



Chapter 2

Drawing Sketches for Solid Models

Learning Objectives

After completing this chapter, you will be able to:

- *Understand the need of Sketcher environment.*
- *Start NX and create a new file in it.*
- *Invoke different NX environments.*
- *Understand the need of datum planes.*
- *Create three fixed datum planes.*
- *Invoke the Sketcher environment.*
- *Use various drawing display tools.*
- *Understand different selection filters.*
- *Select and deselect objects.*
- *Use various sketching tools.*
- *Use different snap points options.*
- *Delete sketched entities.*
- *Exit the Sketcher environment.*

THE SKETCHER ENVIRONMENT

Most designs created in NX consist of sketch-based features and placed features. A sketch is a combination of number of two-dimensional (2D) entities such as lines, arcs, circles, and so on. The features such as extrude, revolve, and sweep that are created by using 2D sketches are known as sketch-based features. The features such as fillet, chamfer, thread, and shell that are created without using a sketch are known as placed features. In a design, the base feature or the first feature is always a sketch-based feature. For example, the sketch shown in Figure 2-1 is used to create the solid model shown in Figure 2-2. In this figure, the fillets and the chamfers are the placed features.

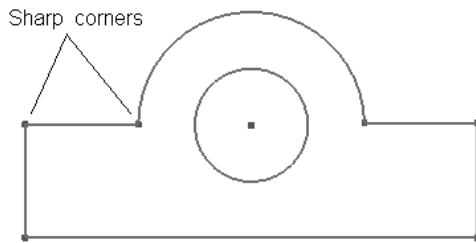


Figure 2-1 Profile for the sketch-based feature of the solid model shown in Figure 2-2

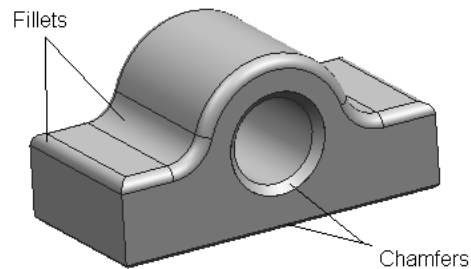


Figure 2-2 Solid model created using the sketch-based and placed features

To create sketch-based features, invoke the Sketcher environment and draw the sketch. Exit the Sketcher environment and then use the solid modeling tools to convert the sketch into a feature.

Unlike other solid modeling software packages where you need to use separate files for starting different environments, NX uses only a single type of file to start different environments. In 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. For example, you can draw sketches and convert them into features, assemble other parts with the current part, and generate drawing views in a single *.prt* file.

STARTING NX 7

Desktop:	NX 7.0 Shortcut Icon
Taskbar:	Start > All Programs (or Programs) > UGS NX 7.0 > NX 7.0

You can start NX 7 by double-clicking on its shortcut icon on the desktop of your computer. Alternatively, you can choose the **Start** button from the left corner of the taskbar to invoke the menu. From this menu, choose **All Programs** (or **Programs**) > **UGS NX 7.0** > **NX 7.0** to start NX 7, refer to Figure 2-3.

The default NX 7 screen is shown in Figure 2-4. The information about NX 7 is displayed on this screen, which helps you learn more about NX 7. You can also view other information by moving the cursor over the topics displayed on the left of the NX 7 screen.

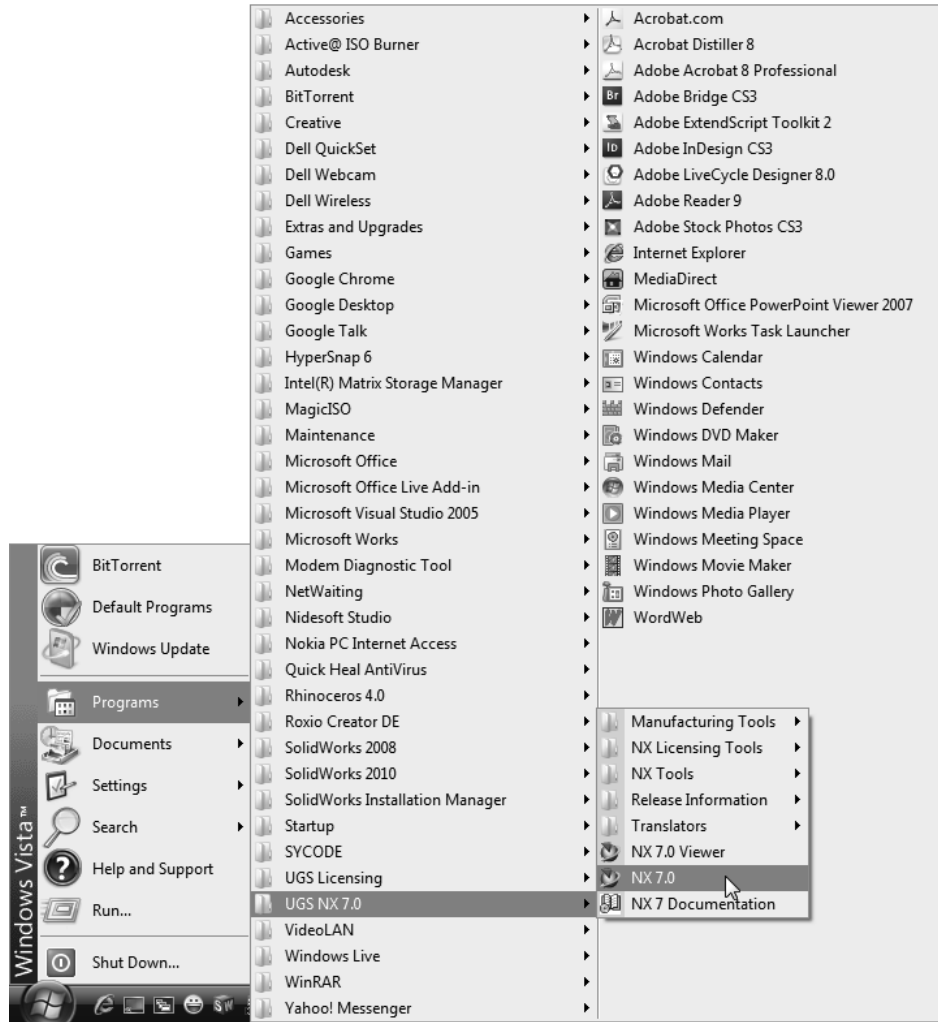


Figure 2-3 Starting NX 7 from the taskbar



Tip: It is advised to read the information on the default initial screen of NX whenever you start a new session. This information will help you learn additional things about NX.

STARTING A NEW DOCUMENT IN NX 7

Menu: File > New
Toolbar: Standard > New



To start a new file, choose the **New** button from the **Standard** toolbar or choose **File > New** from the menu bar; the **New** dialog box will be displayed, as shown in Figure 2-5.

