



# **Chapter 12**

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## **Advanced Surface Modeling**

### **Learning Objectives**

**After completing this chapter, you will be able to:**

- *Create curves from bodies.*
- *Create dart features.*
- *Create emboss sheet features.*
- *Create face blend features.*
- *Create soft blend features.*
- *Create fillet features.*
- *Create bridge features.*

## CREATING CURVES FROM BODIES

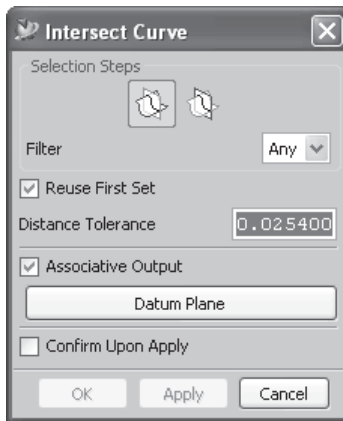
NX allows you to create various types of curves using the existing bodies. These curves are further used to create the surface bodies and are discussed next.

### Creating Intersection Curves

**Menu:** Insert > Curve from Bodies > Intersect  
**Toolbar:** Curve > Intersection Curve

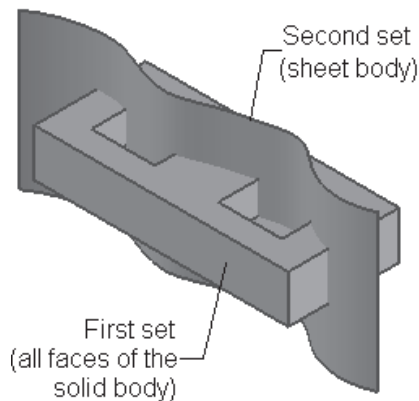


This tool is used to create the intersection curves between two sets of objects. The set of objects could be a solid or a sheet body, one or more faces, or a datum plane. These object sets should intersect fully with each other. To create the intersection curve, choose the **Intersection Curve** button from the **Curve** toolbar; the **Intersect Curve** dialog box will be displayed, as shown in Figure 12-1.



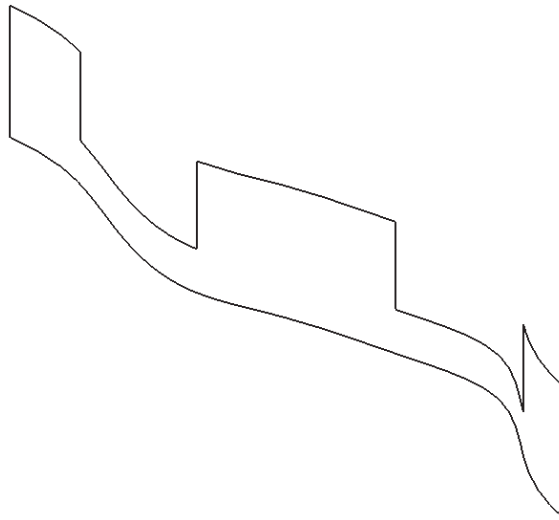
*Figure 12-1 The Intersect Curve dialog box*

The **First Set** button is by default chosen from the **Selection Steps** area. Therefore, you will be prompted to select the first set of objects. Select all the faces of the solid body, as shown in Figure 12-2. To select all the faces of the solid body, drag a box around it. Next, press the SHIFT key and select the surface to remove it from the selection. Choose the **Second Set** button from the **Selection Steps** area; you will be prompted to select the second set of objects. Select the sheet body, as shown in Figure 12-2. The **Filter** drop-down list can be used to aid you in the selection. Selecting the **Reuse First Set** check box ensures that after you choose the **Apply** button, the first set of objects is automatically selected again for creating the



*Figure 12-2 Objects to be selected*

next intersection curve. The **Associative Output** check box allows you to specify whether or not the intersection curve is associative. An associative intersection curve will update automatically when changes are made to its source objects. The **Datum Plane** button allows you to create a datum plane, if required. An intersection curve resulting from the selections made in Figure 12-2 is shown in Figure 12-3.



*Figure 12-3 The resulting intersection curve*

## Creating Section Curves

**Menu:** Insert > Curve from Bodies > Section  
**Toolbar:** Curve > Section Curve



This tool is used to create the intersection curve between a datum plane, solid body, surfaces, and curves. The output curve can be associative. The section curves can be created by four methods: **Select Planes**, **Parallel Planes**, **Radial Planes**, and **Planes Perpendicular to Curve**. The selection steps for each option is different from the other. The methods for creating the section curves are discussed next.

### Creating Section Curves by Selecting Planes

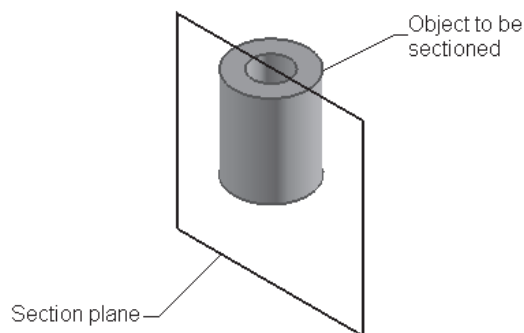


This method is used to create the section curves by specifying the solid or sheet bodies and one or more section planes. To create the select planes section curve, choose the **Section Curve** button from the **Curve** toolbar; the **Section Curve** dialog box will be displayed, as shown in Figure 12-4. By default, the **Select Planes** button is chosen from the **Selection Method** area and the **Objects to Section** button from the **Selection Steps** area. As a result, you will be prompted to select the object to be sectioned. Select the solid body, as shown in Figure 12-5. Choose the **Section Plane** button from the **Selection Steps** area; you will be prompted to select the plane for sectioning. Select the plane, as shown in Figure 12-5. The **Filter** drop-down list can be used to aid you in the selection of the solid body or plane. The

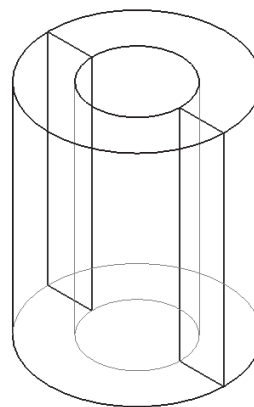


**Figure 12-4** The *Section Curve* dialog box

**Datum Plane** button allows you to create a datum plane, if required. The **Associative Output** check box allows you to specify whether or not the intersection curve is associative. The **Group Section** check box allows you to automatically group the output curves and points that are created for each plane. If the associative output check box is selected, the group section objects check box will not be available. The **Join** drop-down list allows you to join the chain of curves to create a single B-spline curve. The result is either a polynomial spline, a general spline, or a polynomial quintic. The section curves resulting from the selections made in Figure 12-4 are shown in Figure 12-6.



**Figure 12-5** Objects to be selected

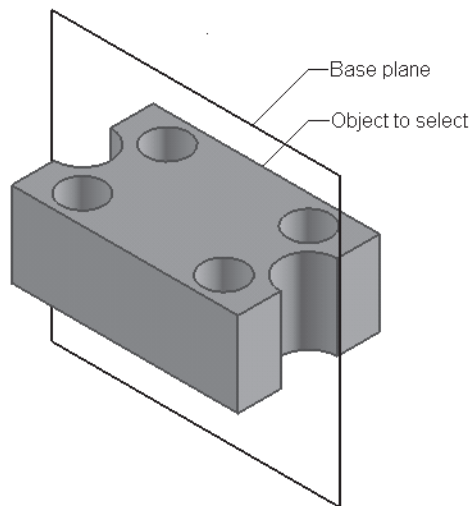


**Figure 12-6** The section curves created by selecting the section plane

### Creating Section Curves Using the Parallel Plane



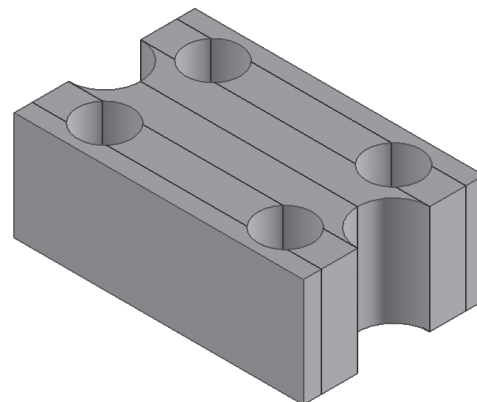
This method is used to create the section curves by specifying the base plane, step value, and the start and end distances. The series of parallel planes are spaced at the equal distances and are further used to create the section of the selected object. To create the section curves using this method, choose the **Section Curve** button from the **Curve** toolbar; the **Section Curve** dialog box will be displayed. Choose the **Parallel Planes** button from the **Selection Method** area; you will be prompted to select the object to be sectioned. Select the solid body, as shown in Figure 12-7.



*Figure 12-7 Objects to be selected*

Choose the **Base Plane** button from the **Selection Steps** area; you will be prompted to select the base for the set of parallel planes. Select the plane, as shown in Figure 12-7. Next, you need to enter the step, start, and end distance values as 10, -25, and 25 in their respective edit boxes. The step distance is the distance between two parallel planes. The start and end distances are measured from the base plane. Between the start and end distance, the software generates as many possible planes.

The positive distance is in the direction of the vector displayed. Choose the **Apply** button and then the **Cancel** button to create the curves. The section curves created using the selections made in Figure 12-7 are shown in Figure 12-8.

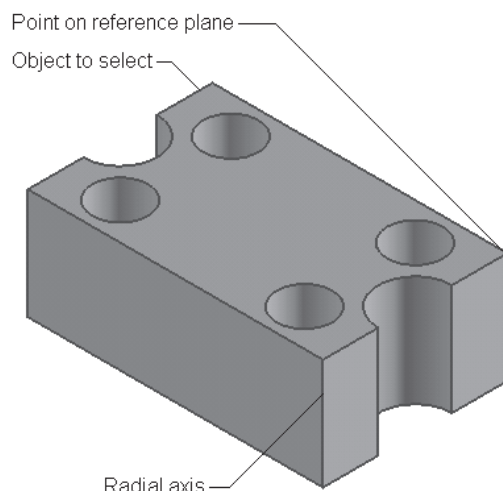


*Figure 12-8 The section curves created by the parallel plane method*

### Creating Section Curves Using the Radial Plane



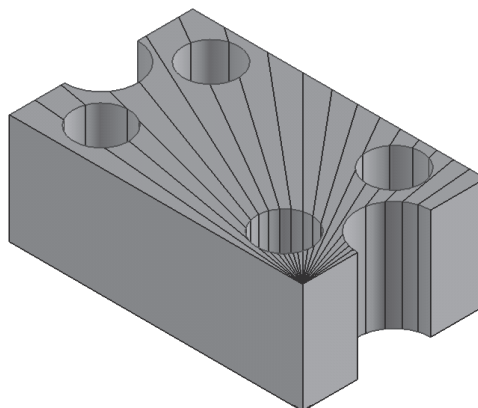
This method is used to specify the planes spaced at equal angles that are used further for sectioning the selected bodies. The planes are pivoted around a common axis. To create the section curves using this method, choose the **Section Curve** button from the **Curve** toolbar; the **Section Curve** dialog box will be displayed. Choose the **Radial Planes** button from the **Selection Method** area; you will be prompted to select the object to be sectioned. Select the solid body, as shown in Figure 12-9. Choose the **Radial Axis** button from the **Selection Steps** area; you will be prompted to select the radial axis. Select the edge of the object, as shown in Figure 12-9.



*Figure 12-9 Objects to be selected*

Alternatively, you can use the **Vector** drop-down list. Choose the **Point on Reference Plane** button from the **Selection Steps** area; you will be prompted to select the radial reference point. Select the radial reference point, as shown in Figure 12-9. Next, enter the values of the step, start, and end angles as 5, 0, and 90 in their respective edit boxes. The step angle is the angle between two radial planes. The start and end angles are measured from the base plane. The base plane is the one that passes through the radial axis and the point on the reference plane. Between the start and end angles, the software generates as many planes as possible.

The section curves resulting from the selections made in Figure 12-9 are shown in Figure 12-10.

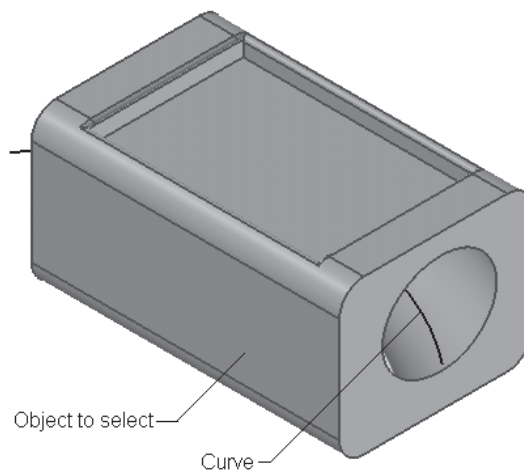


*Figure 12-10 The section curves created by the radial plane method*

### Creating Section Curves Using the Planes Perpendicular to Curves



This method is used to create the section curves along the planes perpendicular to the selected curve. You need to specify the solid or sheet body, curve, and the spacing method. To create the section curves using this method, choose the **Section Curve** button from the **Curve** toolbar; the **Section Curve** dialog box will be displayed. Choose the **Planes Perpendicular to Curve** button from the **Selection Method** area; you will be prompted to select the object to be sectioned. Select the solid body, as shown in Figure 12-11. Choose the **Curve or Edges** button from the **Selection Steps** area; you will be prompted to select the curve or edge. Select the curve, as shown in Figure 12-11, along which the perpendicular planes will be created.



