Chapter 11

Surface Modeling

Learning Objectives

After completing this chapter, you will be able to:

- *Create extrude, revolve, and sweep surfaces.*
- *Create ruled surfaces.*
- *Create surfaces using the Through Curves and Through Curve Mesh tools.*
- *Create a surface by using 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 tools.*
- *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 the assembly created using the surface body type possesses a surface area but not the volume or mass properties. In NX, surfaces are created in the form of single or multiple patches. With the increase in the patches, the control over the shape of the surface also increases. In NX, surfaces are known as sheets and surface modeling is known as sheet modeling.

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

INVOKING THE SHEET MODELING ENVIRONMENT

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

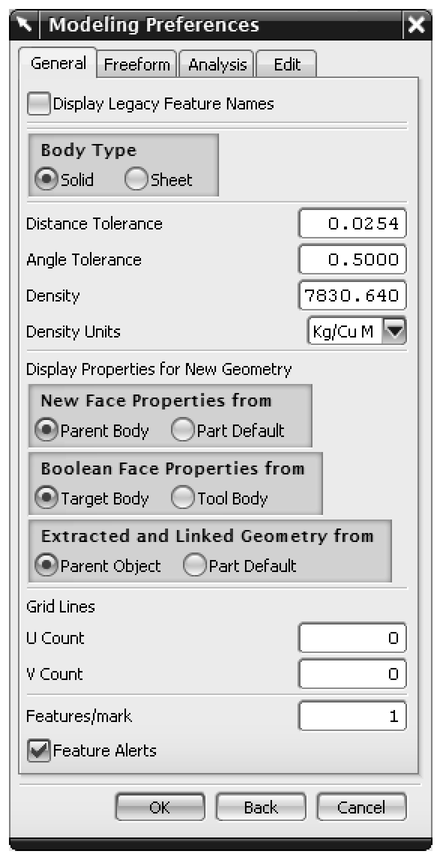

***Figure 11-1** The **Modeling Preferences** dialog box*

Creating an Extruded Surface

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

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

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the **Start** and **End** edit boxes. Choose the **OK** button; the sheet will be created. The options in the **Extrude** dialog box are the same as discussed in Chapter 4. The surface extrude operations performed on open and closed sketches are displayed in Figures 11-2 and 11-3, respectively.

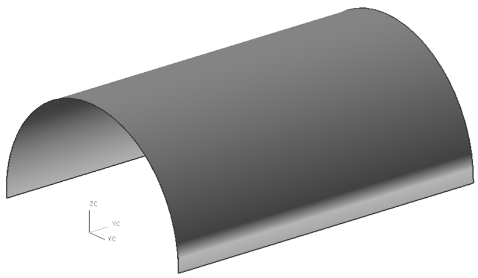


Figure 11-2 *Surface extrude created on an open sketch*

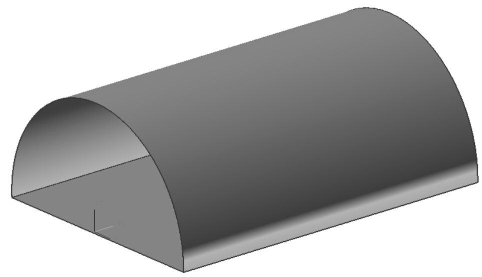
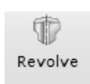


Figure 11-3 *Surface extrude created on a closed sketch*

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

Creating a Revolved Surface

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



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

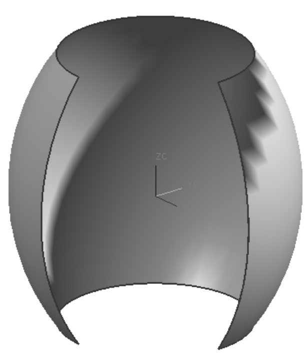


Figure 11-4 *Revolved surface created using an open sketch*

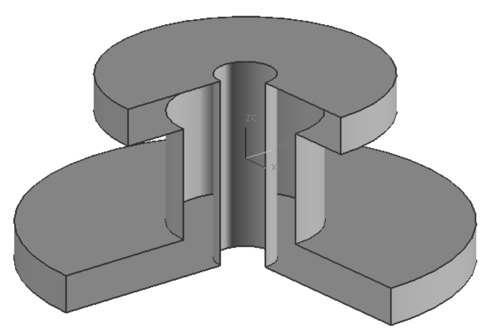


Figure 11-5 *Revolved surface created using a closed sketch*

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Creating a Ruled Surface

Toolbar:	Surface > Ruled	*(Customize to add)*

The **Ruled** tool is used to create ruled surfaces. These surfaces are always created between two similar or dissimilar cross-sections created on different planes. The sketches for this feature may be open or closed. Initially, isoparametric lines are formed to create patches, which are then converted into surfaces. The options to create isoparametric curves are discussed later in this chapter. To create the ruled surface, create two cross-sections on two different planes. Choose the **Ruled** button from the **Surface** toolbar; the **Ruled** dialog box will be displayed, as shown in Figure 11-6. Figure 11-7 shows two cross-sections on two different planes to create a ruled surface.

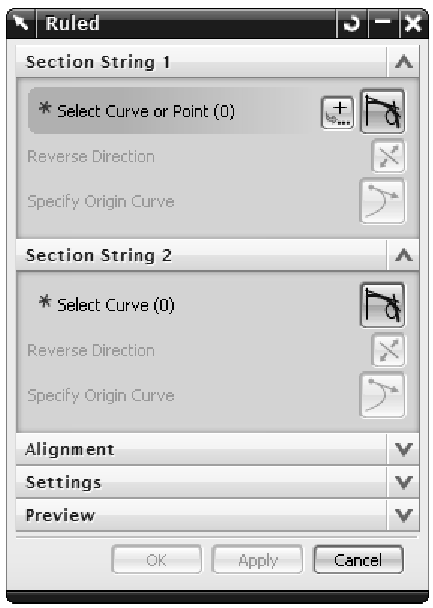

*Figure 11-6 The **Ruled** dialog box*

In the **Section String 1** rollout of **Ruled** dialog box, the **Section 1** button is chosen by default. As a result, you are prompted to select the curves for section 1. Select the curves for the first cross-section; an arrow will be displayed on the first section string indicating the direction of surface formation. Next, choose the **Section 2** button from the **Section String 2** rollout; you will be prompted to select the curves for Section 2. Select the second section string; an arrow will be displayed on the second section string. The arrows on the first and second section strings should point in the same direction, as shown in Figure 11-8.

The curve of the section string on which the direction arrow is displayed is known as origin curve. The first isoparametric line is generated by joining the start points of the origin curves of section string 1 and section string 2. The consecutive control points of the section string 1 and section string 2 are joined by isoparametric lines to generate the ruled surface. You can control the shape of the ruled surface by changing the origin curve. To do so, choose the **Specify Origin Curve** button from the **Ruled** dialog box and select the curve from the

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respective section string that you want to make as origin curve. Note that the **Specify Origin Curve** button will be available only you select the closed sections as section strings. Note that you can use different rollouts in the **Ruled** dialog box to modify a surface. These rollouts are discussed next.

Alignment Rollout

The **Alignment** drop-down list in the **Alignment** rollout is used to specify different methods to distribute control points on section strings for creating isoparametric lines that form patches. If you select the **Parameter** option, the control points will be distributed such that the isoparametric lines are formed at uniform intervals. If you select the **By Points** option from the **Alignment** drop-down list, then isoparametric lines and control points will be displayed along the section string and the **Alignment** rollout will be modified. If you select the control point, the dynamic edit box will be displayed, as shown in Figure 11-8. You can drag the selected control point to change its position as well as isoparametric lines. Alternatively, you can directly enter the arc percentage in the dynamic edit box. To add a new control point in the section, click on the required curve; a control point will be created at that point. To reset control points, choose the **Reset** button from the **Alignment** rollout. If you clear the **Preserve Shape** check box from the **Settings** rollout, some options will be displayed in the **Alignment** drop-down list, which are discussed next. If you select the **Arc length** option from the **Alignment** drop-down list, the entire curve will be divided into two equal segment with respect to arc length. Also, the isoparametric curve will pass through the dividing points. If you select the **Distance** option from the **Alignment** drop-down list, the equally spaced isoparametric lines will be created perpendicular to the direction vector selected. If you select the **Angles** option, the isoparametric curves with angles at equal intervals will be created with respect to the common axis line. If you select the **Spine Curve** option, isoparametric curves will be formed at the intersection points created on the selected curves by perpendicular planes.

Settings Rollout

By default, the **Preserve Shape** check box is selected in this rollout. As a result, sharp corners will be created while modifying control points. If you clear this check box, a smooth curvature will be formed while modifying the position of control points. You can use the **GO (position)** edit box only when the **Preserve Shape** check box is clear. If you select the **GO (position)** edit box, you can enter the value of the maximum distance upto which the shape of the ruled surface can be modified at sharp corners.

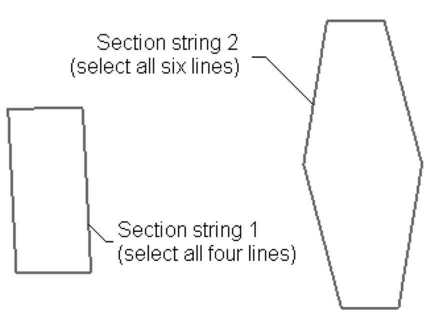

Figure 11-7 *The section strings selected for creating the ruled surface*

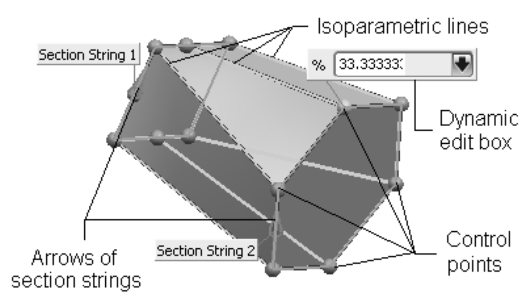

Figure 11-8 *The resulting ruled surface created from the selected section strings*

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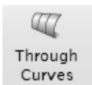
Note
The maximum allowed and the minimum required number of cross-sections for creating the ruled surface are two.

Creating Surfaces Using the Through Curves Tool

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



You can create surfaces with multiple section strings using the **Through Curves** tool. This method allows you to select any number of section strings. To create surfaces using the **Through Curves** tool, choose the **Through Curves** button from the **Surface** toolbar; the **Through Curves** dialog box will be displayed, as shown in Figure 11-9, and you will be prompted to select the section. Select the section and press the middle mouse button; you will be prompted again to select the section. Likewise, you can select any number of section strings. After selecting the section strings, make sure the arrow points in the same direction.

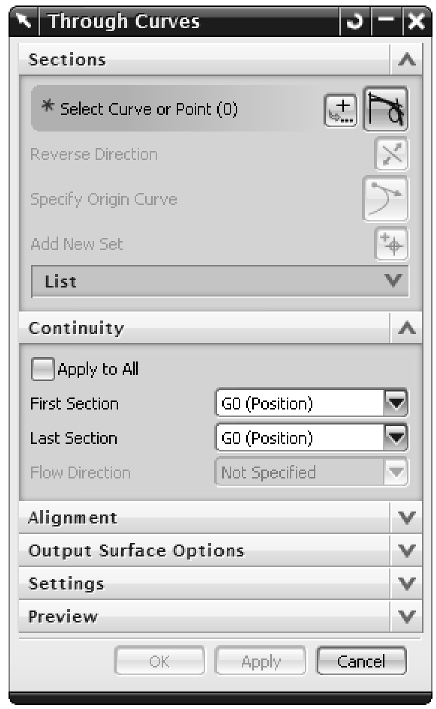


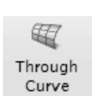
*Figure 11-9 The **Through Curves** dialog box*

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

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

Creating a Surface Using the Through Curve Mesh Tool

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



You can create surfaces by specifying the section strings and the guide strings using the **Through Curve Mesh** tool. You can specify any number of section strings and

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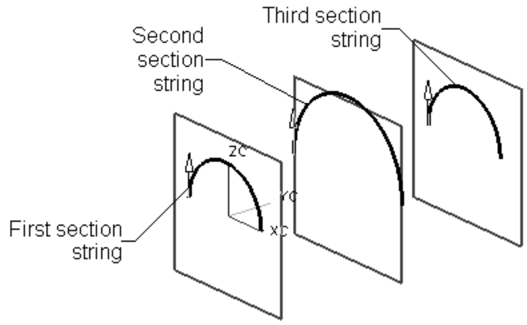

Figure 11-10 *Section strings selected for creating the* ***Through Curves*** *surface*

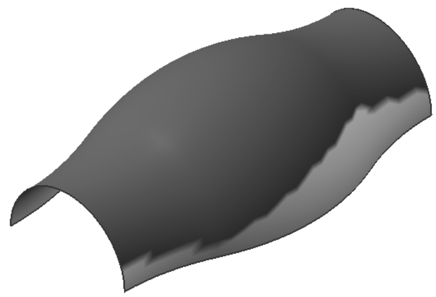

Figure 11-11 *The resulting surface*

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

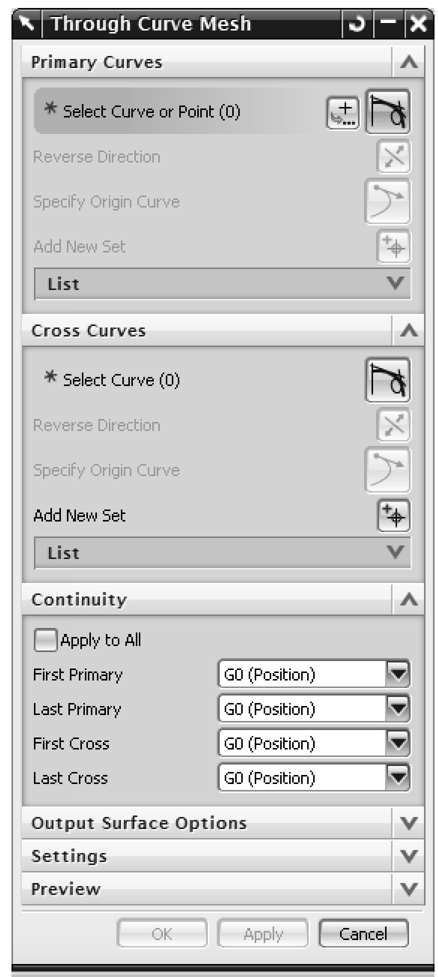

Figure 11-12 *The* ***Through Curve Mesh*** *dialog box*

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

Output Surface Options Rollout

The options from the **Emphasis** drop-down list in the **Output Surface Options** rollout allow you to define the set of curves that effect the shape of the surface to be created. Select the **Both** option from the **Emphasis** drop-down list, the primary curves and cross curves will have an equal effect. If you select the

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Normal option from the **Construction** drop-down list in the **Output Surface Options** rollout, the resulting surfaces will have more number of patches. If you select the **Spline Points** option, the resulting surface will have less number of patches. The surface is formed by reparameterizing curves into temporary curves.