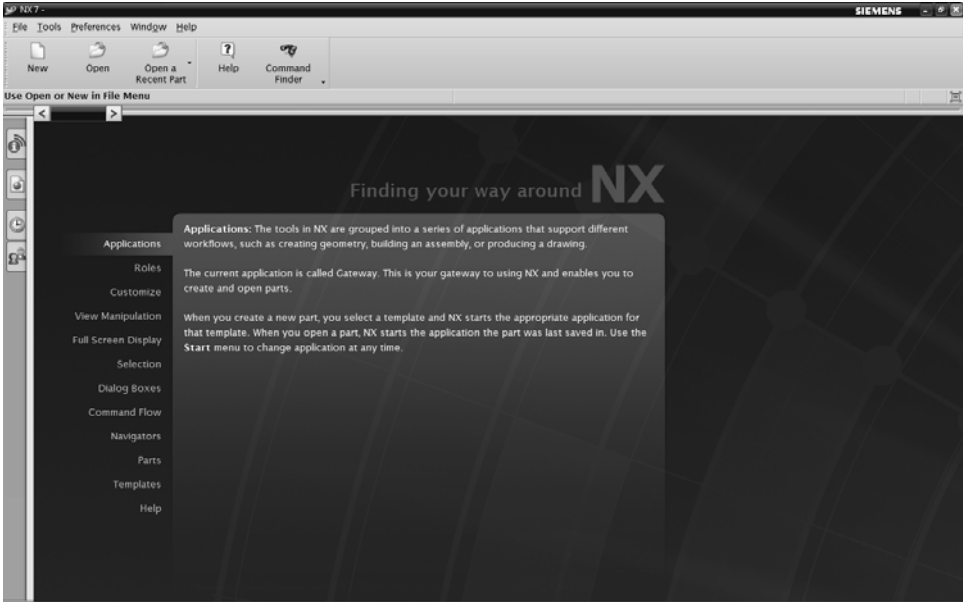


Figure 2-4 Initial default screen of NX 7

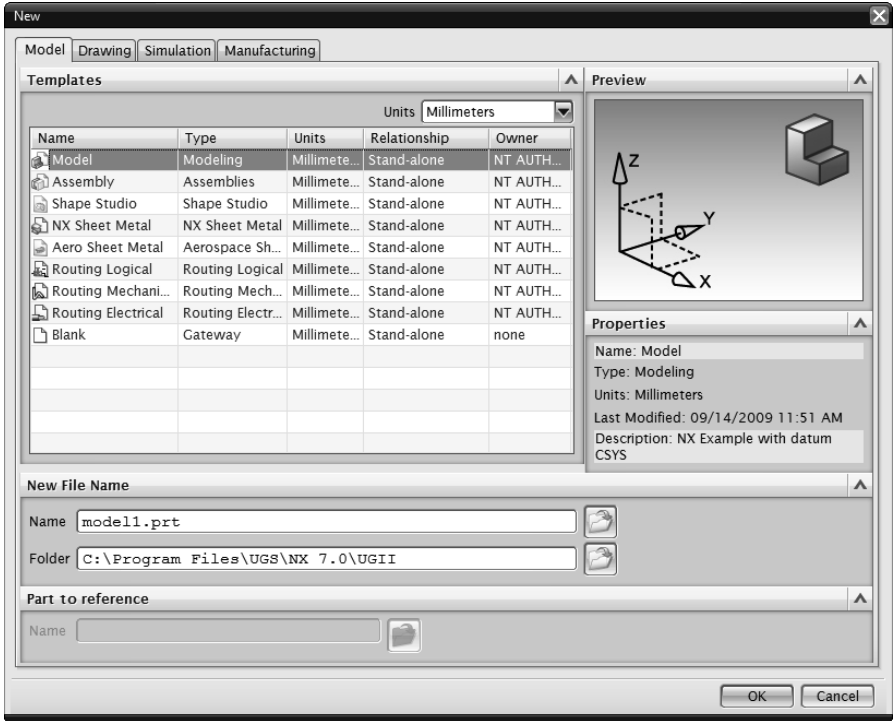


Figure 2-5 The New dialog box

The various tabs and options in this dialog box are discussed next.

Templates Rollout

In the **New** dialog box, templates are grouped together under various environment types such as Modeling, Drawing, Simulation, and Manufacturing. The template files related to these environments are available in their respective tabs. These files are used whenever you start a new file. These template files provide a predefined set of tools with specified environment. This saves a lot of time in setting environment and displaying tools according to your requirements.

Model Tab

By default, this tab will be chosen and the modeling templates are displayed in the **Templates** rollout. Some of the important modeling templates are discussed next.

Model

By default, the **Model** template will be selected. This template is used to start a new part file in the Modeling environment for creating solid and surface models.

Assembly

The **Assembly** template is used to start a new assembly file in the Assembly environment for assembling various parts of the assembly.

Shape Studio

The **Shape Studio** template is used to start a new part file in the Shape Studio environment for creating advanced surface models.

Blank

The **Blank** template is used to start a new file in the Gateway environment. The Gateway environment allows you to examine the geometry and drawing views created. You cannot modify a model in the Gateway environment. However, you can invoke any environment of NX from it.

Drawing Tab

Choose the **Drawing** tab; the drawing templates are displayed in the **Templates** rollout. These templates are used to start a new drawing file in the Drafting environment for generating the drawing views. These templates are arranged according to the sheet size (A0, A1, A2, A3, and A4) in the **Drawing** tab. There are two types of templates for each sheet size, views and no views. If you select the views template, then the drawing views are automatically generated in the drawing sheet. If you select the no views template, then a blank drawing sheet will be opened and you have to create the drawing views manually.

Units

The **Units** drop-down list is used to filter the templates as per the unit. The options in this drop-down list are discussed next.

Millimeters

If you select the **Millimeters** option, the templates only with the millimeters unit will be displayed in the **Templates** area.

Inches

If you select the **Inches** option, the templates only with the inches units will be displayed in the **Templates** area.

All

Select this option to display all the templates (with both millimeters and inches units).

New File Name Rollout

This rollout is used to specify the name and location to save the file. The options in this rollout are discussed next.

Name

Enter the name of the new file in the **Name** text box. Alternatively, choose the button on the right side of the **Name** text box; the **Choose New File Name** dialog box will be displayed. Enter the name in the **File name** edit box. Also, to specify the location to save the new file, browse to the folder where you need to save the file and choose the **OK** button. However, there is a separate option to specify the location, which is discussed next.