*Figure 12-11 Objects to be selected*

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

#### **Equal Arc Length**

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

#### **Equal Parameters**

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

#### **Geometric Progression**

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

a value in the **Ratio** edit box to determine the mathematical ratio for spacing the planes between the start and end percentage points.

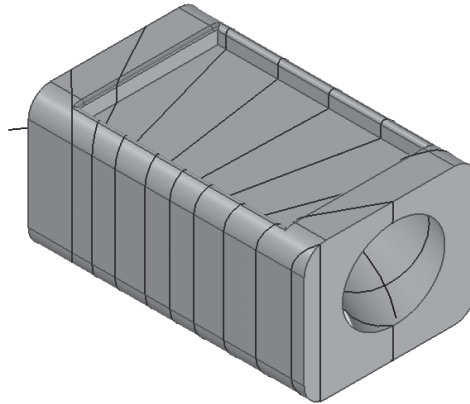
### Chordal Tolerance

This option allows you to create the sections using the planes based on a chordal tolerance. You need to enter the value for the chordal tolerance.

### Incremental Arc Length

This option allows you to create the sections using the planes placed at increments along the curve. You need to enter the value for the arc length.

Select the **Equal Arc Length** method from the **Spacing Along Curve** drop-down list. Next, enter the values of the number, start, and end percentage as 10, 0, and 100 in their respective edit boxes. The section curves created using the selections made in Figure 12-11 are shown in Figure 12-12.



*Figure 12-12 The section curves created using a plane perpendicular to the curve*

## Creating Extract Curves

**Menu:** Insert > Curve from Bodies > Extract

**Toolbar:** Curve > Extract Curve



This tool is used to create the curves using the edges or faces of the solid or sheet bodies. To create the extract curves, choose the **Extract Curve** button from the **Curve** toolbar; the **Extract Curve** dialog box will be displayed, as shown in Figure 12-13. The different extract curve methods are discussed next.

### Edge Curves

This method allows you to extract the curves from the selected edges, or faces, or bodies. The buttons from the **Single Edge Curve** dialog box can be used in the selection.

### Isoparametric Curves

This method allows you to create the isoparametric curves on a selected face. It is used to



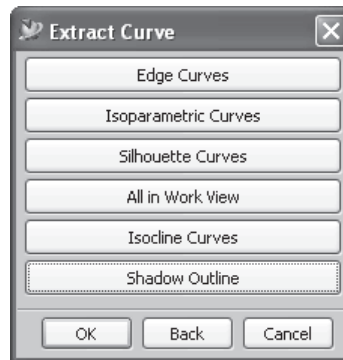


Figure 12-13 The **Extract Curve** dialog box

generate the curves along the U/V parameters on a face.

### Silhouette Curves

This method allows you to create the curves from the silhouette edges.

### All in Work View

This method allows you to create the curves from all the edges of all bodies in the part file.

### Isocline Curves

This method allows you to create the curves where the draft angle on a set of faces is constant. These curves are used to split a surface and to create the parting surfaces on a mold or casting.

### Shadow Outline

This method allows you to create the curves that show only the outline of the bodies in the part file. To execute this option, first you need to set up the **work view** to **Invisible Hidden Edges**. Some of the extract options are discussed next.

### Creating Edge Curves

To create the edge curves, choose the **Edge Curves** button from the **Extract Curve** dialog box; the **Single Edge Curve** dialog box will be displayed, as shown in Figure 12-14, and you will be prompted to select the edge 1. Select the edges that you want to extract. Alternatively, you can use the other options from the **Single Edge Curve** dialog box to extract the curves. These options are discussed next.

#### All in Face

This option allows you to extract the curves from all the edges of the selected face.

#### All of Solid

This option allows you to extract the curves from all the edges of the selected solid.

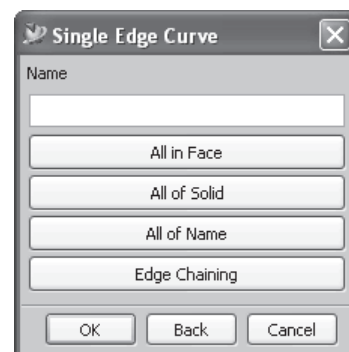


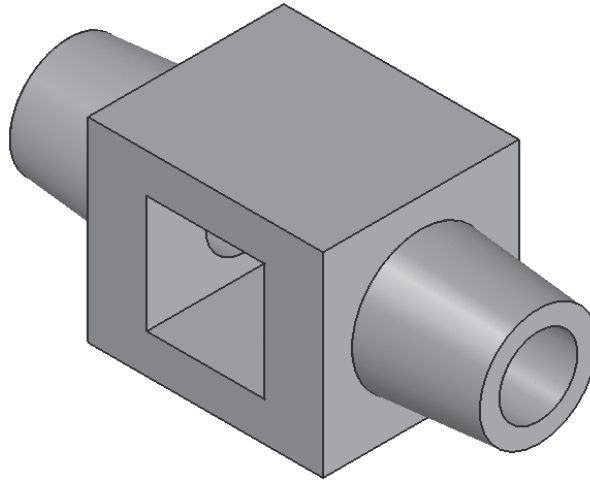
Figure 12-14 The **Single Edge Curve** dialog box

**All of Name**

You need to enter the name of the edges.

**Edge Chaining**

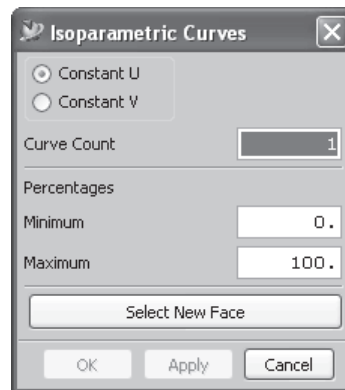
This option allows you to extract the connected curves. Figure 12-15 shows the curves extracted by the **All In Solid** option.



*Figure 12-15 The curves extracted using the All of Solid option*

**Creating Isoparametric Curves**

To create the isoparametric curves, choose the **Isoparametric Curves** button from the **Extract Curves** dialog box; the **Isoparametric Curves** dialog box will be displayed, as shown in Figure 12-16.



*Figure 12-16 The Isoparametric Curves dialog box*

Also, you will be prompted to select the face of the model. Select the side face of the model; you will be prompted to enter the parameters for creating the curve. These parameters are discussed next.

**Constant U & Constant V**

These are the directions in which the curves will be created.

**Curve Count**

It is the number of the isoparametric curves spaced equally between the minimum and maximum percentages.

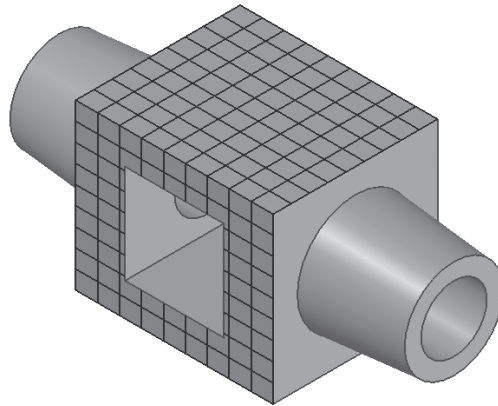
**Percentages**

It is the minimum and maximum value between which the curves will be created.

**Select New Face**

This button allows you to select a new face to create the isoparametric curves.

By default, the **Constant U** radio button will be selected. Enter **10** as the value in the **Curve Count** edit box and choose the **Apply** button. Choose the **Constant V** button and then the **Apply** button. Choose the **Select New Face** button and select the top face of the model. Repeat the same procedure to create the curves in the U & V direction. The resulting curves are shown in Figure 12-17.



*Figure 12-17 The curves created by the Isoparametric method*

## ADVANCED SURFACE MODELING TOOLS

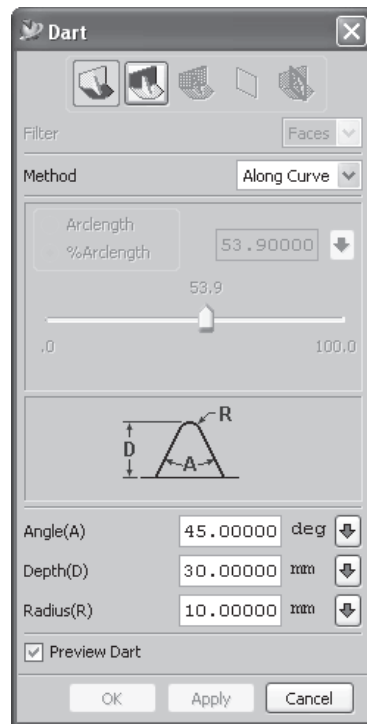
These tools are used to create the basic and advanced surfaces and are discussed next.

### Creating Dart Features

<b>Menu:</b>	Insert > Detail Feature > Dart
<b>Toolbar:</b>	Form Feature > Dart



This tool is used to add a rib along the intersection curve of two faces. The dart feature can be added to the solid body or sheet body. To create the dart feature, you need to select the two faces that intersect each other. The dart will be placed on a plane that is perpendicular to the intersection curve of the two faces. You can define the orientation manually. You need to enter the dimensional values of the dart for angle, depth,

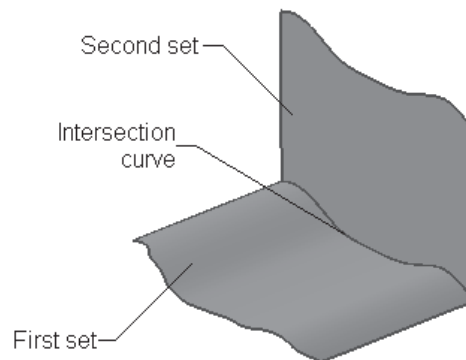


*Figure 12-18 The **Dart** dialog box*

and radius. To create the dart feature, choose the **Dart** button from the **Form Feature** toolbar; the **Dart** dialog box will be displayed, as shown in Figure 12-18. By default, the **First Set** button is chosen and you will be prompted to select the face for the first set. Select the surface, as shown in Figure 12-19. Next, choose the **Second Set** button; you will be prompted to select the face for the second set. Select the surface, as shown in Figure 12-19. The preview of the dart will be displayed on the intersection curve of two faces. Choose the **Location Plane** button, if you want to position the dart feature relative to a plane. This is optional. Choose the **Orientation Plane** button to select a plane for orienting of the dart feature. This is also optional. To position the dart manually, you have two options in the **Method** drop down list. These options are discussed next.

#### **Along Curve**

This option allows you to define a base point for the dart anywhere on the intersection curve. You can enter the value for the arc length in the **Arc length** edit box. Alternatively, you can enter the value for the percentage arc length in the **%Arc length** edit box. The slider can be used to drag the base point along the curve.



*Figure 12-19 Objects to be selected*

**Position**

This option allows you to specify the location for the dart by using the WCS or the absolute coordinate systems. You need to enter the values for the X, Y, Z positions.

Select the **Along Curve** option from the **Method** drop-down list and enter the value in the **%ArcLength** edit box. The dart will be placed on the intersection curve. Now, enter the dimensional values of the dart in the angle, depth, and radius edit boxes. The **Preview Dart** check box allows you to see the preview of the dart. Choose the **Apply** button and then the **Cancel** button. The dart feature created between the two selected surfaces is shown in Figures 12-20 and 12-21.

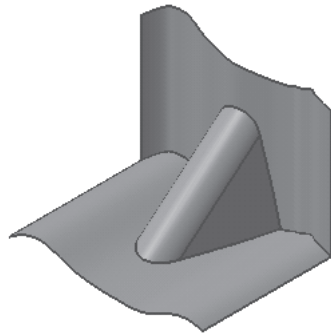


Figure 12-20 The resulting dart feature

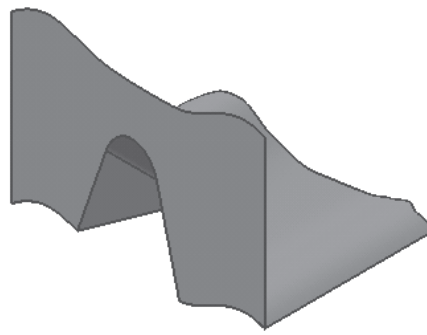


Figure 12-21 The resulting dart feature

## Creating Emboss Sheet Features

**Menu:** Insert > Combine Bodies > Emboss Sheet  
**Toolbar:** Feature Operation > Emboss Sheet



The **Emboss Sheet** tool is used to convert the shapes of the solid bodies into a sheet, using a sheet body as the cutting object. To create the emboss sheet feature, choose the **Emboss Sheet** button from the **Feature Operation** toolbar; the **Emboss Sheet** dialog box will be displayed, as shown in Figure 12-22.

