Chapter 4

Working with Fixed and Relative Datum Planes, Coordinate Systems, and Datum Axes

After completing this chapter, you will be able to:

• *Understand the usage of reference geometries in NX.*
• *Understand the different types of datum planes in NX .*
• *Create fixed and relative datum planes using various methods.*
• *Understand the usage of CSYS (coordinate system).*
• *Create coordinate systems using various methods.*
• *Understand the usage of the datum axis.*
• *Create the datum axis using various methods.*
• *Use additional extrude and revolve options.*
• *Project existing elements on the current sketching plane.*

ADDITIONAL SKETCHING AND REFERENCE PLANES

In previous chapters, you learned to create basic models, which had features placed on one of the fixed datum (principle) planes. All these models were created by selecting any of the fixed datum planes. Yet, most real world models consist of multiple sketched features, reference geometries, and placed features. In NX, features can be added to the base features by using the boolean operations such as create, subtract, unite and intersect, and so on. These operations are available in all tools. When you enter the **Sketcher Task** environment by choosing the **Sketch** button from the **Form Feature** toolbar, the **Sketch Icon Options** are displayed and you are prompted to select a sketching plane. You can select the available fixed datum buttons for creating the sketch. Alternatively, you can choose the **Datum Plane** button and create the additional planes. On the basis of the design requirement, you may select any plane as the sketching plane for the base feature. Also, you can create additional planes by taking the reference of the existing principle planes, faces, surfaces, sketches, or a combination of these objects. Figure 4-1 shows the model with multiple features.
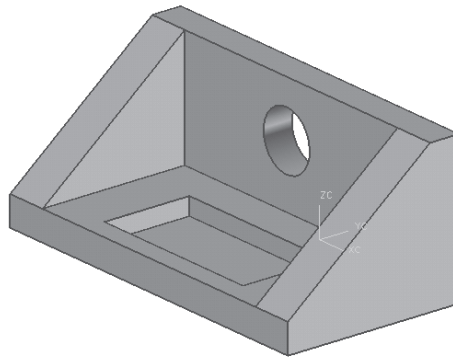

Figure 4-1 *Model with multiple features*

The base feature for the this model is shown in Figure 4-2. The sketch for the base feature is drawn on the XC-YC plane. As mentioned earlier, after creating the base feature, you need to create the other sketched features, placed features, and reference features, refer to Figure 4-3. The extrude features shown in Figure 4-3 requires additional sketching planes on which the sketch for the other features will be created.
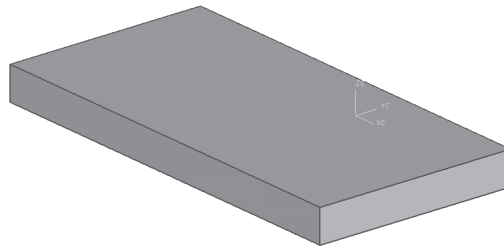

Figure 4-2 *The base feature for the model*

It is evident from Figure 4-3 that the additional features created on the base feature do not lie on the same sketching plane. They are created by defining additional sketching planes. Also, appropriate boolean operations are selected at the time of creating these features.
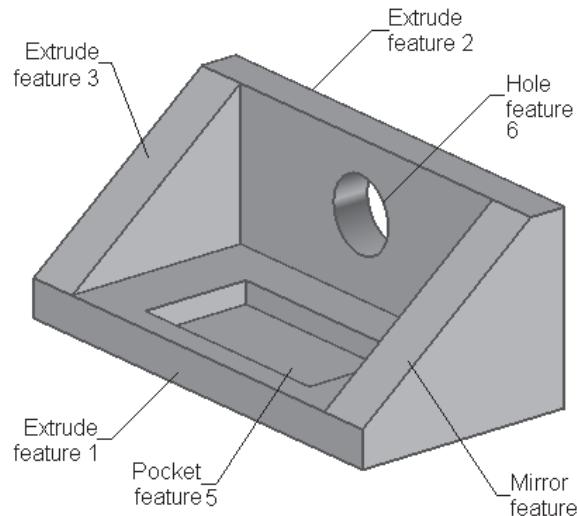

Figure 4-3 *Various features in the model*

As mentioned in earlier chapters, there will be no datum plane in a new part file that is started in the **Modeling** application of NX. Therefore, after starting a file in the **Modeling** application, it is recommended to create the three fixed datum planes as the first feature.

TYPES OF DATUM PLANES

There are two types of datum planes in NX: fixed datum planes and relative datum planes. Fixed datum planes are the three default planes that pass through the origin. Relative datum planes are created in addition to the fixed datum planes by taking the reference of objects such as curves, sketches, edges, faces, surfaces, and point. The methods of creating these datum planes are discussed next.

Creating the Fixed (Principle) Datum Planes

In NX, the principle datum planes are called fixed datum planes. By default, three fixed datum planes are available: XC-YC, YC-ZC, and XC-ZC. As mentioned earlier, you need to create the fixed datum planes once you start a new file. To create the fixed datum planes, choose the **Datum Plane** button from the **Form Feature** toolbar; the **Datum Plane** dialog box will be displayed, as shown in Figure 4-4, and you will be prompted to use the middle mouse button to create the datum plane. Choose the **Fixed Datum** button from the **Datum Plane** dialog box, if it is not already chosen; a drop-down list will be available below the button. By default, the **3 Planes of the WCS** option is selected from the drop-down list. Also, the preview of the fixed datum planes is displayed in green in the drawing window, as shown in Figure 4-5. To create the three fixed datum planes, choose the **OK** button.
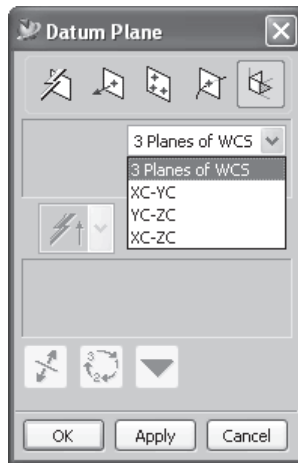
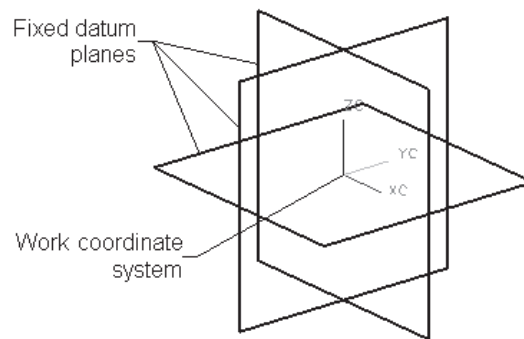*Figure 4-4* The **Datum Plane** *dialog box*

Figure 4-5 The fixed datum planes created

You can also create any one of the fixed datum planes. For example, to create only the XC-YC fixed datum plane, select the **XC-YC** option from the drop-down list below the **Fixed Datum** button. Similarly, to create the YC-ZC or XC-ZC fixed datum planes, select the respective option from the drop-down list.

Creating Relative Datum Planes

As mentioned earlier, relative datum planes are the additional planes that are created to assist you in completing the design. In NX, it is mandatory to have a model or the fixed datum planes for creating the relative datum planes. You can select objects such as curves, sketch, edge, face, surface, and points as the reference to create the relative datum planes. The **Datum Plane** tool from the **Form Feature** toolbar is used to create the relative datum planes. In NX, there are eleven different options for creating the relative datum planes. Most of these options can be

invoked by choosing the buttons in **the Datum Plane Icon Options**. The remaining options can be invoked using the **Datum Plane** dialog box. You can invoke this dialog box by choosing the **Datum Plane Dialog** button from the **Datum Plane Icon Options**. Various options for creating the relative datum planes are discussed next.

Creating Relative Parallel Planes

Menu:	Insert > Datum/Point > Datum Plane
Toolbar:	Form Feature > Datum Plane

The **Parallel** option is used to create a plane parallel to an existing planar face or surface, or an existing datum plane. To create a parallel plane, choose the **Datum Plane** tool from the **Form Feature** toolbar; the **Datum Plane Icon Options** will be displayed. By default, the **Inferred** button is active in the C**onstraints** flyout of the **Datum Plane Icon Options**. Also, you are prompted to select an edge, face, datum plane, datum axis, curve or point.

You need to follow two steps to create the parallel plane. First, choose the **Parallel** button from the **Constraint**s flyout in the **Datum Plane Icon Options**. Next, you need to select a planar face or surface, or a datum plane to which the new plane will be placed parallel. After you select the reference object, the preview of the parallel plane will be displayed in green, as shown in Figure 4-6. Also, an arrow will be displayed from the center of the newly created plane, along with the **Offset** edit box.

The second step is to locate the plane at a distance parallel to the reference objects selected. To do so, enter the offset value for the new plane in the **Offset** edit box. The arrow displayed from the center of the plane indicates the positive direction of the offset. If you need to offset the plane in the direction opposite to that of the displayed direction, enter a negative offset value. After entering the appropriate offset value, choose the **OK** button from the **Datum Plane Icon Options** to create the plane, as shown in Figure 4-7.
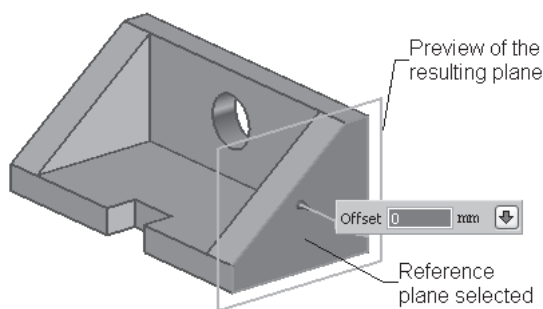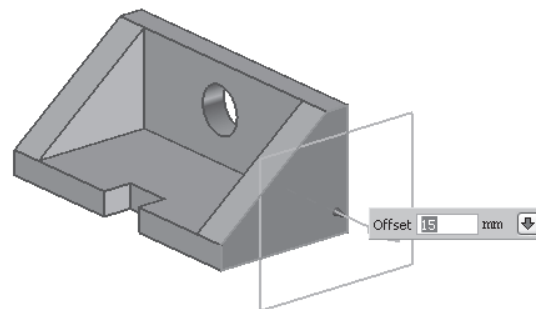

Figure 4-6 *The preview of the plane created*

Figure 4-7 *The preview of the parallel plane after entering the offset value*

Note
*While creating parallel planes, you can only select a planar face or surface as the reference object.
You cannot select a nonplanar face or surface as the reference face.*

Creating Relative Coincident Planes

Menu:	Insert > Datum/Point > Datum Plane
Toolbar:	Form Feature > Datum Plane

The **Coincident** option is used to create a relative datum plane coincident to a point, vertex, or a linear entity. To create a coincident plane, choose the **Datum Plane** tool from the **Form Feature** toolbar; the **Datum Plane Icon Options** are displayed. Choose the **Coincident** button from the **Constraint** drop-down list**.** Note that you need to select at least two geometries to create the datum plane using this method. You can select a point or vertex, or a linear entity as the first and second geometry. However, note that if you select points or vertices as the first and second geometries, you need to specify another entity to complete the plane creation. To enable easy selection of a reference object, you can clear all buttons from the **Snap Point** toolbar. After selecting the second reference object, choose the **OK** button from the **Datum Plane Icon Options** to create the coincident plane. Figure 4-8 shows the reference entities selected and the preview of the resulting plane and Figure 4-9 shows the resulting plane.
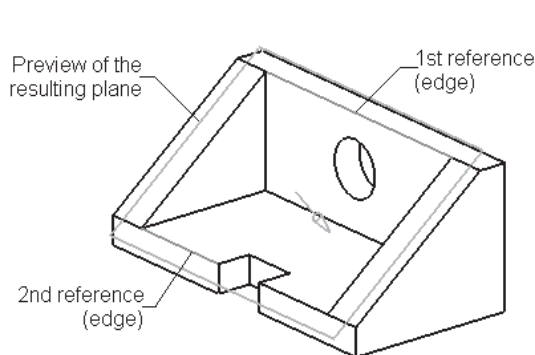

Figure 4-8 Entities selected to create a coincident plane and the preview of the resulting plane
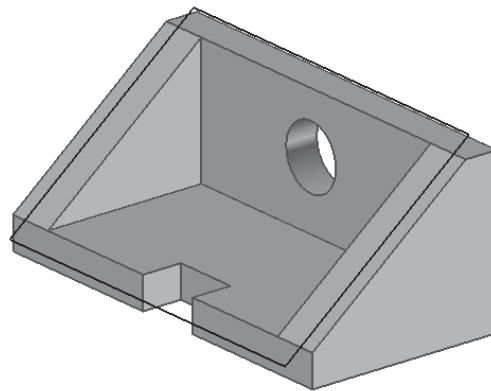
Figure 4-9 The resulting coincident plane

Note
You cannot select a face, surface, or a curve as the reference object for creating the coincident plane.

