



Chapter 1

Drawing Sketches for Solid Models

Learning Objectives

After completing this chapter you will be able to:

- *Understand the need for the Sketcher Task environment.*
- *Understand the datum planes that can be selected to draw sketches.*
- *Understand various drawing display tools.*
- *Understand various sketching tools.*
- *Use various selection methods.*
- *Delete sketched entities.*

THE SKETCHER TASK ENVIRONMENT

Most designs created in NX consist of sketch-based features and placed features. A sketch is a combination of a number of two-dimensional (2D) entities such as lines, arcs, circles, and so on. The sketch-based features are those that are created using these entities. Placed features are the ones that may not require a sketch and are added to the existing features to complete the design. Generally, the sketch-based feature is the base feature or the first feature in most designs. For example, refer to the solid model shown in Figure 1-1.

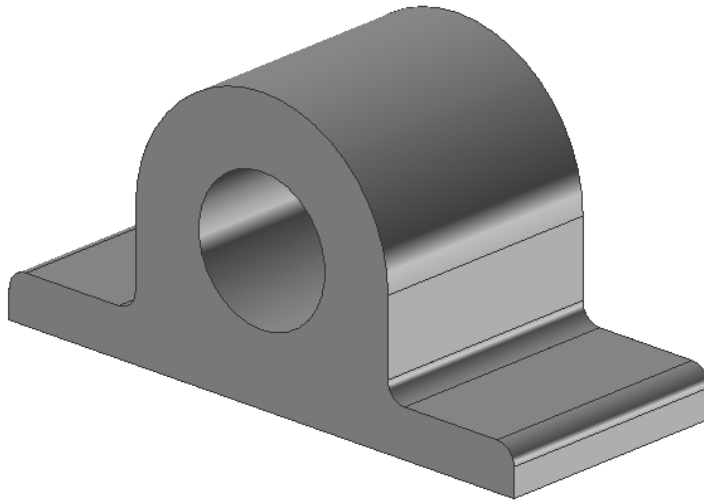


Figure 1-1 Solid model

This model is created using the profile shown in Figure 1-2.

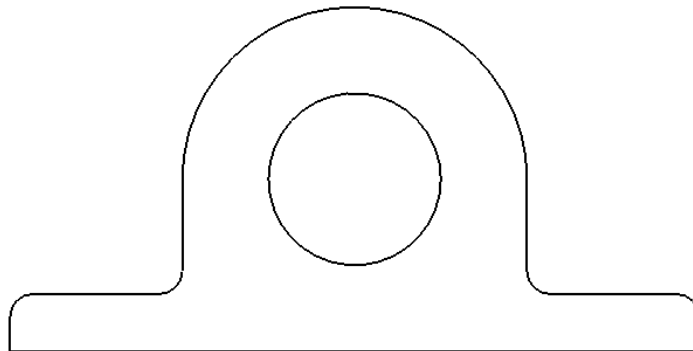


Figure 1-2 Profile of the solid model shown in Figure 1-1

Because you first need to draw a sketch in most of the designs, the **Sketcher Task** environment needs to be invoked first where you can draw the sketch. After drawing the sketch, you need to exit the **Sketcher Task** environment and then use the solid modeling tools to convert the sketch into a feature.

Unlike in other solid modeling tools where you need to use separate files for starting separate environments, in NX, only a single type of file is used to start various kinds of environments. For example, in the same file, you can start separate environments to draw the sketches, convert them into features, generate drawing views, and assemble other parts with the current part. The NX files are saved in the *.prt* format and all the environments required to complete a design can be invoked in the same *.prt* file.

STARTING NX 3

You can start NX 3 by double-clicking on its shortcut icon on the desktop of your computer. Alternatively, you can choose the **Start** button at the lower left corner of the screen to invoke the menu. From this menu, choose **All Programs** (or **Programs**) > **NX 3.0** to display the cascading menu with the NX 3 options. From the cascading menu, choose **NX 3.0** to start NX 3, see Figure 1-3.

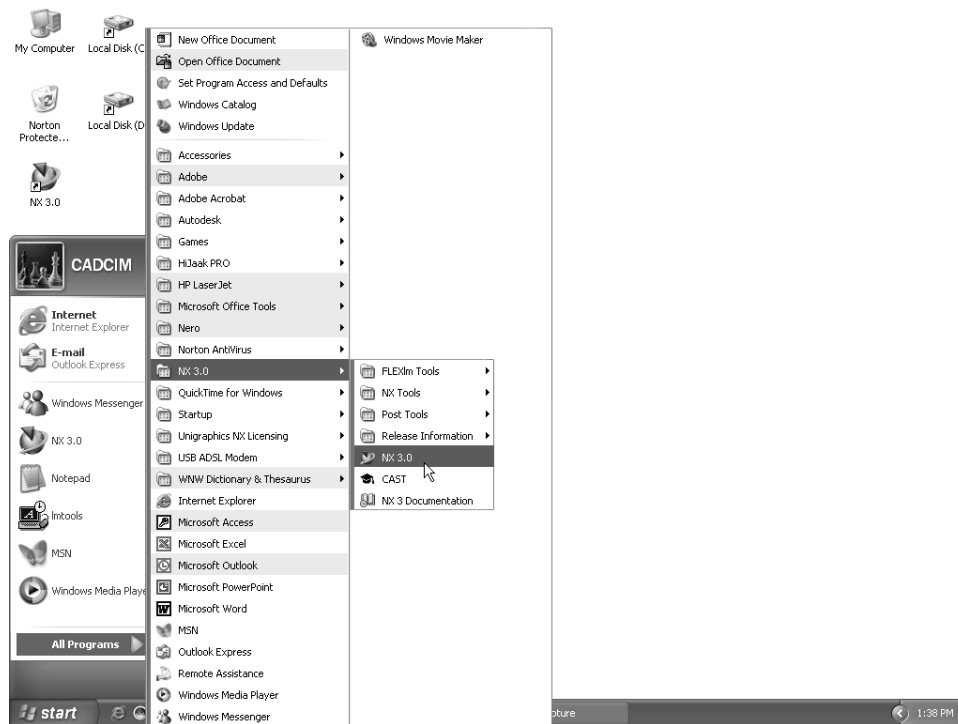


Figure 1-3 Starting NX 3

The default NX 3 screen is shown in Figure 1-4. As you can see, a tip is displayed by default, which helps you to learn more about NX 3. You can view more tips by continuously choosing the **Next Tip** button.



Figure 1-4 Default initial screen of NX 3



Tip. It is a good practice to view a couple of tips whenever you start a new session of NX. These tips will help you learn additional things about working with NX.

STARTING A NEW DOCUMENT IN NX 3

When you start a new file in NX 3, you first need to save it with a name. Only after saving the file, the new document will be started. To start a new file, choose the **New** button from the **Standard** toolbar or choose **File > New** from the menu bar; the **New Part File** dialog box will be displayed, as shown in Figure 1-5.

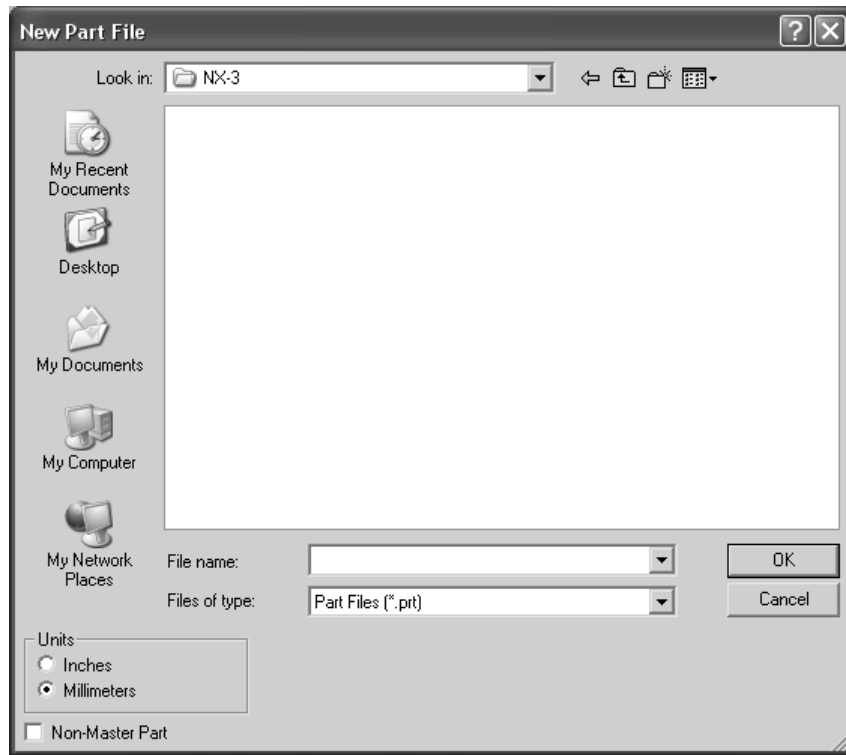


Figure 1-5 The New Part File dialog box

Browse to the folder where you need to save the file and then specify the name of the file in the **File name** edit box. After entering the name of the file, choose the **OK** button; a new part file will be started. The default screen appearance of a new part file of NX 3 is shown in Figure 1-6. You can set the units in inches or millimeters in this dialog box.



Tip. It is recommended that you create a folder with the name *NX 3* in the primary drive of your computer and then create individual folders of each chapter. You can save the part files of each chapter in the folder of that particular chapter. This will ensure a better organization of the part files that you create.

Starting the Modeling Environment for Creating Models

As mentioned earlier, you can invoke any environment of NX in the same part file. As a result, you need to start the required environment after starting a new part file. For example, to create a model, you first need to start the **Modeling** application. You can start the **Modeling** application by choosing **Application > Modeling** from the menu bar or by choosing the **Modeling** button from the **Application** toolbar, as shown in Figure 1-7.

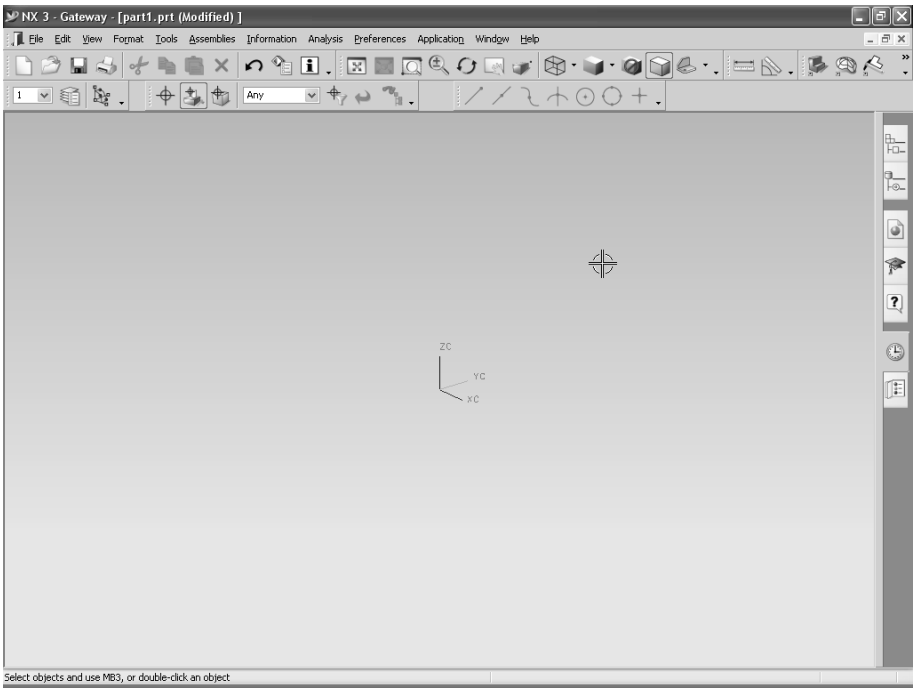


Figure 1-6 Default initial screen of a new part file

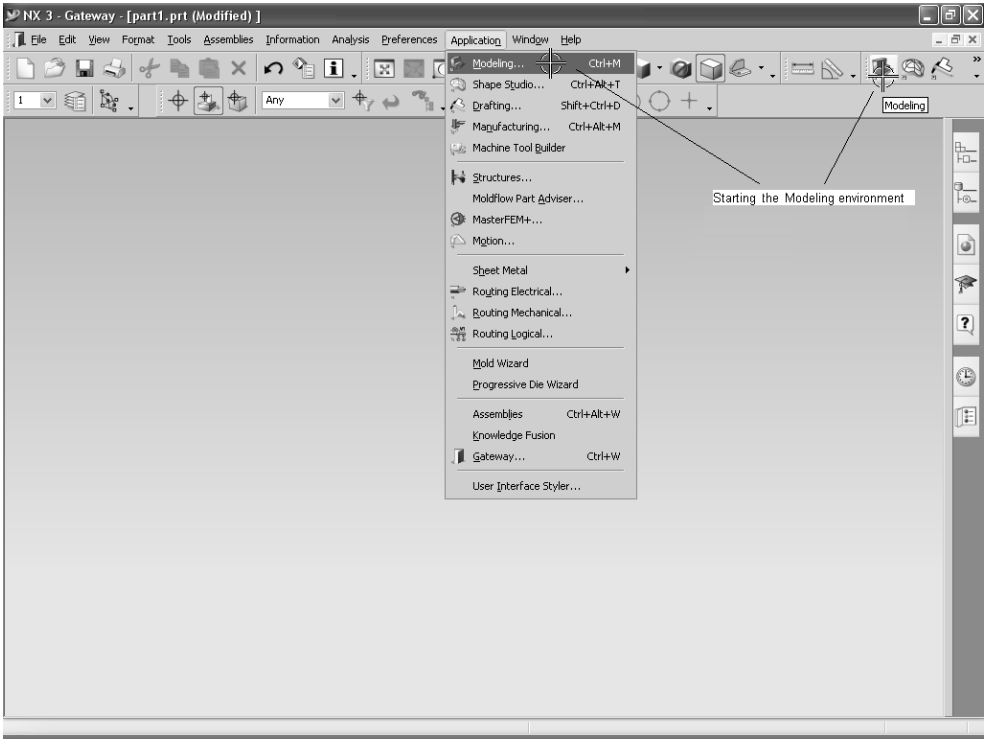


Figure 1-7 Starting the **Modeling** environment

The default screen appearance of the **Modeling** application of NX is shown in Figure 1-8.