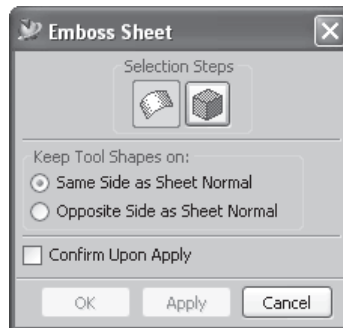


Figure 12-22 The Emboss Sheet dialog box

By default, the **Target Sheet** button will be chosen from the **Selection Steps** area and you will be prompted to select the sheet to be embossed. Select the sheet, as shown in Figure 12-23. Choose the **Tool Solid(s)** button from the **Selection Step** area; you will be prompted to select the tool solids. Select the solid bodies, as shown in Figure 12-23. The radio buttons in the **Keep Tool Shapes** area allow you to specify which part of the tool bodies are added to the sheet. These radio buttons are discussed next.

#### Same Side as Sheet Normal

The portions of the tool bodies above the sheet body and in the direction of the sheet normal vector are added to the shape of the sheet. The other portions are discarded.

#### Opposite Side as Sheet Normal

The portions of the tool bodies below the sheet body and in the direction opposite to the sheet normal vector are added to the shape of the sheet. The other portions are discarded.

The **Confirm Upon Apply** check box allows you to preview the result. The resulting emboss sheet is shown in Figure 12-24.

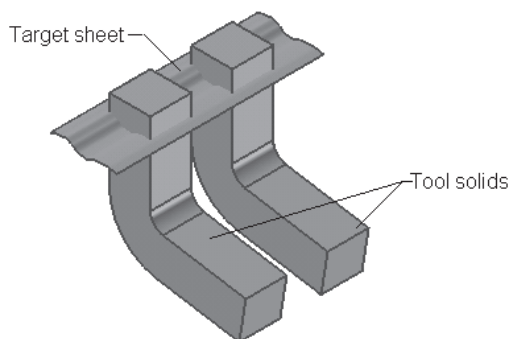


Figure 12-23 Objects to be selected

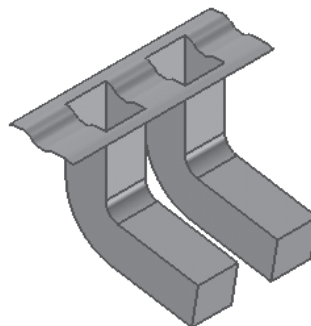


Figure 12-24 The resulting emboss sheet feature

## Creating Face Blend Features

<b>Menu:</b>	Insert > Design Feature > Face Blend
<b>Toolbar:</b>	Feature Operation > Face Blend



This tool is used to create the complicated blends tangent to the specified sets of faces. A face blend can be created between the faces of the solid or sheet bodies. The wall faces of the blend can be trimmed automatically and can also be attached to the blend. To create the face blend feature, choose the **Face Blend** button from the **Feature Operation** toolbar; the **Face Blend** dialog box will be displayed, as shown in Figure 12-25. The face blend dialog box options are discussed next.

### Selection Steps Area

The buttons in the **Selection Steps** area are used to select the faces, cliff edges, and tangency control objects for the blend. These buttons are discussed next.

**First Set**

This button allows you to select the first set of faces for blending. You can also select edges instead of the faces. After selecting the face, a vector will be displayed. This vector should point toward the center of the blend. Choose the **Reverse Normal** button to reverse the direction of the vector.

**Second Set**

This button allows you to choose the second set of faces for blending. After selecting the face, a vector will be displayed. This vector should point toward the center of the blend

**Cliff Edges**

This button allows you to apply a constant-radius blend at a cliff edge along the first and second blend faces.

**Tangency Control**

This button allows you to control the radius of the sphere, or an offset of the conic by maintaining a tangency between the blend surface and an underlying face set along a specified curve or edge.

**Roll Over Smooth Edges**

This check box is used modify the radius of the fillet in order to retain the shape and the sharpness of the edges of the adjacent faces. If this check box is cleared, the adjacent faces will extend in case the fillet radius is more than what can be adjusted in the current face.

**Remove Surface Self-Intersections**

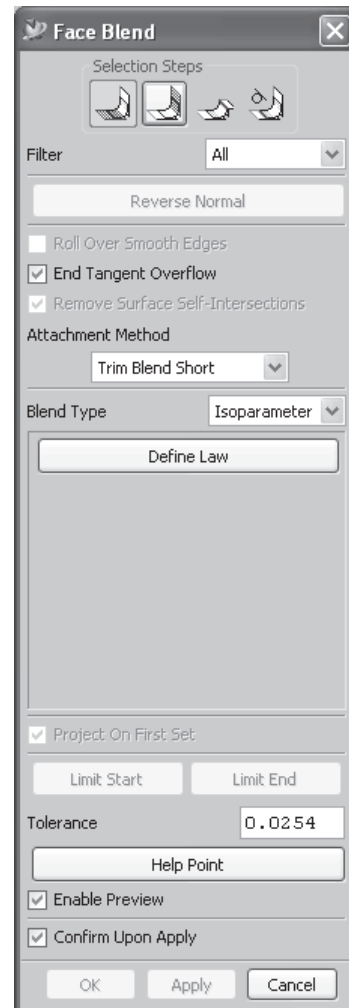
This check box is used to automatically replace those areas with a patch where a self-intersection prevents a blend from being created. This option will be highlighted only for the **Sphere** blend type.

**Attachment Method**

The options in the **Attachment Method** drop-down list are used for trimming and attaching the face blend.

**Blend Type**

The **Blend Type** drop-down list is used to select the type of blend that you want to create. These blend types are discussed next.



**Figure 12-25** The *Face Blend* dialog box

**Sphere**

In this type, the blend cross-section lies in a plane normal to the two selected sets of faces.

**Conic**

This blend type has a conic cross-section. You can control the cross-section with two offsets and a rho value. You need to define a spine string to define the plane of the conic sections.

**Disc**

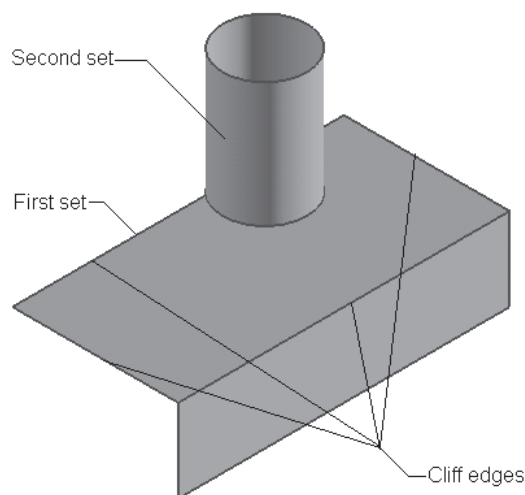
It is a variable radius blend whose cross-section lies in a plane orthogonal to a spine string. The blend radius is defined by the law subfunction. A disc blend has a tighter curvature than a spherical blend.

**Isoparameter**

It is a specialized blend used primarily for the turbine blades. For these type of surfaces, an isoparametric blend may produce good results when the other blend types fail. The blend radius is defined by the law subfunction.

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

1. Choose the **Face Blend** button from the **Feature Operation** toolbar; the **Face Blend** dialog box will be displayed. By default, the **Sphere** is selected from the **Blend Type** drop-down list and the **First Set** button is chosen from the **Selection Steps** area. Therefore, you will be prompted to select the face for the first set.
2. Select the first set face, as shown in Figure 12-26. Make sure the normal vector points in the upward direction.



**Figure 12-26** Objects to be selected

3. Choose the **Second Set** button from the **Selection Step** area; you will be prompted to



select the face for the second set. Select the second set face, as shown in Figure 12-26.

4. Choose the **Cliff Edges** button from the **Selection Step** area; you will be prompted to select the cliff edges. Select the cliff edges, as shown in Figure 12-26.
5. Select the **Trim and Attach All** from the **Attachment Method** drop down list and enter the value of the radius in the radius edit box. Choose the **OK** button. The resulting face blend feature is shown in Figure 12-27.

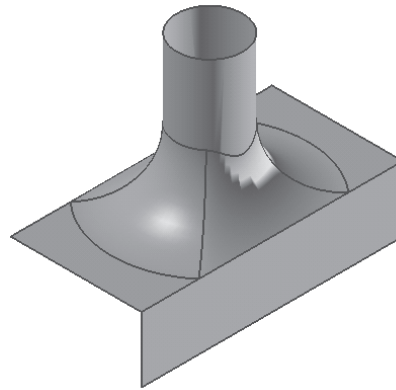


Figure 12-27 The resulting face blend feature

## Creating Soft Blend Features

**Menu:** Insert > Detail Feature > Soft Blend  
**Toolbar:** Feature Operation > Soft Blend



This tool is used to create the blended cross-section other than the circular blend and will give you more control over the cross-sectional shape. You will have more control on the edges of the blend. It allows you to create the designs that are more aesthetically pleasing than the other types of blends. Some of the options in the soft blend dialog box are the same as in the face blend dialog box. To create the soft blend feature, choose the **Soft Blend** button from the **Feature Operation** toolbar; the **Soft Blend** dialog box will be displayed, as shown in Figure 12-28. The soft blend dialog box options are discussed next.

### Selection Steps Area

The buttons in the **Selection Steps** area are used to select the faces and tangency curves for the blend. These buttons are discussed next.

#### First Set

This button allows you to select the first set of faces for blending.

#### Second Set

This button allows you to select the second set of faces for blending. After selecting a face, a vector will be displayed. This vector should point toward the center of the blend. Choose the **Reverse Normal** button to reverse the direction of the vector.

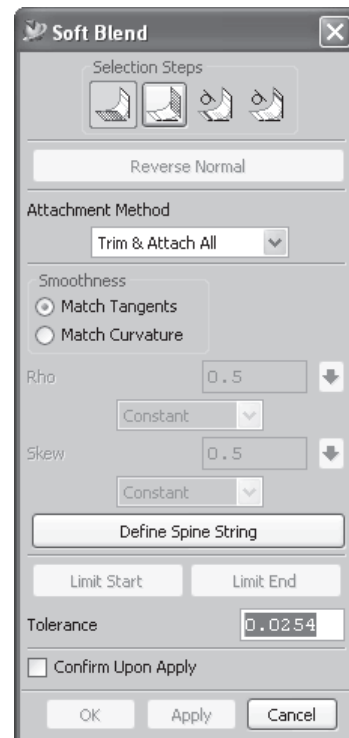


Figure 12-28 The Soft Blend dialog box

**First Tangency Curve**

This button allows you to select a curve that lies on the first set and becomes the edge of the blend.

**Second Tangency Curve**

This button allows you to select a curve that lies on the second set and becomes the edge of the blend.

**Attachment Method**

The options in the **Attachment Method** drop-down list are used for trimming and attaching the soft blend.

**Smoothness**

The options in the **Smoothness** area allow you to match only the tangents of the blend and faces, or match the curvature. These options are discussed next.

**Match Tangents**

By selecting this radio button, the selected faces will be matched with tangency. The cross-sectional shape of the blend is an ellipse.

**Match Curvature**

By selecting this radio button, the selected faces will be matched with a tangency and curvature. You need to enter the values for the two shape control parameters: rho and skew.

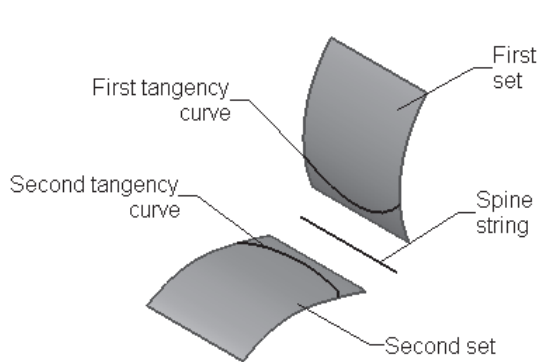
**Define Spine String**

This button allows you to select a spine string for the soft blend.

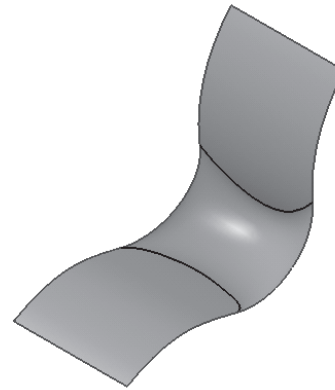
The following steps explain the procedure of creating a soft blend:

1. Choose the **Soft Blend** button from the **Feature Operation** toolbar; the **Soft Blend** dialog box will be displayed, as shown in Figure 12-28. By default, the **First Set** button is chosen from the **Selection Steps** area and you will be prompted to select the face for the first set.
2. Select the first set face, as shown in Figure 12-29.
3. Choose the **Second Set** button from the **Selection Step** area; you will be prompted to select the face for the second set. Select the second set face, as shown in Figure 12-29.
4. Choose the **First Tangency Curve** button from the **Selection Step** area; you will be prompted to select the first tangency curve. Select the curve, as shown in Figure 12-29.
5. Choose the **Second Tangency Curve** button from the **Selection Step** area; you will be prompted to select the second tangency curve. Select the curve, as shown in Figure 12-29.
6. Choose the **Define Spline String** button; you will be prompted to select the spine string.

Select the curve, as shown in Figure 12-29. Choose the **OK** button. The resulting soft blend feature is shown in Figure 12-30.