Creating Relative Perpendicular Planes

Menu:	Insert > Datum/Point > Datum Plane
Toolbar:	Form Feature > Datum Plane

The **Perpendicular** option is used to create a relative datum plane perpendicular to an existing planar face or datum plane. To create a perpendicular plane, choose the **Datum**

Plane tool from the **Form Feature** toolbar; the **Datum Plane Icon Options** will be displayed. Choose the **Perpendicular** button from the **Constraint** drop-down list**.**

You need to select a linear entity and a planar face or datum plane to create a relative datum plane using this option. The linear entity can be a sketched line or an edge. After you select the reference object for placing the plane, the **Angle** edit box will be displayed, along with the angular ruler, as shown in Figure 4-10. By default, the value in the **Angle** edit box is 90. You can enter any value in the same edit box, in case you do not want to create a perpendicular plane. Figure 4-11 shows a relative datum plane created using these parameters.
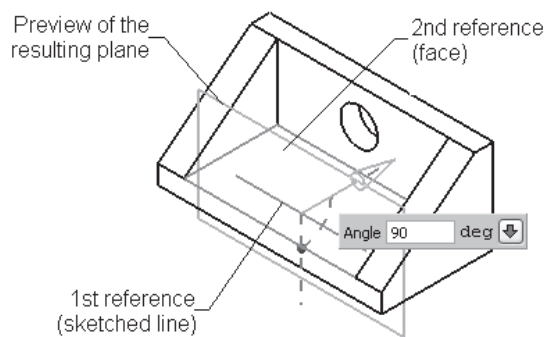

Figure 4-10   *The preview of the perpendicular plane along with the **Angle** edit box*
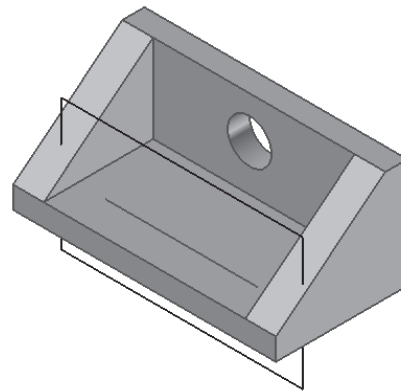
Figure 4-11   *The resulting perpendicular plane created*

Creating Relative Angular Planes

Menu:	Insert > Datum/Point > Datum Plane
Toolbar:	Form Feature > Datum Plane

The **Angle** option is used to create a plane at an angle to another plane passing through an edge, linear sketched segment, or axis. To create an angular plane, invoke the **Datum Plane Icon Options** and choose the **Angle** button from the **Constraints** flyout. Now, select a planar face or a datum plane from which the resulting plane will be at an angle. Next, you need to select an edge or a linear sketch through which the resulting plane will pass. On selecting both references, the **Angle** edit box will be displayed along with the angular ruler, as shown in Figure 4-12. By default, the value is displayed as **90** in the **Angle** edit box. You can enter any value in the same edit box. Figure 4-13 shows the datum plane created at an angle of 65- degree.
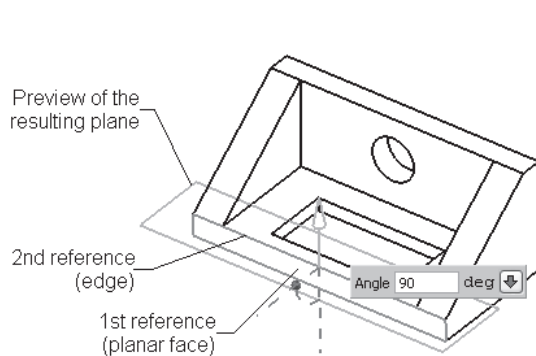
*Figure 4-12 The preview of the angular plane displayed along with the **Angle** edit box*
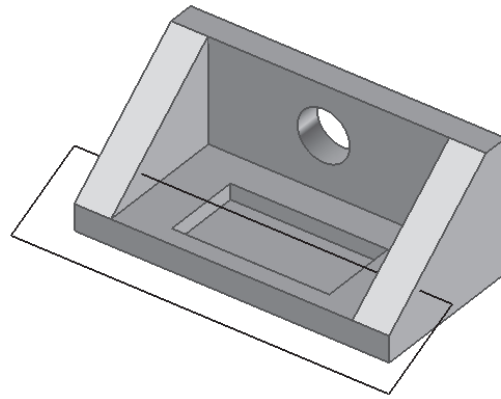
Figure 4-13 The resulting plane created at an angle of 65-degree

Creating Relative Offset Planes

Menu:	Insert > Datum/Point > Datum Plane
Toolbar:	Form Feature > Datum Plane

The **Offset** option is used to create a plane that is at an offset from an existing planar face, planar surface, fixed datum plane, or an existing relative datum plane. To create this type of relative plane, choose the **Offset** option from the **Constraints** flyout of the **Datum Plane Icon Options.** Next, select a planar face, surface, or a datum plane; the preview of the offset plane will be displayed, along with the **Offset** edit box, as shown in Figure 4-14. By default, the value 0 is displayed in the **Offset** edit box. You can enter the desired offset value in this edit box. Note that to offset in reverse direction, and so you need to enter a negative value. Figure 4-15 shows an offset datum plane.
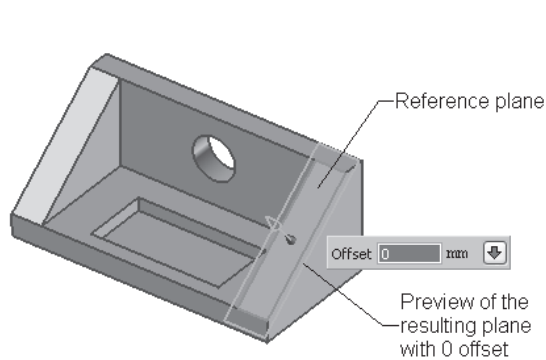

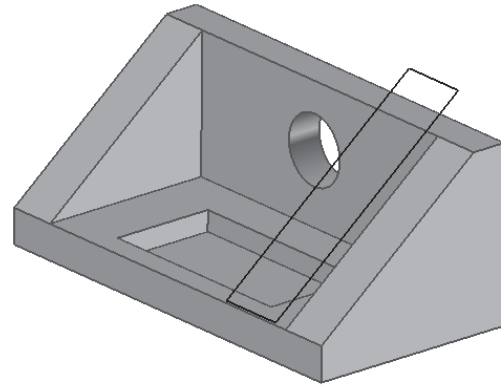*Figure 4-14 The preview of the offset plane displayed along with the **Offset** edit box*

*Figure 4-15 The resulting offset plane created after entering the offset value in the **Offset** edit box*

Creating Relative Tangential Planes

Menu:	Insert > Datum/Point > Datum Plane
Toolbar:	Form Feature > Datum Plane

The **Tangent** option is used to create a tangential plane. You need to select a cylindrical face or a cylindrical surface to create this type of plane. To create a tangential plane, choose the **Tangent** option from the **Constraints** flyout of the **Datum Plane Icon Options**. Now, select a cylindrical face or surface; the preview of the plane will be displayed, as shown in Figure 4-16.

Next, you need to position the tangential plane. To do so, specify a quadrant point from the edge of the reference face selected. To enable an easy selection of the quadrant point from the edge of the reference surface selected, you can turn on the **Quadrant Point** and **Point on Curve** buttons and turn off the **Arc Center** button from the **Snap Point** toolbar. Figure 4-17 shows a tangent datum plane created at one of the quadrant points.
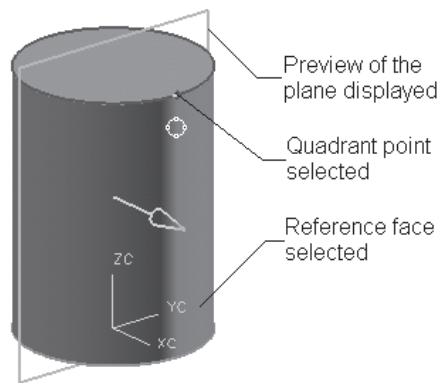

Figure 4-16 *The preview of the tangential plane displayed along with the quadrant point selected*

Figure 4-17 *The resulting tangential plane created after specifying the quadrant point*

Creating Relative Planes in the Center of Two Planes or Planar Faces

Menu:	Insert > Datum/Point > Datum Plane
Toolbar:	Form Feature > Datum Plane

The **Center** option is used to create a relative plane that is in the center of two specified planes or planar faces. To create a tangential plane, choose the **Center** option from the **Constraints** flyout of the **Datum Plane Icon Options**. Then, one by one, select two planes or planar faces; the preview of the resulting plane placed at the center of the two selected planes or planar faces will be displayed, as shown in Figure 4-18. Choose **OK** from the **Datum Plane Icon Options** to create this plane.
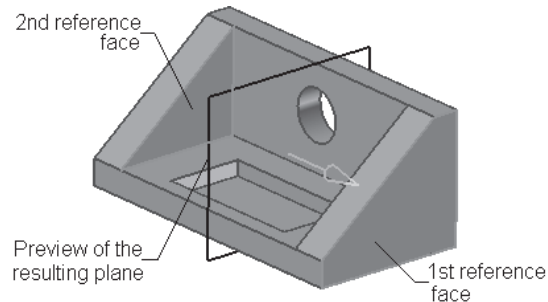
Figure 4-18 *References selected to create the center plane and the resulting plane*

Creating Planes Using the Inferred Tool

Menu: Insert > Datum/Point > Datum Plane
Toolbar: Form Feature > Datum Plane

The **Inferred** option is used to create various types of planes, depending on the reference objects selected and the sequence, in which they are selected.

To create planes using this tool, invoke the **Datum Plane Icon Options**. By default, the **Inferred** option is selected from the flyout in the **Datum Plane Icon Options**. Depending on the reference objects selected and the sequence, in which you select the reference objects, appropriate planes will be created. For example, if you select two parallel planes, then the center plane will be created.

Creating Planes by Specifying a Point and the Direction

Menu: Insert > Datum/Point > Datum Plane
Toolbar: Form Feature > Datum Plane

The **Point and Direction** button in the **Datum Plane** dialog box is used to create a plane at a specified point and oriented along a selected direction. You need to specify a fixing point and define the direction to create the plane. The direction can be defined using an edge, linear sketch, or datum plane. If you select an edge or a linear sketch to define the direction, the plane will be created normal to it and will pass through the specified point. However, if you select a datum plane or a planar face, the plane will be created parallel to it and will pass through the specified point.

To create this type of plane, invoke the **Datum Plane Icon Options**. Next, choose the **Datum Plane Dialog** button from the same; the **Datum Plane** dialog box will be displayed. By default, the **Inferred** button is chosen from the **Datum Plane** dialog box. Choose the **Point and Direction** button; you will be prompted to select a point. Specify the point at which the plane is to be

created. Before specifying the point, you may need to turn on the required button from the **Snap Point** toolbar for enabling an easy selection of the point. After specifying the point, the preview of the plane will be displayed.

Next, you need to define the direction for the plane by selecting a linear edge, sketch, planar face, or surface. You can also select a cylindrical face to define the direction. If you select a cylindrical face as a reference for defining the direction, the plane will be placed perpendicular to the axis of the cylindrical face selected. Figure 4-19 shows the reference selected to create this type of plane and Figure 4-20 shows the resulting plane.
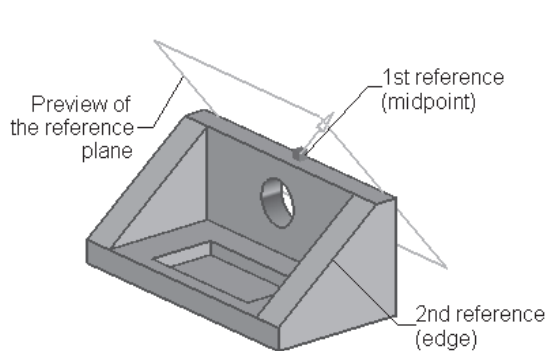

Figure 4-19 *The preview of the plane created using the* *Point and Direction* *button*
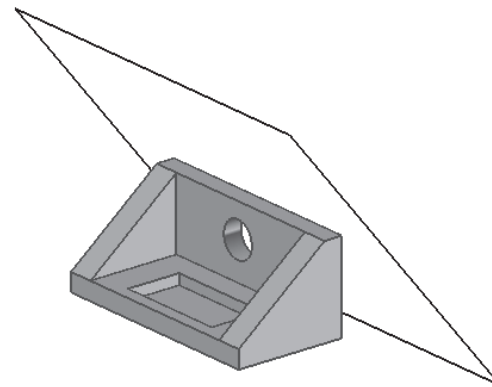
Figure 4-20 *The resulting plane created using the* *Point and Direction* *button*

Note
After specifying the point, a drop-down list will be available in the *Datum Plane* *dialog box. This drop-down list contains the options for defining the direction. By default, the* *Inferred* *option is selected from this drop-down list. To define the direction by constructing a new vector, select the* *Vector Constructor* *option from this drop-down list; the* *Vector Constructor* *dialog box will be displayed, using which you can construct the new vector for defining the direction.*

Creating Planes Using the Three Points Tool

Menu:	Insert > Datum/Point > Datum Plane
Toolbar:	Form Feature > Datum Plane

The **Three Points** button in the **Datum Plane** dialog box is used to create a plane by specifying three points. You need to specify three fixing points for creating the plane. To create a plane using this method, invoke the **Datum Plane** dialog box and choose the **Three Points** button; you will be prompted to select a point. Before specifying the point, you may need to turn on the required button from the **Snap Point** toolbar for enabling an easy selection. Specify the first point. Similarly, specify the second and third points. After specifying all three points, the preview of the plane will be displayed, as shown in Figure 4-21. Figure 4-22 shows the resulting plane.
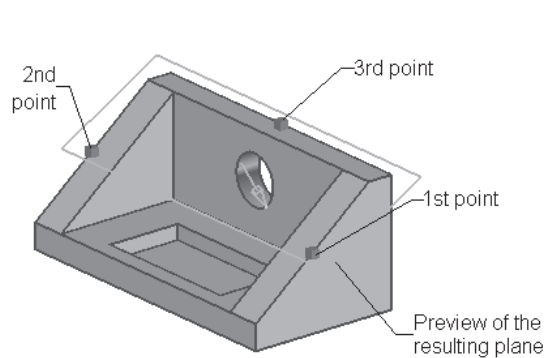
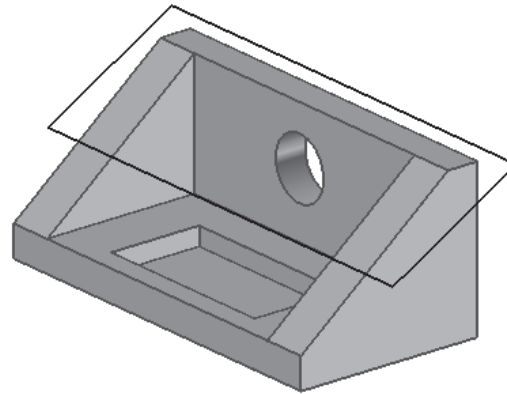*Figure 4-21* *The preview of the plane created using the* **Three Points** *option*

Figure 4-22 *The resulting plane created using the* **Three Points** *option*

Creating Planes Using the Point on Curve Tool

Menu:	Insert > Datum/Point > Datum Plane
Toolbar:	Form Feature > Datum Plane

The **Point on Curve** button from the **Datum Plane** dialog box is used to create a plane passing through a selected curve and is located at a specified point. To create a plane using this option, choose the **Point on Curve** button from the **Datum plane** dialog box. Select a curve or an edge to create the plane; the preview of the plane, along with the **Arclength** edit box will be displayed, as shown in Figure 4-23. By default, the current positional value of the plane will be displayed in the **Arclength** edit box.