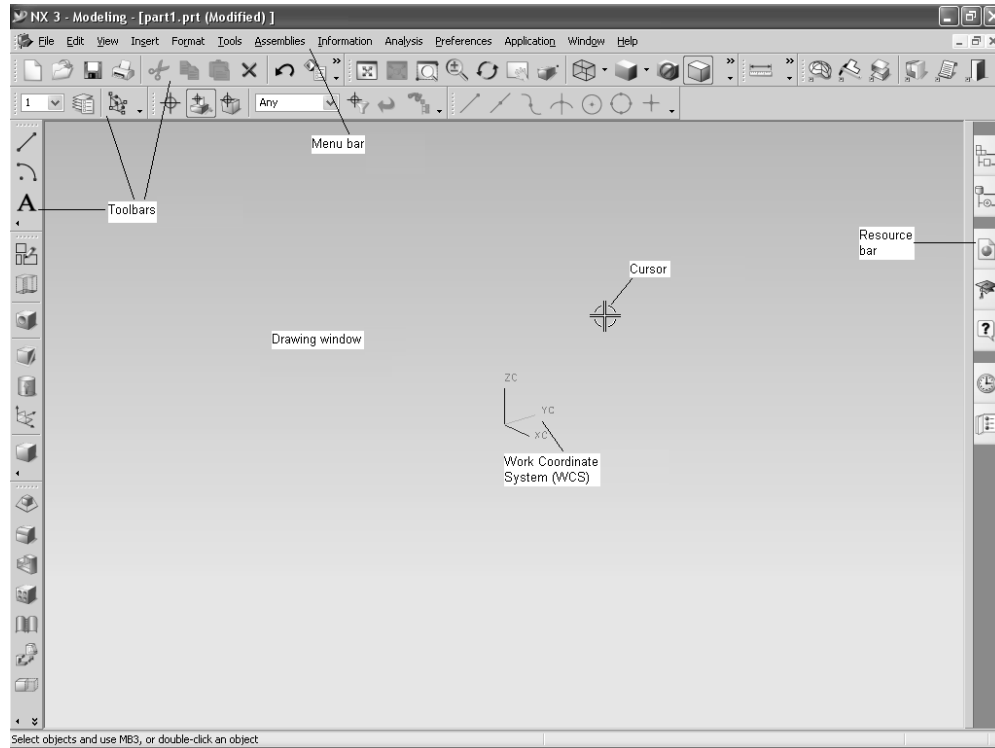


Figure 1-8 The **Modeling** environment of NX

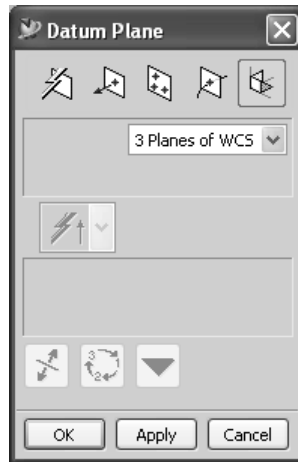
As evident in Figure 1-8, the modeling tools are provided in the toolbar available on the left of the drawing window. Also, a triad displaying the XC, YC, and ZC axes is displayed at the center of the screen. This triad is termed as the Work Coordinate System (WCS). On the right of the drawing window is the resource bar. This bar provides shortcuts to various resources of NX such as the assembly navigator, part navigator, help, history, and so on.

Invoking the Sketcher Task Environment

As mentioned earlier, the base feature (first feature) of most designs is the sketch-based feature. The profiles of the sketch-based features are defined using a sketch. Therefore, to create the base feature in most designs, you first need to invoke the **Sketcher Task** environment.

In NX, the **Sketcher Task** environment can be invoked using an existing datum plane or by selecting a planar face of an existing feature. By default, no datum plane is available in NX. Therefore, you need to first create the datum planes and then select any one of them to start the **Sketcher Task** environment.

It is recommended that you create the three default datum planes first and then use one of them to start the **Sketcher Task** environment. To create the default planes, choose **Insert > Datum/Point > Datum Plane** from the menu bar; the **Datum Plane** dialog box will be displayed, as shown in Figure 1-9.



*Figure 1-9 The **Datum Plane** dialog box*

By default, the **3 Planes of WCS** option is selected from the drop-down list in this dialog box. This option creates the three default datum planes using the current WCS (Work Coordinate System). Choose the **OK** button to create the default datum planes and exit the dialog box. Figure 1-10 shows the default screen appearance after creating the three datum planes of the WCS.

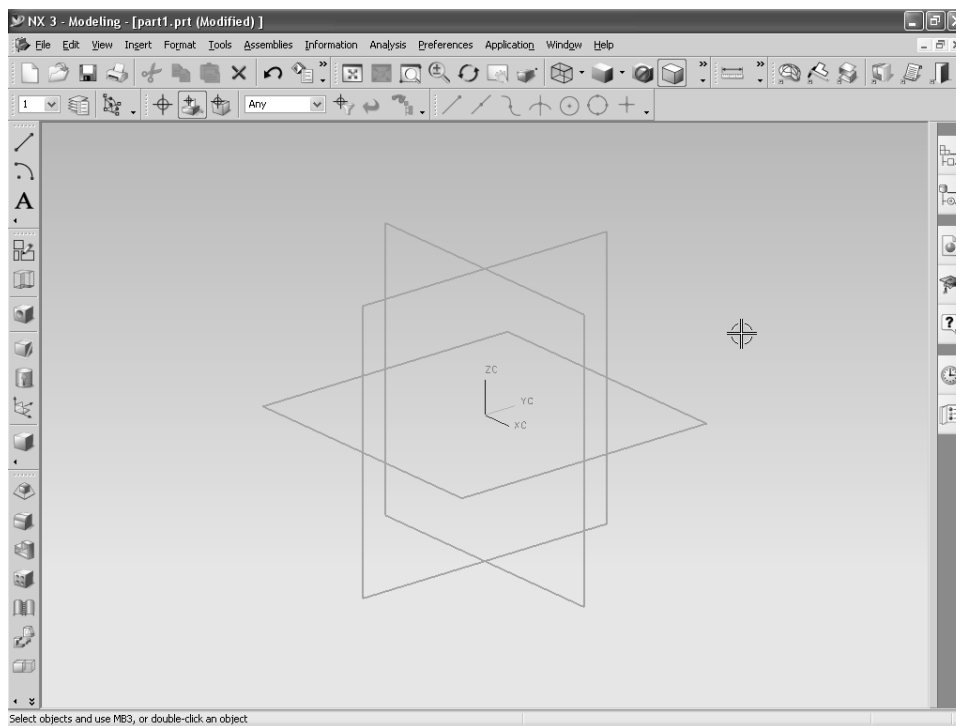


Figure 1-10 Screen appearance after creating the datum planes of the WCS

Now, to start the **Sketcher task** environment, choose the **Sketch** button from the **Form Features** toolbar on the left of the drawing window. Alternatively, you can choose **Insert > Sketch** from the menu bar. On doing so, the **Sketch Icon Options** will be displayed. The lower left area of the NX window has the prompt area where you are prompted to select an object for the sketching plane, select the sketch axis to be aligned, or choose a curve function to begin sketching. Choose the **XC-YC Plane** button from the **Sketch Icon Options**; the **Create Sketch** dialog box will be displayed. Choose **Yes** from this dialog box. Next, choose the **OK** button from the **Sketch Icon Options**; the plane will be oriented parallel to the screen and the **Sketcher Task** environment will be invoked. Figure 1-11 shows the default screen display in the **Sketcher Task** environment of NX.

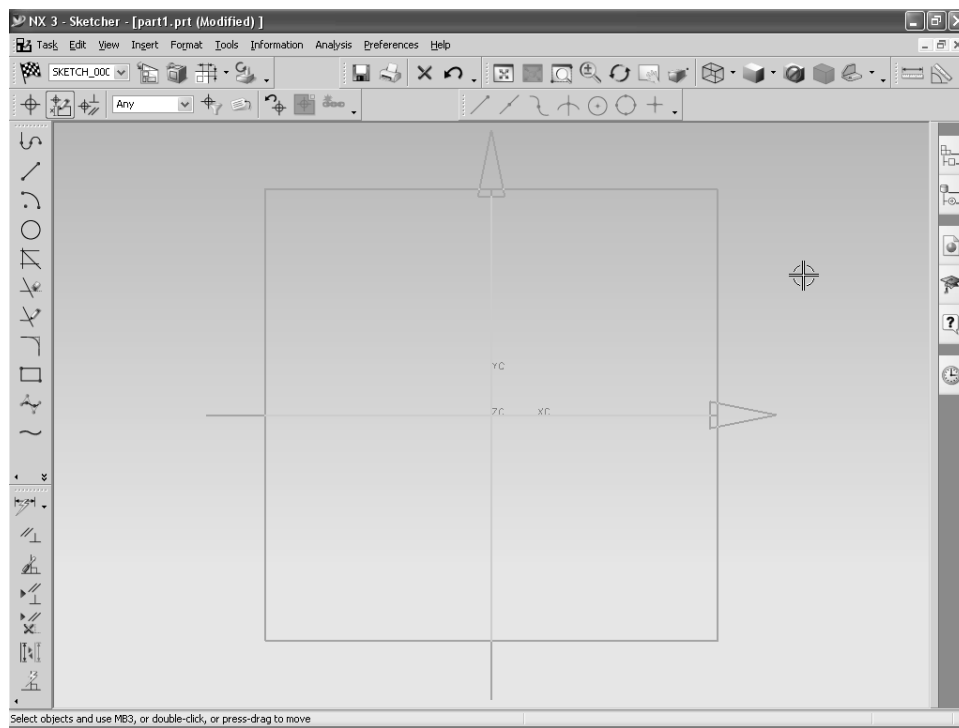


Figure 1-11 Default screen appearance in the **Sketcher Task** environment of NX 3



Tip. If the toolbar icons appear large, you can reduce their size. To do so, right-click on any toolbar to display the shortcut menu and then choose the **Customize** option at the bottom; the **Customize** dialog box will be displayed. Choose the **Options** tab and then select the **Extra Small (16)** radio button from the **Toolbar Icon Size** area. The size of the toolbar icons will be reduced.

THE DRAWING DISPLAY TOOLS

The drawing display tools are an integral part of any solid modeling tool. They enable you to zoom and pan the drawing so that you can view it clearly. The drawing display tools in NX are discussed next.

Fitting all Entities in the Current Display

Menu: View > Operation > Fit
Toolbar: View > Fit



The **Fit** tool enables you to modify the drawing display area such that all entities in the drawing fit in the current display. You can use the CTRL+F keys as the keyboard shortcut to fit the entities in the current display.

Zooming to an Area

Menu: View > Operation > Zoom
Toolbar: View > Zoom



The **Zoom** tool allows you to zoom on to a particular area by defining a box around it. When you choose this button, the default cursor is replaced by the magnifying glass cursor and you will be prompted to drag the cursor to indicate the zoom rectangle. Specify a point on the screen to define the first corner of the zoom area. Next, hold the left mouse button and drag the cursor. Now, release the left mouse button and specify another point to define the opposite corner of the zoom area. The area defined inside the rectangle will be zoomed and displayed on the screen.

Dynamic Zooming

Toolbar: View > Zoom In/Out



The **Zoom In/Out** tool enables you to dynamically zoom in or out of the drawing. When you invoke this tool, the default cursor is changed into a magnifying glass cursor with + and - sign at the center of the cursor. To zoom in, press and hold the left mouse button in the center of the screen and then drag the cursor down. Similarly, to zoom out, press and hold the left mouse button in the center of the screen and drag the cursor up.



Tip. NX allows you to restore the view that was current before it was modified using the **Zoom** or the **Zoom In/Out** tool. This can be done by choosing **View > Operation > Unzoom** from the menu bar.

Panning Drawings

Toolbar: View > Pan



The **Pan** tool allows you to dynamically pan drawings in the drawing window. When you invoke this tool, the arrow cursor will be replaced by a hand cursor and you will be

prompted to click to select the origin or drag for dynamic pan. Press and hold the left mouse button down in the drawing window and then drag to pan the drawing.

Fitting View to Selection

Toolbar: View > Fit View to Selection



This tool zooms the display such that the selected entity fits in the current display area. This tool will be available only when an entity is selected in the current sketch.