Folder

Specify the location to save the new file in the **Folder** text box. Alternatively, choose the button to the right side of the **Folder** text box; the **Choose Directory** dialog box will be displayed. Next, browse to the folder where you want to save the file and choose the **OK** button.



Note

It is recommended that you create a folder with the name NX 7 in the primary drive of your computer and then create individual folder for each chapter within the NX 7 folder. Now, you can save the part files of all chapters in their respective folders. This will ensure a better organization of the part files that you create.

*In this textbook, the **Model** template has been used for starting a new file for illustrations.*

After specifying all required options in the **New** dialog box, choose the **OK** button; the new file will start in the specified environment. Figure 2-6 shows the default initial screen of a new file invoked by using the **Model** template.

INVOKING DIFFERENT NX ENVIRONMENTS

You can invoke any NX environment in the same part file at any time. To invoke different environments, choose the **Start** button from the **Standard** toolbar; a flyout will be displayed, as shown in Figure 2-7. Choose the environment that you want to invoke from this flyout.

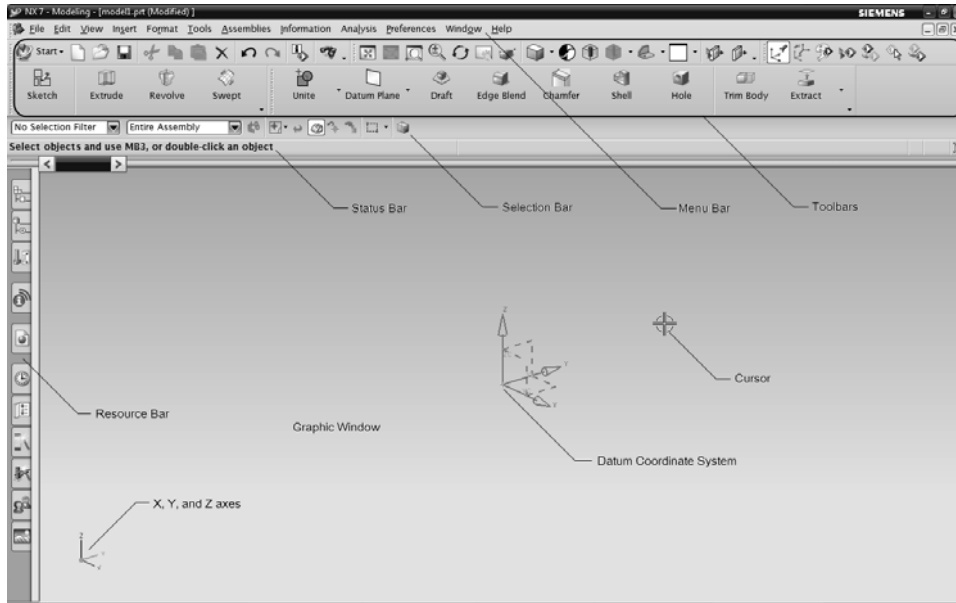


Figure 2-6 Default Initial screen of a new part file

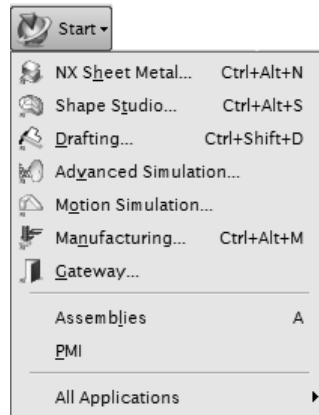


Figure 2-7 Different environments of NX

CREATING THREE FIXED DATUM PLANES (XC-YC, YC-ZC, XC-ZC)

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



You can select the datum coordinate system plane (XC-YC, YC-ZC, or XC-ZC) from the drawing window to create the base feature. You also create three fixed datum planes (YC-ZC, XC-ZC, and XC-YC) first and then use one of them to start the Sketcher environment. To create three fixed datum planes, choose **Insert > Datum/Point > Datum Plane** from the menu bar. Alternatively, choose the **Datum Plane**

button from the **Feature Operation** toolbar; the **Datum Plane** dialog box will be displayed, as shown in Figure 2-8. Next, select the **YC-ZC plane** option from the drop-down list in the **Type** rollout; the preview of the plane will be displayed in the drawing window. Choose the **Apply** button; the YC-ZC plane will be created.

Similarly, select the **XC-ZC plane** and **XC-YC plane** option from the drop-down list in the **Type** rollout to create the XC-ZC and XC-YC planes, respectively. Figure 2-9 shows the three fixed datum planes created.

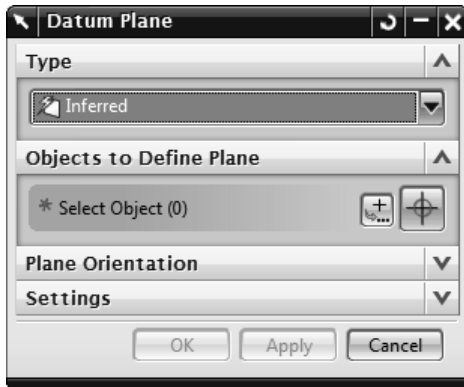


Figure 2-8 The *Datum Plane* dialog box

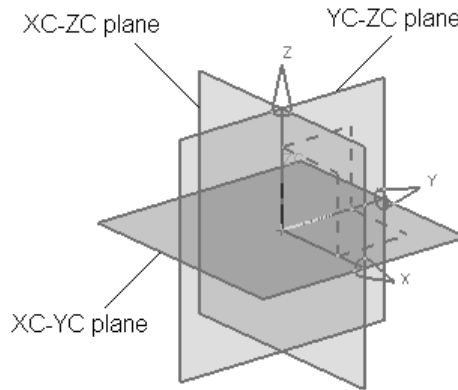


Figure 2-9 Three fixed datum planes

DISPLAYING THE WCS (WORK COORDINATE SYSTEM)

Menu: Format > WCS > Display
Toolbar: Utility > Display WCS



The display of WCS (Work Coordinate System) is important in selecting the planes for drawing sketches. When you start a new file, by default, the display of WCS is turned on. It is recommended to keep the display of WCS turned on while drawing sketches and creating features.

If the display of WCS is turned off, then to turn it on, choose the **Display WCS** button from the **Utility** toolbar; the WCS will be displayed at the origin of the drawing window. Figure 2-10 shows the WCS with the datum coordinate system hidden for better visualization. The **Display WCS** button is the toggle button. Choose this button again to turn off the display of WCS.

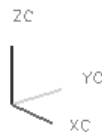


Figure 2-10 The WCS (Work Coordinate System)

INVOKING THE SKETCHER ENVIRONMENT



As mentioned earlier, the base feature or the first feature in a design is always a sketch-based feature. The profiles of the sketch-based features are defined by using a sketch. Therefore, to create the base feature, first you need to invoke the Sketcher environment.