Settings Rollout

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

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

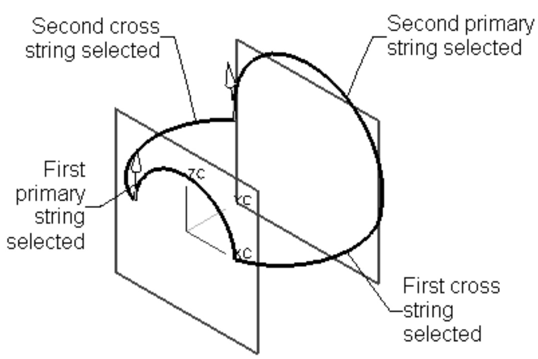

***Figure 11-13** The control strings selected for creating the through curve mesh surface*

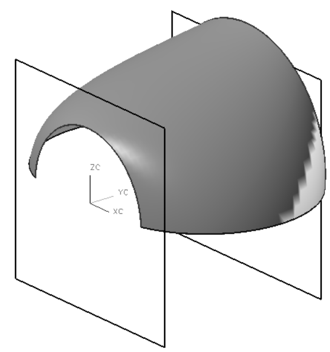

***Figure 11-14** The resulting through curve mesh surface*

Creating a Surface using the Four Point Surface tool

Menu:	Insert > Surface > Four Point Surface
Toolbar:	Freeform Shape > Four Point Surface

The **Four Point Surface** tool is used to create a planar (2D) or non-planar (3D) surface. To create a surface by using this method, choose the **Four Point Surface** button from the **Freeform Shape** toolbar; the **Four Point Surface** dialog box will be displayed, as shown in Figure 11-15, and you will be prompted to specify the point. Specify the point for the first surface corner. Similarly specify the other three surface corners and choose the **OK** button; the four point surface will be created. You can also redefine the previously selected corner points. To do so, choose the button corresponding to the point to which you want to redefine from the **Surface Corners** rollout; the respective point will be highlighted in the graphic window. Again, specify the point for

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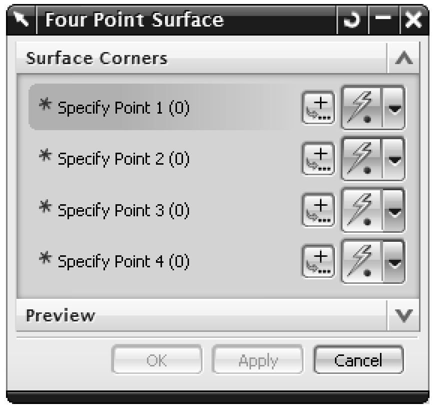

***Figure 11-15** The **Four Point Surface** dialog box*

the corner. Figure 11-16 shows the corner points to be selected for creating the surface. Figure 11-17 shows the resulting surface formed by enclosing the specified corner points.

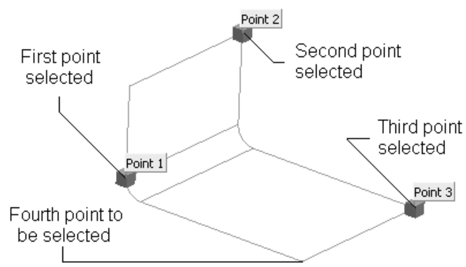

***Figure 11-16** The four corner points to be selected for creating a surface*

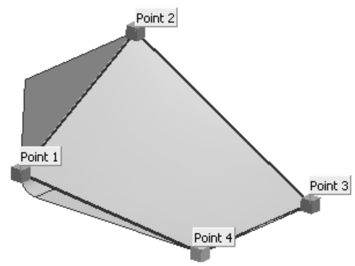

***Figure 11-17** The resulting surface*

Creating a Swoop Surface

Menu:	Insert > Surface > Swoop
Toolbar:	Freeform Shape > Swoop

The swoop surfaces are created as rectangular or square shaped planar (2D) surfaces, and later modified into 3D surfaces by using the options in the swoop tool. To create a swoop surface, choose the **Swoop** button from the **Freeform Shape** toolbar; the **Point** dialog box will be displayed and you will be prompted to select the point. Specify the point for the first corner of the rectangle; you will be prompted to select the point. Specify the second corner of the rectangle; the planar surface will be created. The vertical and horizontal axes will be displayed over the planar surface. Also, the **Swoop Shape Control** dialog box will be displayed, as shown in Figure 11-18. The **Swoop Shape Control** dialog box is 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

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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. You can use the 3 degree splines to form a surface by selecting the **Cubic** radio button from the **Degree** area. The use of 3 degree splines is convenient while transferring surface data from one CAD package to the other. You can select the **Quintic** radio button to make the resulting surface comparatively smoother.

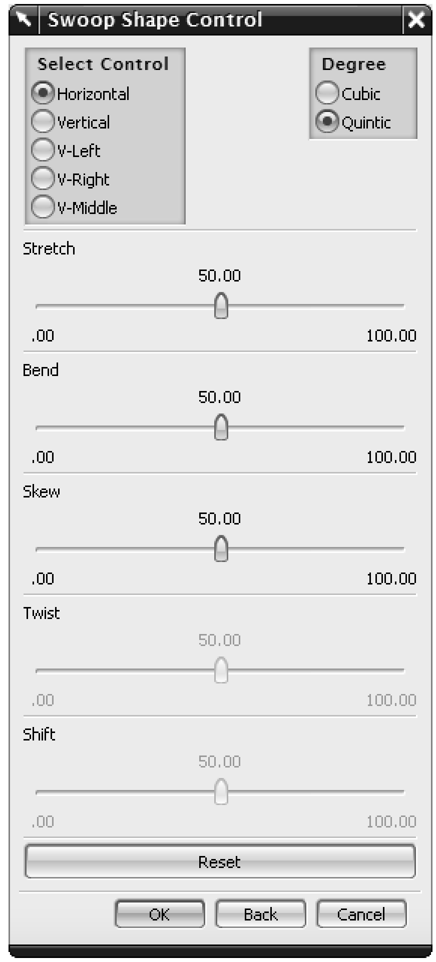

***Figure 11-18** The **Swoop Shape Control** dialog box*

Sliding Bars

Using the **Stretch** slider 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 sliders. Using the **Bend** slider 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** slider 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** slider bar, you can provide a twisting effect to the surface in the positive or negative direction along the reference position selected from the **Select Control** area. Using the **Shift** slider 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 11-19 shows the planar surface created after specifying both the corners of the rectangle. Figure 11-20 shows the 3D surface modified from the planar surface using the shape modification sliding bars.

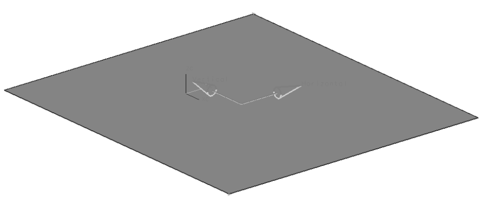

***Figure 11-19** The planar surface created after specifying both corners of the rectangle*

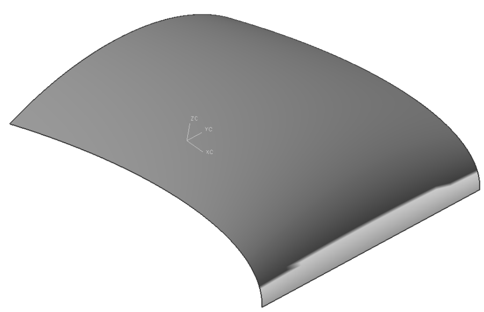

***Figure 11-20** The resulting 3D surface modified from the planar surface created*

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Creating Planar Surfaces from 2D Sketches and Edges of Solid

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

The **Bounded Plane** tool can be used to create a surface from the 2D sketches or the closed coplanar edges. If you need to enclose a 2D sketch or a closed coplanar edges with a surface, choose the **Bounded Plane** button from the **Feature** toolbar; the **Bounded Plane** dialog box will be displayed, as shown in Figure 11-21 and you will be prompted to select curves for bounded plane. You can select the solid face, solid edge, or closed coplanar sketch and choose the **OK** button; the **Bounded Plane** surface will be created. Figure 11-22 shows the bounded plane surface enclosing a 2D sketch and Figure 11-23 shows the bounded plane surface created from a circular edge. You can also create the bounded plane surface by selecting a solid face. You can only select 2D faces. The resulting surface, after selecting the solid face, will remain on the same surface.

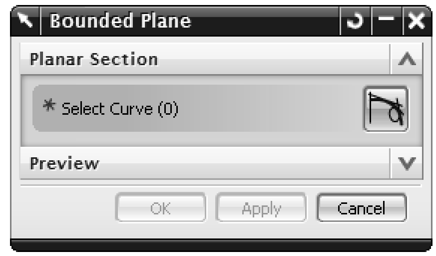

*Figure 11-21 The **Bounded Plane** dialog box*

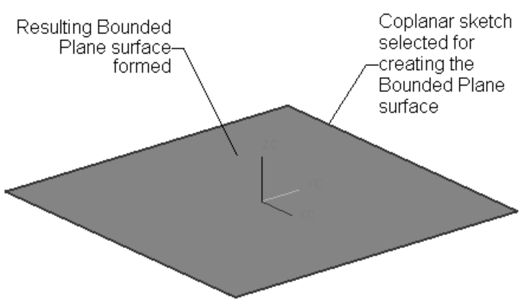

Figure 11-22 The bounded plane surface formed from a 2D sketch

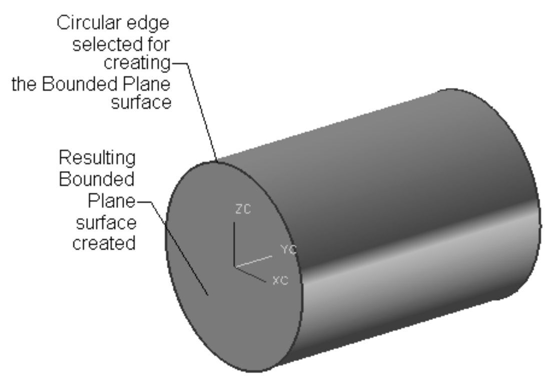

Figure 11-23 The bounded plane surface formed from a circular edge

Creating a Transition Surface Using the Transition Tool

Toolbar:	Surface > Transition

Generally, creation of a transition surface involves with the selection of required cross-sections and mapping the intersected surface formed between the selected cross-sections automatically. You can also define the shape constraint for the connecting (intersecting) surface. To create a **Transition** surface, you need to create more than two cross-sections. Generally, three cross-sections are created. After creating cross-sections, choose the **Transition** button from the **Surface** toolbar; the **Transition** dialog box will be displayed, as shown in Figure 11-24 and you will be prompted to select curves/ edges to section. The **Sections** button in the **Sections** rollout is chosen by default. As a result,

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you will be prompted to select curves to section. Select sections and choose the **OK** button. Note that you need to press the middle mouse button after selecting sections. After selecting sections, wireframe preview of the resultant model will be displayed. Figure 11-25 shows the wireframe view of the resultant model and Figure 11-26 shows the resulting surface.

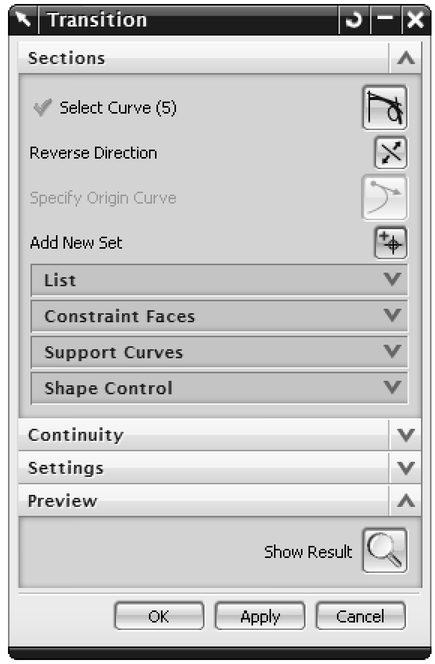

***Figure 11-24** The **Transition** dialog box*

Constraint Faces sub-rollout, and Continuity and Preview rollouts

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

Support Curves

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

Shape Control

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

Depth and **Skew** slider bars will be available in this sub-rollout to control the depth and the skew angle of the selected bridge.

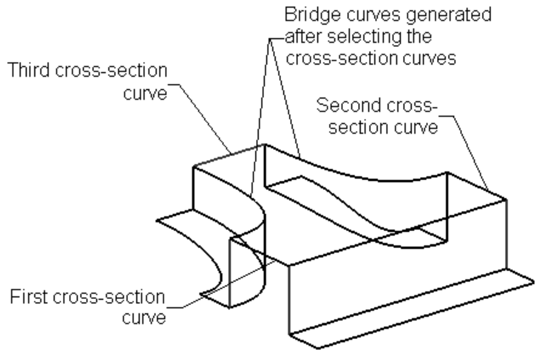

***Figure 11-25** The bridge curves generated after selecting cross-section curves*

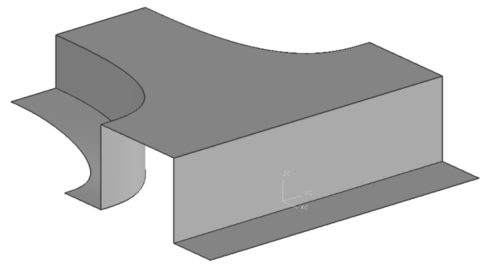

***Figure 11-26** The resulting transition surface created from cross-sections*

Creating a 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 multi-patches triangular surfaces that enclose a closed 2D sketch or a closed 3D curve. While doing so, an existing surface can be optionally selected as the reference for maintaining the shape of the surface to be created. For creating the N-sided surface, choose the **N-sided Surface** button from the **Surface** toolbar; the **N-sided Surface** dialog box will be displayed, as shown in Figure 11-27, and you will be prompted to select a closed loop of curves or edges. By default, the **Trimmed** option will be chosen from the **Type** drop-down list. This selection will create a surface with a single patch. To create a surface with multiple triangular patches, you need to choose the **Triangular** button from the **Type** drop-down list. Both options for creating the N-sided surface are discussed next.

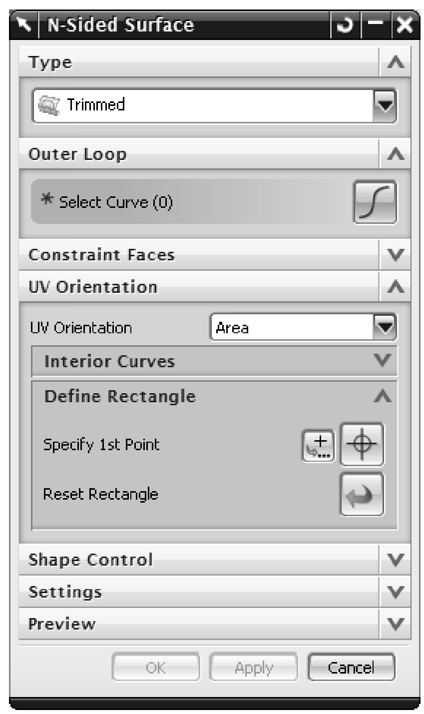

***Figure 11-27** The **N-Sided Surface** dialog box*

Trimmed

By default, the **Curve** button is chosen in the **Outer Loop** rollout. As a result, you will be prompted to select a closed loop. Select a closed loop. After selecting a closed boundary of a 2D sketch or a 3D curve;

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the preview of the new surface will be displayed. Next, select the **Trim to Boundary** check box from the **Settings** rollout; the surface created will automatically be trimmed with respect to closed loop of the curve, as shown in Figure 11-28.

By default, the **Area** option is selected from the **UV Orientation** drop-down list in the **UV Orientation** rollout. Choose the **Select Curve/Edge** option from the **Interior Curves** sub-rollout; you will be prompted to select the curve. Select the curve; the surface will be modified such that it passes through the selected interior curves; thereby, deforming the shape of the surface accordingly, as shown in Figure 11-29. For better understanding of the deformation of the surface, the **Trim to Boundary** check box is kept clear in this figure. You can also define a rectangle by specifying two points as diagonally opposite corners of the rectangle so that the resultant surface is created in the specified rectangle. To do so, choose the **Specify 1st Point** option from the **Define Rectangle** sub-rollout and click in the graphic window; a rectangle will be attached to the cursor. Also, the **Specify 2nd Point** option will be activated in the **Define Rectangle** sub-rollout. Again, click in the graphic window to specify the second point of the rectangle; a square surface will be created in the graphic window. To reset the created rectangle, choose the **Reset Rectangle** button from the **Define Rectangle** sub-rollout.