Restoring the Original Orientation of the Sketching Plane

Menu: View > Orient View to Sketch
Toolbar: Sketcher > Orient View to Sketch



Sometimes, while using the drawing display tools, you may change the orientation of the sketching plane. The **Orient View to Sketch** tool restores the original orientation that was active when you invoked the **Sketcher Task** environment. Note that this tool is available only in the **Sketcher Task** environment.

SETTING SELECTION FILTERS IN THE SKETCHING ENVIRONMENT

NX provides you with various object selection filters in the **Sketcher Task** environment. These filters allow you to define the type of entities you want to select. All these filters are available in the **Selection** toolbar available above the drawing window, on the left side. Some of these filters are discussed next.

Selecting Sketch Objects



This is the default object selection filter that is activated in the **Sketcher Task** environment. This filter allows you to select only the sketch objects and dimensions. However, you cannot select other objects such as datum planes, constraints, and so on using this filter. When you select this filter, more sub-filters will be enabled in the **Type Filter** drop-down list that allow you to make more precise selections.

Selecting General Objects



This filter can be activated by choosing the first button in the **Selection** toolbar and allows you to select any object displayed in the drawing window. When you select this filter, more sub-filters enabled in the **Type Filter** drop-down list that allow you to make more precise selections.

Selecting Constraints



This filter can be activated by choosing the third button in the **Selection** toolbar. If this filter is activated, you can select only the constraints applied to the sketched objects. You will not be allowed to select any other entity in the drawing window. You can also select the sub-filters in the **Type Filter** drop-down list to make more precise selections.

**Note**

You will learn more about constraints in later chapters.

SELECTING OBJECTS

After setting the selection filters, you are now ready to select objects in the **Sketcher Task** environment of NX. When there is no tool active in the **Sketcher Task** environment, the select mode will be invoked. Ensure that the select mode is active by pressing the ESC key. In this mode, you can select individual sketched entities available in the drawing window by clicking on them.

To select multiple entities, press and hold the left mouse button down and drag a box around them. All entities that lie partially or fully inside the box are selected.

DESELECTING OBJECTS

By default, the selected objects are red in color. If you want to deselect individual objects for the selection, press and hold down the SHIFT key and then drag a box around the entity or click on it; the entity will be deselected.



If you want to deselect all selected entities, press the ESC key. Alternatively, you can choose the **Deselect All** button from the **Selection** toolbar.

SKETCHING TOOLS

Most of the tools required to draw a sketch in the **Sketcher Task** environment of NX are available in the **Sketch Curve** toolbar located on the left of the drawing window. These tools are discussed next.

Drawing Sketches Using the Profile Tool

Menu:	Insert > Profile
Toolbar:	Sketch Curve > Profile



The **Profile** tool is the most commonly used tool to draw sketches in NX. This tool allows you to draw continuous lines and tangent/normal arcs. When you invoke this tool, the **Profile Icon Options** will be displayed with four buttons close to the top left corner of the drawing window. Also, the dynamic input boxes will be displayed below the cursor and you will be prompted to select the first point of the line or press and drag to begin the arc creation. The dynamic input boxes allow you to enter the coordinates or the length and angle of the line. The methods of creating lines and arcs using this tool are discussed next.

Drawing Lines



The option to draw straight lines is active by default when you invoke the **Profile** tool. This is because the **Line** button is chosen by default in the **Profile Icon Options**. NX allows you to draw lines using two methods. These methods are discussed next.

Drawing Lines by Entering Values

In this method of drawing lines, you can enter the coordinate values or the length and angle

of the line. The values are entered in the dynamic input boxes that are available below the cursor when you invoke the **Profile** tool. After you enter the coordinates of the start point of the line, a rubber-band line is displayed with the start point fixed at the point you specified and the endpoint attached to the cursor. Also, you will be prompted to select the second point of the line or drag to start drawing the arc.



Tip. You can toggle between the two dynamic input boxes by pressing the **TAB** key. Note that once you specify a value in one of the boxes and press the **TAB** key, the value will be locked. After specifying the value in the second box, press the **ENTER** key or the **TAB** key to register the value and draw the line using these values.

Note that on specifying the start point of the line, the dynamic input boxes are changed to **Length** and **Angle** mode, as shown in Figure 1-12.

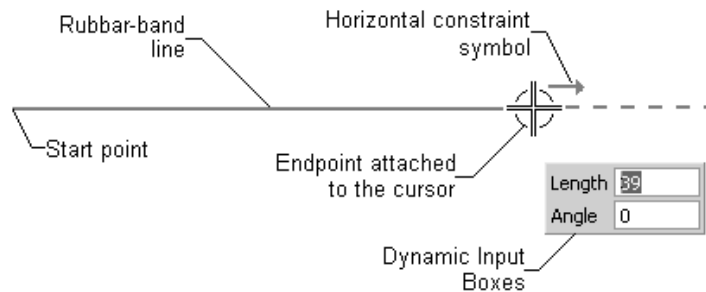


Figure 1-12 Drawing a horizontal line

This is because the **Parameter Mode** button is automatically chosen from the **Profile Icon Options**. Choosing the **Coordinate Mode** button will restore the coordinate mode option of specifying the endpoint of the line. As you move the cursor in the drawing window, the length and angle of the line are modified accordingly in the dynamic input boxes. You can draw a line by specifying its length and angle in these boxes.

After specifying the second point of the line, the line drawing process does not end. Instead, another rubber-band line starts with the start point at the endpoint of the last line and the endpoint attached to the cursor. You can repeat the above mentioned process to draw as many continuous lines.

Drawing Lines by Picking Points in the Drawing Window

This is the most convenient method of drawing lines and is extensively used in sketching.

The parametric nature of NX ensures that irrespective of the length of the line that is drawn, you can modify it to the required values using dimensions. To draw lines using this method, invoke the **Profile** tool and pick a point in the drawing window; a rubber-band line appears. Specify the endpoint of the line by picking a point in the drawing window; another rubber-band line appears with the start point as the endpoint of the last line and the endpoint attached to the cursor. You can continue specifying the endpoints of the lines to draw chain of continuous lines.



Tip. To restart drawing lines using the **Profile** tool or to break the sequence of continuous lines, right-click and choose **Back** from the shortcut menu. Alternatively, you can press the **ESC** key once. Remember that pressing the **ESC** key twice will terminate the **Profile** tool.

While drawing a line, you will notice that some symbols are displayed on the right of the cursor. For example, after specifying the start point of the line, if you move the cursor in the horizontal direction, an arrow pointing toward the right will be displayed, refer to Figure 1-12. This symbol is of the **Horizontal** constraint that is applied to the line. This constraint will ensure that the line you draw is horizontal. These constraints are automatically applied to the sketch while drawing. You will learn more about the constraints in later chapters.

Drawing Arcs



The option to draw arcs can be activated by choosing the **Arc** button in the **Profile Icon Options**. Alternatively, you can press and hold the left mouse button down and drag the cursor to invoke the arc mode. Generally, the arcs that are drawn using this tool are in continuation with the lines. Therefore, the start point of the arc is taken as the endpoint of the last line. As a result, when you invoke the arc mode, you are required to specify only the endpoint of the arc.

When you are drawing the arc in continuation with the lines, you will notice that a circle with four quadrants is displayed at the start point of the arc, as shown in Figure 1-13.

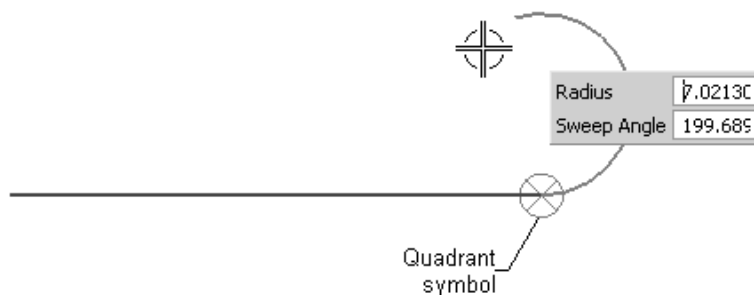


Figure 1-13 Quadrant symbol displayed while drawing the arc using the **Profile** tool

This symbol is called the **quadrant symbol** and it helps you in defining whether you want to draw a tangent arc or a normal arc. This symbol also helps you in specifying the direction of the arc.

As evident in Figure 1-13, there are four quadrants in the quadrant symbol. The movement of the cursor in these quadrants will determine whether the arc will be tangent to the line or normal to the line. If you want to draw a tangent arc, move the cursor to the start point of the arc and then move it in the quadrants along the line through a small distance; the tangent arc appears. Now, move the cursor to size the arc, as shown in Figure 1-13.

To draw a normal arc, move the cursor through a small distance in the quadrant normal to the line; the normal arc appears. Move the cursor to size the arc, as shown in Figure 1-14.

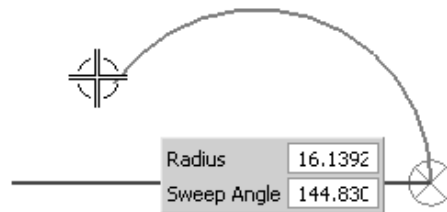


Figure 1-14 Normal arc being drawn

As soon as you invoke the arc mode, the current dynamic input boxes are changed to the **Radius** and **Sweep Angle** input boxes. These boxes allow you to specify the radius and the sweep angle to draw the arc.



Note

*If you are not drawing the arc in continuation with a line, this tool will work similar to the **Arc by 3 Points** tool, which is discussed later in this chapter.*

Using Help Lines to Locate Points

You will notice that when a sketching tool is active while drawing sketches, some dotted lines are displayed from the keypoints of the existing entities. The keypoints include the endpoints, midpoints, center points, and so on. These dotted lines are called **help lines**. If the help lines are not displayed automatically, move the cursor to the keypoints and then move the cursor away; the help lines will be displayed. The help lines are used to locate the points with reference to the keypoints of the existing entities. Figure 1-15 shows the use of the help lines to locate the start point of a new line.

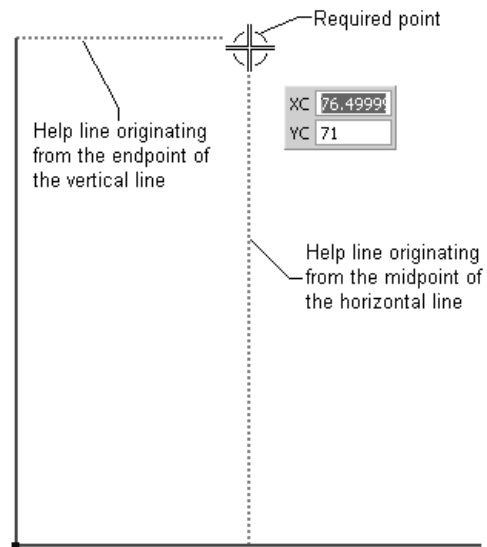


Figure 1-15 Using help lines to locate a point

Drawing Individual Lines

Menu: Insert > Line
Toolbar: Sketch Curve > Line



NX also allows you to draw individual lines. This can be done using the **Line** tool. The working of this tool is similar to working of the line mode of the **Profile** tool. The only difference is that this tool allows you to draw only one line. As a result, after you specify the endpoint of the line, no rubber-band line is displayed. Instead, you are prompted to specify the start point of the line. You can specify the start point and the endpoint of the lines by picking points on the screen or by entering values in the dynamic input boxes. You can use this tool to draw as many individual lines as required.

Drawing Arcs

Menu: Insert > Arc
Toolbar: Sketch Curve > Arc



NX allows you to draw arcs using two methods. These methods can be activated by choosing their respective buttons from the **Arc Icon Options** that are displayed when you invoke this tool. Both these methods of drawing arcs are discussed next.

Drawing Arcs Using Three Points



This method is used to draw an arc by specifying its start and endpoint, and a point on the arc. When you invoke the **Arc** tool, this method is activated by default and you will be prompted to specify the start point of the arc. You can specify the endpoint by clicking in the drawing window or by entering the coordinates in the dynamic input boxes. After specifying one of the endpoints of the arc, you will be prompted to specify the other

endpoint of the arc. You can also specify the radius of the arc by entering its value in the dynamic input box.

Note that the next prompt will depend on how you specify the second point. If you specify the endpoint of the arc by clicking a point in the drawing window, you will be prompted to select a point on the arc and the **Radius** dynamic input box will be displayed. However, if you specify the radius of the arc in the dynamic input box after specifying the first endpoint, then you will be prompted to specify the endpoint of the arc. You can click anywhere in the drawing window to draw the arc. Figure 1-16 shows a three-point arc being drawn by specifying the two endpoints and a point on the arc.

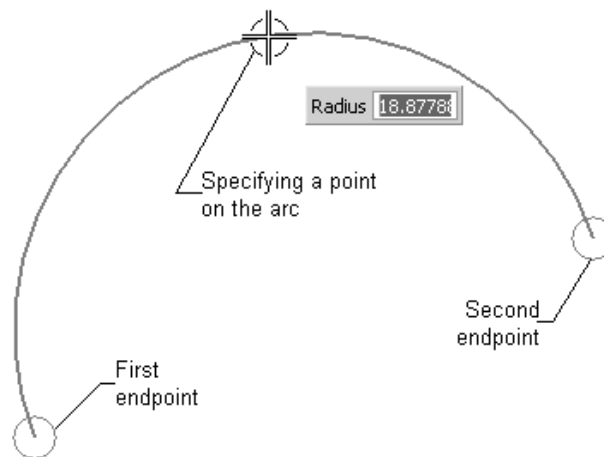


Figure 1-16 Drawing a three point arc



Tip. If the start point of the arc drawn using three points is at the endpoint of an existing entity, the resultant arc can be drawn tangent to the selected entity. To do this, while defining the point on the arc, move the cursor such that the resulting arc is tangent to the selected entity.