*Figure 12-29 Objects to be selected*



*Figure 12-30 The resulting soft blend feature*

## Creating Fillet Features

**Menu:** Insert > Detail Feature > Fillet  
**Toolbar:** Surface > Fillet Surface

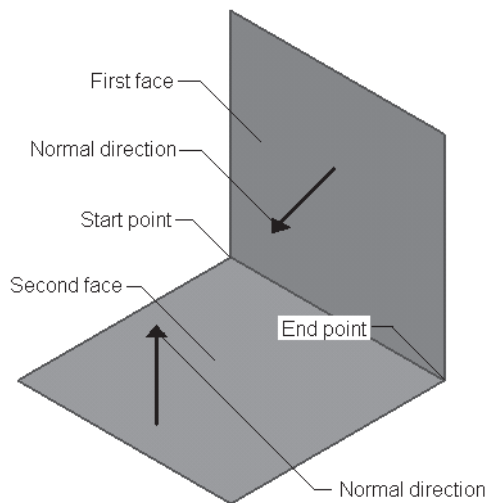


This tool is used to create the fillets of a constant or variable radius between two faces of the solid or sheet bodies. The fillet created will be tangent to the two faces. The faces need to intersect or be close enough so that the fillet touches both the faces. You can create four types of fillets, namely: constant, linear, S-shaped, and general. This feature is not parametric, which means that you cannot edit the parameters after the creation of the feature. The procedure for creating the fillets is discussed next.

### Creating Constant Fillet Features

To create the constant fillet, you need to enter a radius, start point, and end point. The following steps are required to create the constant fillet:

1. Choose the **Fillet** button from the **Surface** toolbar; the **Fillet** dialog box will be displayed and you will be prompted to select the first face to fillet. Select the first face, as shown in Figure 12-31; the selected face will be highlighted with the normal direction vector and you will be prompted to decide if the normal direction is OK. This vector should point toward the center of the blend. Choose the **No** button to flip the direction.
2. Next, you will be prompted to select the second face. Select the second face, as shown in Figure 12-31. Flip the direction by choosing the **No** button.
3. Next, you will be prompted to select the spine curve. This is optional for the constant fillet. You need to select the spline curve only for the general fillet type. Choose the **OK** button.



**Figure 12-31** Objects to be selected

4. Next, you will be prompted to choose the creation options from the **Fillet** dialog box, as shown in Figure 12-32. These options are discussed next.

#### **Create Fillet**

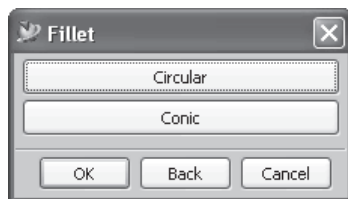
This button enables you to specify whether or not you want to create a fillet. By default, the **Yes** option is chosen.

#### **Create Curve**

This button allows you to create the offset intersection curve between the two faces. By default, the **No** option is chosen.

Choose the **OK** button to accept the default options.

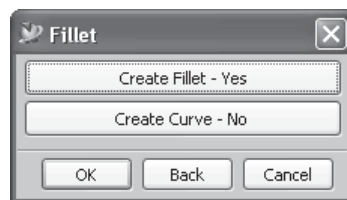
5. The options in the **Fillet** dialog box will be modified and you will be prompted to choose the cross-section type, as shown in Figure 12-33.



**Figure 12-33** The **Fillet** dialog box

#### **Circular**

The cross-section will be circular in shape and will have a specified radius value.



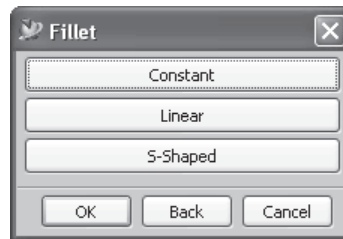
**Figure 12-32** The **Fillet** dialog box

**Conic**

The cross-section will be conic in shape and will have specified radius, ratio, and rho values.

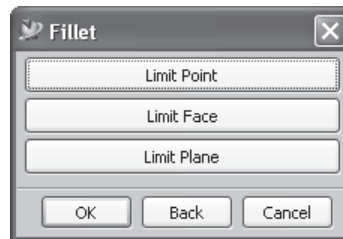
Choose the **Circular** button.

6. The options in the **Fillet** dialog box will be modified and you will be prompted to select the fillet type, as shown in Figure 12-34. Choose the **Constant** button.



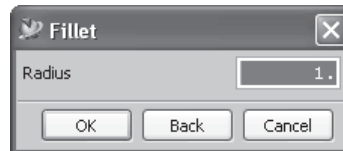
*Figure 12-34 The Fillet dialog box*

7. Next, you will be prompted to select the fillet start point in the **Fillet** dialog box, see Figure 12-35. Choose the **Limit Point** button; the **Point Constructor** dialog box will be displayed and you will be prompted to select the inferred point. Select the endpoints of one of the faces as the start point of the fillet, as shown in Figure 12-31.



*Figure 12-35 The Fillet dialog box*

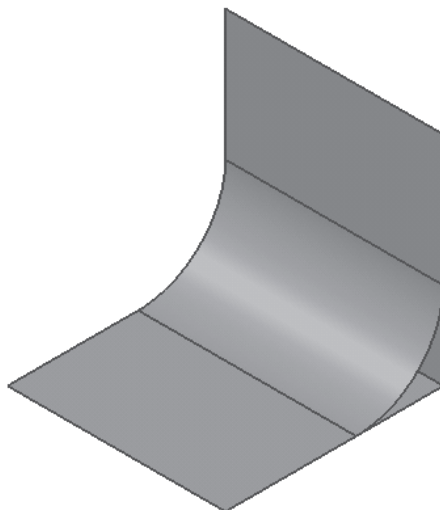
8. The options in the **Fillet** dialog box will be modified and you will be prompted to enter the radius. Enter the radius in the radius edit box, as shown in Figure 12-36, and choose the **OK** button.



*Figure 12-36 The Fillet dialog box*

9. Next, you will be prompted to check the direction. Choose the **Yes** button to flip the direction.

10. You will be prompted to select the endpoint. Choose the **Limit Point** button from the **Fillet** dialog box, as shown in Figure 12-36. The **Point Constructor** dialog box will be displayed and you will be prompted to select the inferred point. Select the endpoint, as shown in Figure 12-31. Choose the **OK** button to finish the procedure. The resulting constant fillet is shown in Figure 12-37.

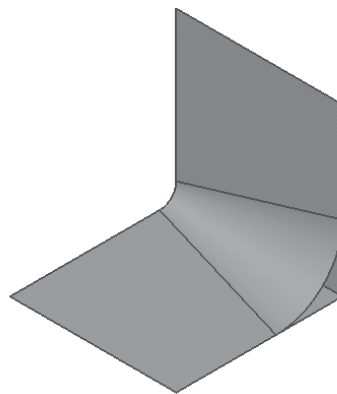


*Figure 12-37 The resulting constant fillet*

### Creating Linear Fillet Features

This type of fillet has two radius values. The radius will be linear through out the fillet. The procedure to create the linear fillet is the same as the constant fillet, with a few exceptions that are discussed next.

1. Choose the **Linear** button as the fillet type from the **Fillet** dialog box.
2. You need to enter the radius values at the start and end points of the fillet. The resulting linear fillet is shown in Figure 12-38.



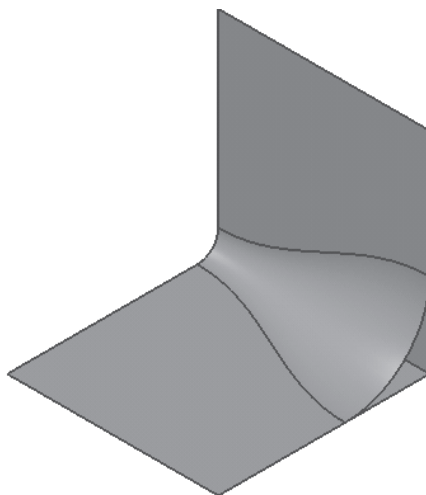
*Figure 12-38 The resulting linear fillet*

### Creating S-Shaped Fillet Features

This type of fillet is used to create a variable radius fillet of an S-shaped curvature. The radius will be variable through out the fillet. The procedure to create the S-shaped fillet is the same as the constant fillet, with a few exceptions that are discussed next.

1. Choose the **S-Shaped** button as a fillet type from the **Fillet** dialog box.

2. You need to enter the radius values at the start and end points of the fillet. The resulting S-shaped fillet is shown in Figure 12-39.

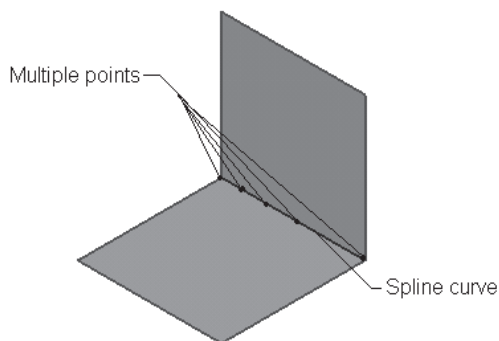


*Figure 12-39 The resulting S-shaped fillet*

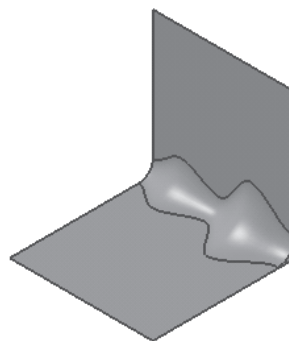
### Creating General Fillet Features

This type of fillet is used to create a variable radius fillet by specifying the multiple points on the spine curve. The procedure to create the general fillet is the same as the constant fillet, with a few exceptions that are discussed next.

1. Select the spline curve when you are prompted to do so, as shown in Figure 12-40. Note that you need to create a curve at the intersection edge of the two surfaces.
2. Choose the **General** button as a fillet type from the **Fillet** dialog box. You need to select multiple points on the spine curve, as shown in Figure 12-40, with the help of the **Point Constructor** dialog box and enter the radius values at these points. The resulting general fillet is shown in Figure 12-41.



*Figure 12-40 Objects to be selected*



*Figure 12-41 The resulting general fillet*

## Creating Bridge Features

**Menu:** Insert > Detail Feature > Bridge

**Toolbar:** Surface > Bridge

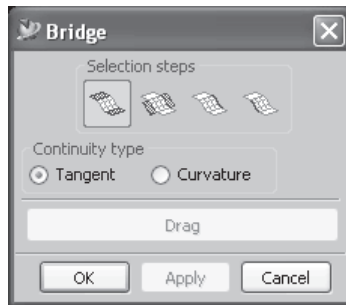


This tool is used to create the bridge surface that joins the edges of the surfaces. You can specify a tangent or a curvature continuity between the bridge surface and the defining faces. You can control the bridge surface by side faces, side strings, or by dragging. These methods are discussed next.

### Creating Bridge Features by Side Faces

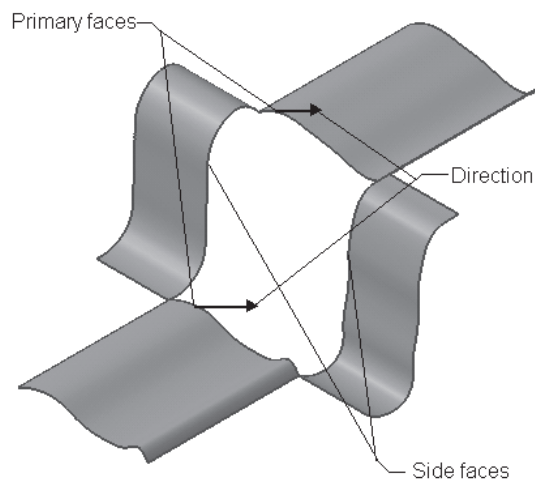
The following steps are required to create the bridge feature by side faces:

1. Choose the **Bridge** button from the **Surface** toolbar; the **Bridge** dialog box will be displayed, as shown in Figure 12-42.



*Figure 12-42 The Bridge dialog box*

2. By default, the **Primary Faces** button is chosen from the **Selection Steps** area and you will be prompted to select the primary faces. Select the primary faces, as shown in Figure 12-43.



*Figure 12-43 Objects to be selected*



Note that you need to select the faces by specifying a point close to the edge, along which you want to create the bridge surface. Also, the arrowheads of the selected edges must be in the same direction.

3. Choose the **Side Faces** button from the **Selection Steps** area; you will be prompted to select the side faces. Select the side faces, as shown in Figure 12-43.
4. Next, you need to choose the continuity type from the **Continuity Type** area. You can specify a tangent or a curvature continuity between the selected faces and the bridge surface.

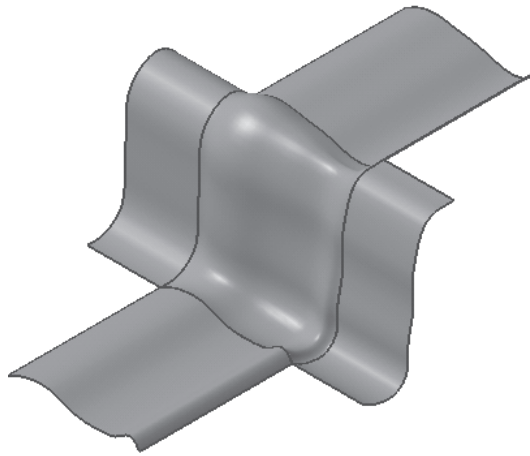
#### **Tangent**

The bridge surface will be tangent to the selected faces.

#### **Curvature**

The bridge surface will have a curvature.

5. Select the **Tangent** radio button and then the **Apply** button. The resulting bridge surface is shown in Figure 12-44.