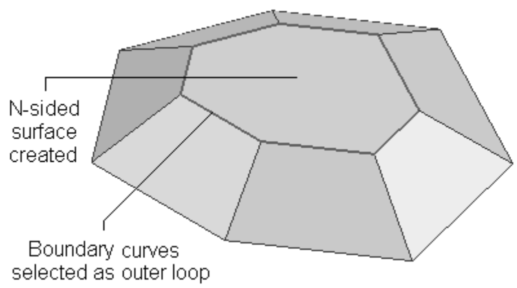

Figure 11-28 Single patch N-sided surface created for the selected boundary curve

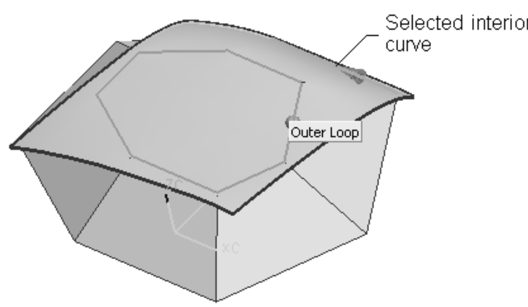

Figure 11-29 Modified part with interior curve selected

If you select the **Spline** option from the **UV Orientation** drop-down list in the **UV Orientation** rollout, then both the **UV Orientation** rollout and the **Shape Control** rollout will be modified. Choose the **Select Curve** option from the **Spine** sub-rollout; you will be prompted to select a curve/edge. Select a curve. The **Center Flat** slider bar in the **Central Control** sub-rollout of the **Shape Control** rollout is used to modify the shape of the surface created with respect to the selected curve, as shown in Figure 11-30. To reset the options in the **Shape Control** rollout, choose the **Reset** button from this rollout.

If you select the **Vector** option from the **UV Orientation** drop-down list in the **UV Orientation** rollout; the **UV Orientation** rollout will be modified. Choose the **Specify Vector** option from the **Vector** sub-rollout; you will be prompted to select the object to infer vector. Select a vector. The **Center Flat** slider bar in the **Central Control** sub-rollout of the **Shape Control** rollout is used to modify the shape of the surface created with respect to the specified vector, as shown in Figure 11-31. To reset the options in the **Shape Control** rollout, choose the **Reset** button from the **Shape Control** rollout.

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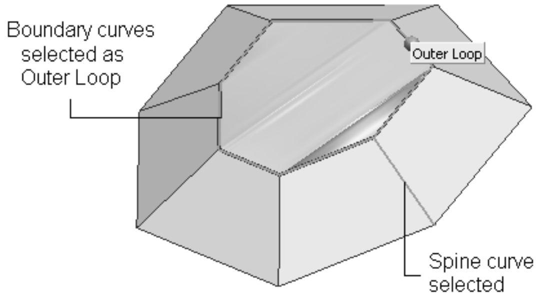

Figure 11-30 Modified part with the spine curve selected

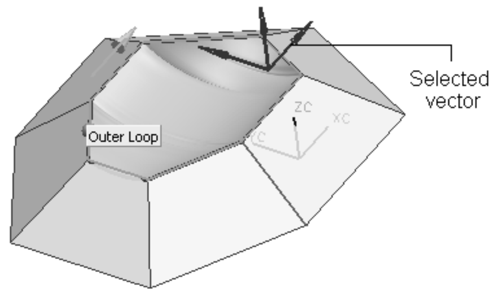

Figure 11-31 Modified part with the selected vector

Triangular

To create a triangular patch surface, select the **Triangular** option from the **Type** drop-down list in the **Type** rollout; you will be prompted to select a curve/edge. Select the closed entity; the preview of the selected surface will be displayed in the graphic window, as shown in Figure 11-32 and the **Shape Control** and **Settings** rollout will be modified.

By default, the **Position** option is selected in the **Control** drop-down list of the **Center Control** sub-rollout in the **Shape Control** rollout. You can move the center point of the new surface in the X, Y, and Z directions by using the **X**, **Y**, and **Z** slider bars, respectively, as shown in Figure 11-33. You can specify the flow direction of the new surface as per your requirement by selecting any one of the following options from the **Flow direction** drop-down list: **Not Specified**, **Perpendicular**, **Iso U/V Line**, or **Adjacent Edge**.

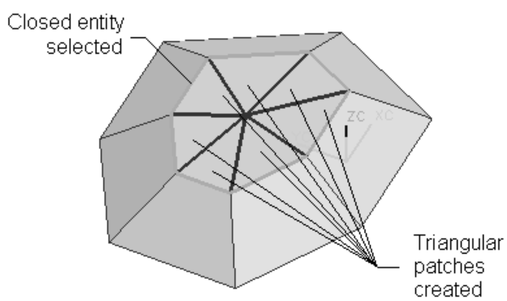

*Figure 11-32 Triangular patches created using the **Triangular** option*

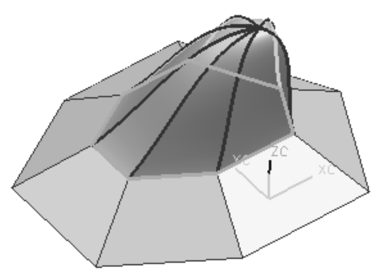

Figure 11-33 Modified surface in the X, Y, and Z directions

Select the **Tilting** option from the **Control** drop-down list and then use the **X** and **Y** slider bar to tilt the created surface in the X and Y directions, respectively, as shown in Figure 11-34. By default, the **Merge Faces if Possible** check box in the **Settings** rollout is clear. As a result, patches are created for each edge of the loop. If you select this check box, then the patches of the loop will be removed by treating the tangent continuous edges as single loop, as shown in Figure 11-35.

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Figure 11-34 *The tilted surface in the X and Y directions*

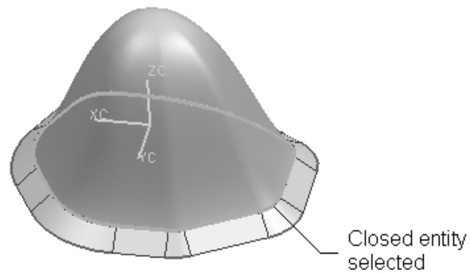

Figure 11-35 *The merged curves of the loop by selecting the **Merge Faces if Possible** check box*

Creating a Silhouette Flange Surface

Menu:	Insert > Flange Surface > Silhouette Flange
Toolbar:	Surface > Silhouette Flange

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

Creating a Silhouette Flange Surface Using the Basic Method

The **Silhouette Flange** tool is used to create silhouette flange surfaces on an edge or a curve by taking any of the adjacent surfaces as reference. To do so, invoke the **Silhouette Flange** tool from the **Surface** toolbar; the **Silhouette Flange** dialog box will be displayed, as shown in Figure 11-36 and you will be prompted to select curve or edges. Select the curve or edge on which you want to create a flange. By default, the **Basic** option is selected from the **Type** drop-down list and the **Curve** button is chosen from the **Base Curve** rollout. Selecting the **Basic** option enables you to create flange without the help of any other existing flange surfaces. Select the edge or a curve for creating the silhouette flange surface and choose the **Face** button from the **Base Face** rollout; you will be prompted to select the face that will act as base face. Select the desired face. The other options in this dialog box are discussed next.

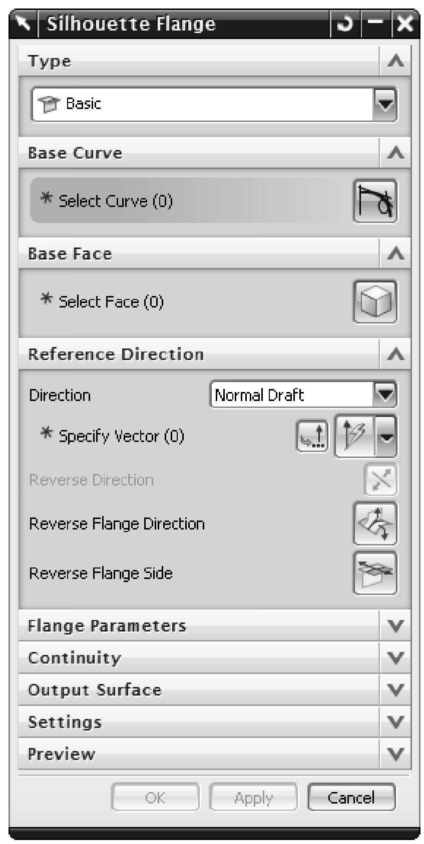

Figure 11-36 *The **Silhouette Flange** dialog box*

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Reference Direction Rollout

In this rollout, you can specify the direction of a flange by selecting any one of the options from the **Direction** drop-down list. These options are **Face Normal**, **Vector**, **Normal Draft**, and **Vector Draft**. The **Normal draft** option is selected by default in the **Direction** drop-down list. Select the **Specify Vecto**r option from this rollout and select the vector; the preview of the flange surface will be displayed, as shown in Figure 11-37. To change the direction of the flange to opposite direction, choose the **Reverse Flange Direction** button from this rollout; the direction of the flange will be reversed. To switch the flange extension to the opposite side of the bend, select the **Reverse Flange Side** button from this rollout. If you do not get the desired result after choosing this button then choose the **Reverse Direction** button from this rollout.

Flange Parameters Rollout

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

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

You can change the transition type of the bend radius of the flange to modify the bend radius by selecting the options **(Constant**, **Linear**, **Bend**, and **Minimum/Maximum)** from in the **Values Along Spine** sub-rollout of the **Flange Parameters** rollout. You can change the transition type of the length of the flange to modify the length by selecting the options **(Constant**, **Linear**, **Bend**, and **Minimum/Maximum)** from in the **Length** sub-rollout, refer to Figure 11-38.

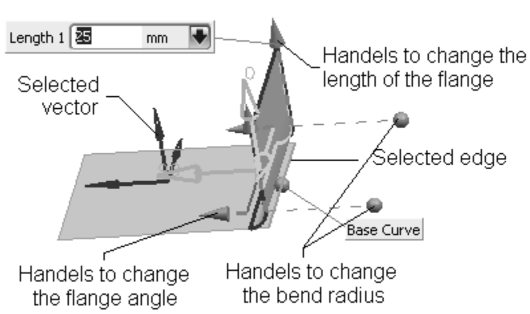

Figure 11-37 *Preview of the component*

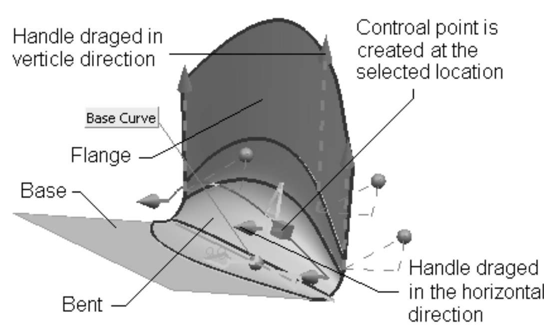

Figure 11-38 *Part modified using the* ***Flange Parameters and Continuity*** *rollout*

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Continuity Rollout

You can control the continuity between the base and the bent portion using the options in the **Base and Pipe** sub-rollout of the **Continuity** rollout. To do so, select the required G1, G2, and G3 continuities in the **Continuity** drop-down list of the **Base and Pipe** sub-rollout. To control the amount of edge shift, you can use the **Lead-in** slider bar. Alternatively, you can use the **Lead-in** edit box to control the edge shift. Similarly, you can control the continuity between the flange and the bent portion in the **Flange and Pipe** sub-rollout of this rollout, refer to Figure 11- 38.

Output Surface Rollout

In this rollout, the **Bend and Flange** option is selected by default in the **Output Options** drop-down list. If you select the **Pipe Only** option in this drop-down list, then only the pipe will be created. If you select the **Flange Only** option in the **Output Options** drop-down lis, then only the flange will be created. By default, the **Trim Base Faces** check box is clear in this rollout. As a result, the portion extended beyond the flange will be retained, as shown in Figure 11-39. If you want to remove the unwanted portion of the flange, select the **Trim Base Faces** check box in this rollout. The **Extended Flange** check box will be available only when the **Trim Base Faces** check box in kept clear. If you select this check box, the flange will be extended to cover the entire span of the base surface.

Settings Rollout

In this rollout, the **Create Curves** check box is clear. If you select this check box; two curves will be created along the center of the bend radius and at the intersection of the bend and flange. If you select the **Show Pipe** check box in this rollout; the pipe of the bend radius will be displayed only in the preview, as shown in Figure 11-40.

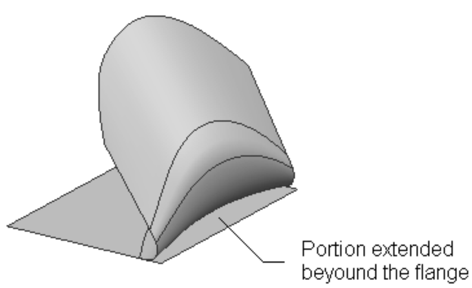

Figure 11-39 *Unwanted extended surface*

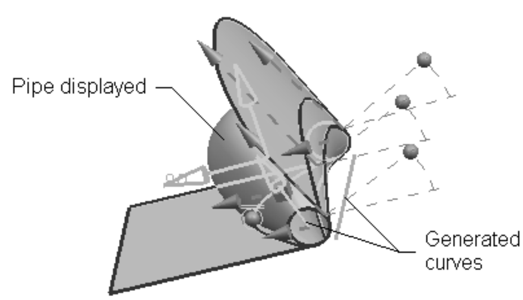

Figure 11-40 *Pipe of the bend radius displayed in the preview*

Creating a Silhouette Flange Surface Using the Absolute Gap Method

By selecting the **Absolute Gap** option from the **Type** drop-down list in the **Type** rollout, 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 center line of bend radius of the two pipes and the nearest tangential distance between them. You can also maintain a predefined gap between two silhouette flange surfaces by entering the gap value in the **Gap** edit box.

If you select the **Absolute Gap** option from the **Type** drop-down list in the **Type** rollout; the **Base Feature** rollout will be displayed. By default, the **Base Face** option is chosen in the **Base Face** rollout. As a result, you will be prompted to select the faces to define the base face. Select the reference face and choose the **Select Feature** option in the **Select Feature** rollout; you will be prompted to select the silhouette flange to define the base flange. Select the existing flange. Next, specify the reference direction in the **Reference Direction** rollout. To do so, choose the **Reverse Flange Side** button in the **Reference Direction** rollout; the preview of the resultant component will be displayed, as shown in Figure 11-41. To change the gap between the created flange and the existing selected flange, you can enter the required value in the **Gap** edit box, which is available at the bottom of the **Flange Parameters** rollout.

Creating a Silhouette Flange Surface Using the Visual Gap Method

The **Visual Gap** option from the **Type** drop-down list in the **Type** rollout is used to create a flange surface in accordance with an existing flange surface by specifying a visual gap attribute between the two flange surfaces. To create the silhouette flange surface using the visual gap method, 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 to create the surface. Figure 11-42 shows the silhouette flange created using the **Visual Gap** method.

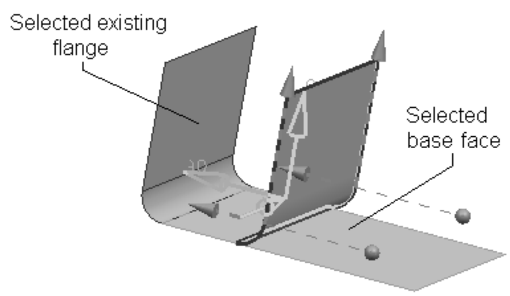

Figure 11-41 *The newly created silhouette flange surface displayed along with handles and pipe*

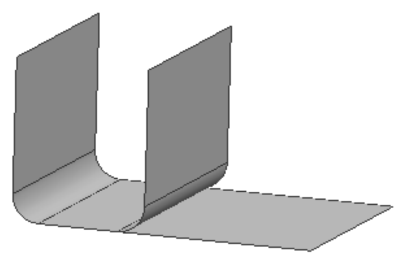

Figure 11-42 *The resulting silhouette flange surface created using the **Visual Gap** method*

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 are discussed next.