Drawing Arc by Specifying the Center Point and the Endpoints



This method is used to draw an arc by specifying its center point, start point, and endpoint. To invoke this option, choose the **Arc by Center and Endpoints** button from the **Arc Icon Options**; you will be prompted to specify the start point of the arc. This point is actually the center point of the arc. Next, you will be prompted to specify its start point and endpoint. Note that when you specify the start point of the arc after specifying the center point, the radius will be automatically defined. Therefore, the endpoint is used only to define the arc length. Figure 1-17 shows an arc being drawn using this method.



Tip. After specifying the center point of the arc, you can also specify its radius and the sweep angle in the dynamic input boxes. In this case, you will next be prompted to specify the start point of the arc and then the endpoint. The endpoint will basically define the direction of the arc.

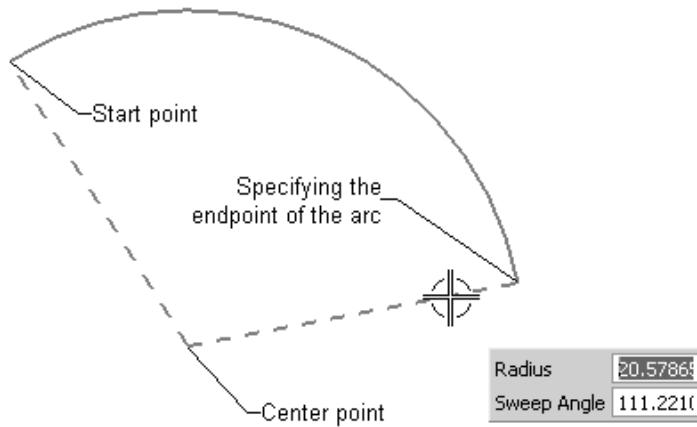


Figure 1-17 Drawing a center point arc

Drawing Circles

Menu: Insert > Circle
Toolbar: Sketch Curve > Circle

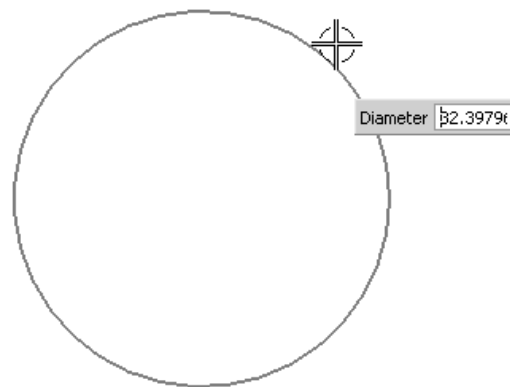


In NX, you can draw circles using two methods. These methods can be activated by choosing their respective buttons from the **Circle Icon Options** that are displayed when you invoke this tool. Both these methods of drawing circles are discussed next.

Drawing a Circle by Specifying the Center Point and Diameter



This method is active by default when you invoke the **Circle** tool and is the most widely used method of drawing circles. In this method, you need to specify the center point of a circle and a point on it or its diameter. The point on the circle defines the radius or the diameter of the circle. To draw a circle using this method, choose the **Circle by Center and Diameter** button from the **Circle Icon Options**; you will be prompted to specify the center point of the circle. Specify the center point of the circle in the drawing window. Next, you will be prompted to specify a point on the circle. Specify a point on the circle to define the radius. Alternatively, you can also enter the value of the diameter in the dynamic input box. Figure 1-18 shows a circle being drawn using this method.

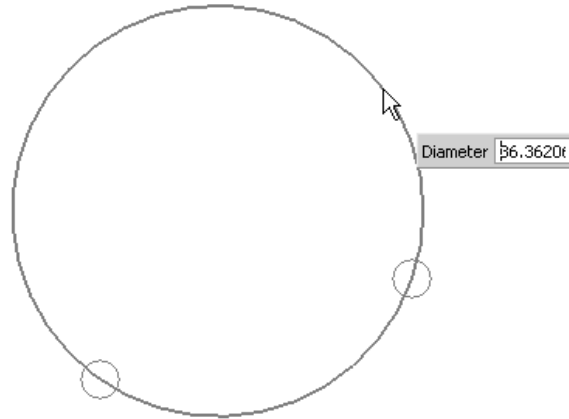


*Figure 1-18 Circle being drawn using the **Circle by Center and Diameter** method*

Drawing a Circle by Specifying Three Points



This method is used to draw a circle using the three points that you need to define on it. To invoke this method, choose the **Circle by 3 Points** button from the **Circle Icon Options**. You will be prompted to specify the center point of the circle. This point is actually the first point on the circumference of the circle. Next, you will be prompted to specify the second point of the circle. On specifying these two points, small reference circles will be displayed on these two points, as shown in Figure 1-19. Now, specify the third point, which is a point on the circle. This completes the circle.



Drawing Rectangles

Menu: Insert > Rectangle
Toolbar: Sketch Curve > Rectangle

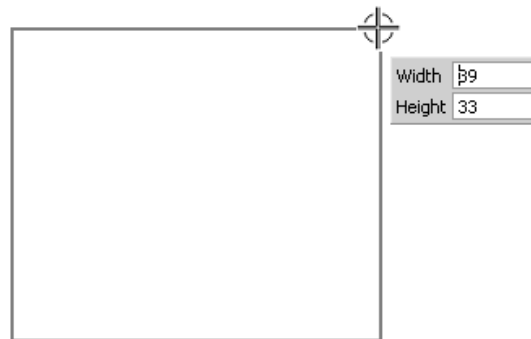


In NX, you can draw rectangles using three methods. These methods can be activated by choosing their respective buttons from the **Rectangle Icon Options** that are displayed when you invoke this tool. All three methods of drawing rectangles are discussed next.

Drawing Rectangle Using Two Opposite Corners



The **By 2 Points** method draws a rectangle by specifying two opposite corners of the rectangle. When you invoke the **Rectangle** tool, the button of this method is chosen by default in the **Rectangle Icon Options** and therefore, you are prompted to specify the first point of the rectangle. This point will work as one of the corners of the rectangle. Next, you are prompted to specify a point to create the rectangle. This point will be diagonally opposite to the point you specified initially. You can click anywhere on the screen to specify the second corner or enter the width and height of the rectangle in the dynamic input boxes. Figure 1-20 shows a rectangle drawn using the **2 Points** method.



Drawing Rectangles Using Three Points



You can invoke this method of drawing a rectangle by choosing the **By 3 Points** button from the **Rectangle Icon Options**. This method draws a rectangle using three points.

The first two points are used to define the length and angle of one of the sides of the rectangle and the third point is used to define the width of the rectangle. When you invoke this method, you are prompted to specify the first point of the rectangle. Once you specify the first point, you are prompted to specify the second point of the rectangle. Both these corners are along the same direction. Therefore, these points are used to define the length of the rectangle. After specifying the second point, you are prompted to specify a point to create the rectangle. This point is used to define the width of the rectangle. Note that if you specify the second point at a certain angle, the resulting rectangle will also be at an angle. After specifying the first point, you can also specify the length, width, and the angle of the rectangle in the dynamic input boxes. Figure 1-21 shows an inclined rectangle drawn using the **By 3 Points** method.

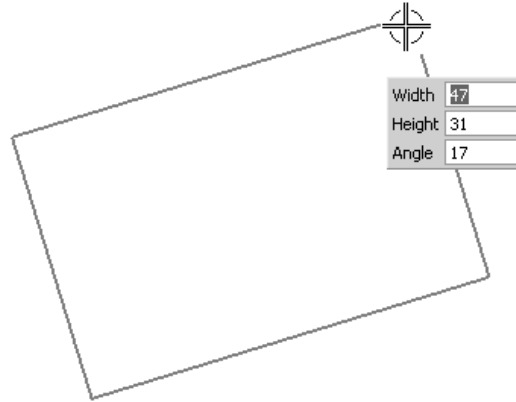


Figure 1-21 Rectangle being drawn using the **By 3 Points** method

Drawing Rectangles From Center



You can invoke this method of drawing a rectangle by choosing the **From Center** button from the **Rectangle Icon Options**. This method also draws a rectangle using three points. However, the first point is taken as the center of the rectangle in this case. When you invoke this method, you are prompted to specify the center point of the rectangle. Once you specify the center point, you are prompted to specify the second point of the rectangle. Both these points are along the same direction. Therefore, these points are used to define the length of the rectangle. After specifying the second point, you are prompted to specify a point to create the rectangle. This point is used to define the width of the rectangle. Note that if you specify the second corner at a certain angle, the resulting rectangle will also be at an angle.



Tip. By default, the buttons of some of the tools are not available in the toolbars. However, you can customize the toolbars to add these buttons. To customize the toolbars, choose the black arrow at the bottom of the vertical toolbar or at the right end of the horizontal toolbar. When you choose this arrow, a cascading menu will be displayed. Choose **Add or Remove Buttons** from the cascading menu; another cascading menu will be displayed with the names of a few toolbars. Move the cursor over the name of the toolbar to which you need to add more buttons; another cascading menu will be displayed with all the buttons that can be added to the selected toolbar. All the buttons that are currently available in the toolbar will have a check mark on their left. Choose the name of the tool whose button you want to add to the toolbar; the button will be added at the end of the toolbar.

Placing Points

Menu: Insert > Point
Toolbar: Sketch Curve > Point (*Customize to add*)



In NX, the points are placed using the **Point Constructor** dialog box (Figure 1-22). This dialog box is displayed when you invoke the **Point** tool.

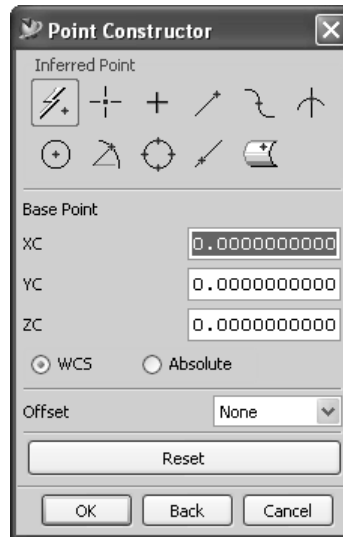


Figure 1-22 The **Point Constructor** dialog box

When this dialog box is invoked, you will be prompted to define point or specify the inferred point. This dialog box is divided into three main areas: **Button** area, **Base Point** area, and **Offset** area. The options in these areas are discussed next.

Buttons Area

This area provides you with various buttons to select the location of the resulting point. The functions of these buttons are discussed next.

Inferred Point



This button is chosen by default when you invoke the **Point Constructor** dialog box. If there are no entities in the drawing window, this option allows you to place a point anywhere in the drawing window. However, if there are some entities in the drawing window, then this option helps you select the keypoints of the entity. For example, if there are a few lines in the drawing window, then this option helps you to select the endpoints or the midpoints of the lines.

Cursor Location



This button allows you to place a point at the current location of the cursor in the drawing window. Even if there are some entities in the drawing window, they will not be considered if the **Cursor Location** button is chosen.

Existing Point

Choosing this button allows you to select the points that exist in the drawing window. As a result, the new point will be placed on top of the existing point.

End Point

Choosing this button allows you to place the point at the endpoint of existing lines, arcs, or splines.

Control Point

Choosing this button allows you to place the point at the control point of the existing sketched entities. The control points include the endpoints and midpoints of lines or arcs, center points of circles, ellipses, control points of splines, and so on.

Intersection Point

Choosing this button allows you to place the point at the intersection point of the two existing sketched entities. When you choose this button, you will be prompted to select the first and the second intersecting entity.

Arc/Ellipse/Sphere Center

Choosing this button allows you to place the point at the center point of an existing arc or circle.

Angle on Arc/Ellipse

Choosing this button allows you to place the point on the circumference of the selected arc, circle, or ellipse such that the resulting point is at some angle. When you choose this button, you are prompted to select an arc or ellipse to use as the angle reference. When you select anywhere on the circumference on the arc, circle, or ellipse, the **Base Point** area changes to the **Enter value (0-360 degrees)** area and the **Angle** edit box is displayed in this area. You can enter the angle value for the point in this edit box.

Quadrant Point

Choosing this button allows you to place the quadrant points of a circle or an ellipse. The quadrant point that is closest to the current location of the cursor will be placed.

Point on Curve/Edge

Choosing this button allows you to place the point on the selected curve or edge. The location of the point is defined in terms of its percentage from the start point of the curve. When you choose this button, you are prompted to select the curve location. When you select anywhere on the selected curve, the **Base Point** area changes to the **Curve Location** area and the **U Parameter** edit box is displayed in this area. You can enter the distance of the point, in terms of percentage, from the start point of the curve in this edit box.

**Note**

*The **Point on Surface** option will be discussed in later chapters.*

Base Point Area

This area displays the X, Y, and Z coordinates of the point that you specified using the buttons in the **Buttons** area. You can modify the coordinates of the point using the edit boxes in this area.