In NX, you can invoke the Sketcher environment by using the datum coordinate system plane (XC-YC, YC-ZC, or XC-ZC), any reference plane, or the existing face of the model.

To invoke the Sketcher environment, choose the **Sketch** button from the **Feature** toolbar. Alternatively, choose **Insert > Sketch** from the menu bar; the **Create Sketch** dialog box will be displayed, as shown in Figure 2-11. Also, the Sketcher environment will be invoked, but no tool will be activated. Enter the name for the sketch in the **Sketch Name** drop-down list of the **Sketcher** toolbar and press the ENTER key. If you do not specify the name, NX will automatically name the sketches as SKETCH_000, SKETCH_001, SKETCH_002, and so on. In this textbook, you will accept the default sketch name given by NX. The **Sketch Name** drop-down list is also used to edit the sketches created earlier; this is discussed later in this textbook.

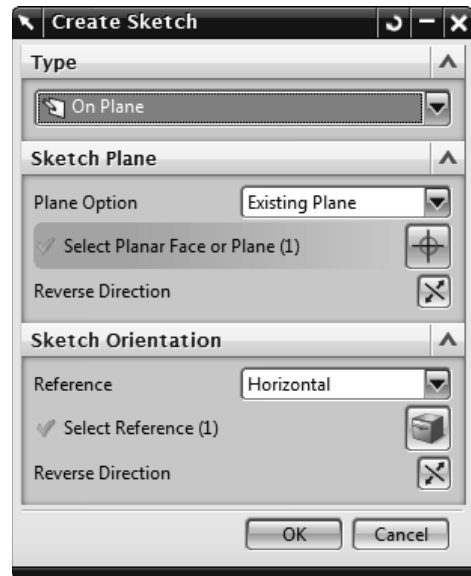


Figure 2-11 The **Create Sketch** dialog box

You will be prompted to select an object for the sketch plane or to select a sketch axis to orient in the prompt area above the drawing window. The options in the various rollouts of the **Create Sketch** dialog box are discussed next.

Type Rollout

The options in this rollout are used to specify whether you want to draw the sketch on existing plane or on a temporary plane defined on the path.

On Plane

By default, this option will be selected from the drop-down list. It is used to specify the existing plane, face, or datum coordinate system plane as a sketching plane.

On Path

Select this option from the drop-down list to specify the sketch plane on the existing path. The temporary sketch plane will be created perpendicular to the path selected.

Depending on the option selected from the drop-down list, the **Create Sketch** dialog box will be modified. The various rollouts in the modified dialog box for both the options are discussed next.

On Plane Options

By default, the rollouts related to the **On Plane** option will be displayed in the **Create Sketch** dialog box, refer to Figure 2-11. The rollouts in this dialog box are discussed next.

Sketch Plane Rollout

The options in this rollout are used to specify the sketch plane by different methods. The options in this rollout are as follows:

Plane Option

This drop down list provides various options to select the sketch plane. Some of these options are discussed below.

Existing Plane

By default, this option will be selected from the **Plane Option** drop-down list and it allows you to select the existing plane or face as the sketch plane.

Create Plane

Select this option to create a new datum plane and use it as the current sketch plane. As you select this option, the **Specify Plane** area is displayed in the **Sketch Plane** rollout. The options in the **Specify Plane** area are used to create a datum plane. The options to create a datum plane are discussed in later chapter.

Create Datum CSYS

Select this option to create a new datum coordinate system.

Reverse Direction



The **Reverse Direction** button in the **Sketch Plane** rollout is used to reverse the direction of the sketching plane.

Sketch Orientation Rollout

The options in this rollout are used to specify the horizontal or vertical reference for the sketch. The sketching plane gets orientated according to the specified references. The options in this rollout are discussed next.

Reference

Select the required option (**Horizontal** or **Vertical**) from the **Reference** drop-down list to specify the reference for the sketch.

Select Reference

This button is used to specify the horizontal or vertical reference by selecting the existing planar face, edge, datum axis, or datum plane. The sketching plane gets oriented according to the specified reference. The horizontal and vertical constraints will be added to the sketch with respect to the specified reference direction.

Reverse Direction

The **Reverse Direction** button in the **Sketch Orientation** rollout is used to reverse the direction of reference specified (horizontal or vertical).

On Path Options

Select this option from the drop-down list in the **Type** rollout to create a sketching plane on a selected path; the rollouts related to the **On Path** option will be displayed in the **Create Sketch** dialog box, as shown in Figure 2-12. The options in these rollouts are discussed next.

Path Rollout

The **Curve** button in this rollout is used to select the path. The path may be a curve or the edge of an existing solid body.

Plane Location Rollout

The options in this rollout are used to specify the location of the sketch plane along the path in terms of arc length or point. These options are discussed next.

Location

This drop-down list contains different options to specify the location of the sketch plane along the path. These options are as follows:

Arc Length

This option allows you to specify the sketch plane distance from the start point of path. Enter the distance in the **Arc Length** edit box.

**Note**

The nearest endpoint of the selected path will be considered as the start point of the path.

% Arc Length

This option allows you to specify the distance of the sketch plane in terms of the percentage of arc length from the start point of path. Enter the % value in the **% Arc Length** edit box.

Through Point

This option allows you to specify the sketch plane by picking a point on the path. You can use **Point Constructor** button or **Inferred Point** drop-down list to create or locate a point.

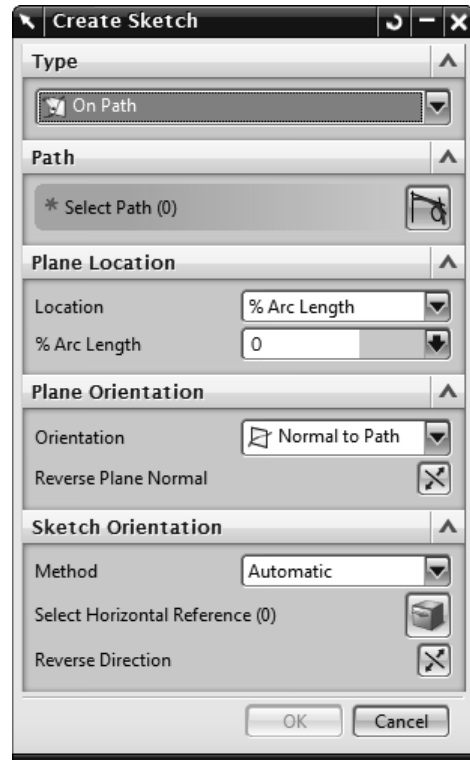


Figure 2-12 The rollouts related to the **On Path** option

Plane Orientation Rollout

The options in this rollout are used to specify the direction of the sketch plane with respect to the selected path. These options are discussed below:

Orientation

This drop-down list contains different options to specify the direction of the sketch plane. These options are discussed next.