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Extending a Surface Dynamically Using the Faces Option

As discussed earlier, you can also extend a surface dynamically by choosing the **Law Extension** button from the **Surface** toolbar. On doing so, the **Law Extension** dialog box will be displayed, as shown in Figure 11-43 and you will be prompted to select the base curve profile. By default, the **Faces** option is selected from the **Type** drop-down list in the **Type** rollout. Using this method, you can extend the surface by taking an existing face as reference. Select the curve string that you want to extend from the surface and choose the **Face** button from the **Reference Faces** rollout; you will be prompted to select reference faces. Select the required face as the reference face and then choose the **Specify New Location** option from the **Length Law** rollout; the preview of the surface will be displayed, as shown in Figure 11-44.

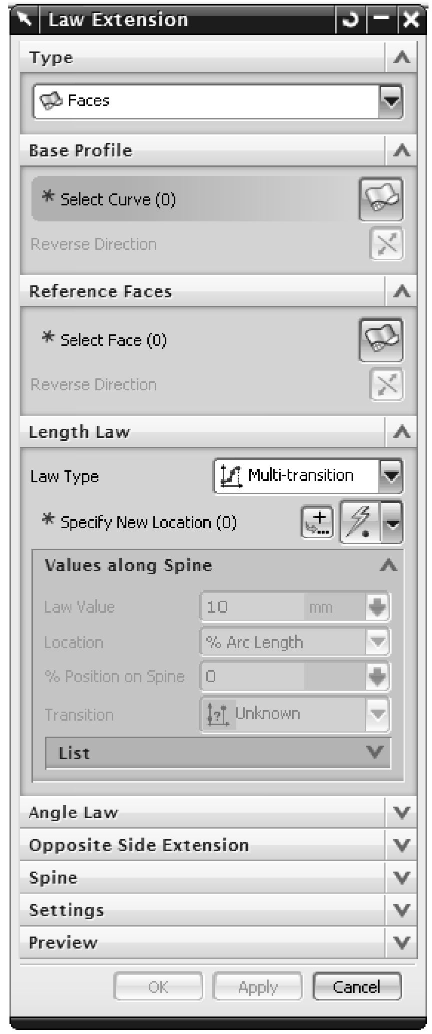

***Figure 11-43** The **Law Extension** dialog box*

Length Law and Angle Law Rollouts

The options in the **Length Law** and **Angle Law** rollouts are same with the only difference that the length law is applicable for the length of the flange, where as the angle law is applicable for the angle of the flange. If you select the point on the selected curve for the **Specify New Location** option, then a new control point will be displayed at that point. By default, the **Multi-transition** option is selected in the **Law Type** drop-down list in the **Length Law** as well as the **Angle Law** rollout. As a result, you can change the length or angle of the flange regardless to the other control points. However, you can change the length and angle of the flange by applying other laws such as **Constant**, **Linear**, **Cubic**, **Linear along Spine**, **Cubic along Spine**, **By equation**, and **By Law Curve**. For example, select the **Linear** option from the **Law Type** drop-down list; the **Start** and **End** edit boxes will be displayed in this rollout. Enter the start and end values in the **Start** and **End** edit boxes, respectively, and then choose the **OK** button; the modified extended surface will be displayed.

Opposite Side Extension Rollout

By default, the **None** option is selected in the **Extension Type** drop-down list. If you select the **Symmetric** option from the **Law Type** drop-down list; a symmetric flange will be created on the opposite side of the created flange. If you select the **Asymmetric** option from the **Law Type** drop-down list; the **Length Law** and **Values along Spine** sub-rollouts will be displayed in this rollout. Also, the new flange will be created on the

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opposite side of the created flange. You can modify this new flange by using the **Length Law** and **Values along Spine** sub-rollouts.

Spine Rollout

This rollout is used to select the curve using the **Curve** button. Select the curve, as shown in Figure 11-45. The imaginary plane will be placed perpendicular to the selected curve, with respect to which the angle of the flange will be measured, refer to Figure 11-45.

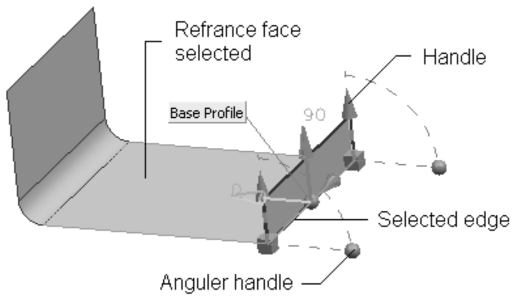

Figure 11-44 *Preview of the component*

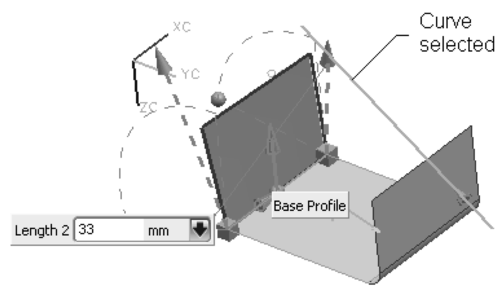

Figure 11-45 *Curve selected using the* ***Curve*** *button in the* ***Spline*** *rollout*

Settings Rollout

In this rollout, by default, the **Lock End Length/Angle Handles** check box is clear. As a result, you can move handles and angular handles regardless to each other, as shown in Figure 11-46. If you select this check box, the end handles of the profile will be locked. As a result, if you drag the handles at the start point, the handle at the end point will be modified simultaneously.

In the **Base Profile** area of the **Settings** rollout, you can redefine the degree and knot points of the curve already selected from **Base Profile** rollout. This helps in rebuilding the flange of desired shape and to maintain continuity between the reference faces and the law extension surfaces.

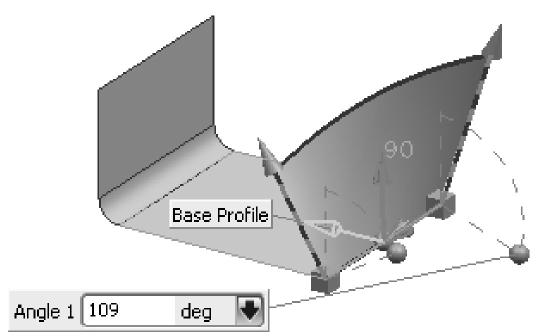

Figure 11-46 *End handles dragged regardless to each other*

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Extending a Surface Dynamically Using the Vector Option

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

Note

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

Creating a Surface Offset Using the Offset Surface Tool

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

The **Offset Surface** tool can be used to offset a surface in the direction normal to the selected surface. For offsetting a surface, choose the **Offset Surface** button from the **Surface** toolbar; the **Offset Surface** dialog box will be displayed, as shown in Figure 11-47. By default, the **Face** button is chosen from the **Face to Offset** rollout and you will be prompted to select the faces for the new set. Select the face, as shown in Figure 11-48. Next, enter the offset value in the **Offset 1** edit box. If you want create a new set, choose the **Add New Set** button from the **Face to Offset** rollout. Choose the **Reverse Direction** button to flip the offset direction. Choose the **OK** button; the resulting offset surface will be created as shown in Figure 11-48.

Figure 11-47 *The **Offset Surface** dialog box*

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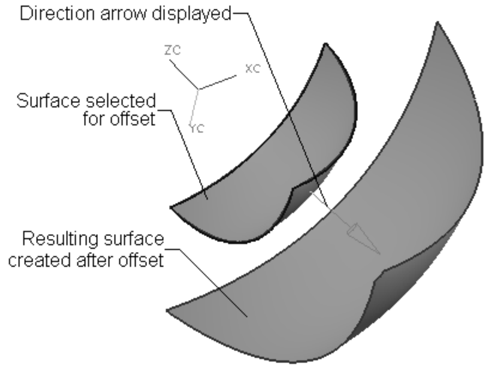

Figure 11-48 *The offset surface created*

Trimming and Extending a Surface Using the Trim and Extend Tool

Menu:	Insert > Trim > Trim and Extend
Toolbar:	Surface > Trim and Extend *(Customize to add)*

The **Trim and Extend** tool is used to trim or extend an open or closed surface. To trim or extend a surface, choose the **Trim and Extend** button from the **Surface** toolbar; the **Trim and Extend** dialog box will be displayed, as shown in Figure 11-49, and you will be prompted to select the target edge to extend. Select a single edge or multiple edges from the surface that is to be extended. When you select multiple edges for extending, ensure that the selected edges are in continuity. If the **Preview** check box is selected in the **Preview** rollout, the preview of the extended surface will be displayed. The different rollouts in the **Trim and Extend** dialog box are discussed next.

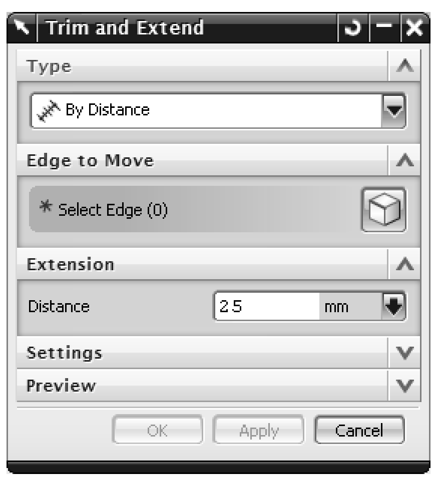

Figure 11-49 *The* ***Trim and Extend*** *dialog box*

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Type Rollout

On selecting the **By Distance** option from the drop-down list in the **Type** rollout, you will be able to define the length of the surface extension by specifying a distance value in the **Distance** edit box. If you select the **Percentage of Measured** option from this rollout, the extension length will be specified in terms of the percentage of the original surface. On selecting the **Until Selected** option, the surface will be extended up to the selected reference object. This option can also be used to trim the selected surface. If you select the **Make Corner** option, a corner will be created at the intersection of the extended surface with the tool body and the tool body will be trimmed.

Settings Rollout

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

Note that if you select the **Until Selected** option from the drop-down list in the **Type** rollout, you need to select the tool body that serves as the boundary object after selecting the edge for extension. Choose the **Tool** button from the **Tool** rollout and select the boundary object. Next, choose the **OK** button to extend the surface up to the selected boundary object. The options from the **Arrow Side** drop-down list in the **Desired Results** rollout are used to retain or discard the selected tool body. If you select the **Retain** option, the selected tool body will be retained after trimming. If you select the **Delete** option, then the material from the tool body will be removed in the direction of the arrow displayed on selecting the tool body. Figure 11-50 shows the preview of the extension surface after selecting the edges. Figure 11-51 shows the surface extended using the **Until Selected** option.

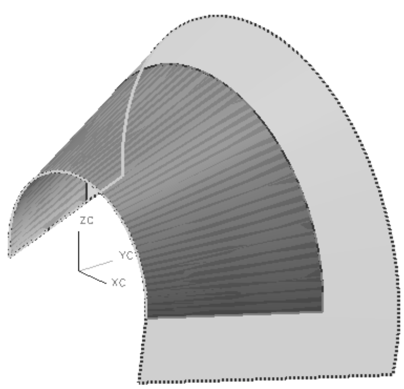

***Figure 11-50** The preview of the extended surface after selecting edges*

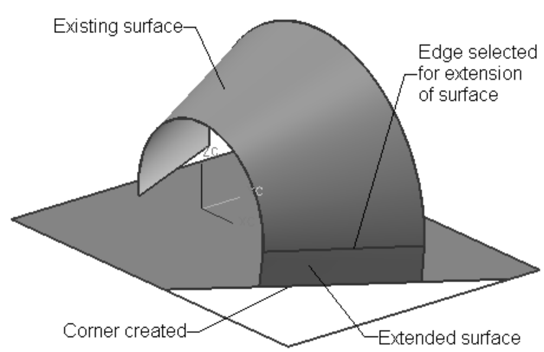

***Figure 11-51** The resulting surface extended using the **Until Selected** option*

Trimming a Sheet by Using the Trimmed Sheet Tool

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

The **Trimmed Sheet** tool can be used to trim a sheet by defining the trim boundary. You can also trim a sheet by projecting a curve and then defining it as trim boundary. If the trim boundary is a surface, then the surface to be trimmed must intersect fully with the trimming surface. Choose the **Trimmed Sheet** button from the **Surface** toolbar; the **Trimmed Sheet** dialog box will be displayed, as shown in Figure 11-52, and you will be prompted to select a target sheet body. By default, the **Sheet Body** button will be chosen from the **Target** rollout. Select the sheet to be trimmed and press the middle mouse button.

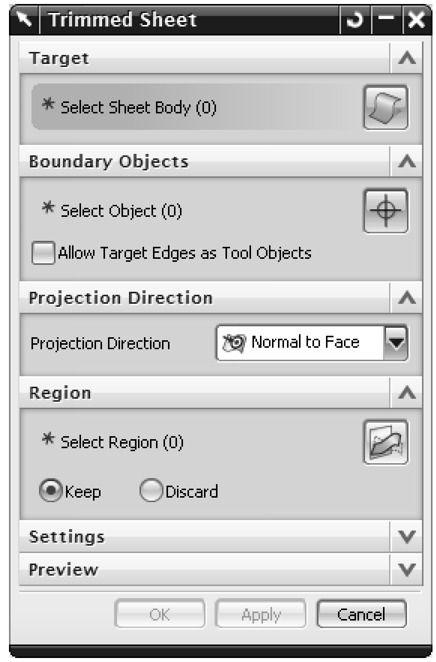

Figure 11-52 *The* ***Trimmed Sheet*** *dialog box*

Next, you will be prompted to select the boundary objects. Select the boundary objects. Choose the **Region** button; the surface to be trimmed will be divided into two colors. The pale green color indicates whether this region is to be kept or discarded. You need to select the **Keep** or **Discard** radio button in the **Region** rollout to specify whether the area highlighted in the pale green color has to be kept or discarded. If you select the **Keep** radio button from this rollout, the region in the pale green color will be retained, and the other region will be removed. If you select the **Discard** radio button, the region in the pale green color will be removed (trimmed) and the other region will be retained.

Figure 11-53 shows the entities selected for trimming a surface. Figure 11-54 shows the resulting trimmed surface after selecting the **Discard** radio button from the **Region** rollout.

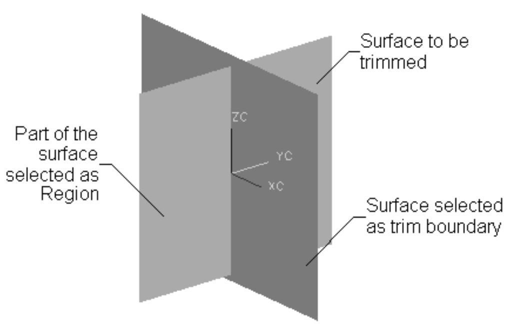

Figure 11-53 *Entities selected for trimming a sheet*

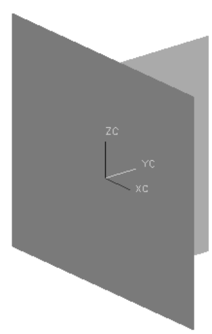

Figure 11-54 *The resulting trimmed sheet after selecting the* ***Discard*** *radio button*

The **Projection Direction** drop-down list contains the options for projecting (imprinting) a curve or a sketch on the surface to be trimmed. The projection curve or sketch can be defined as the trimming boundary. Select the surface to be trimmed and the curve or the sketch as

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the trim boundary. The selected curve or the sketch automatically gets imprinted on the surface to be trimmed and forms the trim boundary. The curve projected as the trim boundary should intersect the surface to be trimmed. Figure 11-55 shows the objects selected when the trim boundary is created by imprinting a curve on the surface to be trimmed. Figure 11-56 shows the resulting trimmed surface after selecting the **Discard** radio button from the **Region** rollout.