Reset Button

This button is used to reset the coordinates to the origin.



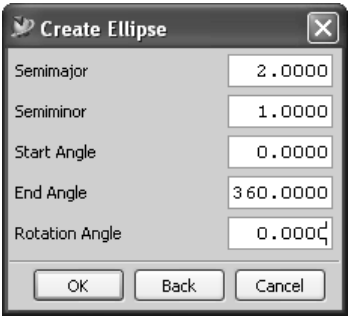
Note
The remaining options in this dialog box are advanced options and will be discussed in later chapters.

Drawing Ellipses or Elliptical Arcs

Menu: Insert > Ellipse
Toolbar: Sketch Curve > Ellipse (Customize to add)



In NX, you can draw ellipses or elliptical arcs using the **Ellipse** tool. When you invoke this tool, the **Point Constructor** dialog box will be displayed, similar to the one shown in Figure 1-22 and you will be prompted to define ellipse center - specify inferred point. In NX, the ellipse or the elliptical arc is created in two steps. In the first step, you need to locate the center point of the ellipse using the **Point Constructor** dialog box. As soon as you locate the center point of the ellipse, the **Create Ellipse** dialog box will be displayed, as shown in Figure 1-23.



This dialog box allows you to define the ellipse by specifying half of the minor and major axes of the ellipse, refer to Figure 1-24. If you want to draw an elliptical arc, you can specify the start angle and the end angle of the arc in their respective boxes. Figure 1-25 shows the parameters related to the elliptical arcs.

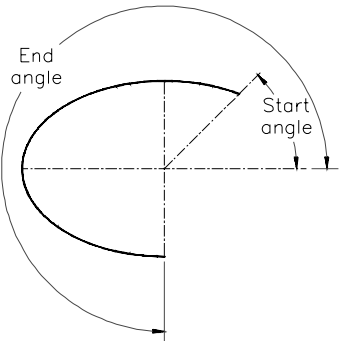
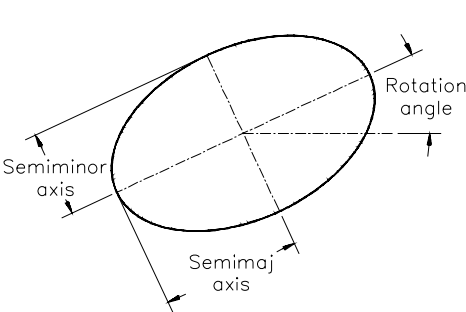


Figure 1-24 Parameters related to an ellipse **Figure 1-25** Parameters related to an elliptical arc



Tip. While placing points or drawing an ellipse, sometimes some red cross marks are displayed on the screen. To remove them, you can refresh the screen by pressing the F5 key.

Drawing General Conics

Menu: Insert > Ellipse > General Conic

Toolbar: Sketch Curve > Ellipse > General Conic (Customize to add)



To invoke the **General Conic** tool from the toolbar, choose the down arrow on the right of the **Ellipse** button and then choose the **General Conic** button from the flyout. This tool allows you to create a conic in the **Sketcher Task** environment using three points. The first two points define the endpoints of the conic and the third point defines the apex of the conic. In addition to specifying these three points, you also need to specify the projective discriminant value, termed as the rho value.

When you invoke this tool, the **Point Constructor** dialog box will be displayed. Using this dialog box, you can locate the endpoints and the apex of the conic. As soon as you specify the third point, the **General Conic** dialog box will be displayed, as shown in Figure 1-26.

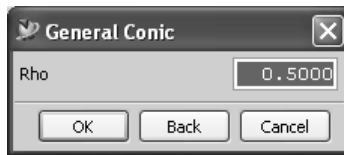


Figure 1-26 The General Conic dialog box

Figure 1-27 shows conics with various rho values.

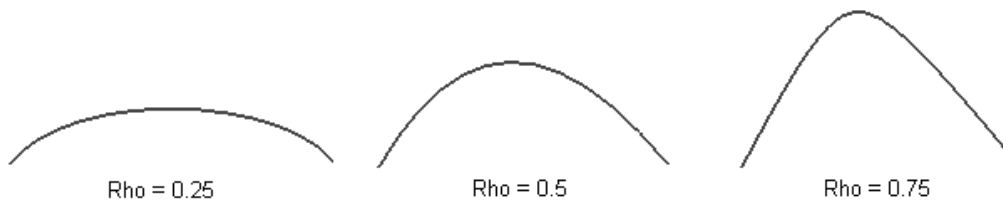


Figure 1-27 Conics with different rho values

Drawing Studio Splines

Menu: Insert > Studio Spline

Toolbar: Sketch Curve > Studio Spline



This tool allows you to create studio splines for creating free form features. When you invoke this tool, the **Studio Spline** dialog box will be displayed, as shown in Figure 1-28. The options available in this dialog box are discussed next.

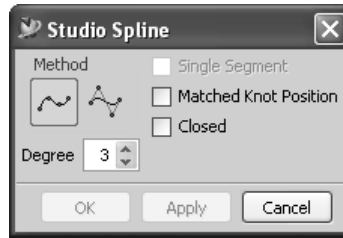


Figure 1-28 The *Studio Spline* dialog box

Method Area

There are two methods of drawing studio splines and the buttons to invoke these methods are available in the **Method** area. These methods of drawing a studio spline are discussed next.

Through Points



This is the default method of drawing splines. In this method, you can specify continuous points in the drawing area by clicking the left mouse button. These points will act as the defining points of the spline. While drawing a spline, you can move these points to change the shape of the spline and then continue drawing the spline. Figure 1-29 shows a spline being drawn using this method.

By Poles



If you use this method, the points that you specify in the drawing window act as the poles of the spline. Figure 1-30 shows a spline being drawn using this method. Remember that the display of poles is automatically removed when you finish drawing the spline.



Figure 1-29 Spline being drawn using the *Through Point* method

Degree Spinner

The **Degree** spinner is used to specify the degree of the spline. Figures 1-31 and 1-32 show splines with various degrees. Note that the degree of the spline cannot be more than the poles used to draw it.

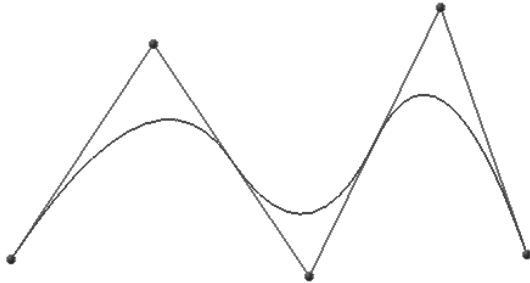


Figure 1-31 Spline with degree = 2

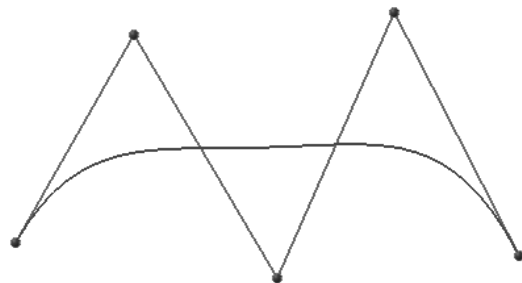


Figure 1-32 Spline with degree = 4

Single Segment

This check box is available only with the **By Poles** method and is used to create a single segment spline. However, you can specify as many number of poles. When you select this check box, the **Degree** spinner is not available.

Match Knot Position

This check box is available only with the **Through Points** method and is used to create a spline by matching the position of the defining points with the knots. In this case, the knots are placed only at the places where the defining points are specified. When you select this check box, the **Closed** check box is not available.

Closed

This check box is available for both the methods and is used to create closed splines. Figure 1-33 shows a closed spline being created.

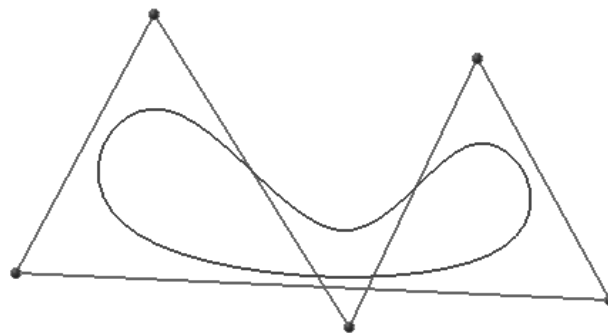


Figure 1-33 A closed spline being drawn

Filleting Sketched Entities

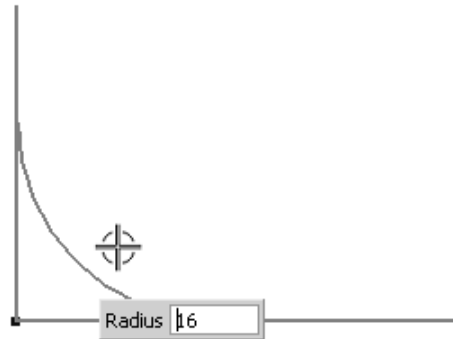
Menu: Insert > Fillet
Toolbar: Sketch Curve > Fillet



Filleting is defined as the process of rounding the sharp corners of a profile to reduce the stress concentration. You can create a fillet by removing the sharp corner and replacing it by the round corner. In NX, you can create a fillet between any two sketched entities.

You can also create a fillet using three sketched entities.

On invoking the **Fillet** tool, you will be prompted to select or drag over curves to create fillet and the **Radius** dynamic input box will be displayed below the cursor. You do not need to necessarily specify the fillet radius in advance. Instead, you can select the two entities to fillet and then move the cursor to define the radius of the fillet. Figure 1-34 shows the preview of a fillet being created between two lines. In this case, the radius value is not defined in advance. As a result, as you move the cursor, the fillet radius is modified dynamically.



Tip. When you fillet two entities, there are two solutions. The best solution is displayed by default. If you want to view the alternate solution, press the PAGE UP key on the keyboard.

The **Fillet Icon Options** that are displayed when you invoke the **Fillet** tool have two buttons. The first button in these options is the **Trim Input** button and is chosen by default. As a result, the sharp corners are automatically trimmed after filleting, as shown in Figure 1-35. If you clear this button, the sharp corners are not trimmed after filleting, as shown in Figure 1-36.

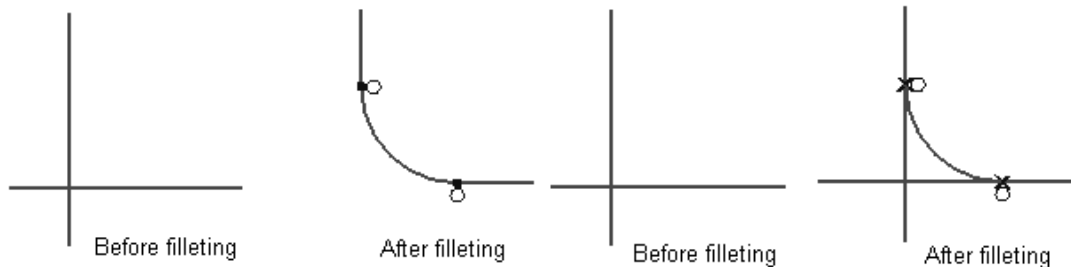


Figure 1-35 Before and after filleting with trimming

Figure 1-36 Before and after filleting without trimming

**Note**

Ideally, the profiles that have the fillet created with the sharp corners retained may not give the desired result when used to create features. Therefore, they should be avoided.

The second button in the **Fillet Icon Options** is the **Delete Third Curve** button. This button is useful if you are creating fillet using three entities. Remember that to use this option, the middle entity should be selected last. In this case, choosing this button ensures that if the fillet is tangent to the middle entity, it is automatically deleted, as shown in Figure 1-37. If you clear this button, the middle entity will not be deleted, as shown in Figure 1-38.

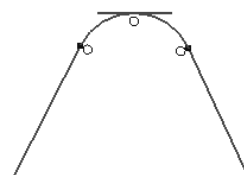
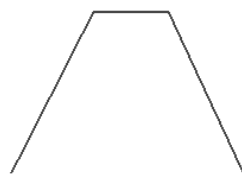
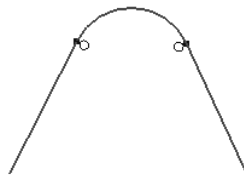
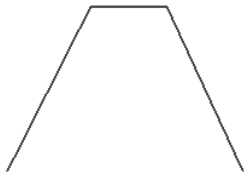


Figure 1-37 Before and after filleting with the third curve deleted

Figure 1-38 Before and after filleting with the third curve retained



Tip. In NX, you can create fillets by simply dragging the cursor across the entities that you need to fillet. For example, if you want to fillet two lines, invoke the **Fillet** tool and drag the cursor across them; the corner of these two lines will be filleted. The radius of the fillet will depend on how far from the corner you dragged the mouse.

USING SNAP POINTS WHILE SKETCHING

While drawing in the **Sketcher Task** environment, you will notice that the cursor automatically snaps to some keypoints of the sketched entities. For example, if you are specifying the center point of a circle and you move the cursor close to the endpoint of an existing line, the cursor snaps to the endpoint of the line and changes its shape to the snap cursor. Also, the endpoint snap symbol is displayed below the cursor. This suggests that the endpoint of the line has been snapped and if you click now, the center point of the circle will be coincident with the endpoint of the line.