Next, you need to specify the location of the resulting plane on the selected curve. You can locate the plane on the curve by entering the distance value in the **Arclength** edit box. If you want to enter the value in terms of percentage of the total length of the curve, select the **% Arclength** check box below the **Arclength** edit box in the **Datum Plane** dialog box. When you select this check box, the **Arclength** edit box will be changed to **% Arclength** edit box. The length of the reference object selected will be taken as 100% and the value entered thereafter will be taken in terms of percentage of the total length and the plane will be located at that point. For example, if you enter the value as **0** inside the **% Arclength** edit box, then the plane is placed on the start points of the reference object selected. Note that you can also drag the cube in the preview of the plane to modify its location on the curve. Figure 4-24 shows a plane created using this method.

The **Alternate Solution** button in the **Datum plane** dialog box will be enabled after selecting the curve. This button can be used to invoke the other possible orientations for the resulting plane. This button will be automatically disabled after choosing the **Apply** button from the **Datum Plane** dialog box.
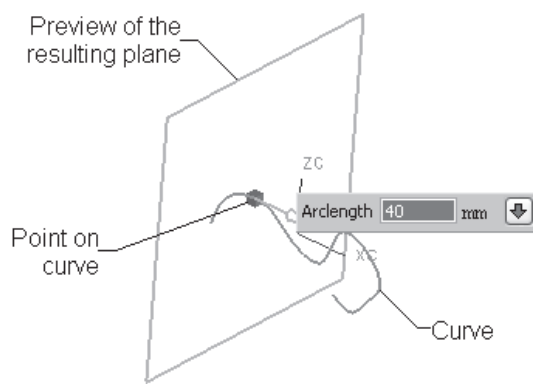
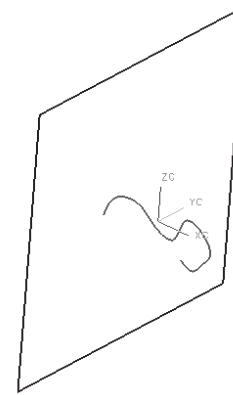*Figure 4-23* *The preview of the plane created using the* **Point on Curve** *button*

Figure 4-24 *The resulting plane created using the* **point on Curve** *button*

Creating Associative Datum Coordinate Systems

As mentioned earlier, when you open a new file in NX, you will only have the default handles for the file. These handles represent the X, Y, and Z directions on the 3D space and form the basics in creating both the fixed and relative datum planes. Other than these handles, you can also create new coordinate systems for the separate features in a part. A part may contain any number of coordinate systems. The coordinate system can be used as a reference for creating features, sketches, and curves. Also, it can be used for assembling the component parts in the assembly. The coordinate system is also treated as a feature and is displayed in the **Unused** section of the **Part Navigator**. The coordinate system is always associative with the object members or feature operation to which it is related. There are five methods by which you can create an associative coordinate system. The methods for creating the coordinate system are discussed next.

Note
The coordinate system can only be created when there is a reference object existing in the file. The reference objects may be an existing coordinate system, face, surface, or point.

Creating A Coordinate System Using the Origin, X-Point, Y-Point Tool

Menu:	Insert > Datum/Point > Datum CSYS
Toolbar:	Form Feature > Datum CSYS

To create the coordinate system using this tool, choose the **Datum CSYS** button from the **Form Feature** toolbar; the **CSYS Constructor** dialog box will be displayed, as shown in Figure 4-25. By default, the **Inferred** button is chosen from the **Inferred** area of the **CSYS Constructor** dialog box and you will be prompted to define the associative coordinate system, choose the method, or select objects.

Choose the **Origin, X-point, Y-Point** button from the **Inferred** area of the **CSYS Constructor**
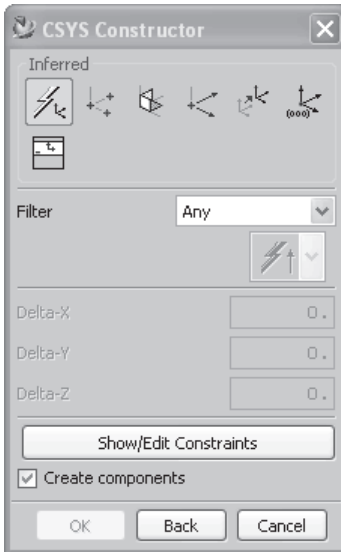
*Figure 4-25 The **CSYS Constructor** dialog box*

dialog box; you will be prompted to specify the origin of the coordinate system**.** For creating the coordinate system using this method, you first need to specify the origin point. Select the point on which you need to fix the coordinate system. At the time of specifying the origin point, turn on the required snap button from the **Snap Point** toolbar to enable an easy selection of points. After specifying the origin point, you need to specify a point along both X and Y directions. Now, specify the X and Y points. Note that the X point is used to specify the X axis direction of the coordinate system and the Y point is used to define the orientation of the XY plane.

After you specify the origin, X, and Y points, the preview of the coordinate system will be displayed, as shown in Figure 4-26. Choose **Apply** and then the **OK** button from the **CSYS Constructor** dialog box to accept the coordinate system. If the **Create Components** check box is selected, the three datum planes will also be created with the coordinate system, as shown in Figure 4-27.

Creating a Coordinate System Using the Three Planes Tool

Menu:	Insert > Datum/Point > Datum CSYS
Toolbar:	Form Feature > Datum CSYS

To create the coordinate system using this tool, choose the **Datum CSYS** button from the **Form Feature** toolbar; the **CSYS Constructor** dialog box will be displayed. By default, the **Inferred** button is chosen from the **Inferred** area of the **CSYS Constructor** dialog box and you will be prompted to define the associative CSYS or choose the method or select objects. Choose the **Three Planes** button from the **Inferred** area of the **CSYS Constructor** dialog box; you will be prompted to select a plane**.** Also, the **Filter** drop-down list is displayed in the **CSYS Constructor** dialog box**.** For creating the coordinate system using this method, you
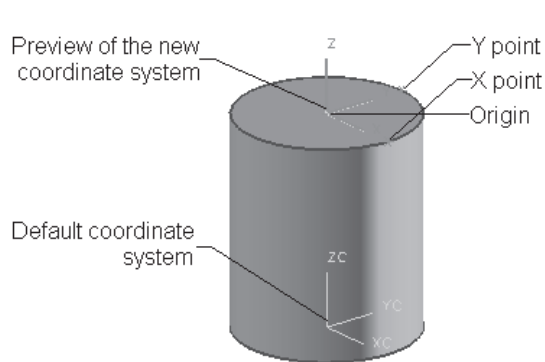
Figure 4-26 The preview of the CSYS created using the **Origin, X point, Y point tool**
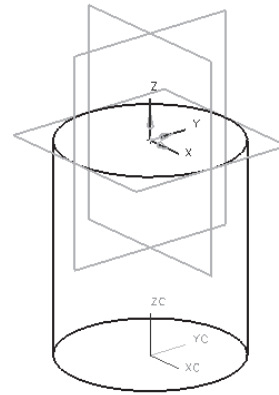

Figure 4-27 Wireframe display of the model with the new coordinate system and the related datum planes

need to select three reference objects that are mutually perpendicular to each other. Only, faces or planes can be selected as the reference object. The following is the procedure of creating this type of coordinate system:

Select the type of reference object you need to select from the **Filter** drop-down list. This allows you to customize the process of selecting a particular type of reference object. Next, you need to select three reference objects. If the reference object selected are not mutually perpendicular, then the **Message** window will be displayed and you will be informed that no intersection could be found. After selecting the three reference objects, the preview of the CSYS will be displayed, as shown in Figure 4-28. Choose **Apply** and then the **OK** button from the **CSYS Constructor** dialog box to accept the coordinate system. The resulting coordinate system is shown in Figure 4-29.
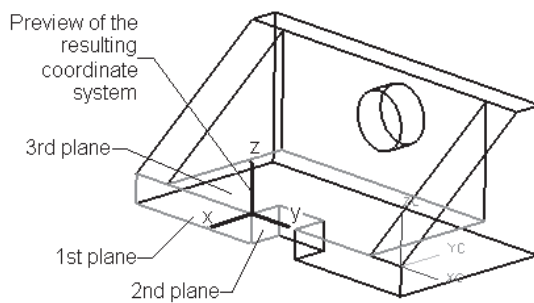

Figure 4-28 The preview of the CSYS created using the **Three Planes** tool
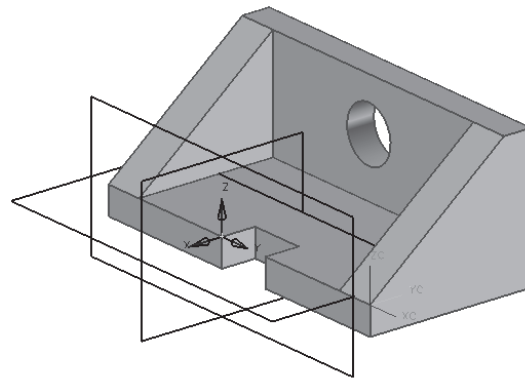

Figure 4-29 The resulting CSYS created using the **Three Planes** tool

Creating Coordinate Systems Using the Offset from CSYS Tool

Menu:	Insert > Datum/Point > Datum CSYS
Toolbar:	Form Feature > Datum CSYS

To create a coordinate system using this tool, choose the **Datum CSYS** button from the **Form Feature** toolbar; the **CSYS Constructor** dialog box will be displayed. Choose the **Offset from CSYS** button from the **Inferred** area of the **CSYS Constructor** dialog box; you will be prompted to select an existing coordinate system**.** Select an existing coordinate system; the **Delta-X, Delta-Y**, and **Delta-Z** edit boxes will be enabled in the **CSYS Constructor** dialog box. By default, the values will be displayed as **0** in all edit boxes. Also, the coordinate system will be created over the one that has been selected as the reference. Next, you need to position the coordinate system. To do so, you need to enter the offset values along X, Y, and Z directions in the corresponding edit boxes. You can also directly locate the origin point for the coordinate system by dynamically clicking in the graphics window. After specifying the origin point for the coordinate system, choose the OK button from the **CSYS Constructor** dialog box. The resulting coordinate system will be displayed, as shown in Figure 4-30.
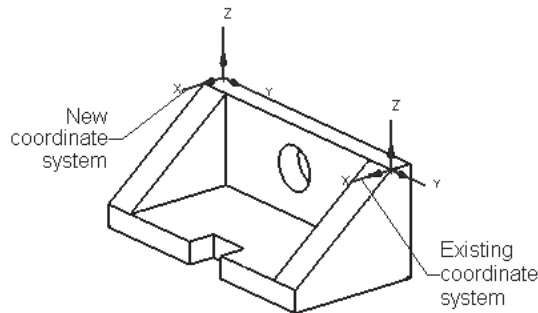

Figure 4-30 *The resulting coordinate system created using the **Offset from CSYS** tool*

Note
*To modify the location of the coordinate system, after it is created, double-click on it. On doing so, the **CSYS Constructor** dialog box will be displayed. Make the changes in the values entered in the **Delta-X**, **Delta-Y**, and **Delta-Z** edit boxes and choose the **OK** button; the modifications in the coordinate system will be reflected in the drawing window.*

Creating Coordinate Systems Using the CSYS of Current View Tool

Menu:	Insert > Datum/Point > Datum CSYS
Toolbar:	Form Feature > Datum CSYS

The **CSYS of Current View** tool can be used to create a coordinate system for a part. The coordinate system is always created such that the Z axis is normal to the current view on the screen. To create a coordinate system using this tool, invoke the **CSYS**

Constructor dialog box. Now, orient the model to the required position and then choose the **CSYS of Current View** button; you will be prompted to define the associative coordinate system or choose OK to create a coordinate system of the current view. Note that you can change the view of the model and update the coordinate system for the modified view by invoking the tool again. You can perform this upgradation any number of times before choosing the **OK** button. Orient the view of the model and choose the **OK** button from the same dialog box to update the coordinate system for the modified orientation.

Creating Coordinate Systems Using the Inferred Tool

Menu:	Insert > Datum/Point > Datum CSYS
Toolbar:	Form Feature > Datum CSYS

You can use the **Inferred** tool to create different types of coordinate systems. The resulting coordinate system will depend on the reference objects selected and their sequence of selection.

To create the coordinate system using this tool, invoke the **CSYS Constructor** dialog box. Depending on the reference objects selected and their sequence of selection, the appropriate coordinate system will be created.

Creating Fixed and Relative Datum Axes

The datum axis can be used as a reference object while creating a sketch based features such as a revolved feature or while creating the feature based operations such as the draft feature. There are two types of datum axes in NX: fixed datum axes and relative datum axes. A fixed datum axis can be created without specifying any reference object. However, for creating a relative datum axis, you need to select the reference object. The procedure for creating these datum axes are discussed next.

Creating Fixed Datum Axes Using the Fixed Datum Tool

Menu:	Insert > Datum/Point > Datum Axis
Toolbar:	Form Feature > Datum Axis

As mentioned earlier, the fixed datum axis can be created without any reference object selected. To create the fixed datum axis, choose the **Datum Axis** button from the **Form Feature** toolbar; the **Datum Axis Icon Options** will be displayed and you will be prompted to select an edge, face, datum plane, datum axis, curve, or point. Choose the **Datum Axis Dialog** button from the **Datum Axis Icon Options**; the **Datum Axis dialog** box will be displayed, as shown in Figure 4-31.