Normal to Path

This option allows you to orient the sketch plane normal to the selected path.

Normal to Vector

This option allows you to orient the sketch plane normal to the specified vector. You can use the **Vector Constructor** button or the **Inferred Vector** drop-down list to create or specify the vector.

Parallel to Vector

This option allows you to specify the sketch plane parallel to the specified vector. You can use the **Vector Constructor** button or the **Inferred Vector** drop-down list to create or specify the vector.

Through Axis

This option aligns the sketch plane so that it passes through the specified axis. Specify the axis using the **Vector Constructor** button or the **Inferred Vector** drop-down list.

Reverse Direction



The **Reverse Direction** button is used to reverse the direction of sketch plane normal.

Sketch Orientation Rollout

The options in this rollout are used to specify the reference for a sketch. The sketching plane will be orientated according to the specified reference. The options in this rollout are discussed next.

Method

The options in this drop-down list are used to specify references for the orientation of a sketch. These options are discussed next.

Automatic

The **Automatic** option is selected by default in this drop-down list. This option allows you to select the horizontal reference by using the **Select Horizontal Reference** button. Specify the horizontal reference for the sketch; the sketching plane will be oriented based on the specified reference.



Note

*In the **Automatic** option, if you select an existing curve as a path, the sketch will be oriented*

using the curve parameters and if you select an existing edge as a path, the sketch will be oriented relative to face.

Relative to Face

This option allows you to orient a sketch with respect to a selected face.

Use Curve Parameters

This option allows you to orient a sketch by using curve parameters.

Reverse Direction



The **Reverse Direction** button in this rollout is used to reverse the direction of the specified reference.

All the options in the **Create Sketch** dialog box have already been discussed. Now, for illustration purpose, select the **On Plane** option from the drop-down list. By default, the XC-YC plane will be selected. Next, choose the **OK** button from the **Create Sketch** dialog box; the plane will be oriented parallel to the screen and the Sketcher environment will be invoked. By default, the **Profile** tool will be invoked. Figure 2-13 shows the default screen display of the Sketcher environment of NX.

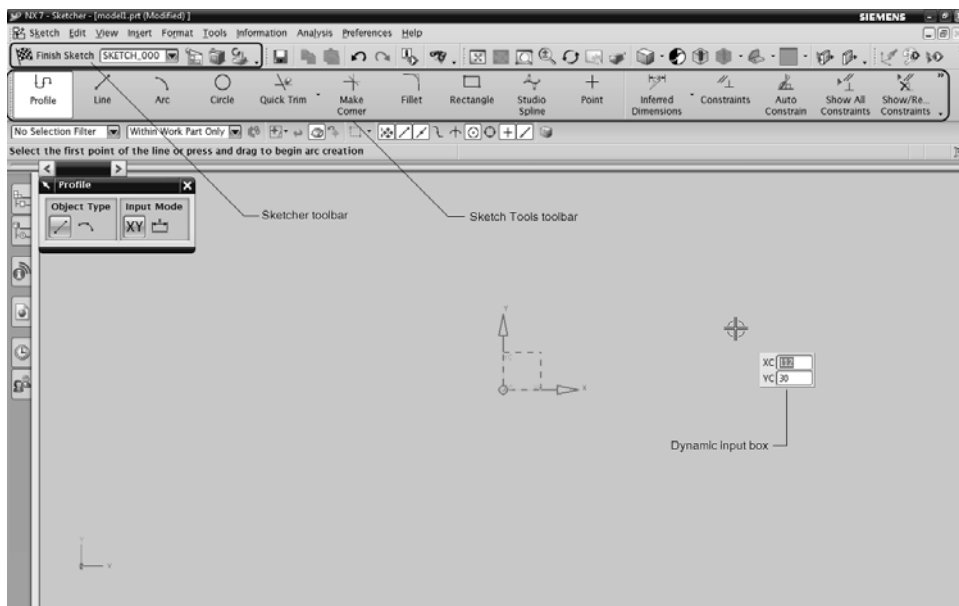


Figure 2-13 The default screen display of the Sketcher environment of NX



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.

SKETCHING TOOLS

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

By default, some of the tools are not available in the toolbars. However, you can customize the toolbars to add these tools. You will learn more about customizing the toolbars later in this chapter.

Drawing Sketches Using the Profile Tool

Menu: Insert > Curve > Profile
Toolbar: Sketch Tools > Profile



By default, the **Profile** tool is invoked in the **Sketch Tools** toolbar, when you invoke the Sketcher environment. 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** dialog box will be displayed with four buttons on the top left corner of the drawing window, refer to Figure 2-14.



Figure 2-14 The **Profile** dialog box

Also, the dynamic input boxes are displayed below the cursor and you are prompted to select the first point of the line or press and drag the left mouse button 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** dialog box. 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 available below the cursor when you invoke the **Profile** tool. After you have entered the coordinates of the start point of the line, a rubber-band line will be 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. On specifying the start point of the line, the dynamic input boxes will change into the length and angle modes, as shown in Figure 2-15.

This happens because the **Parameter Mode** button is automatically chosen in the **Profile** dialog box. As you move the cursor in the drawing window, the length and angle of the line is modified, based on the relative position of the cursor with respect to the point specified earlier in the dynamic input boxes. You can draw a line by specifying its length and angle in these boxes.

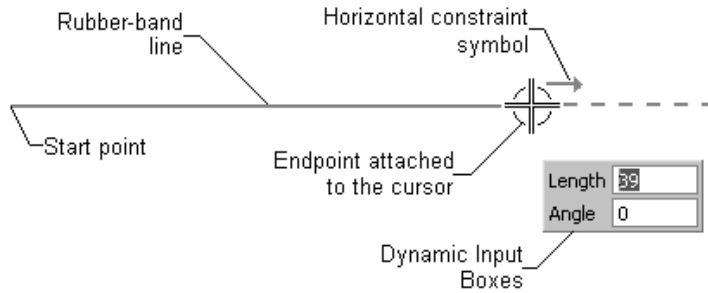


Figure 2-15 Drawing a horizontal line



Note

After specifying the start point of the line, if you choose the **Coordinate Mode** button from the **Profile** dialog box, the coordinate mode option for specifying the endpoint of the line will be restored.

The line drawing process does not end after you specify the second point of the line. Instead, another rubber-band line starts with its 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 a chain of continuous lines.



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 second dynamic input box will be activated. Specify the value in the second box and then press the ENTER key or the TAB key to register the values and draw the line using these values.