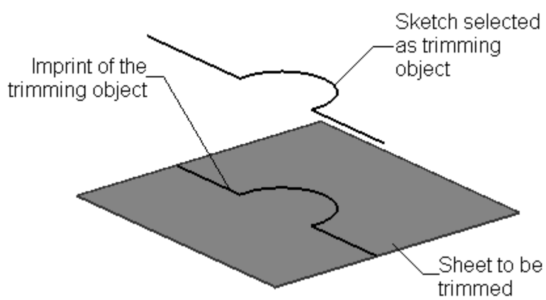


Figure 11-55 *Sketch selected for trimming a sheet*

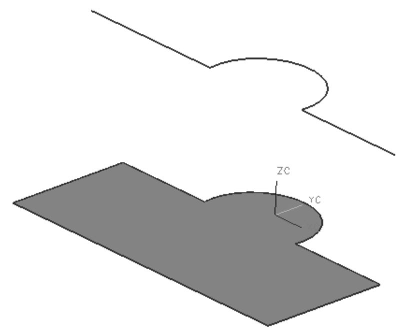
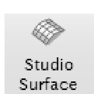


Figure 11-56 *The resulting trimmed sheet after selecting the **Discard** radio button*

Creating a Surface Using the Studio Surface Tool

Enhanced

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



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

Choose the **Studio Surface** button from the **Freeform Shape** toolbar; the **Studio Surface** dialog box will be displayed, as shown in Figure 11-57, and you will be prompted to select a section. By default, the **Section (Primary) Curves** button will be chosen from the **Section (Primary) Curves** rollout. Select the section curves one by one. After selecting one section curve, press the middle mouse button to continue selecting other section curves. Note that all section curves should point in one direction. After selecting the section curves, choose the **Guide (Cross) Curves** button from the **Guide (Cross) Curves** rollout; you will be prompted to select a guide curve. Select the guide curves one by one in the same way as you did for section curves. Note that all the guide curves should also point in one direction.

Other options in the dialog box have been discussed in earlier tools. After selecting all the parameters, choose the **OK** button; the surface will be created. Figure 11-58 shows the section and the guide curve selected for creating the studio surface. Figure 11-59 shows the preview of the resulting studio surface.

Figure 11-60 shows a single section and two guide curves selected and Figure 11-61 shows the resulting studio surface.

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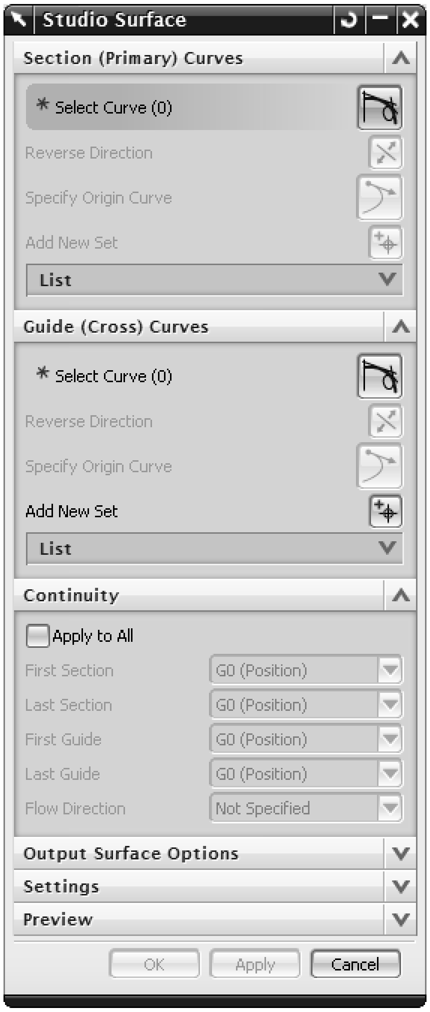

Figure 11-57 *The* ***Studio Surface*** *dialog box*

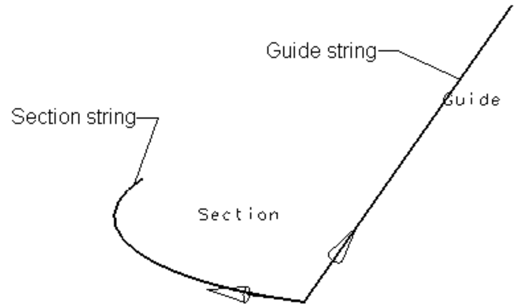

Figure 11-58 *The section string and the guide string selected for creating a studio surface*

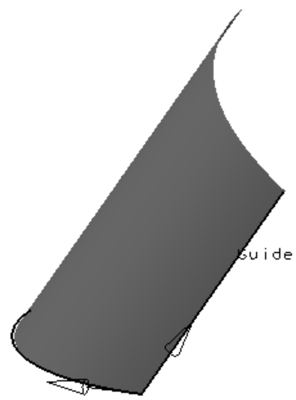

Figure 11-59 *The preview of the studio surface created using the* ***Studio Surface*** *tool*

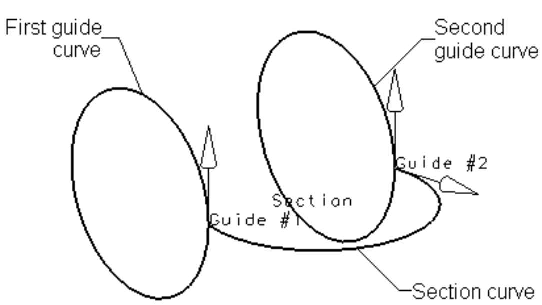

Figure 11-60 *The section curve and the guide curves selected for creating the studio surface*

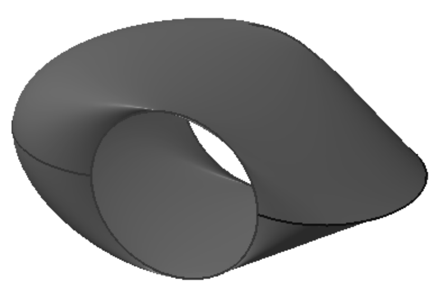

Figure 11-61 *The resulting studio surface*

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Figure 11-62 shows the selected start and end section curves and Figure 11-63 shows the resulting studio surface.

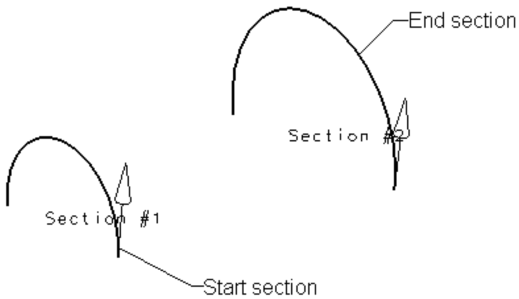

Figure 11-62 The section curves selected for creating a studio surface

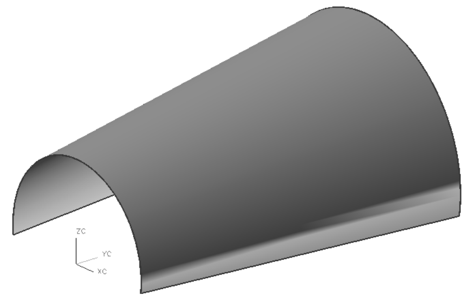

Figure 11-63 The resulting studio surface created using the ***Studio Surface*** *tool*

Figure 11-64 shows two section curves and two guide curves selected. Figure 11-65 shows the resulting studio surface.

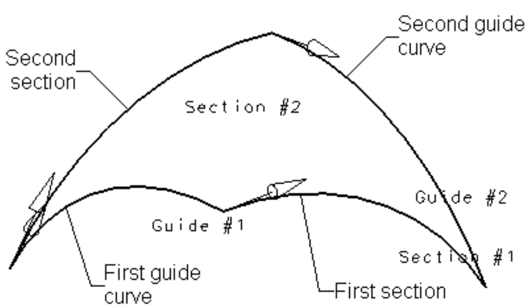

Figure 11-64 The section and guide curves selected for creating a studio surface

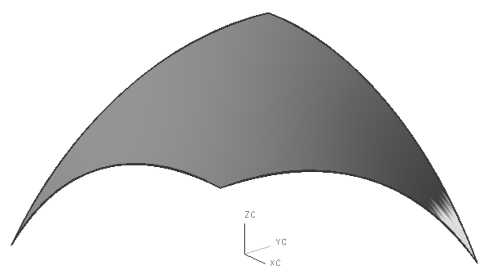

Figure 11-65 The resulting studio surface created using the ***Studio Surface*** *tool*

Figure 11-66 shows the selected section curves and guide curves. Figure 11-67 shows the resulting studio surface.

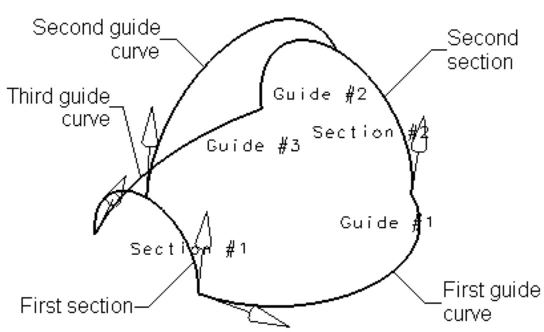

Figure 11-66 The section curves and the guide curves selected to create a studio surface

Figure 11-67 The resulting surface created using the ***Studio Surface*** *tool*

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Creating a Surface between Two Walls Using the Styled Blend Tool

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

The **Styled Blend** tool is used to create a fillet surface between two intersecting walls. While creating fillet surfaces, you can maintain a tangent or curvature continuity among the walls. You can also create variable fillets using this tool. To create a fillet surface using this tool, choose the **Styled Blend** button from the **Freeform Shape** toolbar; the **Styled Blend** dialog box will be displayed, as shown in Figure 11-68 and you will be prompted to select the faces for wall 1.

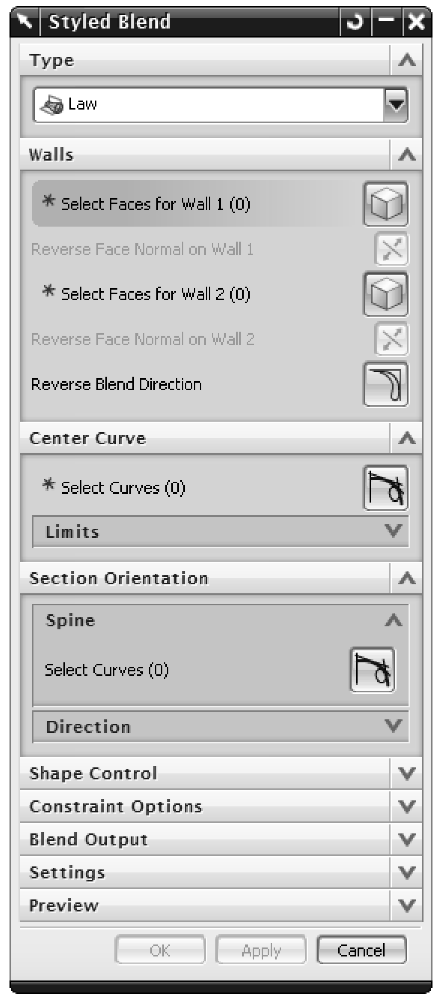

Figure 11-68 *The **Styled Blend** dialog box*

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

Creating a Styled Blend Surface Using the Law Option

By default, the **Law** option is selected from the **Type** drop-down list in the **Type** rollout and the **Select Faces for Wall 1** option is selected from the **Walls** rollout. Select the first wall and then choose the **Select Faces for Wall 2** option from the **Walls** rollout. While selecting both the walls, you need to ensure that the arrow displayed from the walls are facing in the inward direction where the surface is to be created. You can use the **Reverse Direction** button in the **Walls** rollout to flip the direction of the arrow. Select the second wall and press the middle mouse button; the preview of the fillet will be displayed along with the handles and dynamic edit box, as shown in Figure 11-69. Also, the **Specify New Location** option in the **Shape Control** rollout will be activated. You can view the alternate solutions of the displayed fillet by clicking on the **Reverse Blend Direction** button in the **Walls** rollout. If you click on the intersection of two walls, a new control point will be added to that location. You can change the radius of the fillet by dragging handles. By default, the **Single Tube**

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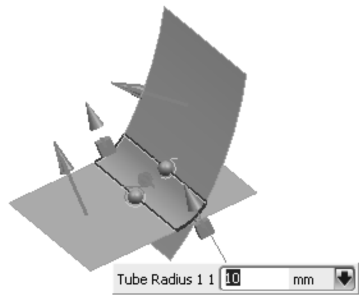

Figure 11-69 Preview of the fillet created between two walls

check box is selected in the **Shape Control** rollout, As a result, the parameters of the fillet change simultaneously for both the walls, as shown in Figure 11-70. If you clear this check box, the parameters of the fillet will change simultaneously only for the wall 1, as shown in Figure 11-71. Also, the **Tube Radius 2** option will be activated in the **Control Type** drop-down list. By default, the **Tube Radius 1** option is selected in the **Control Type** drop-down list. As a result, you can change the radius of fillet using the **Tube Radius** edit box in the **Values along Spline** sub-rollout.

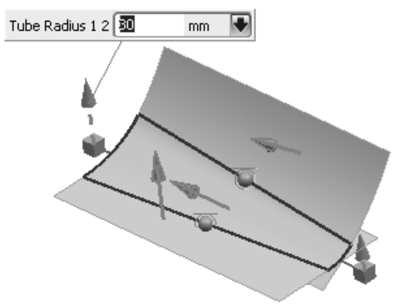

*Figure 11-70 Fillet created with the **Single Tube** check box selected*

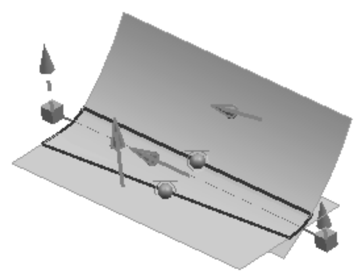

*Figure 11-71 Fillet created with **Single Tube** check box clear*

If you select the **Depth** option in the **Control Type** drop-down list; the **Depth 1** edit box and the **Depth 1** slider bar will be displayed. You can change the depth of the blend using this slider bar. If you select the **Skew** option in the **Control Type** drop-down list, then the **Skew 1** edit box and the **Skew 1** slider bar will be displayed. You can change the skew of the blend using this slider bar. If you select the **Tangent Magnitude** option in the **Control Type** drop-down list; the **Tangent Magnitude 1** edit box and the **Tangent Magnitude 1** slider bar will be displayed. You can change the tangent magnitude of the blend using this slider bar.

If you have already created a curve, as shown in Figure 11-72, then you can use that curve to act as a center of the blend. To do so, choose the **Curve** button from the **Center Curve** rollout and select the curve, the blend will be created using the selected curve as center, as shown in Figure 11-73. By default, line curve will be extended by 10 % of its original length at the start and end points. You can edit this value in the **Limits** sub-rollout.