By default, the **Inferred** button will be chosen in the same dialog box. Choose the **Fixed Datum** button from the **Datum Axis** dialog box; the preview of all the three default datum axes will be displayed, as shown in Figure 4-32, and a drop-down list will be displayed below the **Fixed Datum** button. This drop-down list provides the options to specify the required datum axis or axis to be created. By default, the **3 Axes of WCS** option is selected from this drop-down list. This option will be used to create all the three default datum axes. Also, there are other options available in this drop-down list to create any one of the datum axes required among the three
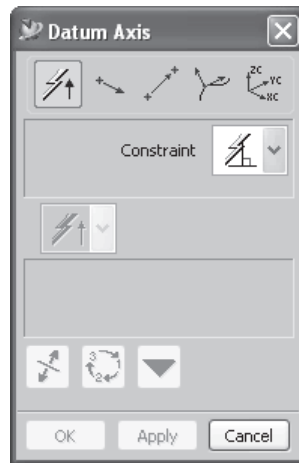
*Figure 4-31   The **Datum
Axis** dialog box*

default datum axes. Select the appropriate option from this drop-down list. Figure 4-33 shows
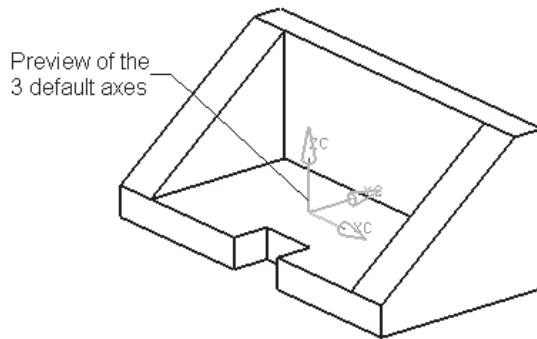the three default datum axes.

*Figure 4-32 The preview of all the three fixed
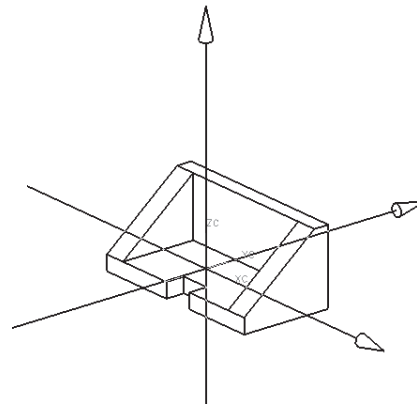datum axis created using the **Fixed Datum** tool*

*Figure 4-33 The resulting fixed datum axis
created using the **Fixed Datum** tool*

Note

*The fixed datum axes that you create are treated as features and are displayed in the **Part Navigator**
cascade menu.*

Creating the Relative Datum Axis Using the Point and Direction Tool

Menu:	Insert > Datum/Point > Datum Axis
Toolbar:	Form Feature > Datum Axis

The **Point and Direction** tool from the **Datum Axis** dialog box is used to create a
datum axis on a point after defining the direction. To create the relative datum axis,
choose the **Datum Axis** button from the **Form Feature** toolbar; the **Datum Axis Icon**

Options will be displayed. Choose the **Datum Axis Dialog** button; the **Datum Axis** dialog box will be displayed. From this dialog box, choose the **Point and Direction** tool; you will be prompted to select a point. Specify the fixing point. You can choose the required buttons from the **Snap Point** toolbar to enable easy selection of the points. After specifying the fixing point, the preview of the datum axis will be displayed, as shown in Figure 4-34. Also, a drop-down list is displayed in the **Datum Axis dialog** box. Next, you need to define the direction along which the datum axis will point. To define the direction, you can use the options in the drop-down list. You can select an edge or a face for defining the direction. If you select an edge as the reference, the axis will be created coincident or parallel to the edge. If you select a face as the reference, the axis will be created perpendicular to it. The resulting datum axis will be displayed, as shown in Figure 4-35.
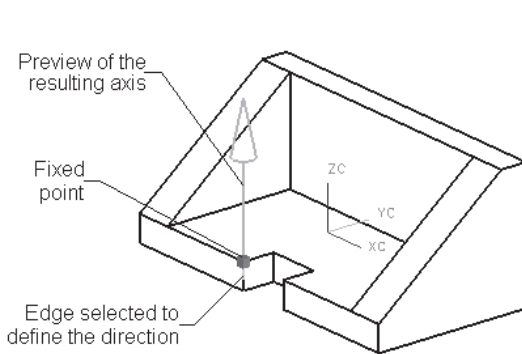

Figure 4-34 *The preview of the relative datum axis created using the* ***Point and Direction*** *tool*
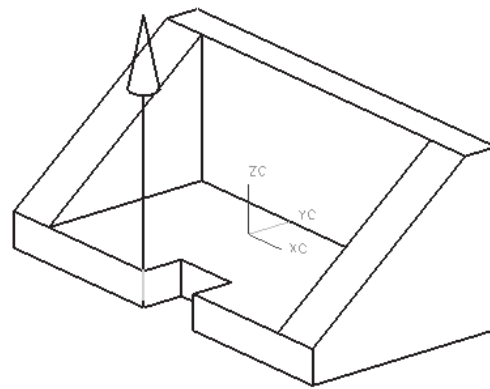
Figure 4-35 *The resulting relative datum axis created using the* ***Point and Direction*** *tool*

Note
Also, you can select an existing axis, linear curve, or a linear sketch member as a reference object for defining the direction for the datum axis.

Creating the Relative Datum Axis Using the Two Points Tool

Menu:	Insert > Datum/Point > Datum Axis
Toolbar:	Form Feature > Datum Axis

The **Two Points** tool is used to create a datum axis between the two selected points. To create the relative datum axis using this tool, invoke the **Datum Axis** dialog box and choose the **Two Points** tool; you will be prompted to select a point. Specify the first point. Next, you need to specify the endpoint toward which the axis will point. After you specify the second point, the preview of the datum axis will be displayed, as shown in Figure 4-36. Choose **Apply** and then the **OK** button from the **Datum Axis** dialog box to accept the axis created. The resulting datum axis will be created, as shown in Figure 4-37.
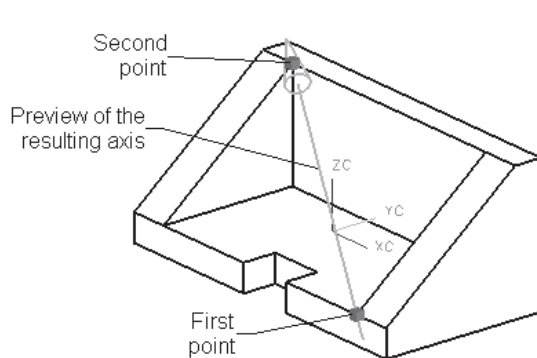
Figure 4-36 *The preview of the relative datum axis created using the* **Two Points** *tool*
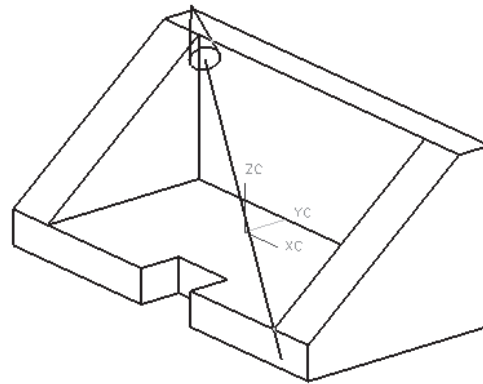

Figure 4-37 *The resulting relative datum axis created using the* **Two Points** *tool*

Creating the Relative Datum Axis Using the Point on Curve Tool

Menu:	Insert > Datum/Point > Datum Axis
Toolbar:	Form Feature > Datum Axis

The **Point on Curve** tool is used to create a datum axis passing through a point on a specified curve. To create the relative datum axis using this tool, choose the **Point on Curve** tool from the **Datum Axis** dialog box; you will be prompted to select a point on the curve or edge. Select a curve or an edge. After you select the curve or an edge, the datum axis will be displayed, along with the **Arclength** edit box, as shown in Figure 4-38. Also, the **Alternate Solution** and the **Reverse Direction** buttons are enabled in the **Datum Axis** dialog box. The **Alternate Solution** button is used to display the other possible solutions available for the selection and the **Reverse Direction** button is used to reverse the direction of the datum axis. Next, you need to locate the datum axis on the selected curve by entering the value in the **Arclength** edit box or by dragging the cube in the preview. Figure 4-39 shows a datum axis created using this option.
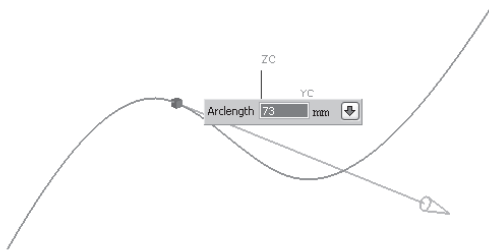

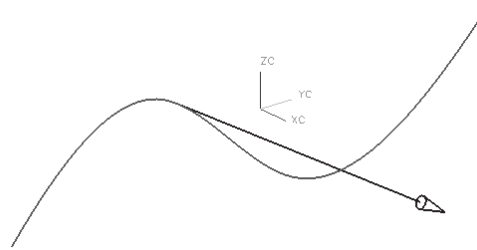**Figure 4-38** *The preview of the relative datum axis created using the* **Point on Curve** *tool*

Figure 4-39 *The resulting relative datum axis created using the* **Point on Curve** *tool*

Creating the Relative Datum Axis Using the Inferred Tool

Menu:	Insert > Datum/Point > Datum Axis
Toolbar:	Form Feature > Datum Axis

The **Inferred** tool is used to create various types of axes, depending on the selection method adopted. The method of creating an axis using this tool is similar to that of creating using the other tools.

OTHER EXTRUSION OPTIONS

You learned about the basic extrude options in the previous chapter. In this chapter, you will learn about additional extrude options.

Specifying the Boolean Operation

After creating the base feature, you can create additional features by specifying four types of boolean operations. These operations are available in the drop-down list in the **Extrude Icon Options** and also in the **Extrude** dialog box. These four types of boolean operations are discussed next.

Create

After drawing the sketch for the additional feature, when you invoke the **Extrude Icon Options**, the **Create** boolean type is selected by default. This option allows you to create a new body, which is independent of the existing feature. The new body is displayed as a separate solid body in the **Part Navigator**, see Figure 4-40. In this figure, the first solid body is the base feature of the model and the second solid body is the new feature created using the **Create** option.
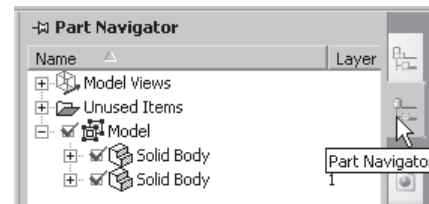
*Figure 4-40   The **Part Navigator** showing the new solid body*

Unite

You can select this boolean type from the drop-down list in the **Extrude Icon Options**, as shown in Figure 4-41. The **Unite** boolean operation allows you to join the new feature with the existing feature. In this case, no additional solid body will be created. Figure 4-42 shows the base feature and the sketch for the additional feature. Note that this sketch is created at a reference plane created at some offset from the top face of the base feature. Figure 4-43 shows an additional feature created using the **Unite** boolean operation.
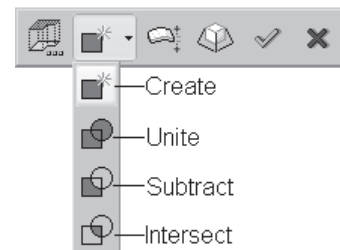
*Figure 4-41   Various boolean operations available in the **Extrude Icon Options***

Subtract

The **Subtract** boolean operation is used to create an extruded feature by removing material from the existing feature. The material to be removed will be defined by the sketch you have drawn. Figure 4-44 shows an extruded cut feature created using the **Subtract** boolean operation.
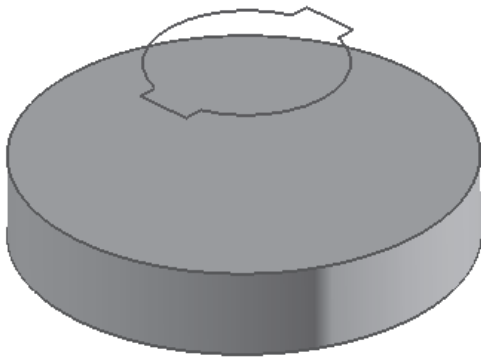
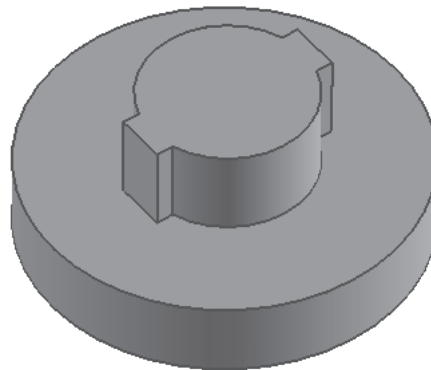*Figure 4-42* Base feature and the sketch for the additional feature

Figure 4-43 3D view of the resulting feature created using the *Unite* boolean operation

Intersect

The **Intersect** boolean operation is used to create an extruded feature by retaining the material common to the existing feature and the sketch, see Figure 4-45. In this case, the material of the base feature that lies outside the boundary of the sketch is removed.
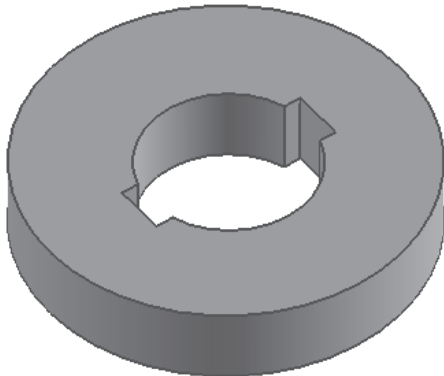

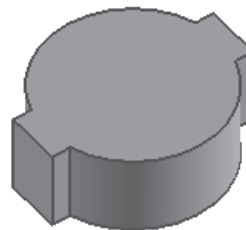*Figure 4-44* Extruded cut feature created using the *Subtract* boolean operation

Figure 4-45 Extruded cut feature created using the *Intersect* boolean operation

Note
The models shown in Figures 4-44 and 4-45 are also created using the sketch shown in Figure 4-42.

Specifying Other Extrusion Termination Options

In the previous chapter, you learned about the **Value** feature termination option. In this chapter, you will learn about the remaining feature termination options available in the **Start** and **End** drop-down lists in the **Limits** area of the **Extrude** dialog box.

