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

### **Self-Evaluation Test**

Answer the following questions and then compare your answers with those given at the end of the chapter:

1. The surfaces are termed as sheets in NX. (T/F)
2. The capped end option is used to close the ends of an extruded sheet. (T/F)
3. You can use the **Until Selected** and **Until Next** options from the **End** drop-down list of **Extrude** dialog box to create the sheet. (T/F)

4. The default tolerance value specified for the creation of the sheet by the manufacturer is 0.00254. (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 close the ends of the closed surface?
  - a. **Bounded Plane**
  - b. **Swept**
  - c. **N-sided Surface**
  - d. None of the above
3. Which tool is used to create the flange surface?
  - a. **Silhouette Flange**
  - b. **N-Sided Surface**
  - b. **Law Extension**
  - d. None of the above
4. Which tool is used to create a surface by using multiple sections only?
  - a. **Through Curve Mesh**
  - b. **Ruled**
  - c. **Studio Surface 1x2**
  - d. None of the above
5. Before adding a fillet at the intersection of two surfaces, the surface has to be
  - a. Stitched using **Sew** tool
  - b. Merged
  - c. Trimmed
  - d. None of the above
6. You can select an open surface to create the bounded plane surface. (T/F)

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. The transition surfaces are the ones that are formed by mapping the intersecting points of the section curves selected. (T/F)
10. You can create a hole feature on a planar surface by using the **Hole** tool. (T/F)

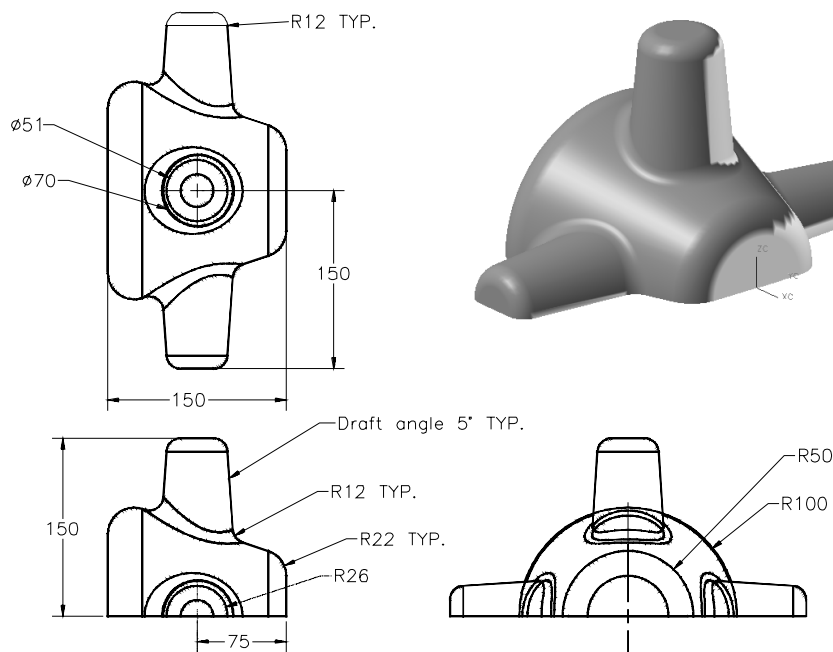
## Exercises

### Exercise 1

Create the surface model shown in Figure 12-105. The drawing views and the dimensions of the surface model are shown in Figure 12-105. Save the model with the following name:

|NX 4|c12|c12exr1.prt

(Expected Time: 30 min)



**Figure 12-105** Drawing views and dimensions of the surface model for Exercise 1

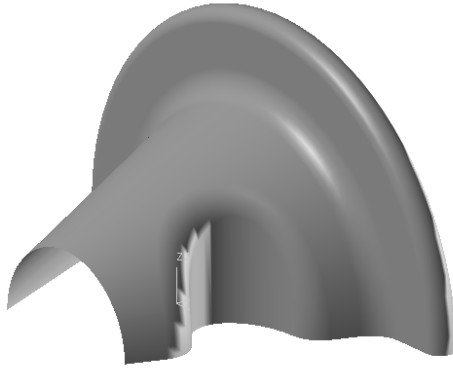


**Exercise 2**

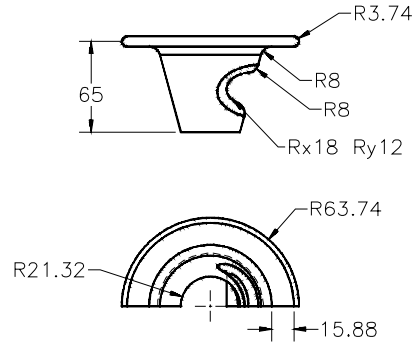
Create the surface model shown in Figure 12-106. The drawing views and the dimensions of the surface model are shown in Figure 12-107. Save the model with the following name:

|NX 4|c12|c12exr2.prt

(Expected Time: 30 min)



**Figure 12-106** The isometric view of the surface model for Exercise 2



**Figure 12-107** The dimensions and drawing views of the surface model

**Answers to Self-Evaluation Test**

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