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 will appear 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 a chain of continuous lines.

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 2-15. This arrow is the symbol 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 the later chapters.



Note

While drawing lines, you can disable the constraints temporarily by pressing the ALT key.

Drawing Arcs



The option to draw arcs can be activated by choosing the **Arc** button in the **Profile** dialog box. Alternatively, you can press and hold the left mouse button and drag the cursor to invoke the arc mode. Generally, the arcs that are drawn by using this tool are in continuation with 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 need to specify only the endpoint of the arc.

When you draw an arc in continuation with lines, you will notice that a circle with four quadrants will be displayed at the start point of the arc, as shown in Figure 2-16. This symbol is called the quadrant symbol and it helps you in defining whether you need to draw a tangent arc or a normal arc. This symbol also helps you in specifying the direction of the arc.

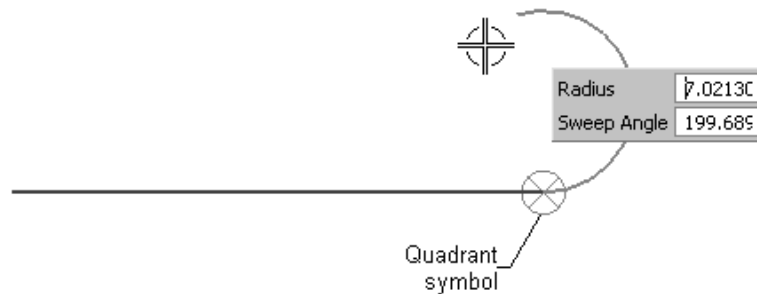


Figure 2-16 Quadrant symbol displayed while drawing an arc using the **Profile** tool

As evident from Figure 2-16, 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. 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 2-16.

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 2-17. As you invoke the arc mode, the current dynamic input boxes change into the **Radius** and **Sweep Angle** input boxes. These boxes allow you to specify the radius and the sweep angle to draw the arc.

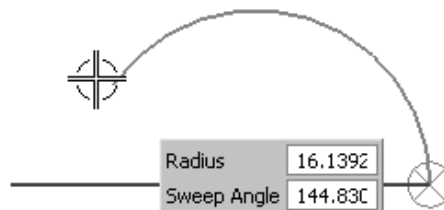


Figure 2-17 Drawing the normal arc



Tip: To restart drawing lines using the **Profile** tool or to break the sequence of continuous lines, press the **ESC** key once.

Press the **ESC** key twice to exit the tool. Alternatively, right-click on the drawing area and choose the **OK** option from the shortcut menu.



Note

If you are not drawing the arc in continuation with a line or an arc, 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 endpoints, midpoints, center points, and so on. These dotted lines are called the 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 2-18 shows the use of the help lines to locate the start point of a new line. You can temporarily disable the help lines by pressing the **ALT** key.

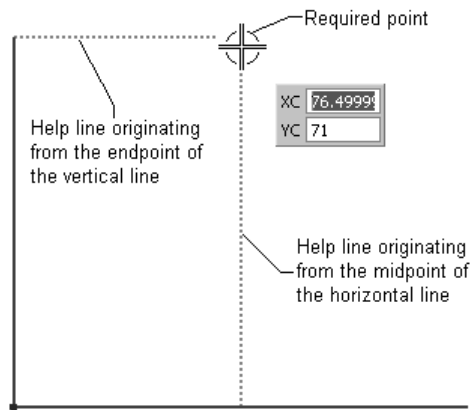


Figure 2-18 Using the help lines to locate a point

Drawing Individual Lines

Menu: Insert > Curve > Line
Toolbar: Sketch Tools > 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 the 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 first point of the line. You can specify the first point and the second point 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 > Curve > Arc
Toolbar: Sketch Tools > Arc



NX allows you to draw arcs using two methods. These methods can be activated by choosing their respective buttons from the **Arc** dialog box that will be displayed when you invoke the **Arc** tool. 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 point, 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 start point by clicking in the drawing window or by entering the coordinates in the dynamic input boxes. After specifying the start point of the arc, you will be prompted to specify the endpoint. 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 endpoint. 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 start point, 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 2-19 shows a three-point arc being drawn by specifying two endpoints and a point on the arc.

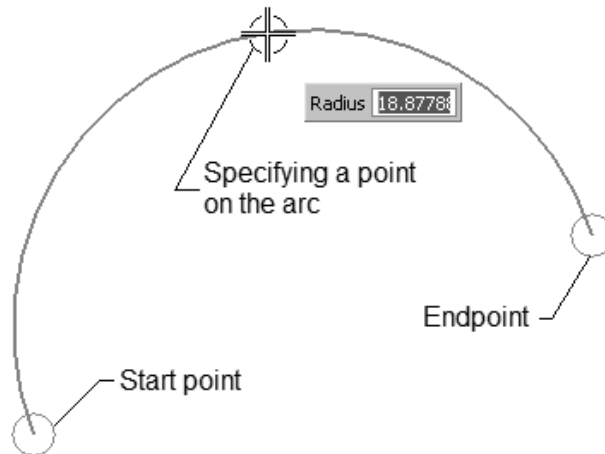


Figure 2-19 Drawing a three-point arc



Tip: While drawing an arc by specifying its three points, if the start point 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 an Arc by Specifying its Center Point and Endpoints



This method is used to draw an arc by specifying its center point, start point, and endpoint. To invoke this method, choose the **Arc by Center and Endpoints** button from the **Arc** dialog box; you will be prompted to specify the center point of the arc. Specify the center point of the arc by clicking in the drawing area or by entering coordinates in the dynamic input boxes. On doing so, you will be prompted to specify the start point of the arc. After specifying the start point of the arc, you will be prompted to specify the endpoint of the arc. Note that when you specify the start point of the arc after specifying the

center point, the radius of the arc will automatically be defined. Therefore, the endpoint is used only to define the arc length. Figure 2-20 shows an arc being drawn using this method.

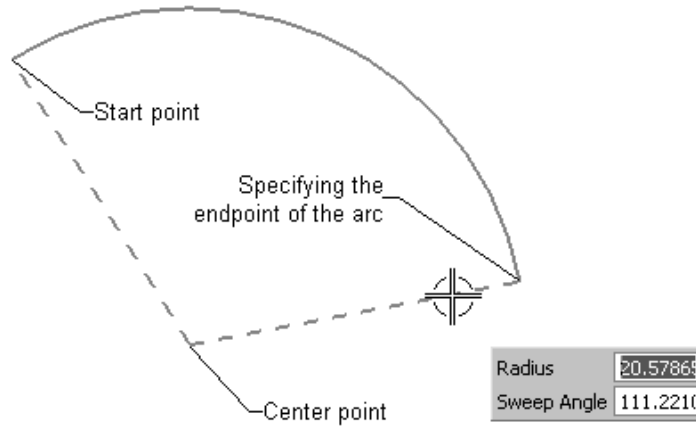


Figure 2-20 Drawing an arc by specifying its center, start, and end points



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 be prompted to specify the start point and then the endpoint of the arc. The endpoint will define the direction of the arc.