Until Next

The **Until Next** option is used to extrude the sketch from the sketching plane to the next surface

that intersects the feature in the specified direction. Figure 4-46 shows the sketch to be extruded and Figure 4-47 shows the sketch extruded up to the next face using the **Until Next** option.
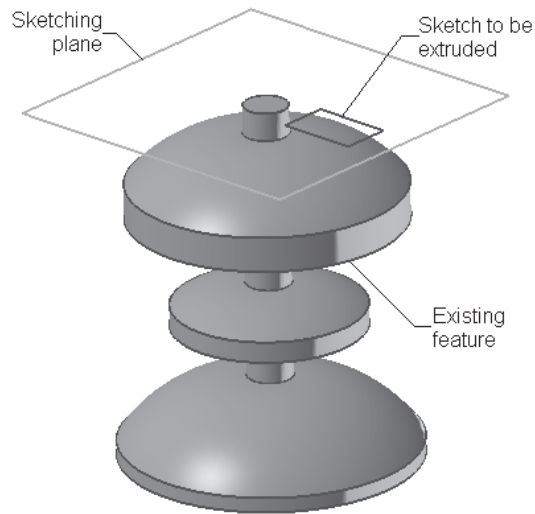

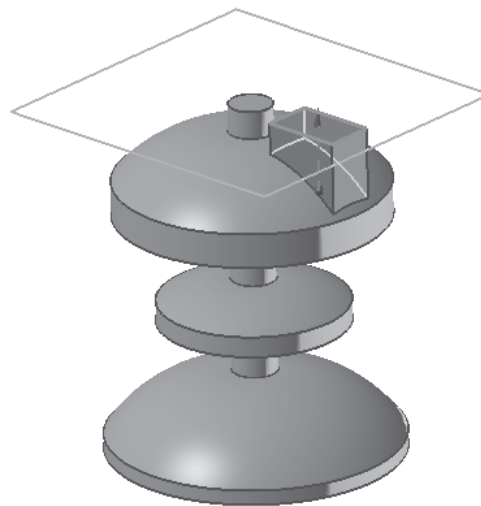*Figure 4-46 Sketch to be extruded and the existing features*

*Figure 4-47 Preview of the sketch being extruded up to the next surface using the **Until Next** option*

Until Selected

The **Until Selected** option is used to extrude the sketch up to a specified face, datum plane, or a body. Figure 4-48 shows the preview of the sketch being extruded up to the selected face.
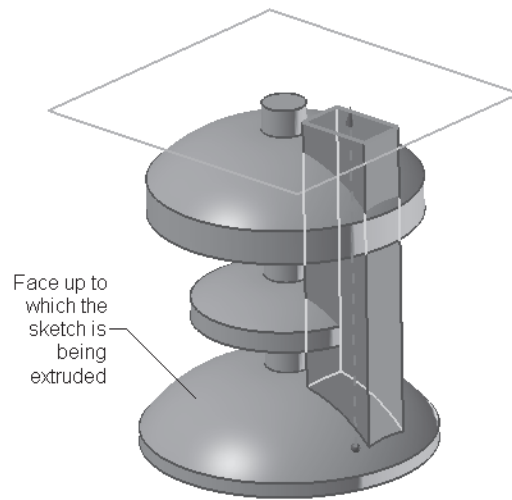
Figure 4-48 *Preview of the sketch being extruded up to a selected surface using the **Until Selected** option*

Until Extended

The **Until Extended** option is used to extrude the sketch up to a specified face, which does not intersect the sketch in its current size and shape. However, when extended, this face will intersect the extruded sketch. Figure 4-49 shows the preview of the sketch being extruded using this option.
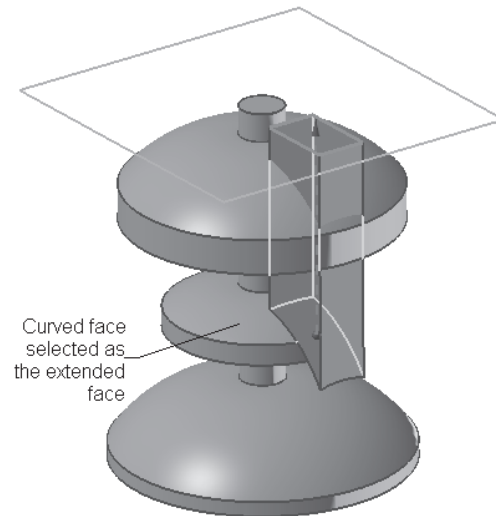

Figure 4-49 *Preview of the sketch being extruded up to the extended surface using the **Until Extended** option*

Through All

The **Through All** option is used to extrude the sketch through all features and bodies that come in the path of the sketch. Figure 4-50 shows the preview of the feature extruded using this method.
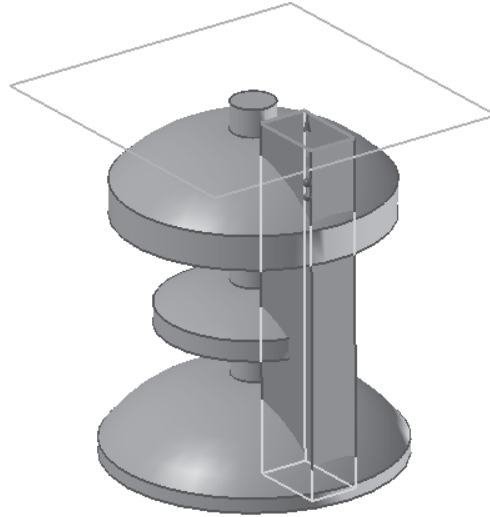

*Figure 4-50 Preview of the sketch being extruded up to the last surface using the **Through All** option*

OTHER REVOLUTION OPTIONS

You learned to create revolved features using the **Axis and Angle** method in the previous chapter. In this chapter, you will learn about the other two options that can be used to create the revolved feature.

Revolving Sketches Using the Trim to Face Method

The **Trim to Face** method is used to create a revolved feature that terminates at a specified face. After you invoke the **Revolved Body** tool and select the sketch to be revolved, you can select the **Trim to Face** option from the **Revolved Body** dialog box. On doing so, the **Trimming Face** dialog box will be displayed and you will be prompted to select a trimming face or a datum plane. You can select the face on which you want the revolved feature to be terminated. Next, you need to specify the revolution axis using the **Vector Construction** dialog box. Figure 4-51 shows a sketch drawn at an angle plane and the face selected to terminate the revolve feature and Figure 4-52 shows the resulting feature. Note that in this revolved feature, the revolution axis is pointing in the upward direction. If the revolution axis point in the downward direction, the revolved feature will be created on the other side of the sketching plane.

Note
*At the final step of creating an additional revolved feature, the **Boolean Operation** dialog box is displayed that allows you to specify the kind of boolean operation you want to use for the revolved feature.*
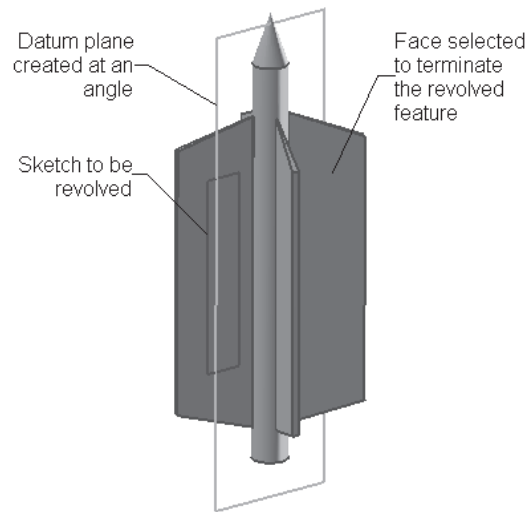
Figure 4-51 *Sketch to be revolved and the face selected to terminate the revolved feature*
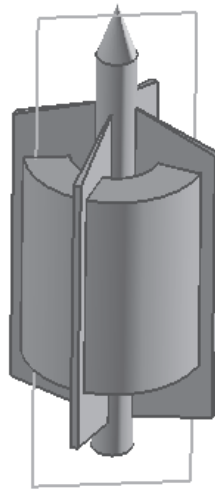

Figure 4-52 *Resulting revolved feature*

Revolving Sketches Using the Trim Between Two Faces Method

The **Trim Between Two Face** method is used to create a revolved feature that starts from a specified face and terminates at another specified face. After you invoke the **Revolved Body** tool and select the sketch to be revolved, you can select the **Trim Between Two Faces** option from the **Revolved Body** dialog box. On doing so, the **Trimming Face** dialog box will be displayed and you will be prompted to select the first trimming face or the datum plane. In this case, select the

face from which you want the revolved feature to start and then choose the **OK** button from the **Trimming Face** dialog box. Again, the **Trimming Face** options will be activated and you will be prompted to select the second trimming face or the datum plane. Select the face at which the revolved feature should terminate and then choose **OK** from the dialog box. Now, define the axis of revolution to complete the revolve feature. Figure 4-53 shows the sketch, the first trimming face, and the second trimming face for creating the revolved feature. Figure 4-54 shows the resulting revolved feature.
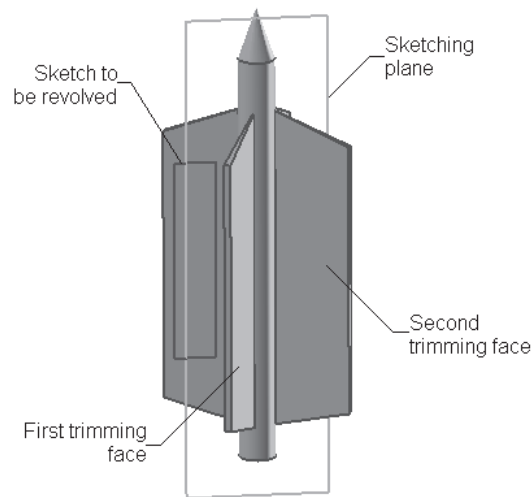

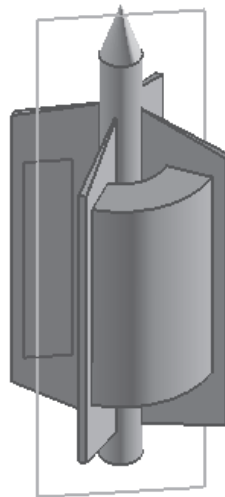**Figure 4-53** *Sketch to be revolved and the first and second trimming faces*

Figure 4-54 *Resulting revolved feature*

PROJECTING EXTERNAL ELEMENTS

While sketching, you may sometimes need to use some elements of the existing features in the current sketch. NX facilitates this by allowing you to project external elements as sketched entities on the current sketching plane. This helps you to create features that are similar to other features created on a different sketching plane. For example, refer to Figure 4-55. The model shown in this figure has a cylindrical base feature and another feature created at the bottom face of the base feature. Now, if you want to create the same feature on the top face of the cylindrical feature, you can simply define a new sketching plane on the top face of the base feature and then project the top face of the second feature. The entities will be automatically placed on the current sketching plane, as shown in Figure 4-56.
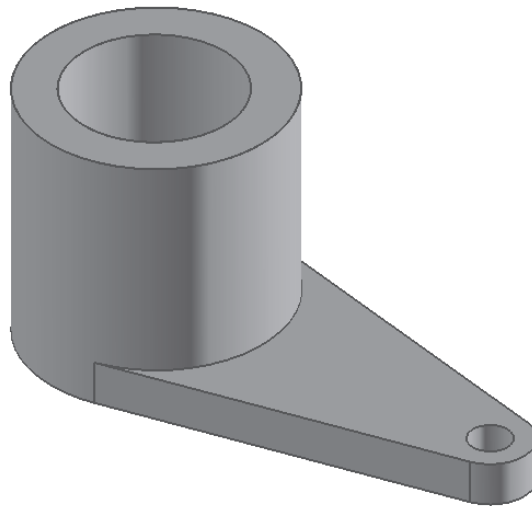
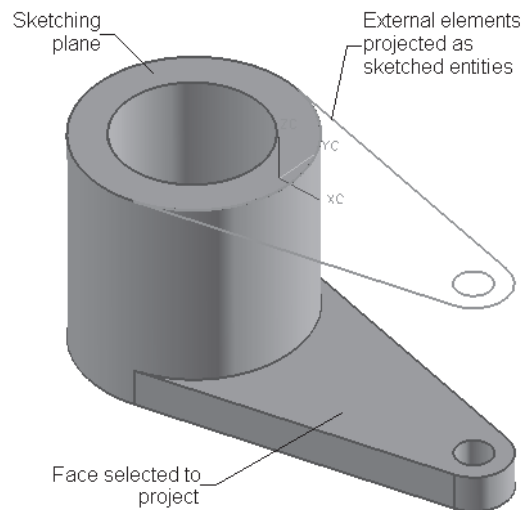*Figure 4-55 Model with two features*

Figure 4-56 Face selected to be projected and the resulting sketch

To project external elements, invoke the sketching environment and then choose **Insert >
Project** from the menu bar; the **Project Icon Options** will be displayed and you will be prompted
to select a geometry to project into the sketch. Choosing the **Project Objects to Sketch Dialog**
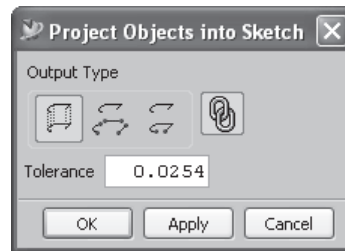button displays the **Project Objects into Sketch** dialog box, as shown in Figure 4-57.

*Figure 4-57 The **Project Objects into Sketch**
dialog box*

The Project Objects into Sketch Dialog Box Options
The options in this dialog box are discussed next.

Output Type
The buttons in this area are used to specify the projection output type. These options are
discussed next.

Original
By default, the **Original** button is chosen. As a result, the entities are projected
similar to the original entities. This means that if the original sketch was a combination
of lines, the projected sketch will also be a combination of lines only.

Spline Segment
Choosing this button projects the external elements as spline segments. Each
external element is represented by a spline segment in the projected sketch. This
means that if you select to project six original elements, the projected sketch will
also comprise of six spline segments.

Single Spline
Choosing this button projects the external elements as a single continuous spline.
As a result, irrespective of the number of entities you select to project, the projected
sketch will have a single continuous spline.

Associative
Choosing this button forces the projected sketched entities to be associative with the
original entities from which they were projected. As a result, if the original entities are
modified, the projected entities are also modified accordingly.

Note

Editing the projected entities may break their associativity with the original elements.

Tolerance

This edit box is used to specify the gap between the entities up to which they will be considered as continuous entities in the projected sketch. If the gap between the original entities is more than that specified in this dialog box, they will appear as noncontinuous entities in the projected sketch.