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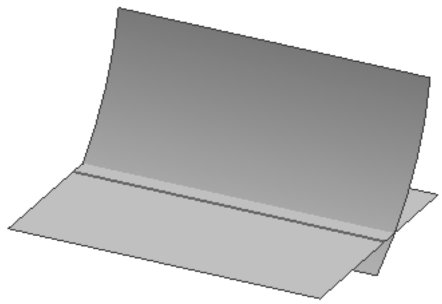

Figure 11-72 Existing curve to be selected for the ***Center Curve*** *option*

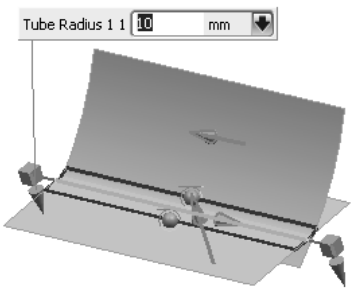

Figure 11-73 Preview of the resultant model

You can restrict the length of the blend using the **Section Orientation** rollout. To do so, select the **Curve** button from the **Spine** sub-rollout; you will be prompted to select the curves for spline. Select a curve, as shown in Figure 11-74; the preview of the resultant model will be displayed, as shown in Figure 11-75. By default, the **Use Center Curve As Spine** check box is clear. If you select this check box, the center curve will be used as spine curve.

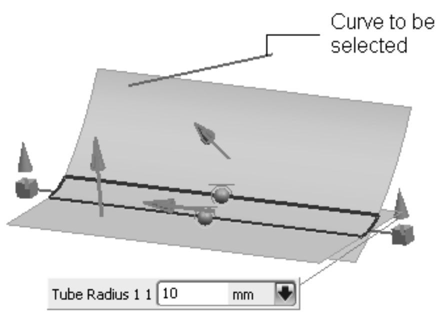

Figure 11-74 The curve to be used to restrict fillet

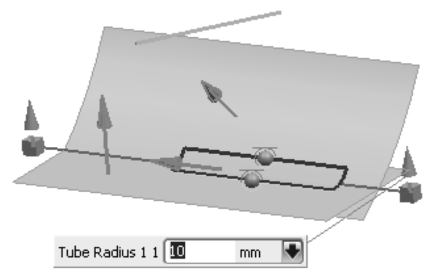

Figure 11-75 Preview of the resultant model

To activate the **Extend Blend** check box in the **Blend Output** rollout, select the **Use Center Curve As Spine** check box from the **Spine** sub-rollout in the **Section Orientation** rollout. If you select the **Extend Blend** check box, the fillet will be extended throughout the center line, as shown in Figure 11- 76. By default, the **No Trim** option is selected in the **Trimming Method** drop-down list of the **Blend Output** rollout, As a result, it keeps the extended portion after the blend in the resultant model. If you select the **Trim & Attach** option in this drop-down list, the extended portion of the wall after the blend and the extended portion of

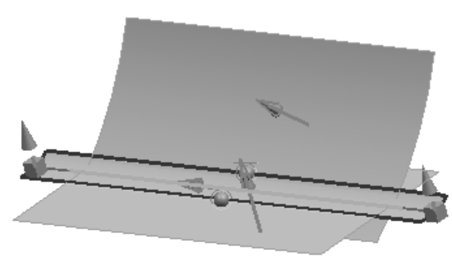

Figure 11-76 Preview of the ***Styled Blend*** *using the* ***Extended Blend*** *check box selected*

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the blended curve after boundary edges in the resultant model will be removed, as shown in Figure 11-77. If you select the **Trim Input Blend** option, only the extended portion of the wall after the blend will be removed, as shown in Figure 11- 78.

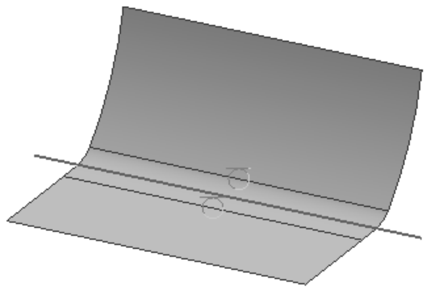

Figure 11-77 The styled blend created using the **Trim & Attach** option

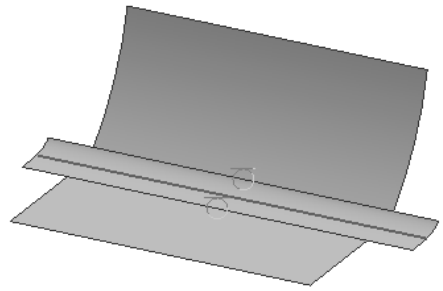

Figure 11-**78** The styled blend created using the **Trim Input Blend** option

By default, the **Show Blend** check box is selected in the **Show** sub-rollout of the **Settings** rollout. If you clear this check box, then the blend will not be displayed in the preview. If you select the **Show Tube** check box, the tube of specified radius will be displayed in the preview. If you select the **Show Labels** check box, labels will be displayed in the preview. If you select the **Show Depth Curve** check box, the depth of the curvature will be displayed in the preview. You can reset the options in the **Settings** rollout by choosing the **Reset All** button in the **Settings** rollout.

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

Creating a Styled Blend Surface Using the Curve Option

To create a styled blend surface using the **Curve** option, select the **Curve** option from the **Type** drop-down list in the **Type** rollout; you will be prompted to select the faces for wall 1. Select the first wall and choose the **Select Faces for Wall 2** option from the **Walls** rollout; you will be prompted to select the faces for wall 2. Select the second wall and press the middle mouse button. Next, select the **Select Curves For Curve Set 1** option in the **Tangent Curves** rollout; you will be prompted to select the curves for curve set 1. Select the first tangential curve from the first wall selected and choose the **Select Curves For Curve Set 2** option; you will be prompted to select the curves for curve set 2. Select the second tangential curve from the selected second wall, refer to Figure 11-79 for selection; the preview of the fillet will be displayed, as shown in Figure 11- 80.

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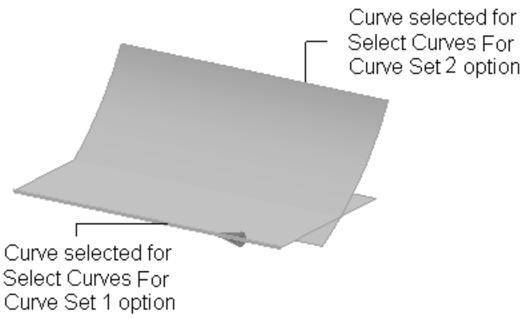

Figure 11-79 Reference selection for the ***Curve*** *option*

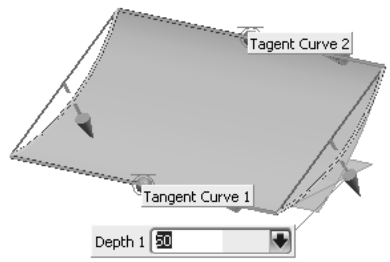

Figure 11-80 Preview of the fillet

Creating a Styled Blend Surface Using the Profile Option

To create a styled blend surface using the **Profile** option, select the **Profile** option from the **Type** drop-down list in the **Type** rollout; you will be prompted to select the faces for wall 1. Select the first wall and choose the **Select Faces for Wall 2** option from the **Walls** rollout. Next, select the second wall and press the middle mouse button; the **Select Curves** option in the **Profile** rollout will be activated. Also, you will be prompted to select the curves for profile curve. Select the curve, as shown in Figure 11-81; the preview of the fillet will be displayed, as shown in Figure 11-82. Remaining options are same as discussed earlier.

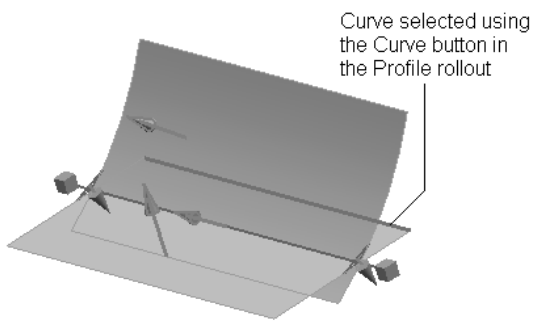

Figure 11-81 The curve selected for creating the ***Styled Blend*** *surface using the* ***Profile*** *option*

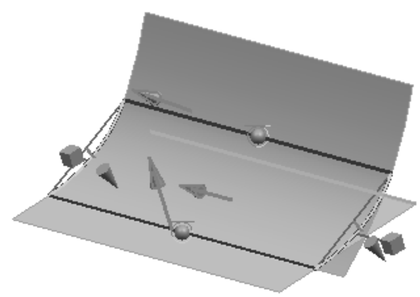

Figure 11-82 The preview of the ***Styled Blend*** *surface using the* ***Profile*** *option*

Creating Surfaces Using the Styled Sweep Tool

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

The **Styled Sweep** tool is used to create surfaces by sweeping cross-sections along one or two guide curves. The surface created using this tool can be modified dynamically by dragging the pivot point by displayed along with the surface. To create the styled sweep surface, invoke the **Styled Sweep** tool from the **Freeform**

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Shape toolbar; the **Styled Sweep** dialog box will be displayed, as shown in Figure 11-83.

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

1 Guide

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

1 Guide, 1 Touch

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

1 Guide, 1 Orientation

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

2 Guides

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

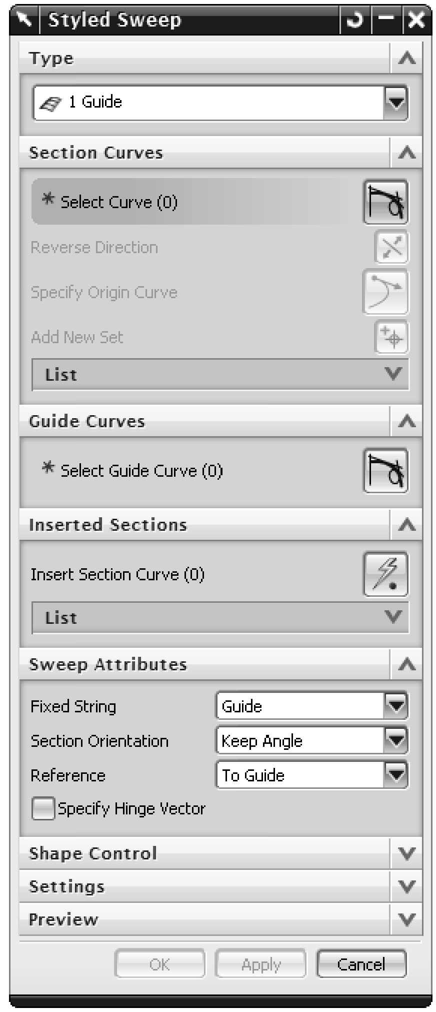

Figure 11-83 *The* ***Styled Sweep*** *dialog box*

Select the required option from the **Type** drop-down list in the **Type** rollout. Next, select the section and guide curves using the **Section Curves** and **Guide Curves** rollout, respectively. Note that you need to press the middle mouse button to add each section and guide string. Figure 11-84 shows the entities to be selected for creating the styled sweep surface using the **1 Guide, 1 Touch** option. After selecting the section and guide strings, the preview of the surface will be displayed along with the pivot point, as shown in Figure 11- 85.

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

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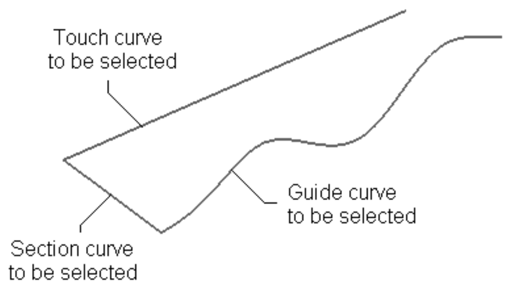


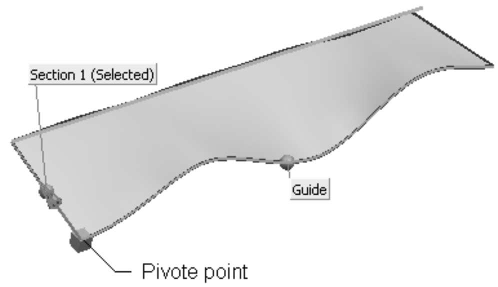
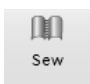


Figure 11-84 *Entities to be selected for creating the styled sweep surface*

Figure 11-85 *Preview of 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 individual surfaces into a single surface with a common edge. When the selected individual surface encloses a volume, a solid body is created. The sheet to which all the other individual sheets are to be stitched is known as the target sheet, and the individual sheets that are to be stitched are known as the tool sheets. You cannot stitch the tool sheets that intersect a target sheet and extend from it.

To stitch the individual surfaces to a single surface, choose the **Sew** button from the **Feature Operation** toolbar; the **Sew** dialog box will be displayed, as shown in Figure 11-86. By default, the **Sheet** option will be selected from the drop-down list in the **Type** rollout and you will be prompted to select the target sheet body. Select the target sheet body; the border of the target sheet will be displayed in cyan color and you will be prompted to select the tool sheet bodies to sew. The border of the selected tool sheets should lie within the cyan boundary. Otherwise, the selected tool sheet will not get stitched to the target sheet. Select the tool sheet bodies and choose the **OK** button; the sheet bodies will be stitched.

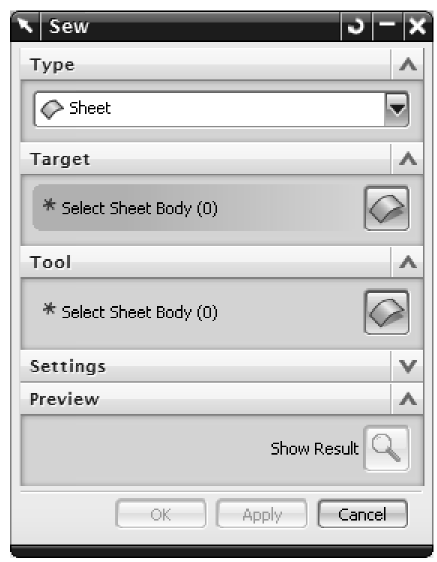


Figure 11-86 *The* ***Sew*** *dialog box*

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

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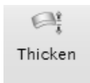
Adding Thickness to a Surface

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

Enhanced



This tool is used to add thickness to the sheet. Once you add thickness to the sheet, it is converted into a solid. To add thickness to the sheet, invoke the **Thicken** tool from the **Feature** toolbar; the **Thicken** dialog box will be displayed, as shown in Figure 11-87 and you will be prompted to select faces to thicken. Select the sheet bodies to which the material is to be added.

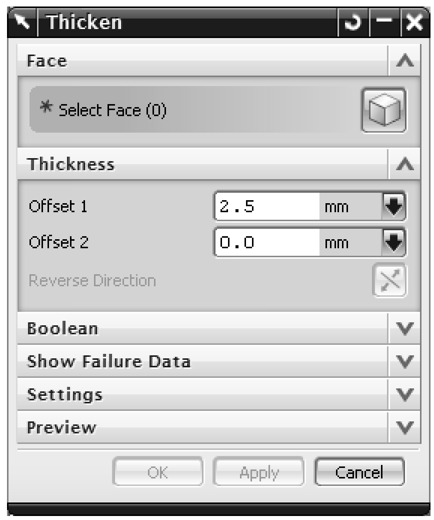


Figure 11-87 *The **Thicken** dialog box*

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

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

Figure 11-88 shows the sheet to add thickness and Figure 11-89 shows the resulting solid body.

Figure 11-88 *The sheet selected for adding material*

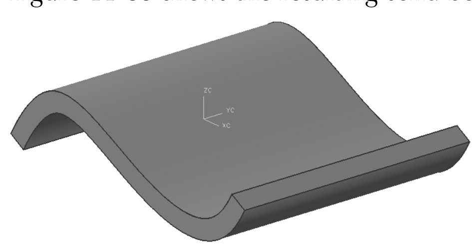


Figure 11-89 *The resulting solid body after adding thickness to the sheet*

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TUTORIALS

Tutorial 1

In this tutorial, you will create the surface model shown in Figure 11-90. The dimensions and orthographic views are shown in Figure 11-91. After creating the surface, save it in the name given below.