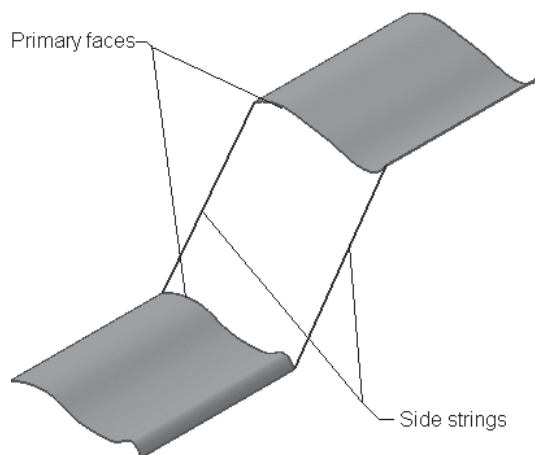


*Figure 12-44 The resulting bridge surface*

### **Creating Bridge Features by Side Strings**

The procedure to create the bridge feature by side strings is the same as for creating the side faces. In this type, instead of selecting the side faces, you have to select the side strings.

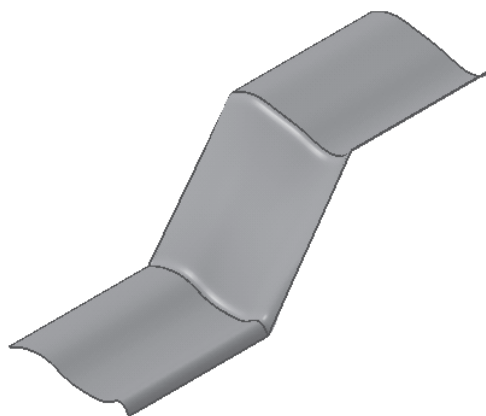
1. Choose the **Bridge** button from the **Surface** toolbar; the **Bridge** dialog box will be displayed and you will be prompted to select the primary faces. Select the primary faces, as shown in Figure 12-45.



**Figure 12-45** Objects to be selected

You need to select each face close to the edge along which you want to create the bridge surface. Also, the arrowheads of the selected edges must be in the same direction.

2. Choose the **First Side String** button from the **Selection Steps** area; you will be prompted to select the first side string. Select the curve, as shown in Figure 12-45.
3. Choose the **Second Side String** button from the **Selection Steps** area; you will be prompted to select the second side string. Select the curve, as shown in Figure 12-45.
4. Next, you need to select the continuity type from the **Continuity Type** area. Select the **Tangent** radio button. Choose the **OK** button. The resulting bridge surface is shown in Figure 12-46.



**Figure 12-46** The resulting bridge surface

## Creating Bridge Features by Dragging

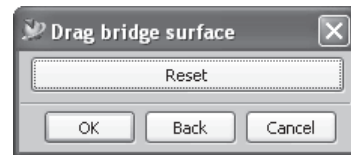
The following steps are required to create the bridge feature by dragging:

1. Choose the **Bridge** button from the **Surface** toolbar; the **Bridge** dialog box will be displayed. By default, the **Primary Faces** button is chosen from the **Selection Steps** area and you will be prompted to select the primary faces. Select the primary faces, as shown in Figure 12-43.

As mentioned earlier, you need to select each face close to the edge along which you want to create the bridge surface. Also, the arrowheads of the selected edges must be in the same direction.

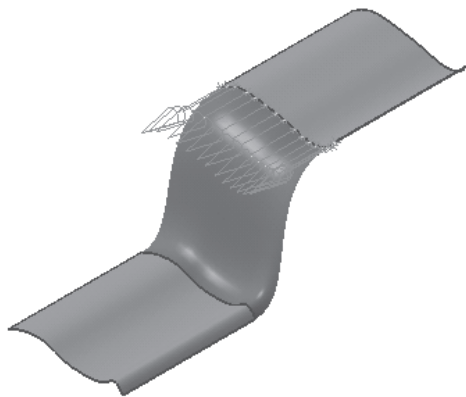
2. Select the **Tangent** radio button, if it is not already selected.

3. Choose the **Apply** button; the preview of the bridge surface will be displayed. Choose the **Drag** button from the **Bridge** dialog box; the **Drag bridge surface** dialog box will be displayed, as shown in Figure 12-47.

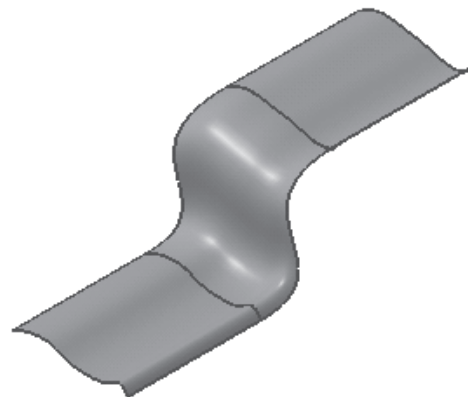


**Figure 12-47** The **Drag bridge surface** dialog box

4. Next, you will be prompted to drag the bridge surface. Press and hold the left mouse button down close to one of the primary surfaces; the series of vectors will appear on that edge, as shown in Figure 12-48. Holding the left mouse button down, drag the cursor to change the shape of the bridge dynamically.
5. Similarly, press and hold the left mouse button down close to the other edge of the bridge and modify it. After making the required modifications, choose the **OK** button. The **Reset** button from the **Drag bridge feature** dialog box is used to retain the bridge surface prior to the dragging. A bridge surface created by dragging is shown in Figure 12-49.



**Figure 12-48** The series of vectors on the edge

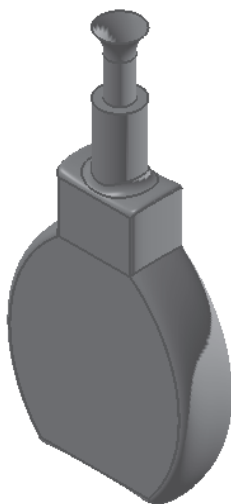


**Figure 12-49** The resulting bridge surface

## TUTORIALS

### Tutorial 1

In this tutorial, you will create the surface model shown in Figure 12-50. The dimensions and orthographic views are shown in Figure 12-51. Assume the missing dimensions. Save the model with the name `\NX 3\c12\c12tut1.prt` (Expected Time: 45 min)



*Figure 12-50 The surface model for Tutorial 1*

The following steps are required to complete this tutorial:

- Start a new file and invoke the **Modeling** application. Set the sheet environment.
- Create the base feature using the **Bounded Plane** tool, refer to Figures 12-52 and 12-53.
- Create the second feature using the **Bounded Plane** tool, refer to Figures 12-54 and 12-55.
- Create the third feature using the **Bridge** tool that will join the two bounded surfaces, refer to Figures 12-56 and 12-57.
- Mirror the bridge surface about the YC-ZC plane using the **Mirror** tool, refer to Figure 12-58.
- Create the bottom surface of the model using the **Bridge** tool, refer to Figures 12-59 and 12-60.
- Extract the curves from the edges and extrude the same, refer to Figures 12-61 and 12-62.
- Create the bridge surface over the extruded surface using the **Bridge** tool, refer to Figures 12-63 and 12-64.
- Create the extrude feature using the **Extrude** tool, refer to Figures 12-65 and 12-66.
- Create the face blend feature using the **Face Blend** tool, refer to Figures 12-67 and 12-68.
- Create the surface using the **Bounded Plane** tool, refer to Figure 12-69.
- Extrude the sketch, refer to Figures 12-70 and 12-71.
- Create the surface by **Through Curve** tool, refer to Figures 12-72, 12-73 and 12-74.
- Save and close the file.

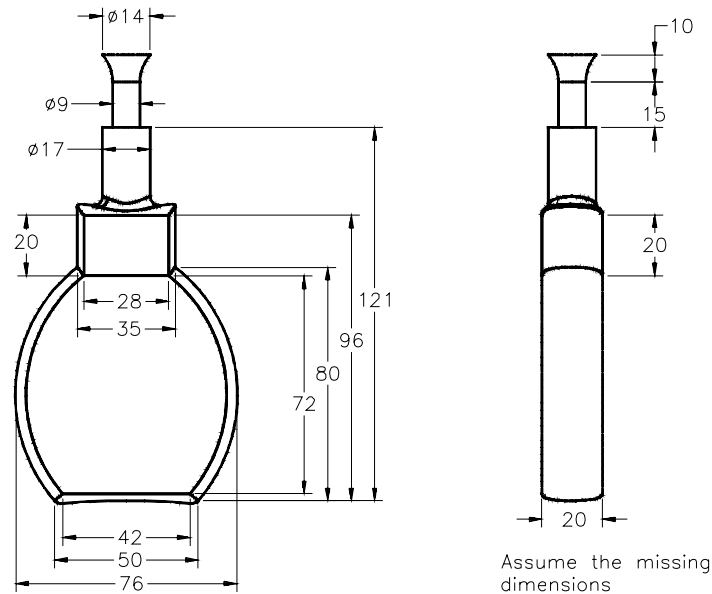


Figure 12-51 Views and dimensions for Tutorial 1

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

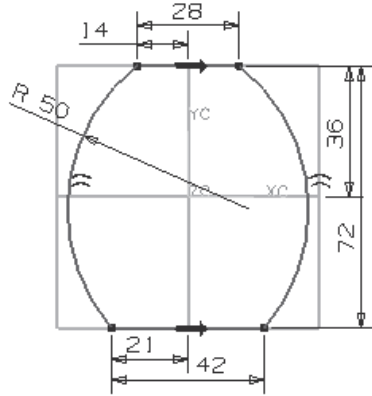
1. Start a new part file and enter the name as **c12tut1**.
2. Invoke the **Modeling** application and choose **Preferences > Modeling** from the menu bar; the **Modeling Preferences** dialog box will be displayed. Select the **Sheet** radio button from the **Body Type** area.
3. Create the three default datum planes.

### Creating the Base Feature Using the Bounded Plane Tool

The base feature for this tutorial needs to be created using the **Bounded Plane** tool. The following steps are required to create the base feature:

1. Draw the fully constrained sketch on the XC-ZC plane, as shown in Figure 12-52. Note the constraints applied to the sketch. Exit the **Sketcher** task environment.
2. Choose the **Bounded Plane** button from the **Form Feature** toolbar. The **Bounded Plane** dialog box is displayed and you are prompted to select the bounding string.

3. Select the sketch and choose the **OK** button. The base surface created, using the **Bounded Plane** tool, is shown in Figure 12-53.



*Figure 12-52 Sketch for the base feature*

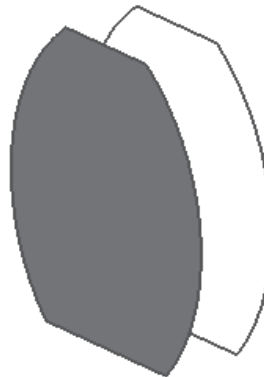


*Figure 12-53 Base feature of the surface model*

### Creating the Second Feature Using the Bounded Plane Tool

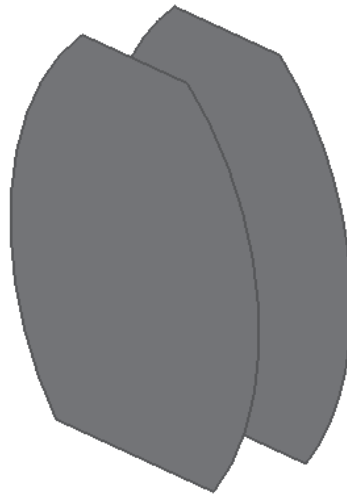
The second feature for this tutorial will be created using the **Bounded Plane** tool. The following steps are required to create the second feature:

1. Create a plane at an offset of 20 from the XC-ZC plane in the backward direction.
2. Select the offset plane as the sketching plane and invoke the **Sketcher** environment.
3. Choose the **Insert > Project** from the menu bar; the **Project Icon Options** are displayed and you are prompted to select the geometry to project. Select the base feature and choose the **OK** button.
4. Choose the **Finish Sketch** button from the **Sketcher** toolbar and exit the **Sketcher** task environment. The projected curves are shown in Figure 12-54.
5. Choose the **Bounded Plane** button from the **Form Feature** toolbar. The **Bounded Plane** dialog box is displayed and you are prompted to select the bounding string.
6. Select the projected curves, as shown in Figure 12-54, and choose the **OK** button.



*Figure 12-54 Sketch for the second feature*

The second feature created is shown in Figure 12-55.