Drawing Circles

Menu: Insert > Curve > Circle
Toolbar: Sketch Tools > Circle



In NX, you can draw circles using two methods. These methods can be activated by choosing their respective buttons from the **Circle** dialog box that are displayed when you invoke the **Circle** tool. 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 the circumference of the circle. The point on the circumference of 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** dialog box; 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. Specify a point to define the radius. Alternatively, you can enter the value of the diameter in the dynamic input box. Figure 2-21 shows a circle being drawn by using this method.



Tip: After specifying the center point of the circle, if you specify the value of diameter in the dynamic input box, the circle of the specified diameter will be created. Also, the preview of the circle of the same diameter will be attached to the cursor. Now, you can place multiple copies of the circle by specifying the center point.

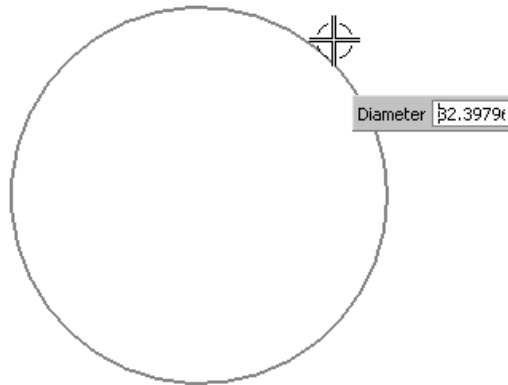


Figure 2-21 Drawing a circle using the Circle by Center and Diameter method

Drawing a Circle by Specifying Three Points



This method is used to draw a circle by specifying three points on circumference.

To invoke this method, choose the **Circle by 3 Points** button from the **Circle** dialog box; you will be prompted to specify the first point of the circle. This point is actually the first point on the circumference of the circle. After specifying the first point, 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 2-22. Now, specify the third point, which is a point on the circle. You can also enter its diameter value in the **Diameter** input box. This completes the creation of the circle.

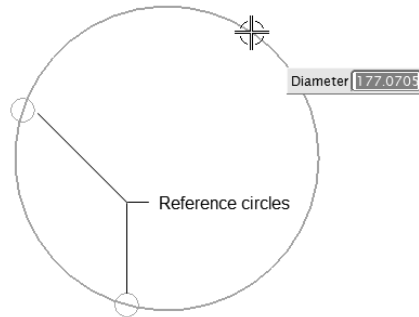
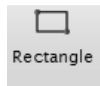


Figure 2-22 Drawing a circle using the Circle by 3 Points method

Drawing Rectangles

Menu: Insert > Curve > Rectangle
Toolbar: Sketch Tools > Rectangle



In NX, you can draw rectangles using three methods. These methods can be used by choosing their respective buttons from the **Rectangle** dialog box. To invoke this dialog box, choose the **Rectangle** button from the **Sketch Tools** toolbar.

Alternatively, choose **Insert > Curve > Rectangle** from the menu bar to display the **Rectangle** dialog box. The three methods of drawing rectangles are discussed next.

Drawing Rectangles by Specifying Corners



The **By 2 Points** method is used to draw a rectangle by specifying the diagonally opposite corners of rectangle. When you invoke the **Rectangle** tool, the **By 2 Points** button will be chosen by default in the **Rectangle Method** area of the **Rectangle** dialog

box. Also, you will be prompted to specify the first point of the rectangle. This point will work as one of the corners of the rectangle. After specifying the first point, you will be prompted to specify the point to create the rectangle. This point will be diagonally opposite to the point that you have specified earlier. 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 2-23 shows a rectangle being drawn by using the By 2 Points method.

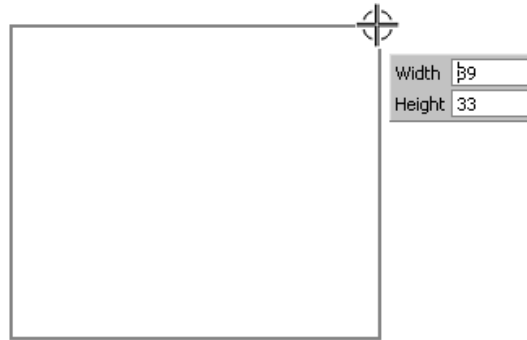


Figure 2-23 Rectangle being drawn by using the By 2 Points method



Tip: If you specify the width and height of a rectangle in the dynamic input boxes after specifying the first point, the preview of the rectangle with the specified width and height will be attached to the cursor. Now, you need to specify a point to define the direction of rectangle.

Drawing Three Points Rectangles



You can draw a three points rectangle by choosing the **By 3 Points** button from the **Rectangle** dialog box. 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 height of the rectangle. When you invoke this method, you will be prompted to specify the first point of the rectangle. Once you specify the first point, you will be prompted to specify the second point of the rectangle. Both these corners are along the same direction. Therefore, these points define the length and orientation 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 second point, you will be prompted to specify a point to create the rectangle. This point is used to define the height of the rectangle. After specifying the first point, you can also specify the height, width, and the angle of the rectangle in the dynamic input boxes. Figure 2-24 shows an inclined rectangle drawn by using the By 3 Points method.

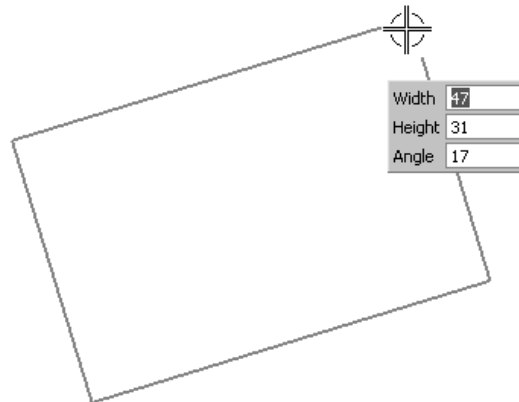


Figure 2-24 Inclined rectangle drawn by using the By 3 Points method



Tip: After specifying the first point of a rectangle, you can toggle between the **By 2 Points** and **By 3 Points** buttons by dragging the left mouse button.

Drawing Centerpoint Rectangles



You can draw a centerpoint rectangle by choosing the **From Center** button in the **Rectangle** dialog box. 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 will be prompted to specify the center point of the rectangle. Once you specify the center point, you will be prompted to specify the second point of the rectangle. Both these points are along the same direction. Therefore, these points 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 second point, you will be prompted to specify a point to create the rectangle. This point is used to define the height of the rectangle.