\NX 6\c11\c11tut1.prt **(Expected time: 30 min)**

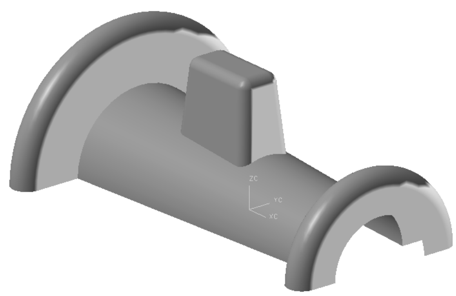

Figure 11-90 *The isometric view of the surface model*

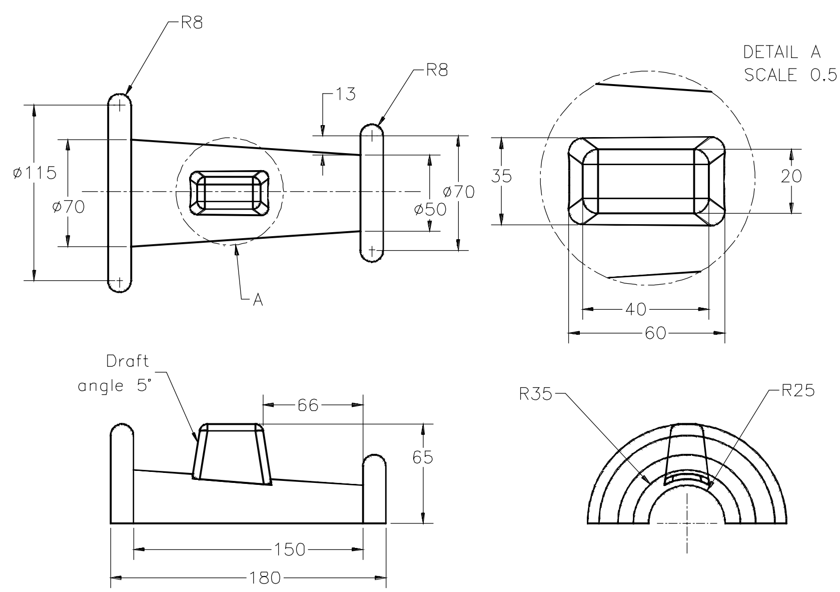

Figure 11-91 *Dimensions and drawing views of the surface model*

The following steps are required to complete this tutorial:

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

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

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Starting a New File and Setting the Sheet Environment

1. Start a new file using the Model template, save the file in the *C:\NX 6\c11 folder* with the name *c11tut1.prt*. Create three fixed datum planes.

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

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

Creating the Base Feature by Revolving the Sketch

1. Create the sketch for the base surface on the XC-YC plane, as shown in Figure 11-92.

2. Revolve the sketch through an angle of 180 degree The resulting revolved base feature created using the **Sheet** option is shown in Figure 11-93.

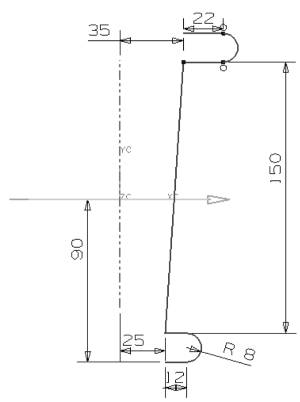

Figure 11-92 *The sketch created for the base feature*

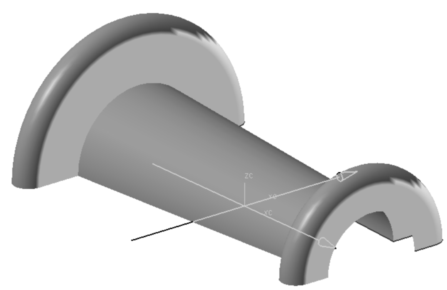

Figure 11-93 *The base feature created by revolving the sketch*

Creating the Second Feature by Extruding the Sketch

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

2. Create the sketch for the second feature by selecting the offset plane as the sketching plane, as shown in Figure 11-94.

3. Extrude the sketch through a distance of 50 in the downward direction and the draft angle of -5. The resulting extruded surface model is shown in Figure 11-95.

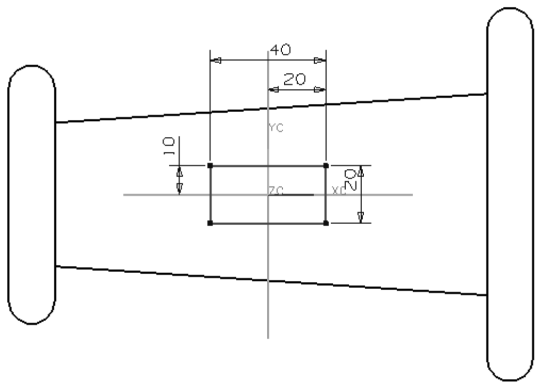

Figure 11-94 The sketch drawn for creating the extruded feature

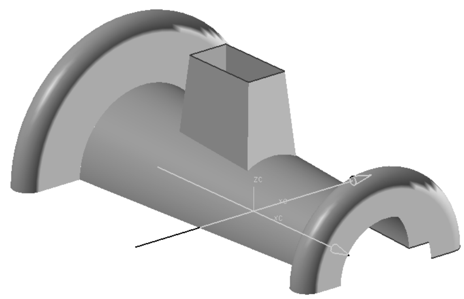

Figure 11-95 The resulting extruded surface

Trimming the Base Surface with respect to the Second Surface

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

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

2. Select the sheet to be trimmed, as shown in Figure 11-96. Make sure that you select the sheet using the selection points, shown in the figure. After selecting the target sheet body, press the middle mouse button; you are prompted to select the trimming objects.

3. Select the trimming surfaces, as shown in Figure 11-96. Make sure that you select the surfaces using the points shown in the figure.

4. Make sure the **Keep** radio button is selected in the **Region** rollout and then choose the **OK** button. The resulting surface model after trimming it, is shown in Figure 11-97.

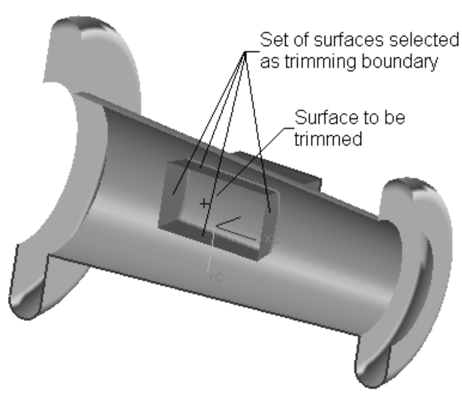

Figure 11-96 The surface to be trimmed and the trimming surfaces selected from the model

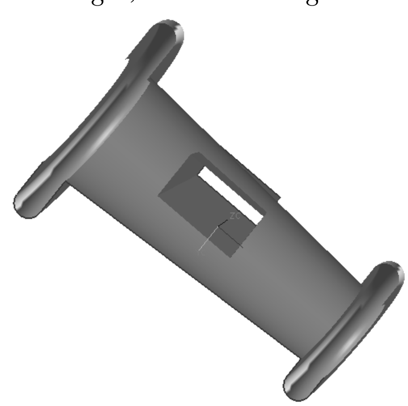

Figure 11-97 The surface model after trimming the base surface with respect to the extruded surface

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Trimming the Second Surface Created with the Base Surface

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

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

2. Select the sheet to be trimmed, as shown in Figure 11-98 and then press the middle mouse button; you are prompted to select the trimming objects.

3. Select the trimming surface, as shown in Figure 11-99. Select the **Keep** radio button from the **Region** rollout and then choose the **OK** button. The resulting surface after trimming the extended portion of the extruded surface is shown in Figure 11-99.

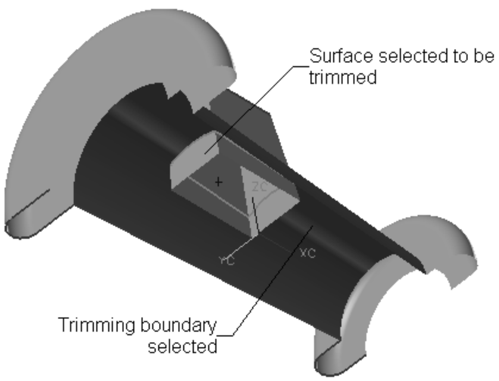


Figure 11-98 *The surface to be trimmed and the trimming surface selected from the model*

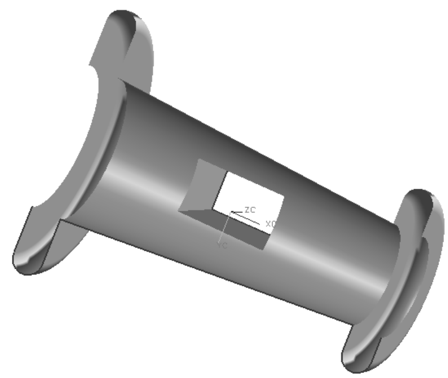
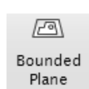


Figure 11-99 *The resulting surface after trimming the extended portion of the extruded surface*

Creating the Bounded Plane Surface

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



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

2. Select the sketch that is used to create the second surface and choose the **OK** button. The resulting bounded plane surface is shown in Figure 11-100.

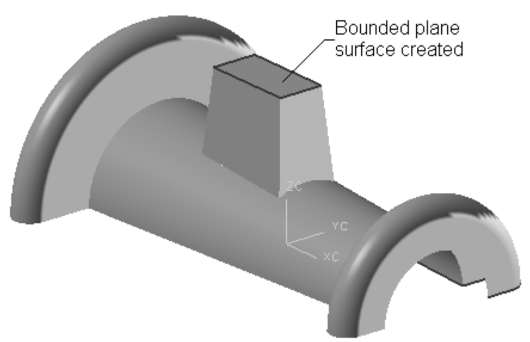
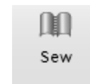


Figure 11-100 *The resulting bounded plane surface*

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

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

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



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

3. Select the bounded plane and the extruded surface.

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

Note

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

Creating Fillets on the Edges Using the Edge Blend Tool

Next, you need to fillet the edges.

1. Choose the **Edge Blend** button from the **Feature Operation** toolbar; the **Edge Blend** dialog box is displayed and you are prompted to select the edges for a new set.

2. Select the edges of the surface, as shown in Figure 11-101. Enter the fillet radius value as **5** in the **Radius 1** edit box.

3. Choose the **OK** button from the **Edge Blend** dialog box. The completed surface model, after adding fillets to edges, is shown in Figure 11-102.

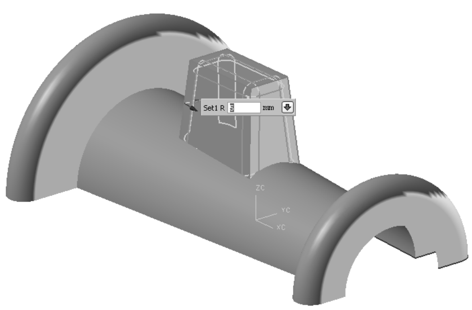


Figure 11-101 *The preview of the fillets displayed after selecting the edges*

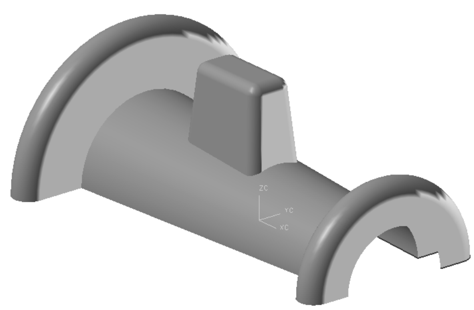


Figure 11-102 *The completed surface model after adding fillets to edges*

Saving and Closing the File

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

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Tutorial 2

In this tutorial, you will create the surface model shown in Figure 11-103. The drawing views and the dimensions of the surface model are shown in Figure 11-104. After creating the model, save it with the name given below.

\NX 6\c11\c11tut2.prt **(Expected time: 45 min)**

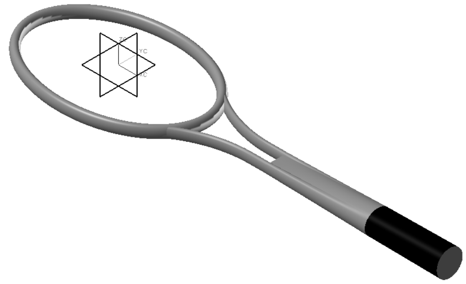

Figure 11-103 *The isometric view of the surface model*

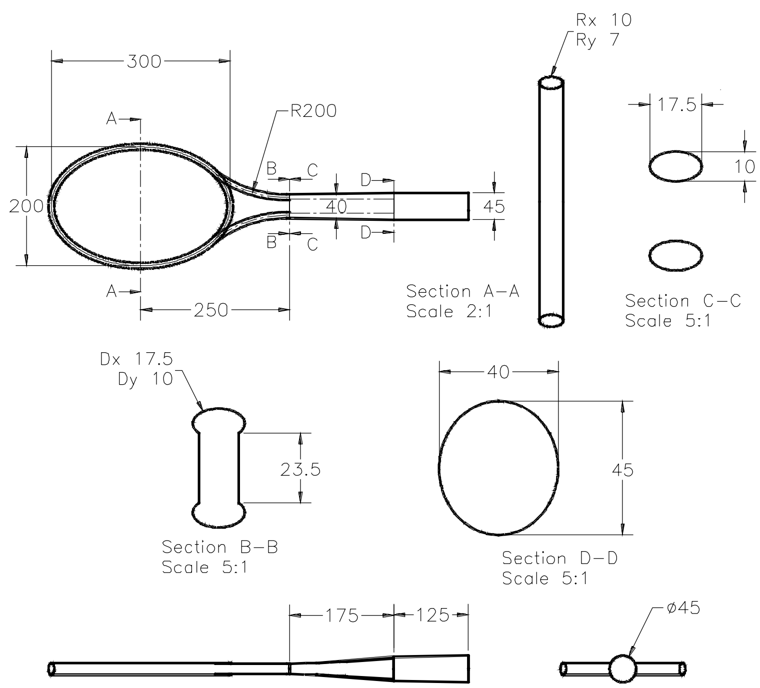

Figure 11-104 *The drawing views and dimensions of the surface model*

The following steps are required to complete this tutorial:

a. Start a new file and set the sheet environment.
b. Create the styled sweep surface as the base feature.
c. Create the sweep surface as the second feature and mirror the surface.
d Create the studio surface as the third feature.

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

Starting the New File and Setting the Sheet Environment

1. Start a new file using the Model template, save the file in the *C:\NX 6\c11 folder* with the name *c11tut2.prt*. Create three fixed datum planes.

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

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

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

Creating the Styled Sweep Surface as Base Feature

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

1. Draw an ellipse on the XC-YC plane as per the dimensions shown in Figure 11-105 and exit the **Sketcher** environment.

2. Draw another ellipse on the XC-ZC plane as per the dimensions shown in Figure 11-106 and exit the **Sketcher** environment.

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

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

5. Choose the **Guide** button from the **Guide Curve** rollout and then select the guide curve drawn on the XC-YC plane. Press the middle mouse button; the preview of the styled sweep surface is displayed.