NX allows you to control these snap settings using the **Snap Point** toolbar, which is available above the drawing window. By default, some buttons are chosen in this toolbar. You can choose more buttons to turn on the snapping using these keypoints. Figure 1-39 shows the **Snap Point** toolbar with all buttons chosen.

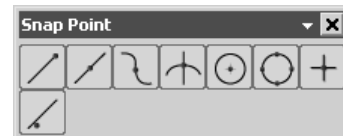


Figure 1-39 The **Snap Point** toolbar

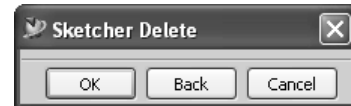
**Note**

The buttons in the **Snap Point** toolbar are available only when you invoke a sketching tool.

DELETING THE SKETCHED ENTITIES



You can delete the sketched entities by selecting them and pressing the DELETE key. You can also choose the **Delete** button from the **Standard** toolbar to delete the sketched entities. If you select the entities and then choose this button, the selected entities are deleted. However, if you choose this button without selecting any sketched entity, the **Sketcher Delete** dialog box will be displayed, as shown in Figure 1-40. You can select the entities to be deleted and then choose the **OK** button from this dialog box. Choosing the **Back** or the **Cancel** button will close this dialog box.



EXITING THE SKETCHER TASK ENVIRONMENT



After drawing the sketch, you need to exit the **Sketcher Task** environment to convert the sketch into a feature. To exit the **Sketcher Task** environment, choose the **Finish Sketch** button from the **Sketcher** toolbar. When you exit the **Sketcher Task** environment, the part modeling environment is invoked and the current view is changed to the trimetric view.

TUTORIALS

As mentioned in the Introduction, NX is parametric in nature. Therefore, you can draw the sketch of any dimensions and then modify its size by changing the values of the dimensions. However, in this chapter, you will use the dynamic input boxes to draw the sketch to the exact dimensions. This will help you in improving your sketching skills.

Tutorial 1

In this tutorial, you will draw the sketch for the model shown in Figure 1-41. The sketch to be drawn is shown in Figure 1-42. Do not dimension the sketch because the dimensions are just for your reference.
(Expected time: 30 min)

The following steps are required to complete this tutorial:

- Start NX and then start a new file.
- Invoke the **Modeling** application and then insert the three default datum planes.
- Invoke the **Sketcher Task** environment using the XC-YC plane as the sketching plane.
- Draw the outer loop of the profile using the **Profile** tool.
- Fillet the sharp corners of the outer loop using the **Fillet** tool.
- Draw the circles using the centers of the fillets to complete the profile.
- Finish the sketch, save the file, and close it.

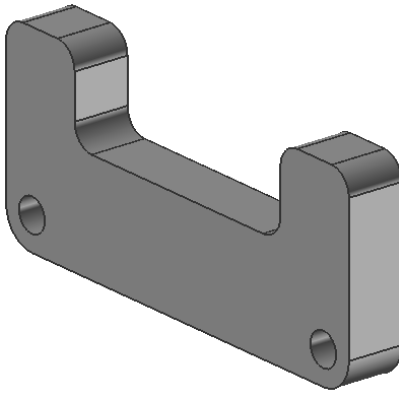


Figure 1-41 Model for Tutorial 1

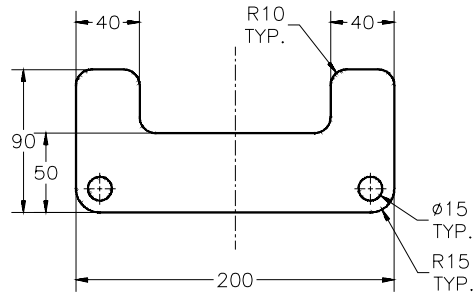


Figure 1-42 Sketch for Tutorial 1

Starting NX 3 and Starting a New File

The sketch of the model will be created in the **Sketcher Task** environment of NX 3. Therefore, you need to start NX 3 and then start a new file.

1. Choose the **Start** button at the lower left corner of the screen to display a menu with additional options.
2. Choose **All Programs** (or **Programs**) > **NX 3.0** > **NX 3.0** from the start menu to start NX 3.

When NX 3 is started, a tip is displayed by default that helps you to learn more about NX 3. You can view more tips by continuously choosing the **Next Tip** button. When you start a new file in NX 3, you first need to save the file with a name. Only after saving the file, the new document will be started.

3. To start a new file, choose the **New** button from the **Standard** toolbar or choose **File > New** from the menu bar; the **New Part File** dialog box is displayed.

It is recommended that you create a folder with the name **NX 3** in the hard drive of your machine and then create separate folders for each chapter in this book.

4. Create a folder with the name **NX 3** in the C: drive (or the base folder of the hard drive of your computer) and then create a folder with the name **c01** in the **NX 3** folder.
5. Browse to the **NX 3\c01** folder and then specify the name of the file as **c01tut1** in the **File name** edit box. After entering the name of the file, choose the **OK** button; a new file is started.

Invoking the Sketcher Task Environment in the Modeling Application

The sketch of the model will be created in the **Sketcher Task** environment of NX, which is invoked in the **Modeling** application. But before you start the **Sketcher Task** environment

in the **Modeling** application, it is recommended that you first create three default datum planes. These planes will help you in creating the model and then later on in assembling it.

1. Choose the **Modeling** button from the **Application** toolbar to invoke the **Modeling** application. Alternatively, you can choose **Application > Modeling** from the menu bar to start the **Modeling** application.



Next, you need to insert the 3 default datum planes.

2. Choose **Insert > Datum/Point > Datum Plane** from the menu bar; the **Datum Plane** dialog box is displayed.

By default, the **3 Planes of WCS** option is selected in the drop-down list available in this dialog box. This option creates three default datum planes using the current WCS (Work Coordinate System).

3. Choose the **OK** button to create the default datum planes and exit the dialog box.

Next, you need to invoke the **Sketcher Task** environment. The base sketch of this model will be created on the plane parallel to the XC-ZC plane. Therefore, you will invoke the **Sketcher Task** environment using this plane.

4. Choose the **Sketch** button from the **Form Feature** dialog box; the **Sketch Icon Options** are displayed.



5. Choose the **XC-ZC Plane** button from the **Sketch Icon Options**; the **Create Sketch** dialog box is displayed. Choose **Yes** from this dialog box.




By default, the Z axis direction of the sketching plane is toward the right of the sketching plane. Because of this, the Y axis direction is also downward. If you draw the sketch using this orientation, the sketch will be drawn upside down. Therefore, you need to reverse the Z axis direction.

6. Double-click on the Z axis, which is displayed in green; the Z axis now points to the left of the sketching plane and the Y axis now points upward.
7. Choose the **OK** button from the **Sketch Icon Options**; the **Sketcher Task** environment is invoked and the sketching plane is oriented parallel to the screen.

Drawing Lines of the Outer Loop

You will draw the lines in the outer loop using the line mode of the **Profile** tool. The line will be started from the origin, which is the point where the XC-YC, YC-ZC, and XC-ZC planes intersect. Its coordinates are 0,0,0. In the current view, the origin is the intersection point of the two planes displayed as horizontal and vertical lines.


1. Choose the **Profile** button from the **Sketch Curve** toolbar to invoke the **Profile** tool; the **Profile Icon Options** are displayed and the line mode is active by default. Also, the dynamic input boxes are displayed below the line cursor. 
2. Move the cursor close to the origin; the coordinates of the point are displayed as 0,0 in the dynamic input boxes. Click to specify the start point of the line at this point.

The point you specify is selected as the start point of the line and the endpoint is attached to the cursor. As you move the cursor on the screen, the line stretches and its length and angle values are dynamically modified in the dynamic input boxes.

Next, you need to specify the endpoint of this line and the points to define the remaining lines. This will be done using the **Length** and **Angle** dynamic input boxes.

3. Type **200** as the value in the **Length** dynamic input box and press the TAB key. Now, type **0** as the value in the **Angle** dynamic input box and press ENTER.

You will notice that the line is drawn, but it is not completely displayed in the current display. To include it in the current display, you need to modify the drawing display area using the **Fit** tool.

4. Choose the **Fit** button from the **View** toolbar; the current drawing display area is modified and the line is displayed completely in the current view. Also, the **Line** tool is still active and you are prompted to specify the second point of the line. 
5. Type **90** as the value in the **Length** in the dynamic input box and press the TAB key. Now, enter **90** as the value in the **Angle** dynamic input box and press ENTER. A vertical line of 90 length is drawn.
6. Choose the **Fit** button again to fit the drawing in the current display.
7. Enter **40** as the value in the **Length** and **-180** as the value in the **Angle** dynamic input box; a vertical line of 40 length is drawn downward.
8. Enter **40** as the value in the **Length** and **-90** as the value in the **Angle** dynamic input boxes; a horizontal line of 40 length is drawn toward the left of the last line.
9. Enter **120** as the value in the **Length** and **180** as the value in the **Angle** dynamic input box; a horizontal line of 120 length is drawn.
10. Move the cursor vertically upward. A rubber-band line is displayed with its starting point at the endpoint of the previous line and the endpoint attached to the cursor.
11. Move the cursor once toward the vertical line of 40 length drawn earlier and then move it back in the vertical direction from the start point of this line. When the line is vertical, the vertical constraint symbol is displayed.

12. Move the cursor vertically upward until the horizontal help line is displayed from the top endpoint of the vertical line of 40 length. Note that at this point, the length in the **Length** dynamic input box is **40** and the angle is **90**. Click to specify the endpoint of this line.
13. Move the cursor horizontally toward the left and make sure that the horizontal constraint symbol is displayed. Click to specify the endpoint of the line when the vertical help line is displayed from the vertical plane. If the help line is not displayed, move the cursor once on the vertical plane and then move it back.
14. Move the cursor vertically downward to the origin. If the first line is not highlighted in purple, move the cursor over it once and then move it back to the origin; the cursor snaps to the endpoint of the first line.
15. Click to specify the endpoint of the line when the horizontal constraint symbol is displayed. Choose the **Fit** button to fit the sketch in the drawing window.
16. Press the ESC key twice from the keyboard to exit the **Profile** tool. The sketch after drawing the lines is shown in Figure 1-43.

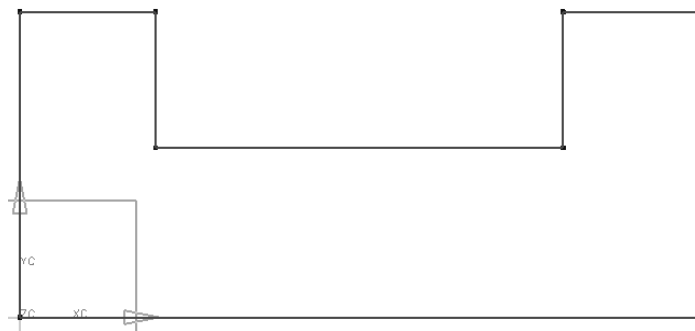


Figure 1-43 Sketch after drawing lines

Filleting the Sharp Corners

Next, you need to fillet the sharp corners so that there are no sharp edges in the final model. You can fillet the corners using the **Fillet** tool.

1. Choose the **Fillet** button from the **Sketch Curve** toolbar; the **Fillet Icon Options** are displayed.



In this tutorial, the bottom left and bottom right corners are filleted with a radius of 15 and the remaining corners with a radius of 10.

2. Type **15** as the value in the **Radius** dynamic input box and press ENTER.

3. Move the cursor over the bottom left corner of the sketch; the two lines comprising this corner are highlighted in purple. Click to select this corner; the fillet is created at the bottom left corner.
4. Similarly, move the cursor over the bottom right corner and click to select it when the two lines that form this corner are highlighted in purple.

Next, you need to modify the fillet radius value and fillet the remaining corners.

5. Type **10** as the value in the **Radius** dynamic input box and press ENTER.
6. Select the remaining corners of the sketch one by one and fillet them with a radius of 10.
7. Right-click and choose **Cancel** from the shortcut menu to exit the **Fillet** tool. The sketch, after creating fillets, is shown in Figure 1-44.

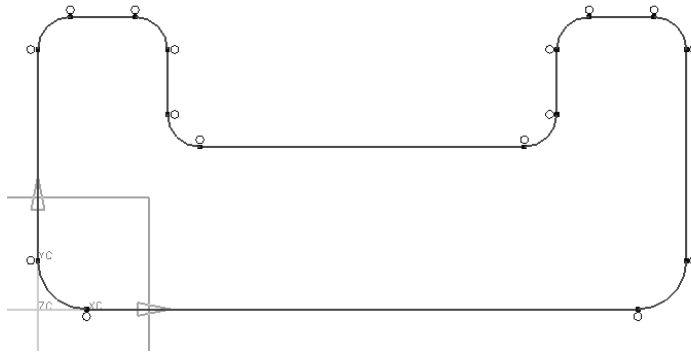



Figure 1-44 Sketch after creating fillets

Drawing Circles

Finally, you need to draw circles to complete the sketch. The circles will be drawn using the **Circle by Center and Diameter** tool. You will use the center points of the fillets as the center points of the circles.

1. Choose the **Circle** button from the **Sketch Curve** toolbar; the **Circle Icon Options** are displayed and **Circle by Center and Diameter** button is chosen by default.  Also, you are prompted to select the center of the circle.
2. Move the cursor on the lower left fillet to a point somewhere between the endpoint of the arc and the midpoint; the cursor snaps to the center point of the arc. Also, the center point snap symbol is displayed above the dynamic input boxes.