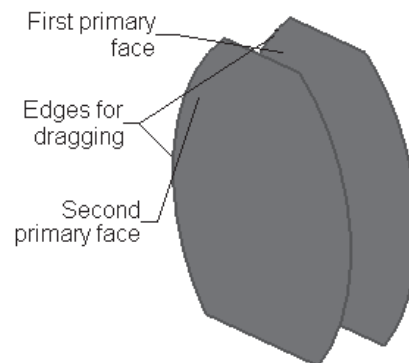


*Figure 12-55 Second feature of the surface model*

### Creating the Bridge Surface Using the Bridge Tool

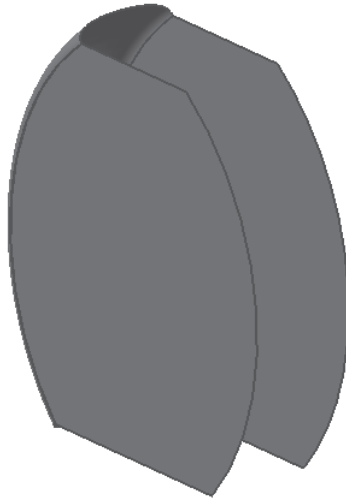
The third feature for this tutorial is a bridge surface. The following steps are required to create the bridge surface:

1. Choose the **Bridge** button from the **Surface** toolbar; the **Bridge** dialog box is displayed. By default, the **Primary Faces** button is chosen from the **Selection Steps** area and you are prompted to select the primary faces.
2. Select the primary faces using the sequence shown in Figure 12-56. Note that you need to select each face close to the edge along which you want to create the bridge surface. Also, the arrowheads of the selected edges must be in the same direction.
3. Next, select the **Tangent** radio button, if it is not already selected.
4. Choose the **Apply** button; the preview of the bridge surface is displayed.
5. Choose the **Drag** button from the **Bridge** dialog box; the **Drag bridge surface** dialog box is displayed and you are prompted to drag the bridge surface.
6. Select the edge close to one of the primary faces, as shown in Figure 12-56. The series of vectors appear on that edge.



*Figure 12-56 Objects to be selected*

7. Hold the left mouse button down and drag the cursor to change the shape of the bridge dynamically.
8. Similarly, modify the bridge at the other surface. The surface created using the **Bridge** tool is shown in Figure 12-57.



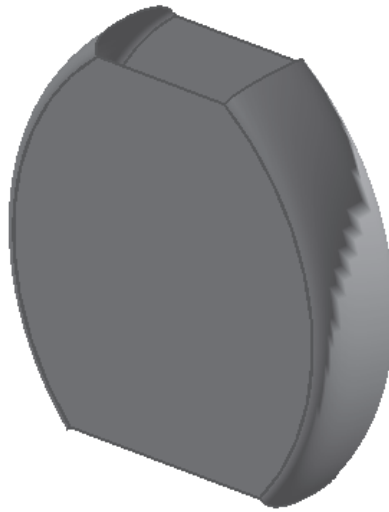
*Figure 12-57 The surface created using the **Bridge** tool*

### **Creating the Mirror Feature**

After creating the bridge surface, you need to mirror it about the datum plane. The following steps are required to create the mirror feature:

1. Choose the **Instance Feature** tool from the **Feature Operation** toolbar; the **Instance** dialog box is displayed.
2. Choose the **Mirror Feature** button; the **Mirror Feature** dialog box is displayed. By default, the **Feature to Mirror** button is chosen from the **Selection Steps** area and you are prompted to select the feature to be mirrored.
3. Select the bridge surface and choose the **Add** button; the feature is added in the **Features in Mirror** area.
4. Choose the **Mirror Plane** button from the **Selection Steps** area; you are prompted to select the plane for mirroring. Turn on the display of the YC-ZC plane by pressing the SHIFT+CTRL+K keys and then select it as the mirror plane.
5. Choose the **OK** button. Turn off the display of the plane. The model, after creating the mirror feature, is shown in Figure 12-58.



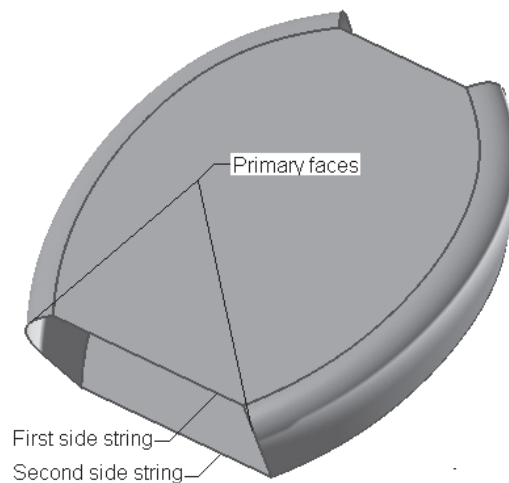


*Figure 12-58 The model after creating the mirror feature*

### Creating the Bottom Face of the Model Using the Bridge Tool

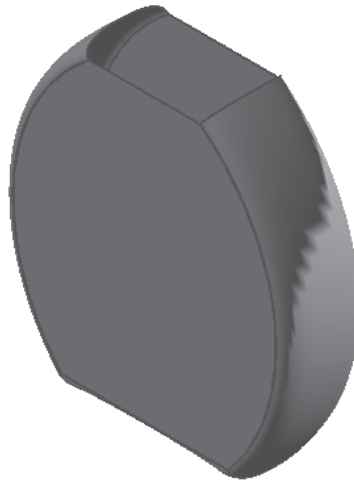
The fifth feature for this tutorial is the bottom surface of the model. The following steps are required to create the bottom surface of the model:

1. Choose the **Bridge** button from the **Surface** toolbar; the **Bridge** dialog box is displayed.
2. Select the primary faces, as shown in Figure 12-59.



*Figure 12-59 Objects to be selected*

3. Choose the **First Side String** button; you are prompted to select the first side string.
4. Select the edge of the model, as shown in Figure 12-59.
5. Choose the **Second Side String** button and select the second edge of the model, as shown in Figure 12-59.
6. Choose the **OK** button. The surface created using the **Bridge** tool is shown in Figure 12-60.

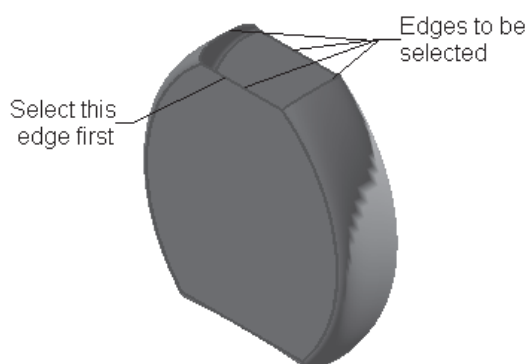


*Figure 12-60 The resulting bridge surface*

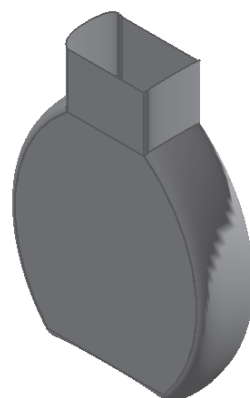
### **Creating the Sixth Feature by Extruding the Edges**

The sixth feature needs to be created by extruding the edges of the model. The following steps are required to create the sixth feature:

1. Choose the **Extrude** button from the **Form Feature** toolbar and select the edges of the model, as shown in Figure 12-61. Note that you need to select the edge of the first surface first. Else, the direction of extrusion will be different than what is required.
2. Enter the value as **20** in the **End** edit box.
3. Choose the **OK** button. The resulting surface model is shown in Figure 12-62.



**Figure 12-61** Edges to be selected

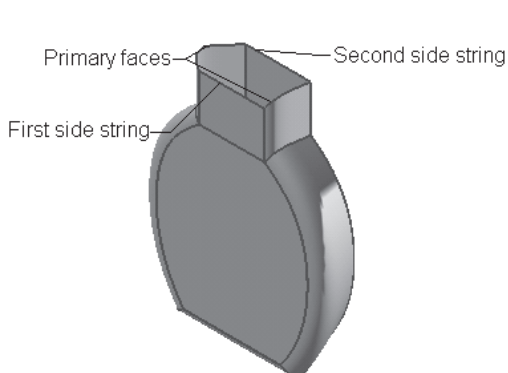


**Figure 12-62** The surface model after creating the extrude feature

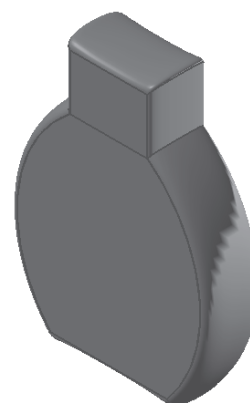
### Creating the Surface Using the Bridge Tool

The seventh feature for this tutorial will also be created using the bridge surface. The following steps are required to create the bridge surface:

1. Choose the **Bridge** button from the **Surface** toolbar; the **Bridge** dialog box is displayed. By default, the **Primary Faces** button is chosen from the **Selection Steps** area and you are prompted to select the primary faces.
2. Select the primary faces, as shown in Figure 12-63. The arrowheads of the selected edges must point in the same direction.
3. Choose the **First Side String** button; you are prompted to select the first side string. Select one of the edges of the model, as shown in Figure 12-63.
4. Choose the **Second Side String** button; you are prompted to select the second side string. Select the edge of the model, as shown in Figure 12-63. Choose the **OK** button. The surface created using the **Bridge** tool is shown in Figure 12-64.



**Figure 12-63** Objects to be selected



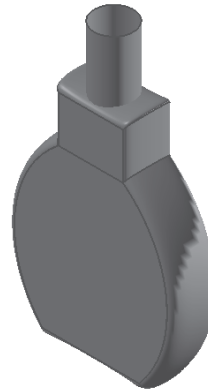
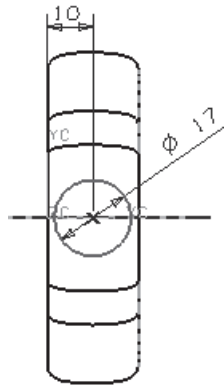
**Figure 12-64** The resulting bridge surface

### Creating the Extrude Feature

The eighth feature is created by extruding the sketch. You need to create the sketch for this feature on a plane that is at an offset of 85 from the XC-YC plane. The following steps are required to create the eighth feature:

1. Choose the **Datum Plane** button from the **Form Feature** toolbar; the **Datum Plane Icon Options** are displayed. Select the XC-YC plane; the **Offset** edit box is displayed.
2. Enter the value as **85** in the **Offset** edit box; a plane is created parallel to the XC-YC plane and at a distance of 85.
3. Create the sketch for the eighth feature by selecting the offset plane as the sketching plane, as shown in Figure 12-65.
4. Extrude this sketch through a distance of 30 in the downward direction.

The resulting surface model is shown in Figure 12-66.



**Figure 12-65** Sketch drawn for the extrude feature

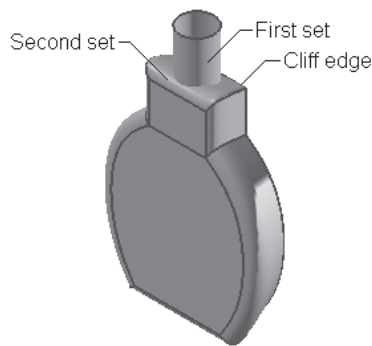
**Figure 12-66** The resulting extruded surface

### Creating the Face Blend Feature

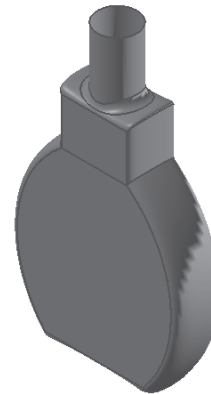
Next, you need to create a face blend feature to blend the extruded surface created in the previous section with the bridge surface. The blending will also ensure that the portion of the extruded surface that extends below the bridge surface is removed. The following steps are required to create the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

1. Choose the **Face Blend** button from the **Feature Operation** toolbar; the **Face Blend** dialog box is displayed.
2. By default, the **Sphere** option is selected from the **Blend Type** drop-down list and the **First Set** button is chosen from the **Selection Steps** area. You are prompted to select the faces for the first set. Select the face, as shown in Figure 12-67. Make sure the arrow points outward. If not, choose the **Reverse Normal** button to flip the direction.

3. Choose the **Second Set** button from the **Selection Step** area and select the second face set, as shown in Figure 12-67. If the arrow does not point upward, reverse the direction.
4. Choose the **Cliff Edges** button from the **Selection Step** area and select the cliff edge, as shown in Figure 12-67.
5. Select the **Trim and Attach All** from the **Attachment Method** drop-down list, if it is not selected already. Enter the value as 3 in the radius edit box and choose the **OK** button. The resulting face blend feature is shown in Figure 12-68.



**Figure 12-67** Entities to be selected

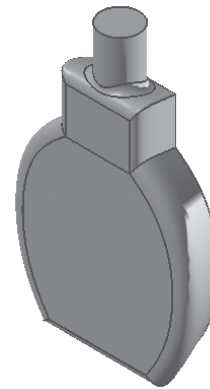


**Figure 12-68** The resulting face blend feature

### Creating the Tenth Feature using the Bounded Plane Tool

The tenth feature is created using the **Bounded Plane** tool. The following steps are required to create the tenth feature:

1. Choose the **Bounded Plane** button from the **Form Feature** toolbar; the **Bounded Plane** dialog box is displayed and you are prompted to select the bounding string.
2. Select the circular edge of the extruded feature created earlier.
3. Choose the **OK** button. The surface model, after creating the bounded plane feature, is shown in Figure 12-69

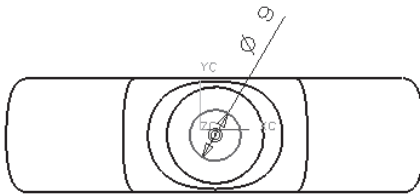


**Figure 12-69** The model after creating the bounded plane feature

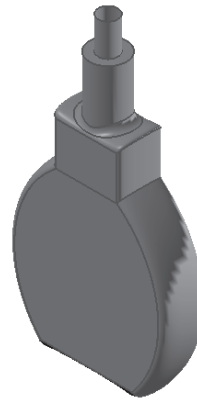
### Creating the Extrude Feature

The eleventh feature is created by extruding the sketch. You need to create the sketch for this feature on the bounded surface created earlier. The steps required to create the eleventh feature are given next.