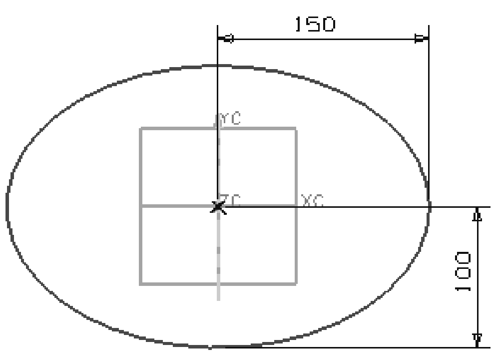

Figure 11-105 *The sketch for the guide string*

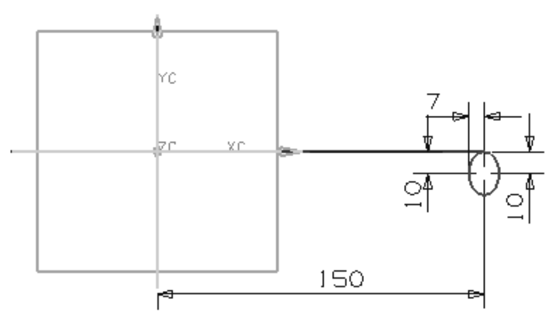

Figure 11-106 *The sketch for the section string*

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6. Choose the **OK** button to create the styled sweep surface. The resulting styled sweep surface created is shown in Figure 11-107.

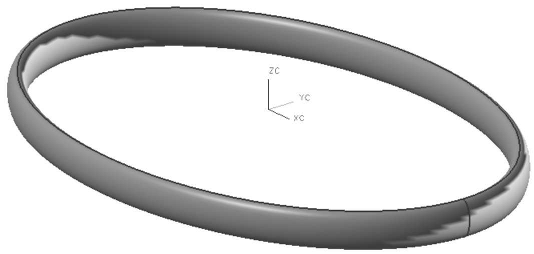

Figure 11-107 *The resulting base feature of the surface model*

Creating the Sweep Surface as the Second Feature

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

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

2. Select the newly created plane as the sketching plane and draw the guide curve, as shown in Figure 11-108.

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

4. Select the newly created plane and draw the ellipse (section curve), as shown in Figure 11-109. Exit the **Sketcher** environment.

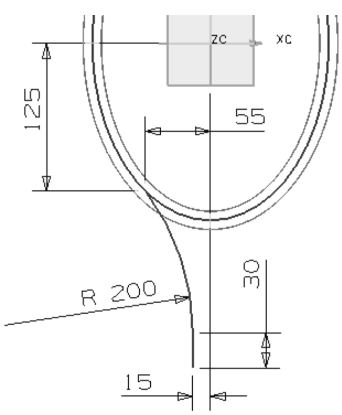

Figure 11-108 *The dimensions for the guide curve of the sweep surface*

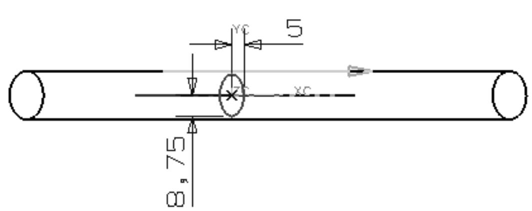

Figure 11-109 *The section curve drawn for creating the sweep surface*

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

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

7. Select the guide curve and choose the **OK** button; the resulting sweep surface is created, as shown in Figure 11-110.

Mirroring the Sweep Surface

1. Mirror the last surface using the XC-ZC plane as the mirror plane. The resulting surface model, after mirroring the sweep surface, is shown in Figure 11-111.

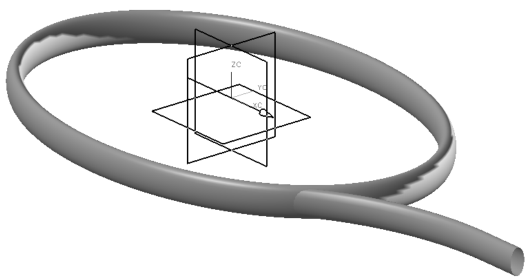

Figure 11-110 The resulting sweep surface created

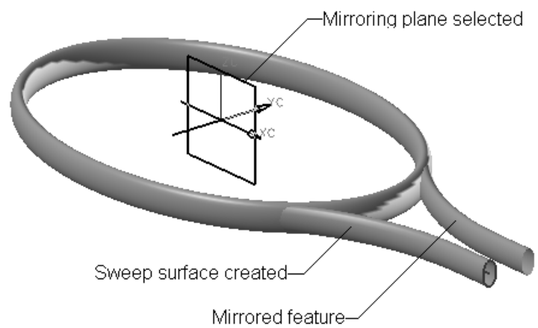

Figure 11-111 The resulting surface model after mirroring the sweep feature

Creating the Studio Surface

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

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

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

3. Select edges and create the profile, as shown in Figure 11-112. Exit the **Sketcher** environment.

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

5. Select the datum plane created as the sketching plane and create the second primary string, as shown in Figure 11-113. The dimensions can be obtained from Figure 11-104.

6. Select the offset datum plane and create the guide strings, as shown in Figure 11-113.

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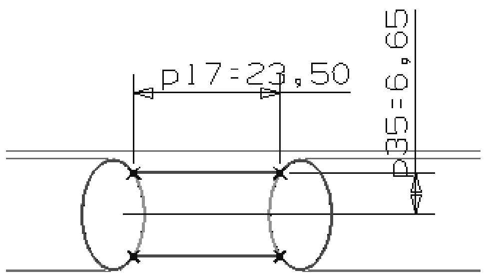

Figure 11-112 The first section curve drawn to create the through curve mesh surface

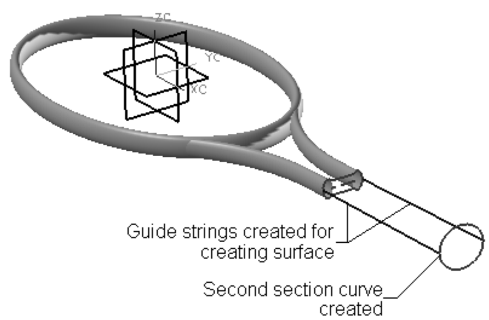

Figure 11-113 The second section curve and the guide strings created to create surface

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

8. Select the first section string, as shown in Figure 11-114. Press the middle mouse button; you are prompted to select the section again.

9. Select all elements from the second section string. Make sure that the arrows in both the section strings point in the same direction, as shown in Figure 11-114. Choose the **Guide (Cross) Curves** button; you are prompted to select the guide string.

10. Select the first guide curve, as shown in Figure 11-114, and press the middle mouse button; you are prompted to select the guide string again.

11. Select the second guide curve, as shown in Figure 11-114. Choose the **Apply** button and then the **Cancel** button to create the surface.

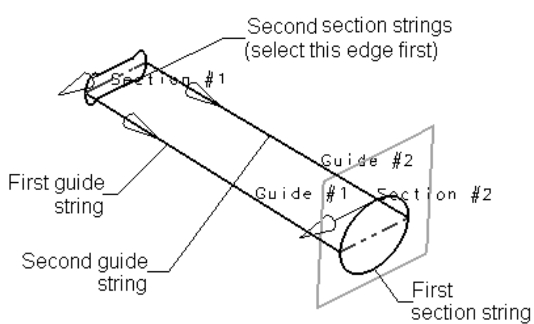

Figure 11-114 Entities to be selected for creating the studio surface

Mirroring the Studio Surface

1. Mirror the studio surface using the XC-YC plane as the mirror plane. The resulting surface model is shown in Figure 11-115.

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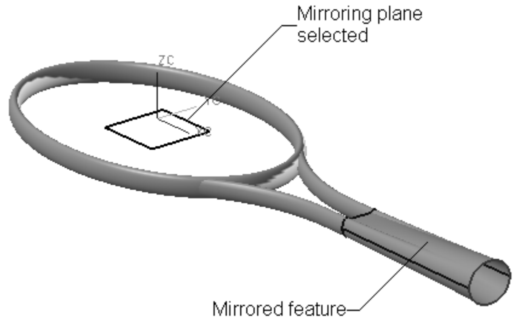

Figure 11-115 The resulting surface model after mirroring the surface

Creating the Ruled Surface

Next, you need to create the ruled surface.

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

2. Select the created plane as the sketching plane and draw the sketch, as shown in Figure 11-116.

3. Exit the **Sketcher** environment and choose the **Ruled** button from the **Surface** toolbar; the **Ruled Surface** dialog box is displayed and you are prompted to select the first section string.

4. Select the first section string, as shown in Figure 11-117. Next, press the middle mouse button; you are prompted to select the second section string.

5. Select the second section string, which is the edge of the surface created earlier, as shown in Figure 11-117. Note that the arrows should point in the same direction. Alternatively, you can control the shape using the **By Points** option in the **Alignment** rollout. Choose the **OK** button; the ruled surface is created.

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

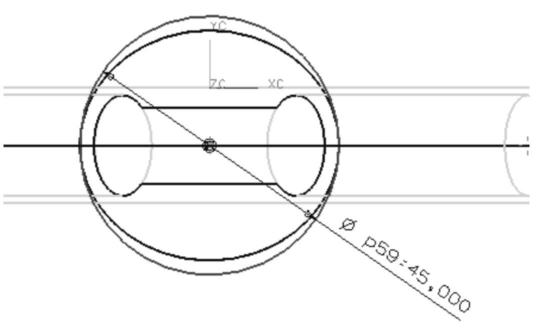

Figure 11-116 The first section curve drawn to create the ruled surface

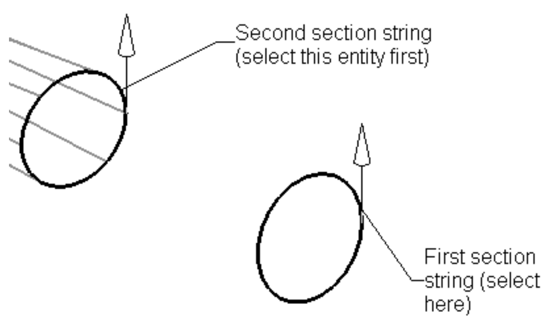

Figure 11-117 The resulting ruled surface created after selecting both cross-sections

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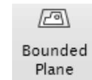
Creating the Bounded Plane Surface



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

2. Select the bounding string for creating the bounded plane surface, as shown in Figure 11-118. The resulting bounded plane surface is displayed, as shown in Figure 11-119.

3. Again, select the sketch shown in Figure 11-119 to create the bounding surface. The resulting bounded plane surface is displayed, as shown in Figure 11-119.

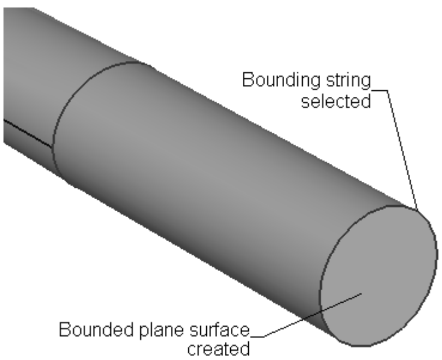


Figure 11-118 *The bounding string selected and the resulting bounded plane surface created*

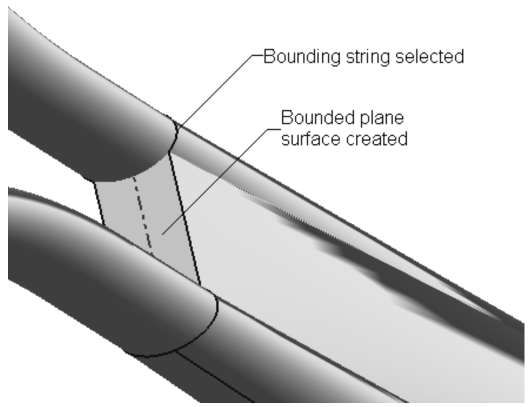


Figure 11-119 *The bounding string selected and the resulting bounded plane surface created*

Saving and Closing the File

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

Self-Evaluation Test

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

1. In NX, Surfaces are termed as sheets. (T/F)

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

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

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

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

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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 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 points are required to create a surface using the **Four Point Surface** tool?

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

2. Which tool is used to create a single patch surface or multi-patches triangular surfaces that enclose a closed 2D sketch or a closed 3D curve?

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

3. Which options are available in the **Type** rollout of the **Law Extension** rollout?

   (a) **Faces, Vector** (b) **Visual Gap, Absolute Gap**
   (c) **Basic, Absolute Gap** (d) None of the above

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

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

5. You can choose **Surface > Bounded Plane** from the menu bar to invoke the **Bounded Plane** tool. (T/F)

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

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

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

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

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

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Exercises

Exercise 1

Create the surface model shown in Figure 11-120. The drawing views and the dimensions of the surface model are shown in same figure. Save the model with the name given below.
\NX 6\c11\c11exr1.prt **(Expected time: 30 min)**

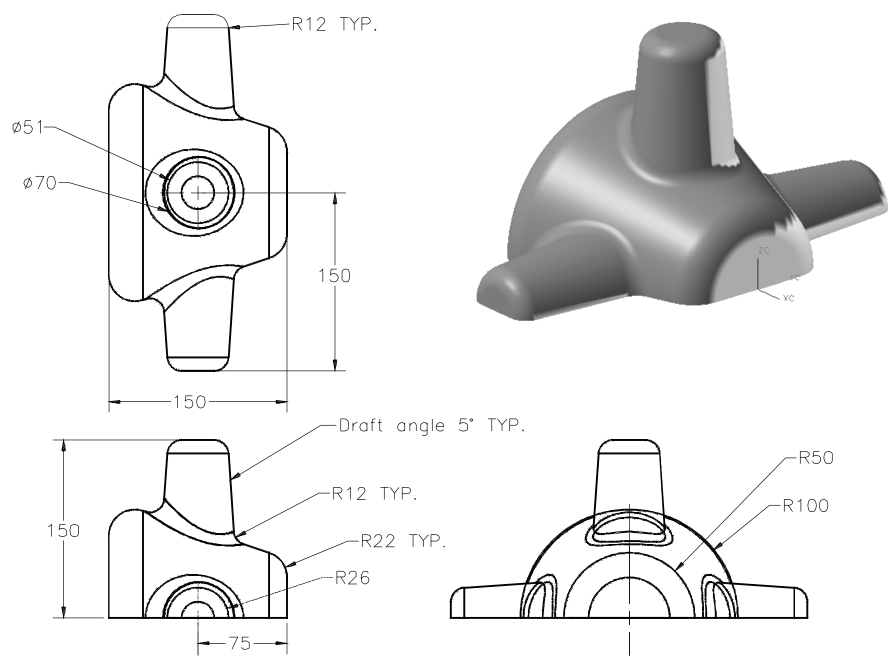

Figure 11-120 *Drawing views and dimensions of the surface model for Exercise 1*

Exercise 2

Create the surface model shown in Figure 11-121. The drawing views and the dimensions of the surface model are shown in Figure 11-122. Save the model with the name given below.
\NX 6\c11\c11exr2.prt **(Expected time: 30 min)**

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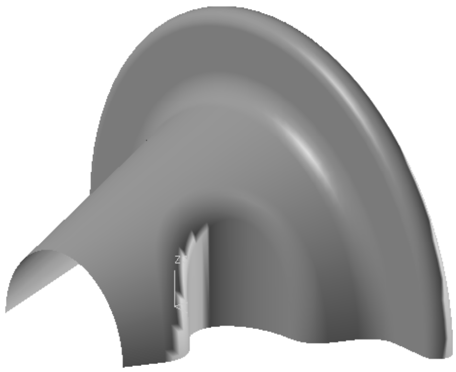

Figure 11-121 The isometric view of the surface model for Exercise 2

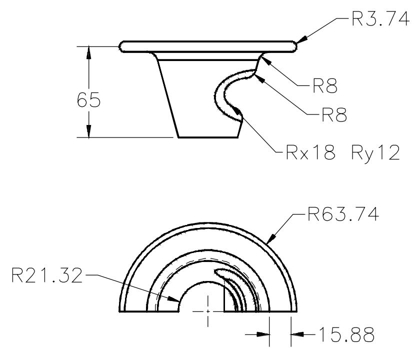

Figure 11-122 The dimensions and drawing views of the surface model

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Answers to Self-Evaluation Test

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

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