3. Click when the cursor snaps to the center point of the arc to specify the center point of the circle.
4. Type **15** as the value in the **Diameter** dynamic input box and press ENTER; a circle of the specified diameter is drawn at the specified center point. Also, another circle of 15 diameter is attached to the cursor.
5. Move the cursor over the bottom right fillet to a point between the endpoint and the mid-point of the arc; the cursor snaps to the center point of the arc and the center point snap symbol is displayed above the dynamic input box.
6. Click when the cursor snaps to the center point of the arc; the circle will be drawn at the required location.

**Note**

*If by mistake you select an incorrect point as the center point of the circle, you can remove the unwanted circle by choosing the **Undo** button from the **Standard** toolbar.*

7. Exit the **Circle** tool by pressing the ESC key twice.

This completes the sketch for the model for Tutorial 1. The final sketch for Tutorial 1 is shown in Figure 1-45.

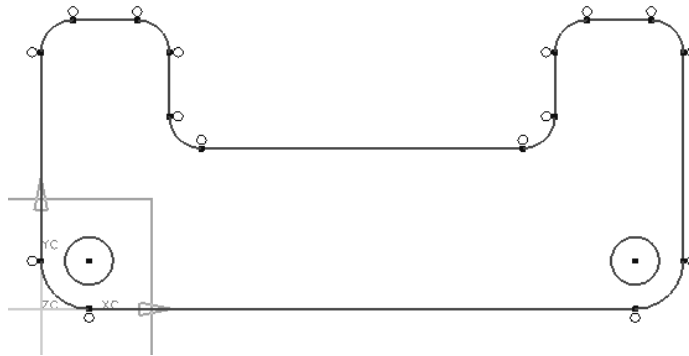



Figure 1-45 Final sketch for Tutorial 1



Tip. For a better visualization, the background color in the graphics in this book are set to white. To change the background color, choose **Preferences > Visualization** from the menu bar to display the **Visualization Preferences** dialog box. Choose the **Color Palette** tab and then choose the **Edit Background** button; the **Edit Background** dialog box is displayed. Choose the **Plain Color** swatch and select the white color. Now, select the **Plain** radio button from the **Shaded Views** area.

Finishing the Sketch and Saving the File

NX allows you to save the sketch file in the **Sketcher Task** environment.

1. Finish the sketch and choose the **Fit** button to fit the sketch in the drawing window.
2. Choose the **Save** button from the **Standard** toolbar to save the sketch. Note that the name and the location of the document were specified when you started the new file.
3. Choose the **Finish Sketch** button from the **Sketcher** toolbar. 
4. Choose **File > Close > Selected Parts** from the menu bar; the **Close Part** dialog box is displayed.
5. Select the name of the current file from the display box and then choose **OK**; the current file is closed.

Tutorial 2

In this tutorial, you will draw the profile of the model shown in Figure 1-46. The profile to be drawn is shown in Figure 1-47. Do not dimension the profile because the dimensions are just for your reference.
(Expected time: 30 min)

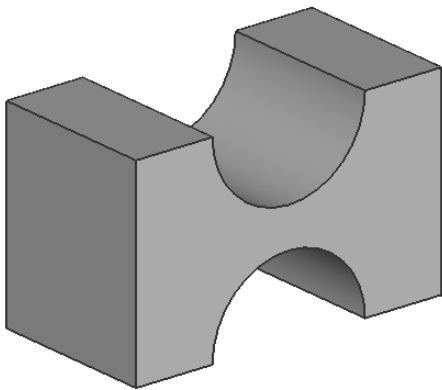


Figure 1-46 Model for Tutorial 2

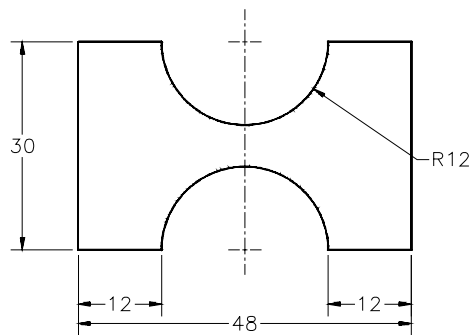


Figure 1-47 Sketch for Tutorial 2

The following steps are required to complete this tutorial:

- a. Start a new file.
- b. Invoke the **Modeling** application and create the three default datum planes.
- c. Select the YC-ZC plane as the sketching plane and invoke the **Sketcher Task** environment.
- d. Draw the sketch of the model using the **Profile** tool.
- e. Finish the sketch, save the file, and close it.

Starting a New File

It is assumed that you are continuing the work after completing Tutorial 1. Therefore, NX session is already open on your computer. You can start a new part file by choosing the **New** button from the **Standard** toolbar, which remains on the screen after you close all the files.

1. To start a new file, choose the **New** button from the **Standard** toolbar or choose **File > New** from the menu bar; the **New Part File** dialog box is displayed.
2. Browse to the **NX 3\c01** folder, if it is not the current folder. Specify the name of the file as **c01tut2** in the **File name** edit box.
3. Choose the **OK** button; a new file is started.

Invoking the Sketcher Task Environment in the Modeling Application

1. Choose the **Modeling** button from the **Application** toolbar to invoke the **Modeling** application. Alternatively, you can choose **Application > Modeling** from the menu bar to start the **Modeling** application.



As mentioned earlier, it is recommended that before you invoke the **Sketcher Task** environment, you should first create the three default datum planes.

2. Choose **Insert > Datum/Point > Datum Plane** from the menu bar; the **Datum Plane** dialog box is displayed.

By default, the **3 Planes of WCS** option is selected in the drop-down list in this dialog box. This option creates three default datum planes using the current WCS (Work Coordinate System).

3. Choose the **OK** button to create the default datum planes and exit the dialog box.

Next, you need to invoke the **Sketcher Task** environment. The base sketch of this model will be created on the plane parallel to the XC-ZC plane. Therefore, you will invoke the **Sketcher Task** environment using this plane.

4. Choose the **Sketch** button from the **Form Feature** dialog box; the **Sketch Icon Options** are displayed.




5. Choose the **YC-ZC Plane** button from the **Sketch Icon Options**; the **Create Sketch** dialog box is displayed. Choose **Yes** from this dialog box.



6. Choose the **OK** button from the **Sketch Icon Options**; the **Sketcher Task** environment is invoked and the sketching plane is oriented parallel to the screen.

Drawing the Sketch

The sketch that you need to draw consists of multiple lines and two arcs. All these entities can be drawn using the **Line** and **Arc** options of the **Profile** tool.

1. Choose the **Profile** button from the **Sketch Curve** toolbar to invoke the **Profile** tool; the **Profile Icon Options** are displayed and the line mode is active by default. Also, the dynamic input boxes are displayed below the line cursor. 
2. Move the cursor close to the origin; the coordinates of the point are displayed as 0,0 in the dynamic input boxes. Click to specify the start point of the line at this point.

The point you specify is selected as the start point of the line and the endpoint is attached to the cursor. As you move the cursor on the screen, the line stretches and its length and angle values are dynamically modified in the dynamic input boxes.

Next, you need to specify the endpoint of this line and the points to define the remaining lines. This will be done using the **Length** and **Angle** dynamic input boxes.

3. Type **12** as the value in the **Length** dynamic input box and press the TAB key. Now, type **0** as the value in the **Angle** dynamic input box and press ENTER.

The first line is drawn and another rubber-band line is displayed with the start point at the endpoint of the previous line and the endpoint attached to the cursor. But because the next entity is an arc, you need to invoke the arc mode.

4. Choose the **Arc** button from the **Profile Icon Options** to invoke the arc mode.

A rubber-band arc is displayed with the start point fixed at the endpoint of the last line and the endpoint attached to the cursor. Also, the quadrant symbol is displayed at the start point of the arc.

5. Move the cursor to the start point of the arc and then move it vertically upward through a small distance. Now, move the cursor toward the right; you will notice that a normal arc starts from the endpoint of the last line.
6. Enter **12** as the radius and **180** as the angle of the arc in the **Radius** and **Sweep Angle** dynamic input boxes, respectively, in the ribbon bar.

The preview of the resulting arc is displayed, but the arc is still not drawn. To draw the arc, you need to specify a point on the screen with the values mentioned in the ribbon bar.

7. Move the cursor horizontally toward the right to the horizontal plane and click when the preview of the required arc is displayed. The arc is drawn and the line mode is invoked again.
8. Enter **12** as the length and **0** as the angle in the **Length** and **Angle** dynamic input boxes,

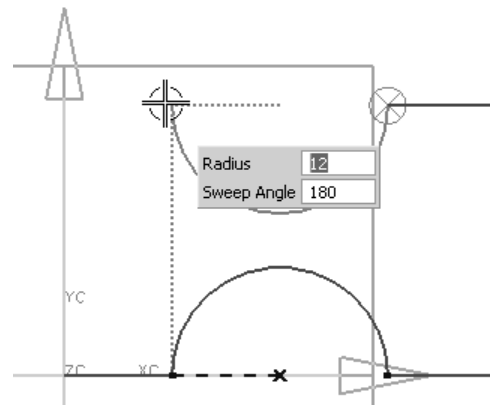
respectively. Choose the **Fit** button from the **View** toolbar to fit the sketch in the drawing window.

9. Enter **30** as the length and **90** as the angle in the **Length** and **Angle** dynamic input boxes, respectively.
10. Move the cursor horizontally toward the left. Make sure the horizontal constraint symbol is displayed. Click to specify the endpoint of the line when the vertical help line is displayed from the endpoint of the arc.

Next, you need to draw the arc. Therefore, you need to invoke the arc mode.

11. Choose the **Arc** button from the **Profile Icon Options** to invoke the arc mode; a rubber-band arc is displayed with its start point fixed at the endpoint of the last line.
12. Move the cursor to the start point of the arc and then move it vertically downward through a small distance. When the normal arc appears, move the cursor toward the left.
13. Move the cursor over the lower arc once and then move it toward the left in line with the upper right horizontal line from where this arc starts.

A horizontal help line is displayed originating from the center of the arc being drawn. At the point where the cursor is vertically in line with the start point of the lower arc, the vertical help line appears from the start point of the lower arc, as shown in Figure 1-48.



14. Click to define the endpoint of the arc when the horizontal and vertical help lines are displayed. The arc is drawn and the line mode is invoked again.
15. Enter **12** as the length and **180** as the angle in the **Length** and **Angle** dynamic input boxes, respectively.
16. Move the cursor to the first line and then move it to the start point of this line; the cursor snaps to the start point of the line.
17. Click to define the endpoint of this line when the cursor snaps to the start point of the first line.
18. Press the ESC key twice to exit the **Profile** tool. The final sketch of the model is shown in Figure 1-49.

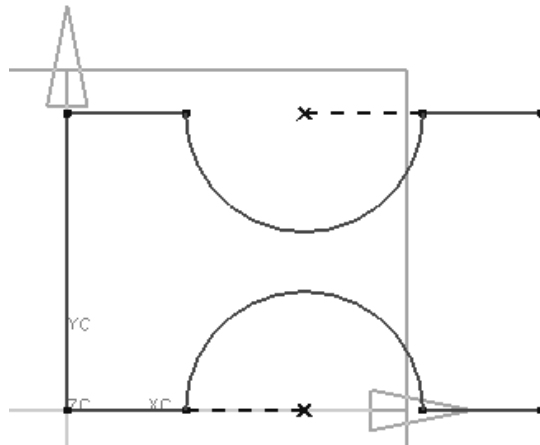



Figure 1-49 Final sketch for Tutorial 2

Finishing the Sketch and Saving the File

NX allows you to save the sketch file in the **Sketcher Task** environment.

1. Finish the sketch and choose the **Fit** button to fit the sketch in the drawing window.
2. Choose the **Save** button from the **Standard** toolbar to save the sketch. Note that the name and the location of the document was specified when you started the new file.
3. Choose the **Finish Sketch** button from the **Sketcher** toolbar. 
4. Choose **File > Close > Selected Parts** from the menu bar; the **Close Part** dialog box is displayed.
5. Select the name of the current file from the display box and then choose **OK**; the current file is closed.

Tutorial 3

In this tutorial, you will draw the profile for the base feature of the model shown in Figure 1-50. The profile to be drawn is shown in Figure 1-51. Do not dimension the profile because the dimensions are just for your reference. **(Expected time: 30 min)**

The following steps are required to complete this tutorial:

- a. Start a new file.
- b. Invoke the **Modeling** application and create the three default datum planes.
- c. Select the XC-YC plane as the sketching plane and invoke the **Sketcher Task** environment.
- d. Draw the sketch of the model using the **Profile** tool and the **Rectangle** tool.
- e. Finish the sketch, save the file, and close it.