1. Create the sketch for the eleventh feature by selecting the bounded surface created earlier as the sketching plane, as shown in Figure 12-70.
2. Choose the **Extrude** button from the **Form Feature** toolbar. The **Extrude Icon Options** are displayed and you are prompted to select the section geometry to be extruded.
3. Select the sketch and enter the value as **15** in the **End** edit box. Choose the **OK** button from the **Extrude Icon Options**. The resulting surface model is shown in Figure 12-71.



**Figure 12-70** Sketch drawn for the eleventh feature

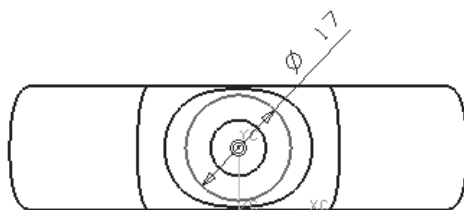


**Figure 12-71** The surface model after creating the extrude feature

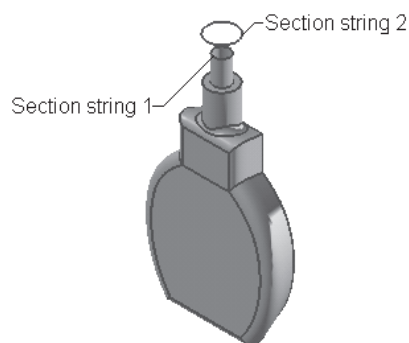
### Creating the Through Curves Surface

The twelfth feature is created using the **Through Curves** tool. You need to create the sketch for this feature on a plane at an offset of 109 from the XC-YC plane. The following steps are required to create the twelfth feature:

1. Create a datum plane at an offset of 109 mm from the XC-YC plane in the upward direction.
2. Create the sketch of the twelfth feature by selecting the offset plane as the sketching plane, as shown in Figure 12-72.
3. Choose the **Through Curves** button from the **Surface** toolbar; the **Through Curves** dialog box is displayed and you are prompted to select the section string 1.
4. Select the edge of the previous extruded feature, as shown in Figure 12-73, and choose the **OK** button; you are prompted to select the section string 2.
5. Select the sketch, as shown in Figure 12-73. Make sure the arrows in both sections point in the same direction. Choose the **OK** button; the **Through Curves** dialog box is displayed.
6. Select the **Tangency** option from the drop-down list in the **First Section String** area and choose the **OK** button. Select the previous extruded surface to define the tangency and choose the **OK** button.

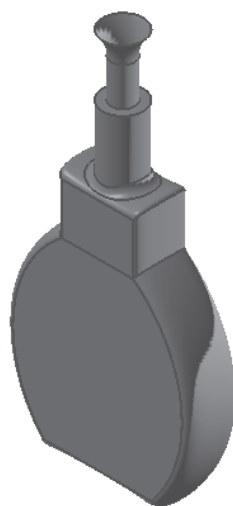


**Figure 12-72** Sketch for the through curve surface



**Figure 12-73** Objects to be selected

This completes the model for Tutorial 1. The final surface model is shown in Figure 12-74.



**Figure 12-74** The resulting through curves surface feature

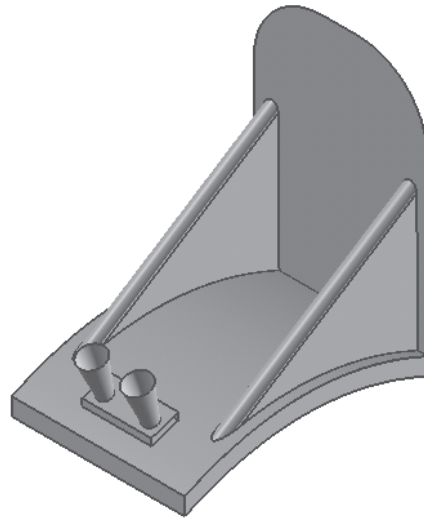
7. Save the model and close the file.

## Tutorial 2

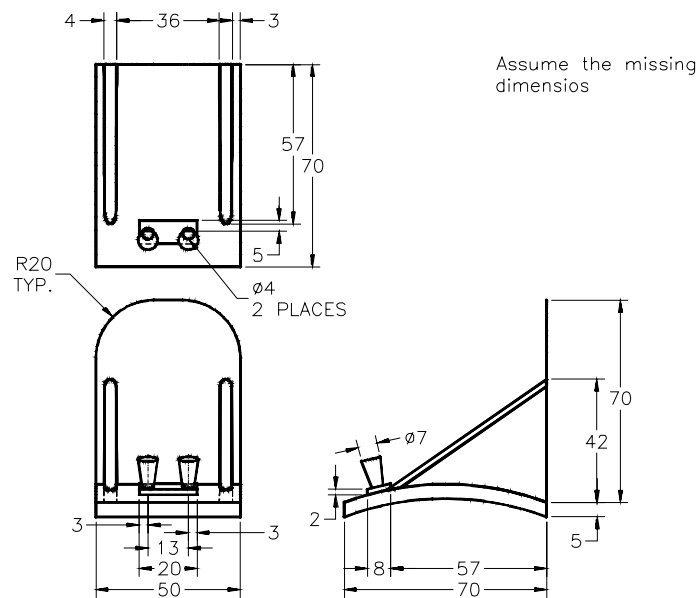
In this tutorial, you will create the surface model shown in Figure 12-75. The dimensions and orthographic views are shown in Figure 12-76. After creating the surface model, save it in the name mentioned below. Assume the missing dimensions.

\\NX 3\\c12\\c12tut2.prt

(Expected Time: 30 min)



**Figure 12-75** The surface model for Tutorial 2



**Figure 12-76** Views and dimensions for Tutorial 2



The following steps are required to complete this tutorial:

- Start a new file in the **Modeling** application of NX and set the sheet environment.
- Create the base feature by extruding the base sketch, refer to Figures 12-77 and 12-78.
- Create the second feature by extruding the edges of the base feature, refer to Figures 12-79 and 12-80.
- Create the third feature by extruding the sketch, refer to Figures 12-81 and 12-82.
- Create the fourth feature by extruding the sketch, refer to Figures 12-83 and 12-84.
- Create the fifth feature by extruding the edge from the base feature, refer to Figures 12-85 and 12-86.
- Trim the fifth feature using the **Trimmed Sheet** tool, refer to Figures 12-87 and 12-88.
- Create the dart feature, refer to Figure 12-89.
- Mirror the dart feature and the fourth extruded feature using the YC-ZC plane, refer to Figure 12-90.

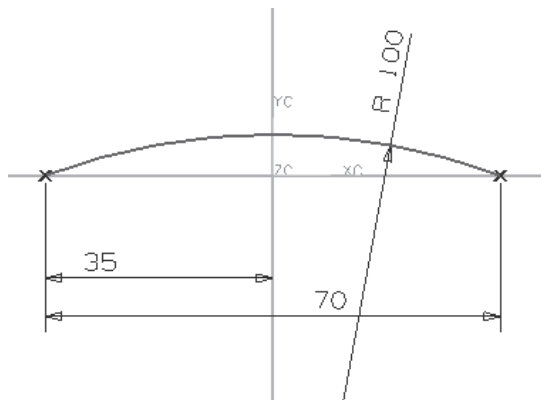
### Starting a New File and Setting the Sheet Environment

- Start a new part file with the name **c12tut2** in the **Modeling** application.
- Choose **Preferences > Modeling** from the menu bar and select the **Sheet** radio button from the **Body Type** area of the **Modeling Preferences** dialog box. Choose the **OK** button.

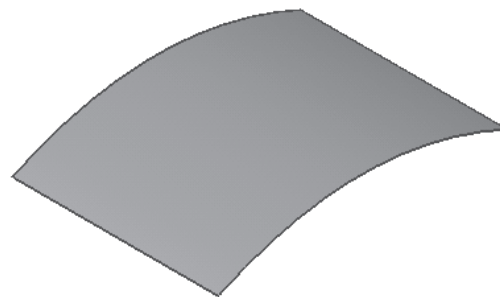
### Creating the Base Feature by Extruding the Sketch

The base feature for this tutorial should be created by extruding the sketch. The following steps are required to create the base feature:

- Create the three default datum planes and then create the sketch of the base feature on the YC-ZC plane, as shown in Figure 12-77.
- Extrude the sketch symmetrically on both sides of the sketching plane through a symmetric distance of 25. The base feature of the surface model is shown in Figure 12-78.



**Figure 12-77** Sketch of the base feature

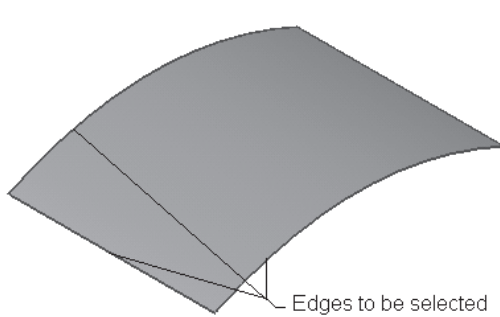


**Figure 12-78** Base feature of the surface model

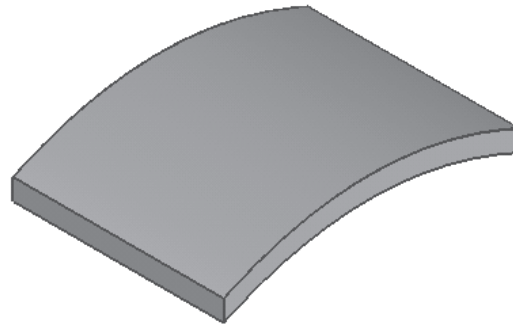
### Creating the Second Feature by Extruding the Edges

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

1. Invoke the **Extrude** tool and select the edges of the base surface, as shown in Figure 12-79.
2. Define the -Z axis direction as the direction of extrusion by choosing the **Direction** button from the **Extrude** dialog box. Enter **5** as the value in the **End** edit box.
3. Choose the **OK** button. The resulting surface model is shown in Figure 12-80.



*Figure 12-79 Object to be selected*



*Figure 12-80 The second feature created by the Extrude tool*

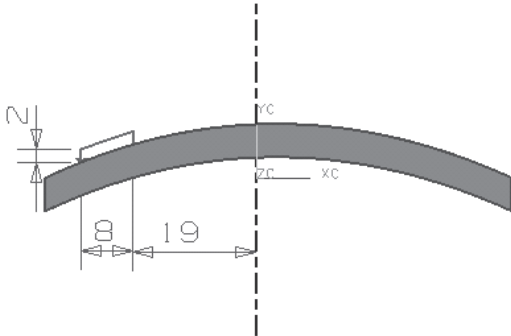
### Creating the Third Feature by Extruding the Sketch

The third feature is created by extruding a sketch. You need to create the sketch for the third feature on the YC-ZC plane. The following steps are required to create the third feature:

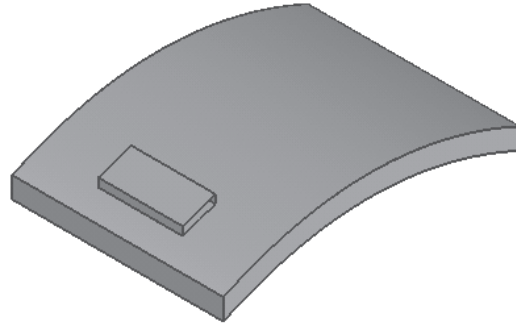
1. Create the sketch for the third feature by selecting the YC-ZC plane as the sketching plane, as shown in Figure 12-81.
2. Extrude the sketch through a symmetric distance of 10. The resulting surface model is shown in Figure 12-82.

### Creating the Fourth Feature by Extruding the Sketch

The fourth feature is also created by extruding the sketch. You need to create the sketch for the fourth feature on the top face of the third feature created. The following steps are required to create the fourth feature:

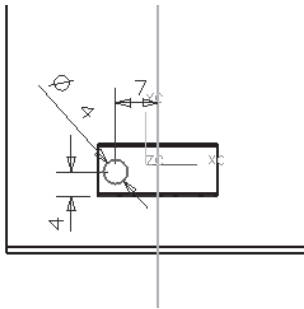


**Figure 12-81** Sketch drawn for extrusion

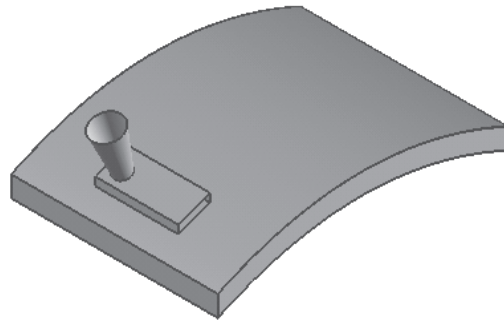


**Figure 12-82** The resulting extrusion feature

1. Create the sketch for the fourth feature by selecting the top face of the third feature as the sketching plane, as shown in Figure 12-83.
2. Extrude this sketch through a distance of 10 with -8 as the taper angle. The resulting surface model is shown in Figure 12-84.