Note

*The output type and associativity can also be defined using the **Project Icon Options**.*

After specifying the projection parameters, select the elements to be projected. You can project individual entities such as edges of existing features or select a face to be projected. In case of a face, all edges that comprise that face are projected. Next, choose the **OK** button from the **Project Icon Options**; the selected elements will be projected.

TUTORIALS

Tutorial 1

In this tutorial, you will create the model shown in Figure 4-58. The dimensions of the model are given in Figure 4-59.   **(Expected time: 30 min)**
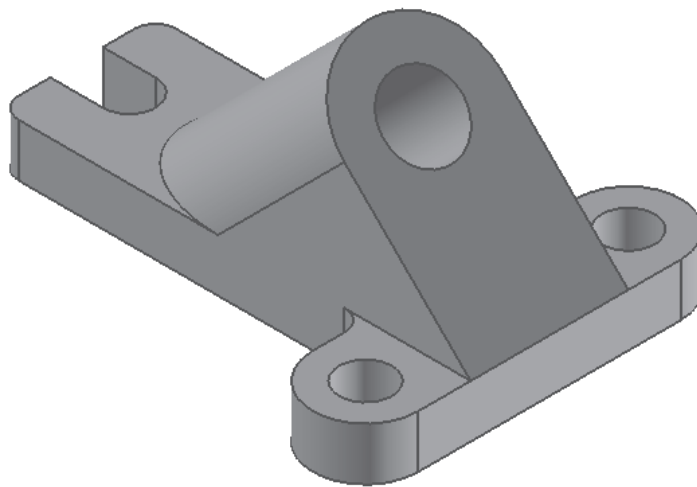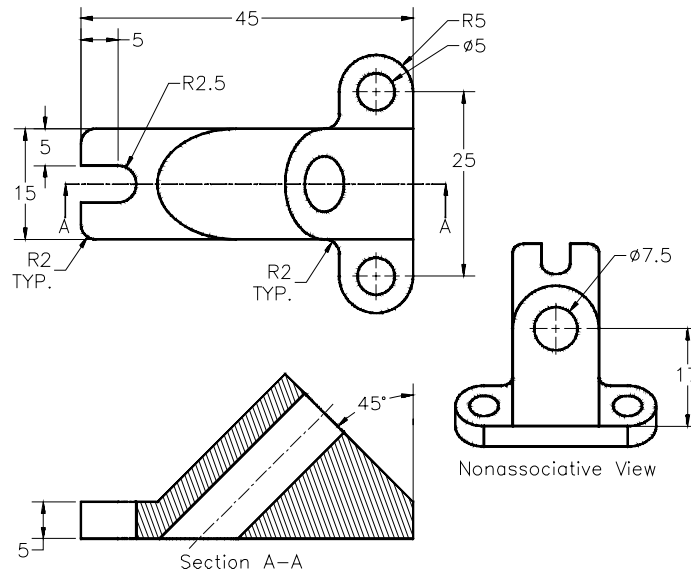

Figure 4-58 *Model for Tutorial 1*

Figure 4-59 *Dimensions of the model for Tutorial 1*

Before creating a model with multiple features, you first need to determine the number of features in that model and then decide the sequence in which they should be created. This will help you determine which feature should be the base feature of the model. The model for this tutorial is a combination of three extrude features. The third extrude feature will be a subtract feature, which will be used to create a hole on the second feature.

The following steps outline the procedure to create this model.

a.   Create the base feature of the model on FIXED_DATUM_PLANE (0), refer to Figures 4-60 and 4-61.
b.   Define a new datum plane at an angle of 45-degree from the right face of the base feature.
c.   Draw the sketch of the next extrude feature using the new datum plane and extrude it using the **Unite** boolean operation, refer to Figures 4-63 and 4-64.
d.   Draw the sketch for the hole on the inclined face of the second feature and then extrude it using the **Subtract** boolean operation to create the hole, refer to Figure 4-65.

Creating the Base Feature

To create the base feature, you first need to create the three default datum planes. Next, you will use FIXED_DATUM_PLANE (0) to create the sketch for the base feature.

1.   Start NX and then start a new file. Create a new folder with the name **c04** inside the **NX 3** folder. Make the **c04** folder the current folder and enter the name of the new file as **c04tut1**. When the new file opens, invoke the **Modeling** environment.

2. Choose the **Datum Plane** button from the **Form Feature** toolbar; the **Datum Plane** dialog box is displayed and the **Fixed Datum** button is chosen by default.

3. Choose the **OK** button from the dialog box to create the three default datum planes.

4. Invoke the sketching environment using FIXED_DATUM_PLANE (0) and draw the sketch similar to the one shown in Figure 4-60.
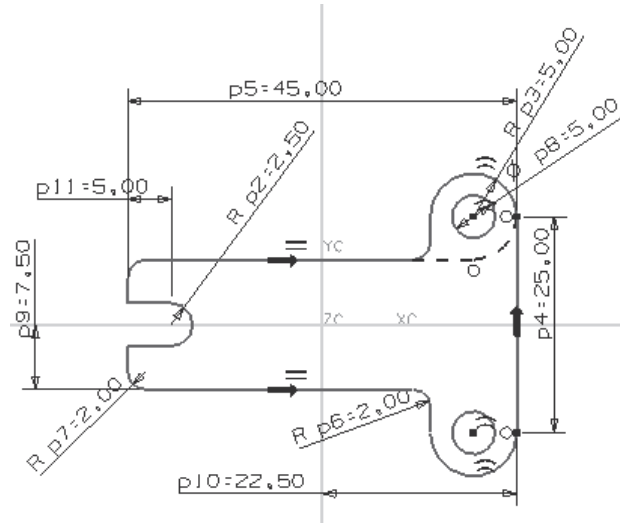

Figure 4-60 Sketch for the base feature

5. Exit the sketching environment and change the current view to the isometric view, if it is not automatically changed.

6. Invoke the **Extrude Icon Options** and select the sketch; the preview of the extruded feature is displayed.

7. Modify the value of extrusion height to 5 and then choose **OK** from the **Extrude Icon Options**.

8. Hide the sketch and the default datum planes. The base feature of the model is shown in Figure 4-61.

Creating the Second Feature

The second feature is also an extruded feature. However, to create the sketch of this feature, you first need to create a new datum plane at an angle of 45-degree to the right face of the base feature.

1. Choose the **Datum Plane** button from the **Form Feature** toolbar; the **Datum Plane Icon Options** will be displayed.
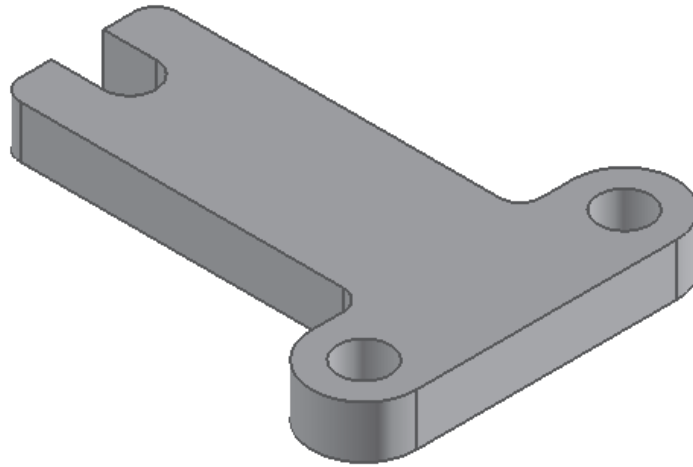
Figure 4-61 *Base feature of the model*

2. Select the right face of the base feature and then the edge shown in Figure 4-62 to define the new datum plane. On doing so, the **Angle** toolbar is displayed.

3. Enter **45** as the value in the toolbar, as shown in Figure 4-62, and choose the **OK** button from the **Datum Plane Icon Options**.
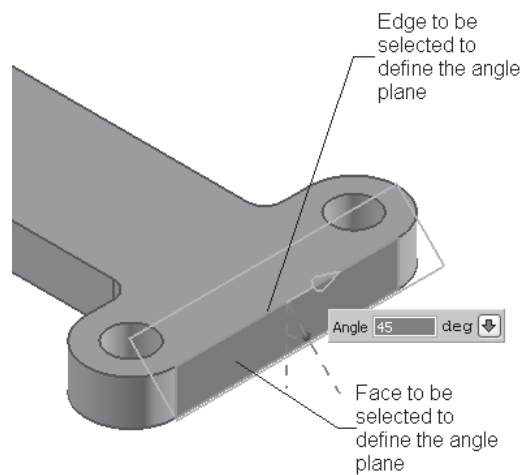

Figure 4-62 *Defining a new datum plane*

4. Draw the sketch of the second feature, as shown in Figure 4-63. You will have to apply collinear constraints between the lines in the sketch and the edges of the base feature to place the sketch at the proper location.
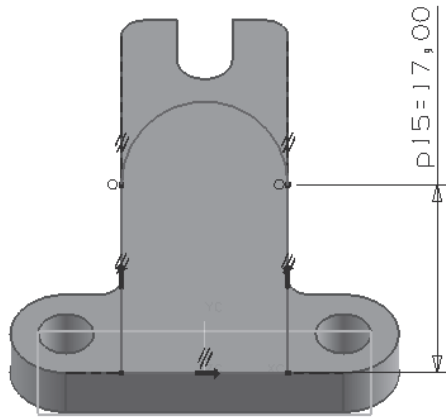
Figure 4-63 *Sketch for the second feature*

5.   Exit the sketching environment and then invoke the **Extrude Icon Options**.

6.   Select the sketch and then invoke the **Extrude** dialog box.

7.   Select **Unite** from the **Boolean** drop-down list in the dialog box.

8.   Select **Until Next** from the **Start** drop-down list in the **Limits** area and then set the value of the **End** edit box to 0.

9.   Choose the **OK** button from the dialog box to close it and create the feature.

10. Hide the sketch and the datum plane. Figure 4-64 shows the model after creating the second feature.

Creating the Third Feature

The third feature is an extruded feature created using the **Subtract** operation. The sketch for this feature will be created on the inclined face of the second feature.

1.   Select the inclined face of the second feature as the sketching plane and draw the circle for the hole. Add the required dimensions and constraints to the circle.

2.   Exit the sketching environment and then invoke the **Extrude Icon Options**.

3.   Select **Subtract** from the drop-down list in the **Extrude Icon Options** and then select the circle.

4.   Next, invoke the **Extrude** dialog box and select **Through All** from the **Start** drop-down list in the **Limits** area. If the options in the drop-down list are not enabled, enter any negative value in the edit box on the right of this drop-down list; the other options will be enabled. Now, select the **Through All** option.

5. Choose **OK** to exit the dialog box and create the feature. Hide the sketch of the circle and then choose the **Fit** button to fit the model on the screen. The final model for this tutorial is shown in Figure 4-65.
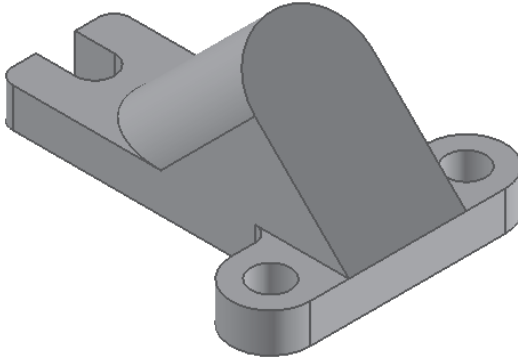

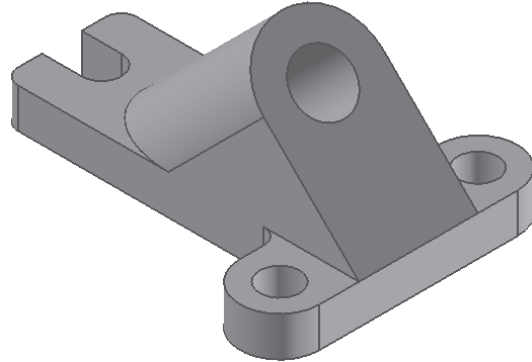***Figure 4-64*** *Model after creating the second feature on the inclined datum plane*

Figure 4-65 *Final model for Tutorial 1*

Saving and Closing the File

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

Tutorial 2

In this tutorial, you will draw the model shown in Figure 4-66. The dimensions of the model are shown in Figure 4-67. **(Expected time: 30 min)**
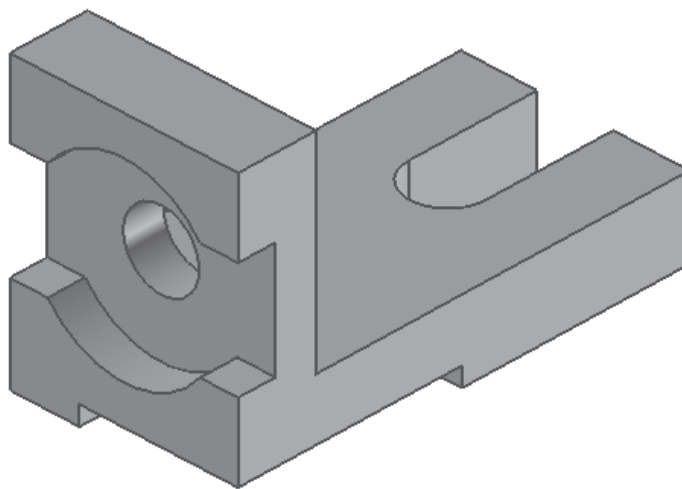

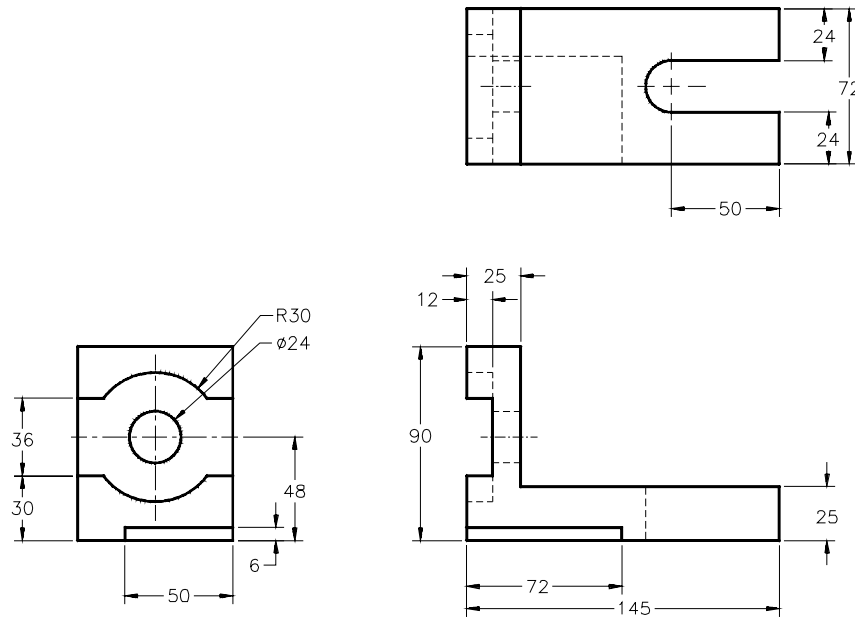***Figure 4-66*** *Model for Tutorial 2*

Figure 4-67 *Dimensions of the model for Tutorial 2*

This model consists of five features. The base feature will be created using an open sketch of L shape, which will be created on FIXED_DATUM_PLANE (1). The next feature will be the subtract feature at the bottom face of the base feature. The third feature is also a cut feature that will be created on the upper horizontal face of the base feature. The fourth and fifth features will be created on the left face of the base feature using the **Subtract** operation.