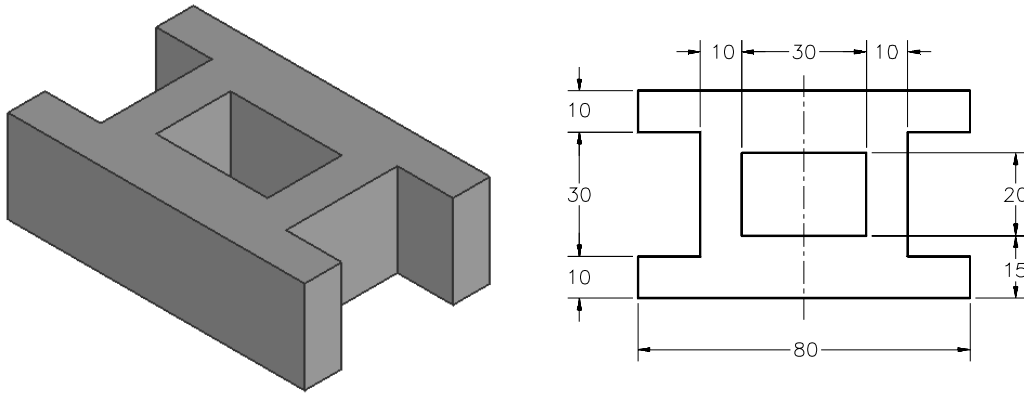


Figure 1-50 Sketch for Tutorial 3

Starting a New File

It is assumed that you are continuing the work after completing Tutorial 2. Therefore, NX session is already open on your computer. You can start a new part file by choosing the **New** button from the **Standard** toolbar, which remains on the screen after you close all the files.

1. To start a new file, choose the **New** button from the **Standard** toolbar or choose **File > New** from the menu bar; the **New Part File** dialog box is displayed.
2. Browse to the **NX 3\c01** folder, if it is not the current folder. Specify the name of the file as **c01tut3** in the **File name** edit box.
3. Choose the **OK** button; a new file is started.

Invoking the Sketcher Task Environment in the Modeling Application

1. Choose the **Modeling** button from the **Application** toolbar to invoke the **Modeling** application. Alternatively, you can choose **Application > Modeling** from the menu bar to start the **Modeling** application.





As mentioned earlier, it is recommended that before you invoke the **Sketcher Task** environment, you should first create the three default datum planes.

2. Choose **Insert > Datum/Point > Datum Plane** from the menu bar; the **Datum Plane** dialog box is displayed.

By default, the **3 Planes of WCS** option is selected in the drop-down list available in this dialog box. This option creates three default datum planes using the current WCS (Work Coordinate System).


3. Choose the **OK** button to create the default datum planes and exit the dialog box.

Next, you need to invoke the **Sketcher Task** environment. The base sketch of this model will be created on the plane parallel to the XC-ZC plane. Therefore, you will invoke the **Sketcher Task** environment using this plane.

4. Choose the **Sketch** button from the **Form Feature** dialog box; the **Sketch Icon Options** are displayed. 
5. Choose the **XC-YC Plane** button from the **Sketch Icon Options**; the **Create Sketch** dialog box is displayed. Choose **Yes** from this dialog box. 
6. Choose the **OK** button from the **Sketch Icon Options**; the **Sketcher Task** environment is invoked and the sketching plane is oriented parallel to the screen.

Drawing the Outer Profile of the Sketch

The outer profile of the sketch consists of lines and therefore can be drawn using the **Profile** tool.

1. Choose the **Profile** button from the **Sketch Curve** toolbar to invoke the **Profile** tool; the **Profile Icon Options** are displayed and the line mode is active by default. Also, the dynamic input boxes are displayed below the line cursor. 
2. Move the cursor close to the origin; the coordinates of the point are displayed as 0,0 in the dynamic input boxes. Click to specify the start point of the line at this point.

As you move the cursor on the screen, the line stretches and its length and angle values are dynamically modified in the dynamic input boxes.

3. Type **80** as the value in the **Length** dynamic input box and press the TAB key. Now, type **0** as the value in the **Angle** dynamic input box and press ENTER.
4. Enter **10** as the length and **90** as the angle in the **Length** and **Angle** dynamic input boxes, respectively.
5. Enter **15** as the length and **180** as the angle in the **Length** and **Angle** dynamic input boxes, respectively.
6. Enter **30** as the length and **90** as the angle in the **Length** and **Angle** dynamic input boxes, respectively.
7. Enter **15** as the length and **0** as the angle in the **Length** and **Angle** dynamic input boxes, respectively.
8. Enter **10** as the length and **90** as the angle in the **Length** and **Angle** dynamic input boxes, respectively.
9. Enter **80** as the length and **180** as the angle in the **Length** and **Angle** dynamic input boxes, respectively.

- Evaluation Chapter. Do not copy. Visit www.cadclm.com for details**



Next, you need to draw the inner profile, which is a rectangle. You can use the **By 2 Points** option of the **Rectangle** tool to draw this rectangle.

-

3. Specify the width and height of the rectangle as **30** and **20** in the **Width** and **Height** dynamic input boxes, respectively; the preview of the rectangle is displayed, but it is not actually drawn yet. As you move the cursor in the drawing window, the rectangle will also move.
4. Move the cursor close to the top left corner of the drawing window and then click to draw the rectangle.
5. Press the ESC key twice to exit the tool. The final sketch for Tutorial 3 is shown in Figure 1-53.

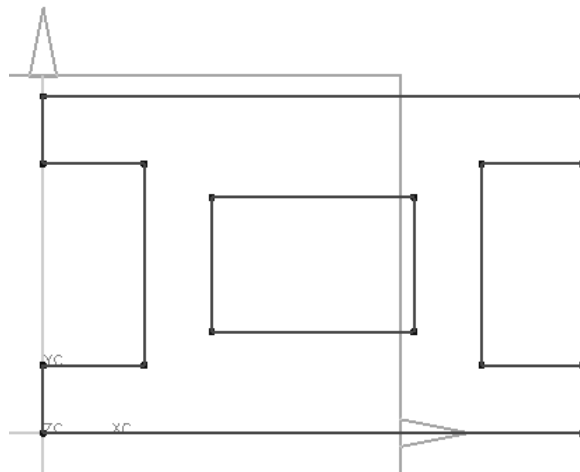



Figure 1-53 Final sketch for Tutorial 3

Finishing the Sketch and Saving the File

NX allows you to save the sketch file in the **Sketcher Task** environment.

1. Finish the sketch and choose the **Fit** button to fit the sketch in the drawing window.
2. Choose the **Save** button from the **Standard** toolbar to save the sketch. Note that the name and the location of the document was specified when you started the new file.
3. Choose the **Finish Sketch** button from the **Sketcher** toolbar. 
4. Choose **File > Close > Selected Parts** from the menu bar; the **Close Part** dialog box is displayed.
5. Select the name of the current file from the display box and then choose **OK**; the current file is closed

Self-Evaluation Test

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

1. Most of the designs created in NX consist of sketch-based features and placed features. (T/F)
2. After starting a new file, you first need to invoke the **Modeling** application before you can draw the sketches. (T/F)
3. You can use the dynamic input boxes to specify the exact values of the sketched entities. (T/F)
4. The **Sketch** button is chosen to invoke the **Sketcher Task** environment. (T/F)
5. You can restore the original orientation of the sketching plane using the _____ tool in the **Sketcher** toolbar.
6. You can invoke the arc mode within the **Profile** tool by choosing the _____ button from the **Profile Icon Options**.
7. You can fillet the corners in the sketch using the _____ tool.
8. You can retain the sharp corners even after filleting them by clearing the _____ button from the **Fillet Icon Options**.
9. Choosing the _____ button from the **Rectangle Icon Options** allows you to draw rectangle from the center.
10. You can exit the **Sketcher Task** environment by choosing the _____ button from the **Sketcher** toolbar.

Review Questions

Answer the following questions:

1. Which one of the following dialog box is displayed when you choose the **New** button from the **Standard** toolbar to start a new file?
 - (a) **New Part File**
 - (b) **New File**
 - (c) **New**
 - (d) **Part File**

2. In NX, the conics are created using which one of the following tools?
- (a) **General Conic** (b) **Conic**
(c) **Round** (d) None
3. Which mode is automatically invoked from the **Profile Icon Options** when you specify the start point of the line?
- (a) **Coordinate Mode** (b) **Angle Mode**
(c) **Parameter Mode** (d) None
4. In NX, how many methods can be used to draw arcs?
- (a) 4 (b) 2
(c) 6 (d) 5
5. Which mode is available in the **Studio Spline** dialog box along with the **By Poles** method to draw splines?
- (a) **No Poles** (b) **From Poles**
(c) **From Points** (d) **Through Points**
6. The files in NX are saved with *.prt* extension. (T/F)
7. You can select the entities by dragging a box around them. (T/F)
8. You can set the selection mode to select only the sketched entities. (T/F)
9. In NX, you can create fillets by simply dragging the cursor across the entities that you want to fillet. (T/F)
10. In NX, you cannot draw a rectangle from its center. (T/F)

Exercises

Exercise 1

Draw the sketch of the base feature of the model shown in Figure 1-54. The sketch to be drawn is shown in Figure 1-55. Do not dimension the profile because the dimensions are just for your reference.
(Expected time: 30 min)

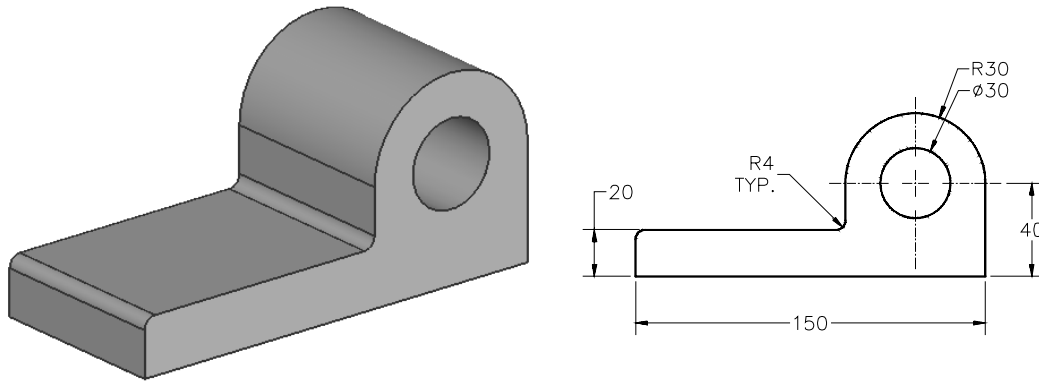


Figure 1-54 Model for Exercise 1

Exercise 2

Draw the sketch of the base feature of the model shown in Figure 1-56. The sketch to be drawn is shown in Figure 1-57. Do not dimension the profile because the dimensions are just for your reference.
(Expected time: 30 min)

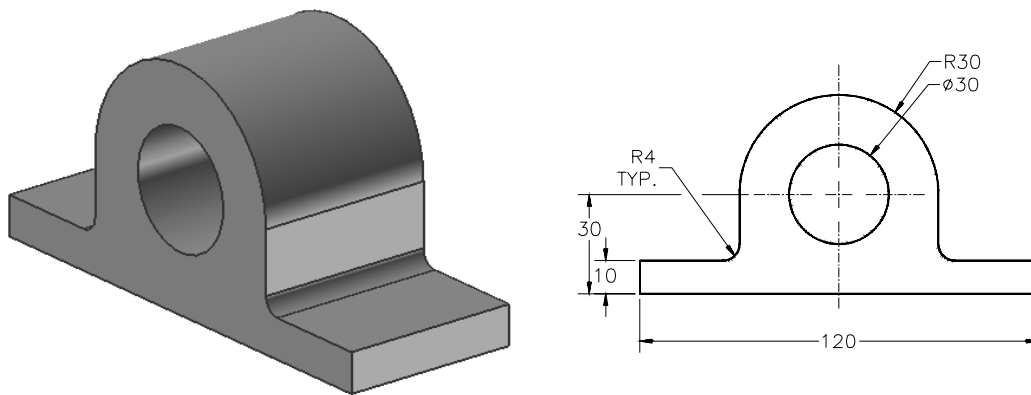


Figure 1-56 Model for Exercise 2

Exercise 3

Draw the sketch of the base feature of the model shown in Figure 1-58. The sketch to be drawn is shown in Figure 1-59. Do not dimension the profile because the dimensions are just for your reference.
(Expected time: 30 min)

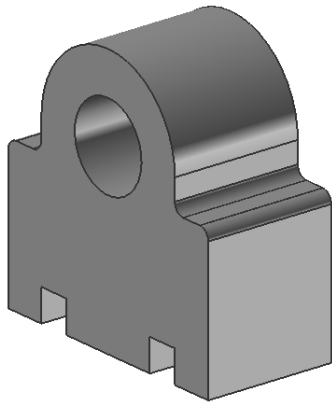


Figure 1-58 Model for Exercise 3

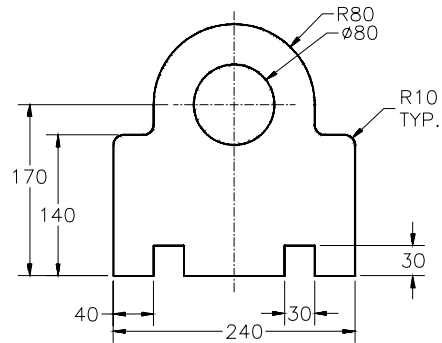


Figure 1-59 Profile for Exercise 3

Answers to Self-Evaluation Test

1. T, 2. T, 3. T, 4. T, 5. Orient View to Sketch, 6. Arc, 7. Fillet, 8. Trim Inputs, 9. From Center, 10. Finish Sketch