**Figure 12-83** Sketch for the fourth feature



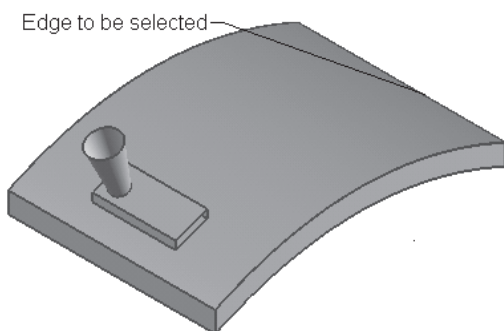
**Figure 12-84** The resulting fourth feature

### Creating the Fifth Feature by Extruding the Edge

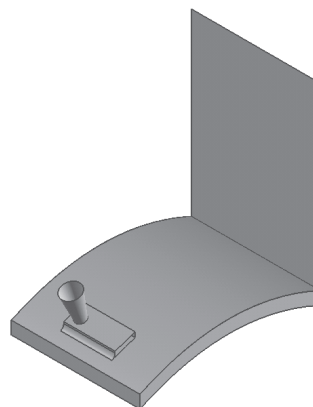
The fifth feature is created by extruding the edge of the base feature. The following steps are required to create the fifth feature:

1. Select the edge shown in Figure 12-85 and extrude it through a distance of 70 in the upward direction.

The resulting surface model is shown in Figure 12-86.



**Figure 12-85** The edge to be selected

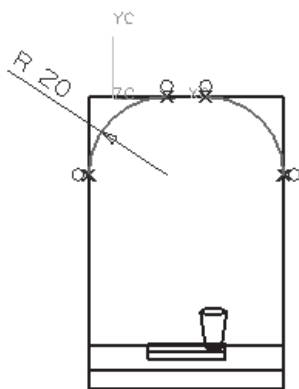


**Figure 12-86** The resulting fifth feature

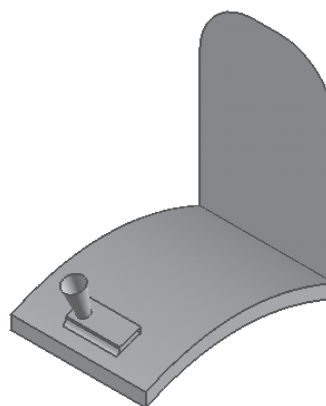
### Trimming the Fifth Feature Using the Trimmed Sheet Tool

The fifth feature will be trimmed using the **Trimmed Sheet** tool. The following steps are required for trimming the fifth feature:

1. Select the surface created earlier as the sketching plane and draw the sketch, as shown in Figure 12-87.
2. 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.
3. Select the fifth feature surface created; you are prompted to select the trimming objects. Select the sketch created.
4. Choose the **OK** button. The resulting surface model is shown in Figure 12-88.



**Figure 12-87** Sketch drawn for trimming

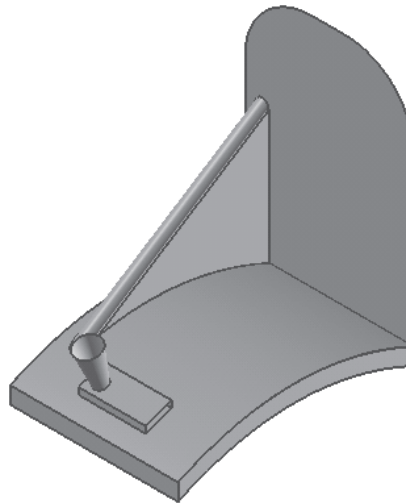


**Figure 12-88** The resulting surface model after trimming the fifth feature

### Creating the Dart Feature

The sixth feature is created using the **Dart** tool. The following steps are required to create the sixth feature:

1. Choose the **Dart** button from the **Form Feature** toolbar; the **Dart** dialog box is displayed. By default, the **First Set** button is chosen and you are prompted to select the faces for the firstset. Select the fifth feature surface.
2. Choose the **Second Set** button; you are prompted to select the faces for the second set. Select the base feature surface; the preview of the dart is displayed along the intersection curve of the two faces
3. Select the **Along Curve** option from the **Method** drop-down list, if it is not selected by default. Enter **10** as the value in the **%ArcLength** edit box.
4. Next, enter the dimensional values of the dart as **1**, **35**, and **2** in the **Angle**, **Depth**, and **Radius** edit boxes, respectively.
5. Choose the **OK** button. The dart feature created between the two selected surfaces is shown in Figure 12-89.



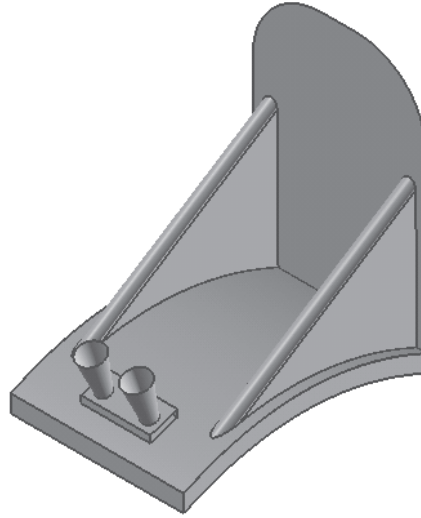
*Figure 12-89 The model after creating the dart feature*

### Creating the Mirror Feature

After creating the dart feature, you need to mirror the features about the datum plane. The following steps are required to create the mirror feature:

1. Invoke the **Mirror Feature** dialog box.

2. Select the dart and the fourth extruded feature and mirror them about the YC-ZC plane. The final model is shown in Figure 12-90.



*Figure 12-90 Final model for Tutorial 2*

### **Self-Evaluation Test**

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

1. The **Emboss Sheet** tool is used to convert the shapes of the solid bodies into a sheet. (T/F)
2. In the **Fillet** tool, you need to select the spline curve only for the general fillet. (T/F)
3. In the linear fillet, the radius will be linear through out the fillet. (T/F)
4. The dart feature can be created on the solids. (T/F)
5. The maximum number of primary faces that can be used to create the bridge surface is \_\_\_\_\_.
6. The \_\_\_\_\_ tool is used to create the fillet that has multiple radii.
7. The \_\_\_\_\_ tool is used to extract the edges from the solid body.

8. In the **Bridge** tool, the \_\_\_\_\_ button is used to retain its original state prior to the drag operation.
9. The \_\_\_\_\_ tool is used to create a fillet.
10. The \_\_\_\_\_ tool is used to create the S-shaped fillet.

### Review Questions

Answer the following questions:

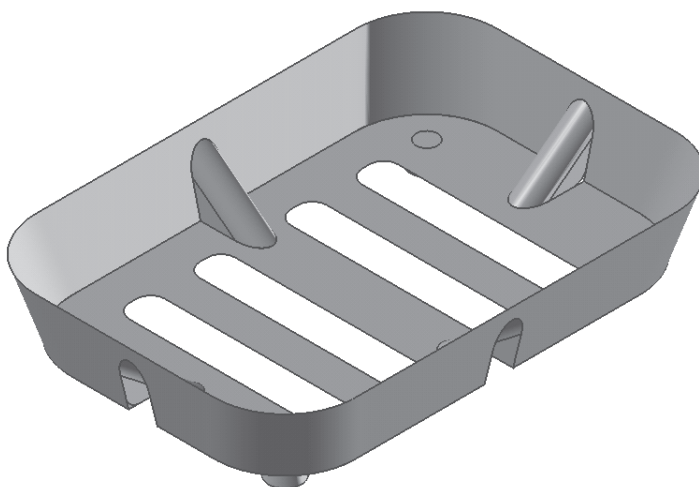
1. How many types fillets can be created?
  - a. Three
  - b. Four
  - c. Five
  - d. None of the above
2. Which toolbar is used to create the curves from the bodies?
  - a. **Intersection Curve**
  - b. **Extract Curves**
  - c. **Curve**
  - d. **Section Curves**
3. Which tool is used to create the intersection curves between two sets of objects?
  - a. **Curve Intersection**
  - b. **Intersection Curve**
  - b. **Extract Curves**
  - d. **Section curves**
4. Which tool is used to create the sheet body that joins the two edges of the faces?
  - a. **Bridge**
  - b. **Fillet**
  - c. **Soft Blend**
  - d. None of the above
5. To create the rib feature, you have to choose the **Dart** button from the **Feature Operation** toolbar. (T/F)
6. The shape of the bridge can be controlled by specifying the side faces or strings. (T/F)
7. The **Isoparameter Blend** tool is used for the turbine blades. (T/F)
8. You can create the surface body from a closed sketch. (T/F)
9. In the **Emboss Sheet** tool, you need to select the solid body as the **Tool Solid(s)**. (T/F)
10. The **Dart** tool is used to add a rib along the intersection curve of two faces. (T/F)

**Exercises****Exercise 1**

Create the surface model shown in Figure 12-91. The drawing views and the dimensions of the surface model are shown in Figure 12-92. Assume the missing dimensions. After creating the surface model, save it in the name mentioned below.

|NX 3|c12|c12exr1.prt

(Expected Time: 45 min)



**Figure 12-91** The surface model for Exercise 1



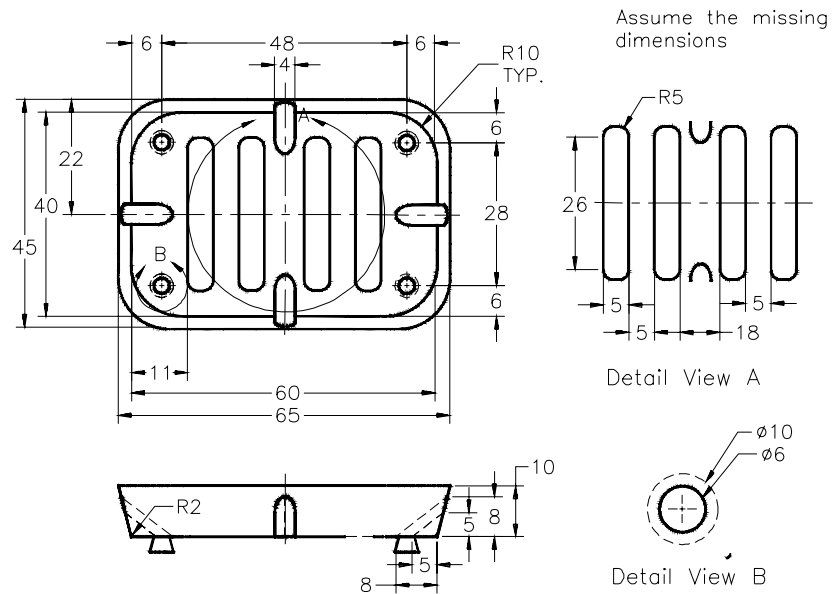


Figure 12-92 Views and dimensions for Exercise 1

## Exercise 2

Create the surface model shown in Figure 12-93. The drawing views and the dimensions of the surface model are shown in Figure 12-94. Assume the missing dimensions. After creating the surface model, save it with the name `\\NX 3\\c12\\c12exr2.prt` **(Expected Time: 45 min)**

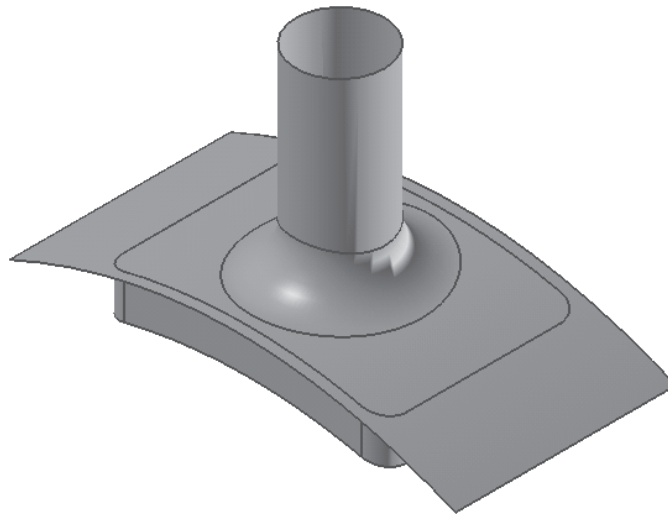
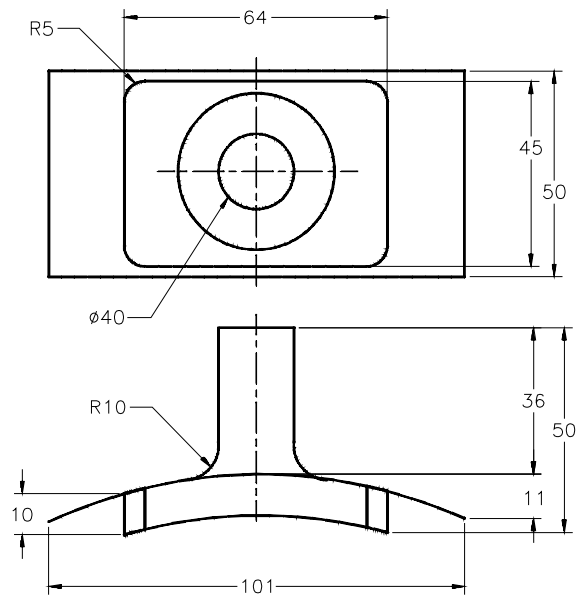


Figure 12-93 The surface model for Exercise 2



*Figure 12-94 Views and dimensions for Exercise 2*

**Answers to Self-Evaluation Test**

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