The following steps outline the procedure to create this model.

a. Create the base feature of the model on FIXED_DATUM_PLANE (1), refer to Figures 4-68 and 4-69.
b. Create the second feature with the **Subtract** operation using the sketch drawn on the front face of the base feature, refer to Figures 4-70 and 4-71.
c. Create the third feature with the **Subtract** operation using the sketch drawn on the upper horizontal face of the base feature, refer to Figures 4-72 and 4-73.
d. Create the fourth and fifth features on the base face of the base feature, refer to Figures 4-74 through 4-76.

Creating the Base Feature

To create the base feature, you first need to create the three default datum planes. Next, you will use FIXED_DATUM_PLANE (1) to create the sketch for the base feature.

1. Start a new file and enter the name of the new file as **c04tut2.** Make sure this file is saved in the *NX 3/c04* folder. When the new file is opened, invoke the **Modeling** environment.

2. Choose the **Datum Plane** button from the **Form Feature** toolbar; the **Datum Plane** dialog box is displayed and the **Fixed Datum** button is chosen by default.

3. Choose the **OK** button from the dialog box to create the three default datum planes.

4. Invoke the sketching environment using FIXED_DATUM_PLANE (1) and draw the open sketch similar to the one shown in Figure 4-68.
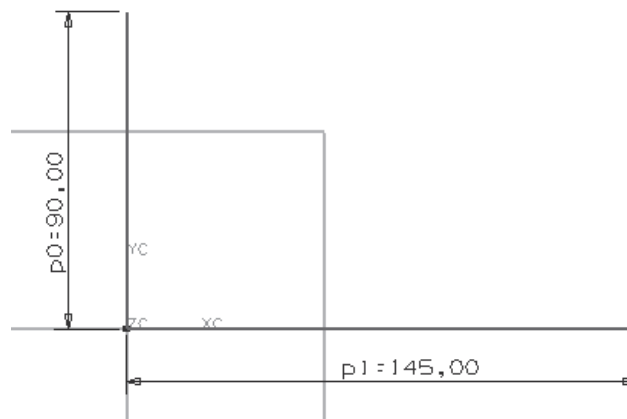

Figure 4-68 Open sketch for the base feature

5. Exit the sketching environment and invoke the **Extrude Icon Options**. Select the sketch and then choose the **Extrude Dialog** button from the **Extrude Icon Options** to invoke the **Extrude** dialog box.

6. Select the **Symmetric Distance** check box in the **Limits** area. Enter **72/2** as the value in the edit box on the right of the **Start** drop-down list; the value in the edit box on the right of the **End** drop-down list also changes automatically to 72/2.

7. Select the **Offset** check box and then enter **-25** as the value in the **Start** edit box. Set the value of the **End** edit box to **0**.

8. Choose the **OK** button to create the base feature. Turn off the display of the sketch. The base feature of the model is shown in Figure 4-69.

Creating the Second Feature

The second feature is also an extruded feature that will be created using the **Subtract** operation. The sketch for this feature will be drawn on the front face of the base feature.

1. Select the front face of the base feature and invoke the sketching plane. Make sure the X axis of the new sketching plane points toward the edge of 145 dimension.
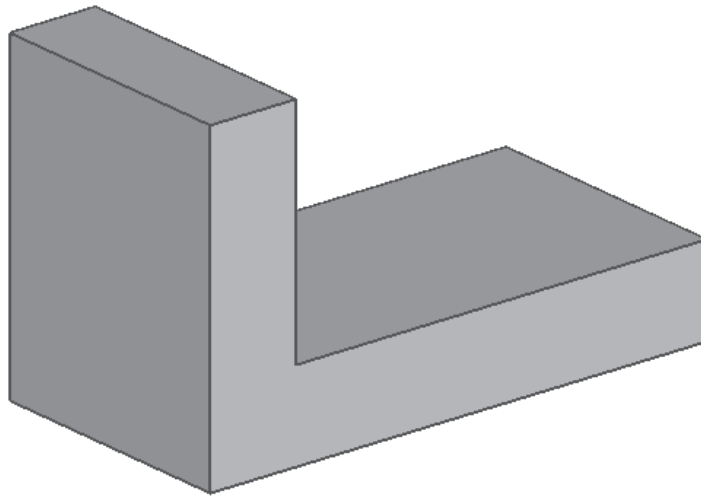
Figure 4-69 *Base feature of the model*

2.   Draw the rectangular sketch for the cut feature. Make the bottom line and the left line of the rectangle collinear with the edges of the base feature. Add the required dimensions, as shown in Figure 4-70.

3.   Exit the sketching environment and then invoke the **Extrude Icon Options**.

4.   Select the sketch and then invoke the **Extrude** dialog box. Select the **Subtract** boolean operation and reverse the direction of extrusion.

5.   Select the value in the **End** edit box as **50** and then choose the **OK** button from the dialog box.

6.   Hide the sketch. The model with the cut feature is shown in Figure 4-71.
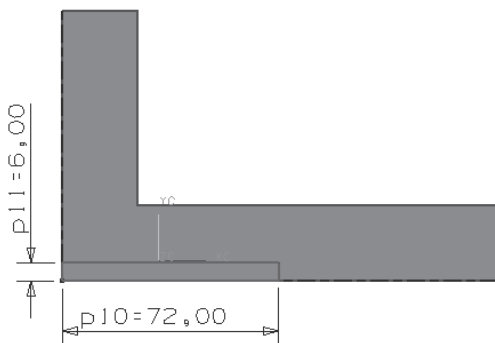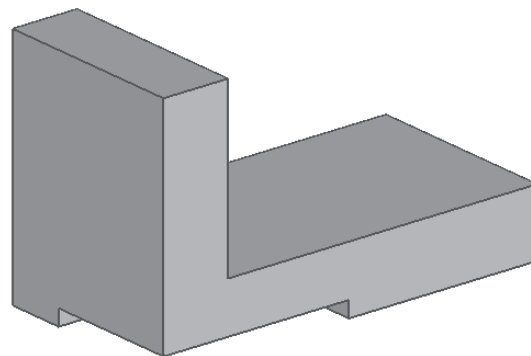

Figure 4-70 *Sketch for the second feature*

Figure 4-71 *Model with the cut feature*

Creating the Third Feature

The third feature is also an extruded feature that will be created using the **Subtract** operation. The sketch for this feature will be drawn on the upper horizontal face of the base feature.

1. Select the upper horizontal face of the base feature and invoke the sketching plane. Make sure the X axis of the new sketching plane points toward the edge of 145 dimension.

2. Draw the sketch of the feature and add the required constraints and dimensions, as shown in Figure 4-72.

3. Exit the sketching environment and then invoke the **Extrude Icon Options**.

4. Select the sketch and then invoke the **Extrude** dialog box. Select the **Subtract** boolean operation and reverse the direction of extrusion.

5. Select **Through All** from the **End** drop-down list and choose **OK** from the dialog box.

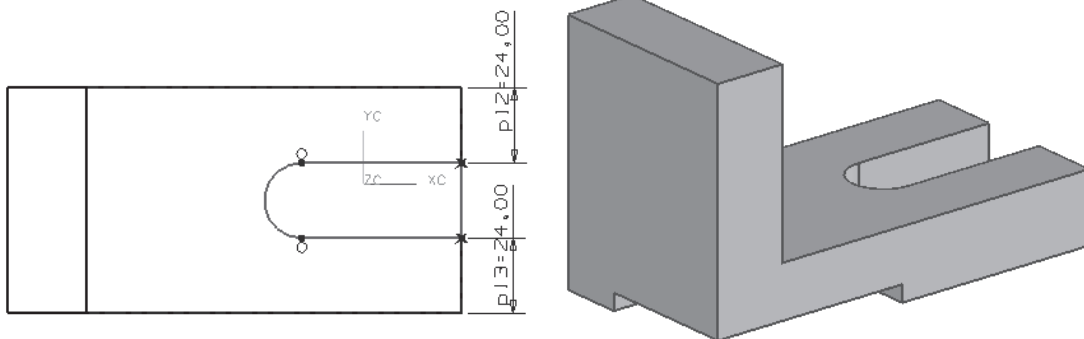6. Hide the sketch. The model with the cut feature is shown in Figure 4-73.

Figure 4-72   *Sketch for the third feature*

Creating the Fourth Feature

The sketch for this feature will be drawn on the left face of the base feature and will be extruded using the **Subtract** operation to a depth of 12.

1. Select the left face of the base feature and invoke the sketching plane. Make sure the X axis of the new sketching plane points toward the right.

2. Draw the sketch of the feature and add the required constraints and dimensions, as shown in Figure 4-74.

3. Exit the sketching environment and then invoke the **Extrude Icon Options**.

4. Select the sketch and then invoke the **Extrude** dialog box. Select the **Subtract** boolean operation and reverse the direction of extrusion.

5. Select the value in the **End** edit box as **12** and then choose the **OK** button from the dialog box.

6. Hide the sketch. The model with the cut feature is shown in Figure 4-75.

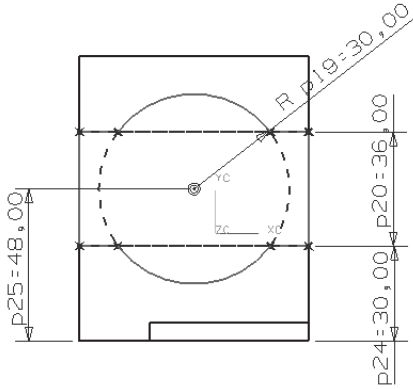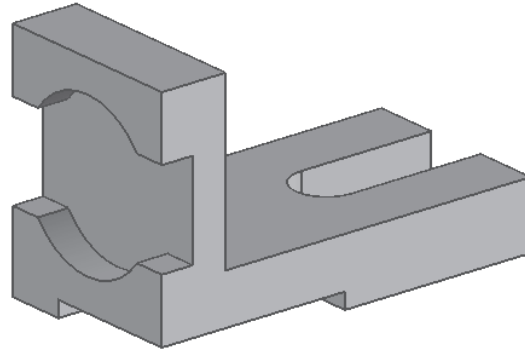*Figure 4-74* *Sketch for the fourth feature*

Figure 4-75 *Model with the fourth feature*

Creating the Fifth Feature

1. Similarly, draw a circle on the new face that is exposed because of the last cut feature and then extrude the circle using the **Subtract** operation and **the Through All** termination. The final model of Tutorial 2 is shown in Figure 4-76.
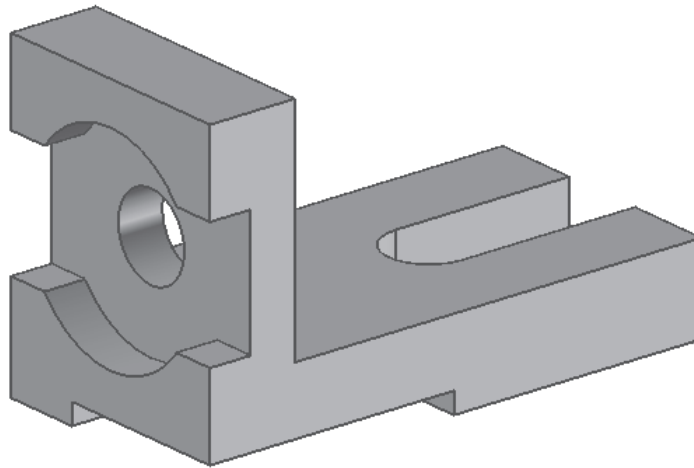

Figure 4-76 *Final model for Tutorial 2*

Saving and Closing the File

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

Tutorial 3

In this tutorial, you will draw the model shown in Figure 4-77. The dimensions of the model are also shown in the same figure.   **(Expected time: 30 min)**

Figure 4-77 *Model for Tutorial 3 and its dimensions*

This model consists of six features. The base feature will be created using the sketch created on FIXED_DATUM_PLANE (1). Next you will create the join feature at the top face of the base feature. The third feature is the cylindrical feature that will be created on a datum plane defined at an offset from the top face of the second feature. The next feature is the hole in the cylindrical feature. The last two features are the holes on the base feature of the model.

The following steps outline the procedure to create this model.

a.   Create the base feature of the model on FIXED_DATUM_PLANE (1), refer to Figures 4-78 and 4-79.

b. Create the second feature with the **Unite** operation using the sketch drawn on the top face of the base feature, refer to Figures 4-80 and 4-81.

c. Create a new datum plane at an offset of 40 from the top face of the second feature.

d. Draw the sketch of the fourth feature on the offset plane and extrude it using the **Unite** operation, refer to Figures 4-82.

e. Create the remaining holes in the model, refer to Figure 4-83.

Creating the Base Feature

To create the base feature, you first need to create the three default datum plane. Next, you will use FIXED_DATUM_PLANE (1) to create the sketch for the base feature.

1. Start a new file and enter the name of the new file as **c04tut3.** Make sure this file is saved in the *NX 3/c04* folder. When the new file opens, invoke the **Modeling** environment.

2. Choose the **Datum Plane** button from the **Form Feature** toolbar; the **Datum Plane** dialog box is displayed and the **Fixed Datum** button is chosen by default. Choose the **OK** button from the dialog box to create the three default datum planes.

3. Invoke the sketching environment using FIXED_DATUM_PLANE (1) and draw the sketch similar to the one shown in Figure 4-78.

4. Exit the sketching environment and invoke the **Extrude Icon Options**. Select the sketch and then choose the **Extrude Dialog** button from the **Extrude Icon Options** to invoke the **Extrude** dialog box.

5. Select the **Symmetric Distance** check box in the **Limits** area. Enter **30** as the value in the edit box on the right of the **Start** drop-down list; the value in the edit box on the right of the **End** drop-down list also changes automatically to 30.

6. Choose the **OK** button to create the base feature. Turn off the display of the sketch. The base feature of the model is shown in Figure 4-79.
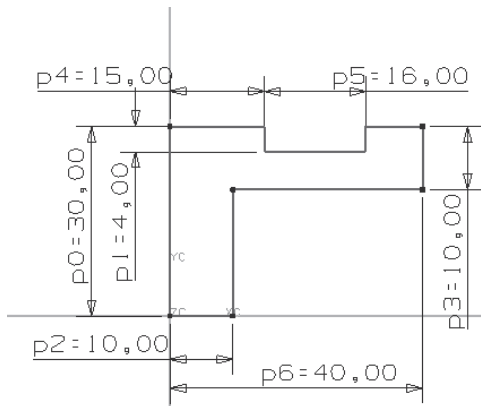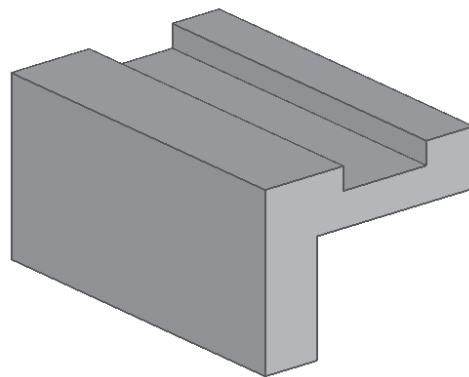

Figure 4-78 *Sketch for the base feature*   ***Figure 4-79*** *Base feature of the model*

Creating the Second Feature

The second feature is also an extruded feature that will be created using the **Unite** operation. The sketch for this feature will be drawn on the top face of the base feature and will be extruded using the **Until Extended** option.

1. Select the top face of the base feature and invoke the sketching plane. Make sure the X axis of the new sketching plane points toward the **Part Navigator**.

2. Draw the sketch for the feature and add the required dimensions and constraints, as shown in Figure 4-80.

3. Exit the sketching environment and then invoke the **Extrude Icon Options**.

4. Select the sketch and then invoke the **Extrude** dialog box. Select the **Unite** boolean operation.

5. Select the **Until Extended** option from the **End** drop-down list. Hold down the middle mouse button and then rotate the view of the model so that its lower faces are visible. Now, select the bottom horizontal face shown in Figure 4-81 to define the termination of the second feature.
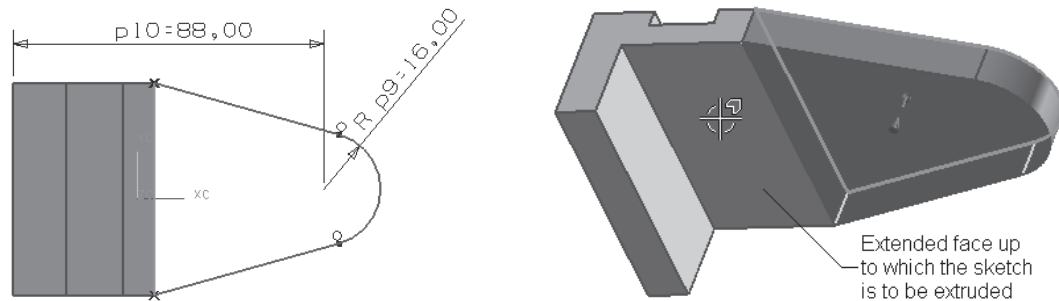

Figure 4-80 *Sketch for the second feature*

6. Choose **OK** to close the **Extrude** dialog box and create the feature.

7. Change the current view to isometric view and hide the sketch.

Creating the Third Feature

The third feature is created on a datum plane created at an offset of 40 from the top face of the second feature. Therefore, you first need to create the datum plane.

1. Choose the **Datum Plane** button from the **Form Feature** toolbar and select the top face of the second feature.

2. Enter **40** as the value in the **Offset** edit box and choose **OK** from the **Datum Plane Icon Options**.

3. Invoke the sketching plane using the new datum plane. You can define the orientation of the X axis of the sketching plane using one of the straight edges of the model.

4. Draw the circle for the third feature. Make the circle concentric with the curve in the second feature and then make the curve and the circle of equal radius. As a result, you will not have to apply any dimension to the sketch.

5. Exit the sketching environment and invoke the **Extrude Icon Options**.

6. Select the sketch and invoke the **Extrude** dialog box. Choose the **Reverse Direction** button and select the **Unite** boolean operation.

7. Enter **60** as the value in the **End** edit box. Choose **OK** to exit the dialog box and create the feature.

8. Change the current view to the isometric view and hide the sketch. The model after creating the cylindrical feature is shown in Figure 4-82.
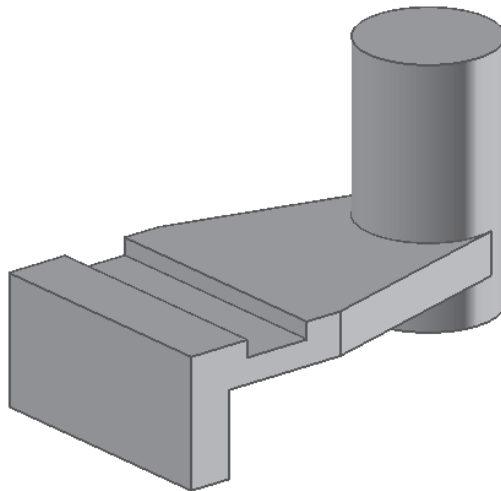
Figure 4-82 Model after creating the cylindrical feature

Creating the Remaining Feature

1. Create the remaining features, which are holes. To create these holes, you need to extrude the sketches using the **Subtract** operations. The final model for Tutorial 3 is shown in Figure 4-83.
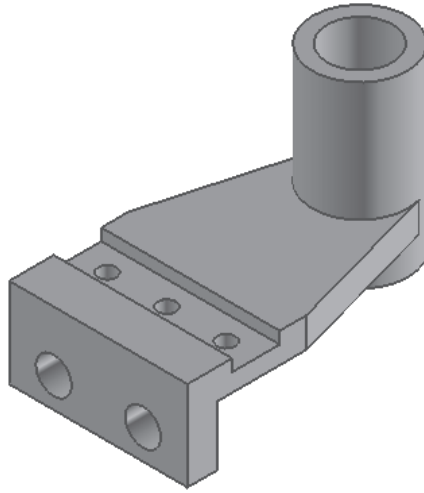
Figure 4-83 *Final model for Tutorial 3*

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 your answers with those given at the end of the chapter:

1.   In mechanical designs, all features are created on a single plane. (T/F)

2.   When you start a new part file, the fixed datum planes are provided to you by default. (T/F)

3.   You can turn off the display of the additional datum planes that you create. (T/F)

4.   In NX, you can extrude a sketch to perform only the join operation. (T/F)

5.   The fixed datum planes are also termed as _____ datum planes.

6.   The _____ boolean operation is used to create an extruded feature by removing material from the existing feature.

7.   The _____ method is used to create a revolved feature that terminates at a specified face.

8.   The _____ option is used to extrude the sketch up to a specified face, datum plane, or a body.

9. Using the _____ option ensures that while projecting, each external element is represented by a spline segment in the projected sketch.

10. The _____ boolean operation is used to create an extruded feature by retaining the material common to the existing feature and the sketch.

Review Questions

Answer the following questions:

1. Which one of the following tools allows you to create additional datum planes?

    (a) **Plane**   (b) **Datum Plane**
    (c) **Reference Plane**   (d) None

2. Which option in the **Insert** menu allows you to project the existing entities on to the current sketching plane?

    (a) **Project**   (b) **Project Edges**
    (c) **Divert**   (d) None

3. Which one of the following operations allows you to join the new feature with the existing feature?

    (a) **Join**   (b) **Unite**
    (c) **Combine**   (d) None

4. Which one of the following operations allows to you to extrude the sketch from the sketching plane to the next surface that intersects the feature in the specified direction?

    (a) **Until Selected**   (b) **Until Next**
    (c) **Value**   (d) None

5. Which one of the following methods is used to create a revolved feature that starts from a specified face and terminates at another specified face?

    (a) **Trim Faces**   (b) **Trim to Faces**
    (c) **Trim**   (d) **Trim Between Two Faces**

6. Which of the following projection output type projects the external elements as a single continuous spline?

    (a) **Spline**   (b) **Spline Segment**
    (c) **Single Spline**   (d) None

7. You cannot turn off the display of datum planes. (T/F)

8. The three default datum planes that you create are the only planes required to create all designs in NX. (T/F)

9. The **Until Selected** option is used to extrude the sketch up to a specified face, datum plane, or a body. (T/F)

10. NX allows you to make the projected sketched entities associative with the original entities from which they were projected. (T/F)

Exercises

Exercise 1

Create the model shown in Figure 4-84. The dimensions of the model are also shown in the same figure. **(Expected time: 30 min)**
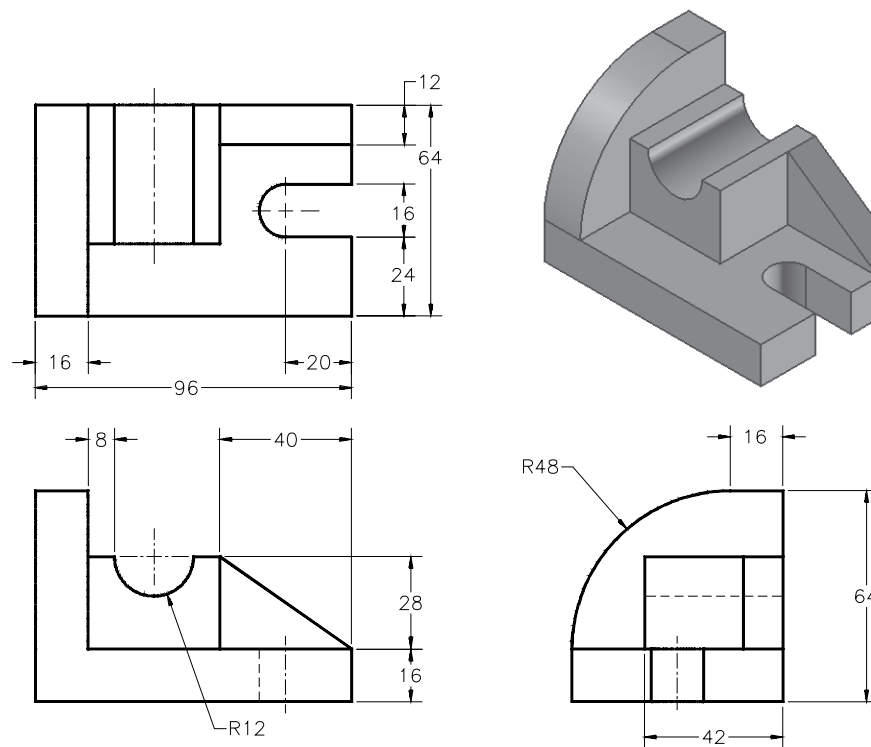

Figure 4-84 *Model for Exercise 1 and its dimensions*

Exercise 2

Create the model shown in Figure 4-85. The dimensions of the model are also shown in the same figure.   **(Expected time: 30 min)**
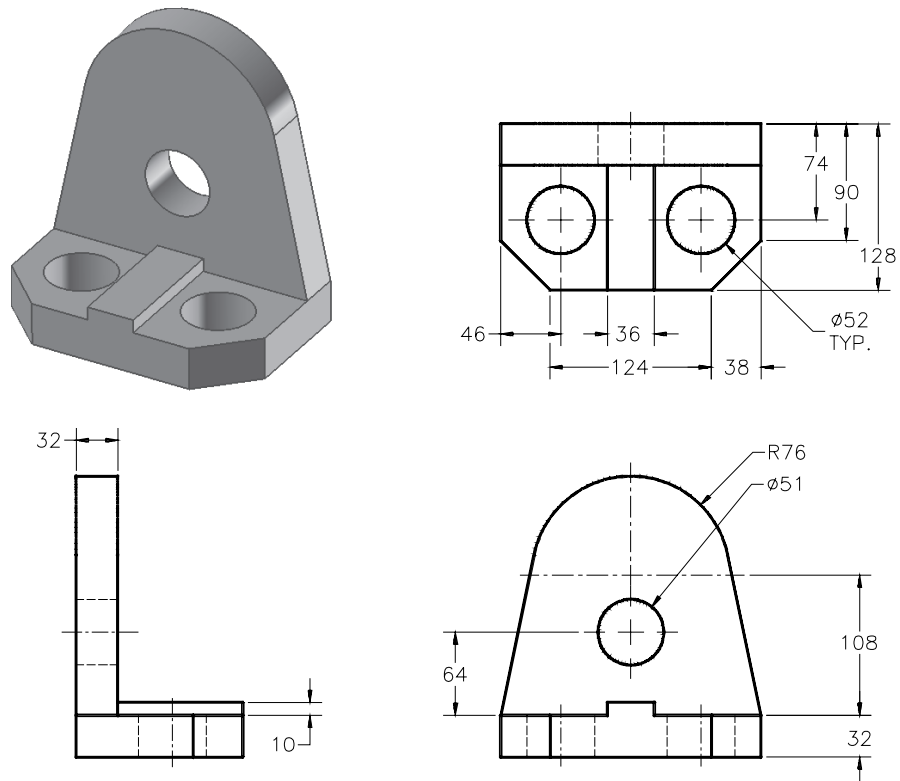

Figure 4-85 *Model for Exercise 2 and its dimensions*