Placing Points

Menu: Insert > Datum/Point > Point
Toolbar: Sketch Tools > Point



In NX, points are placed by using the **Point** dialog box. To invoke this dialog box, choose the **Point** button from the **Sketch Tools** toolbar; the **Point** dialog box will be displayed, refer to Figure 2-25 and you will be prompted to select the object to infer point. This dialog box contains four main rollouts, **Type**, **Point Location**, **Coordinates**, and **Offset**. The options in these rollouts are discussed next.

Type Rollout

This rollout has a drop-down list from which you can select a method to specify the location for the resulting point. Click on the drop-down list; the options for placing a point will be displayed. These options are discussed next.

Inferred Point

This option is selected by default. This option allows you to place a point in the drawing window. However, if there are some entities in the drawing window, then this option helps you to 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 option allows you to place a point at a location where you will click the cursor in the

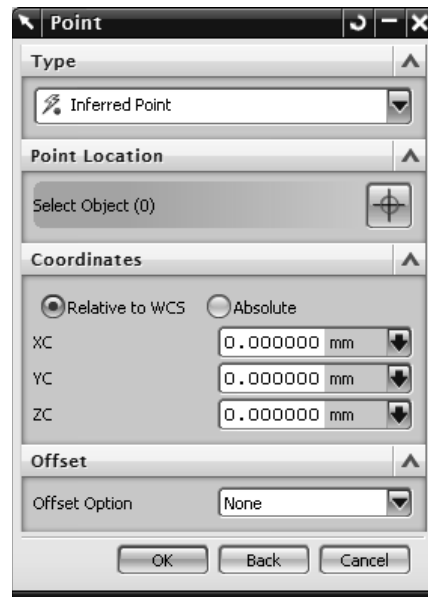


Figure 2-25 The **Point** dialog box

drawing window. If the **Cursor Location** option is selected, then the other entities in the drawing window will not be considered.

Existing Point

This option allows you to select the points that are already placed in the drawing window. As a result, you can place new point on top of the existing point.

End Point

This option allows you to place the point at the endpoint of the existing lines, arcs, or splines.

Control Point

This option 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

This option allows you to place the point at the intersection point of the two existing sketched entities. To do so, select the **Intersection Point** option from the drop-down list in the **Type** rollout; you will be prompted to select the first and second intersecting entities. Specify the two intersecting entities in the drawing area; a point will be placed at the intersection point of the two existing entities.

Arc/Ellipse/Sphere Center

This option allows you to place the point at the center of an existing arc, circle, ellipse, or sphere.

Angle on Arc/Ellipse

This option allows you to place the point on the circumference of the selected arc, circle, or ellipse such that the resulting point is at the specified angle with respect to X-axis. When you choose this option, the **Point** dialog box will be modified and you will be prompted to select an arc or ellipse. Select the arc or ellipse in the drawing; the point will be placed on the circumference of the selected entity. Next, enter the angle value for the point in the **Angle** edit box of the **Angle on Curve** rollout.

Quadrant Point

This option allows you to place the point at the quadrant of a circle, arc, or an ellipse. The point will be placed at the quadrant that is closest to the current location of the cursor.

Point on Curve/Edge

This option allows you to place the point on the selected curve or edge. The location of the point is defined in terms of its curve parameter percentage from the start point of the curve. When you select the **Point on Curve/Edge** option, the **Point** dialog box will be modified and you will be prompted to select the curve to specify the point location. Click anywhere on the curve or the edge; you will be prompted to specify the curve parameter percentage. Next, enter the distance of the point, in terms of curve parameter percentage (0 to 1), from the start point of the curve in the **U Parameter** edit box of the **Location on Curve** rollout.

Point on Face

This option allows you to place the point on the selected face. The location of the point is defined by specifying values in the **U Parameter** and **V Parameter** edit boxes. The **U Parameter** edit box is used to specify the horizontal position of the point, where as the **V Parameter** is used to specify the vertical position of the point. The values of these edit boxes must be between 0.0 and 1.0. As value of the **U Parameter** edit box increases, the position of the point shifts from right to left; and if the value of the **V Parameter** edit box increases, the position of the point shifts from bottom to top.

Between Two Points

This option allows you to create a point between two existing points or between two keypoints of an entity. When you select this option from the **Type** drop-down list, the **Point** dialog box will be modified and you will be prompted to select object to infer point. Select the first point from the drawing window; you will be prompted again to select object to infer point. Select the second point; a point will be created between the two selected points. You can change the location of this point by entering the percentage value in the **%Location** edit box of the **Location Between Points** rollout.

By Expression

This option allows you to specify a point expression by using the X, Y, and Z coordinates. When you select this option from the **Type** drop-down list, the **Point** dialog box will be modified with new rollouts such as **Choose Expression**, **Coordinates**, and **Offset**. The **Choose Expression** rollout is used to display the point expression created already in the part. To create a new expression, choose the **Create Expression** button; the **Expressions** dialog box will be displayed. Enter the name of the point expression in the **Name** edit box, and then edit the point formula as per your requirement in the **Formula** edit box. Once you have edited the values of the X, Y, and Z coordinates in the **Formula** edit box, the **Accept Edit** button will be available. Choose the **Accept Edit** button and then choose the **OK** button from this dialog box; the **Point** dialog box will be displayed. The newly created point expression will be listed in the **Expression** list area of the **Choose Expression** rollout. Select the point expression from the list and then choose the **OK** button from the **Point** dialog box; a point will be created with the specified coordinates in the expression.

Point Location Rollout

This rollout is used to select a point and will not be available for the **Intersection Point**, **Angle on Arc/Ellipse**, **Point on Curve/Edge**, and **Point on Face** options.

Coordinates Rollout

This rollout is used to enter the X, Y, and Z coordinates to specify the location of the point. Also, you can specify or determine the 3D location of the points using this rollout. You can specify the point relative to the Work Coordinate System (WCS) or Absolute Coordinate System by selecting their respective radio buttons.

Offset Rollout

This rollout is used to create a point at a specified distance from a pre-selected point. You can select an option to specify the distance of the required point from the **Offset Option** drop-down list in this rollout. The options in this drop-down list are discussed next.

Rectangular

This option allows you to create a point by specifying its X, Y, Z coordinates with respect to the pre-selected point in the **Delta XC**, **Delta YC**, and **Delta ZC** edit boxes, respectively.

Cylindrical

This option allows you to create a point according to the cylindrical coordinate system with respect to the pre-selected point by specifying the radius, angle, and Z direction in the **Radius**, **Angle**, and **Delta Z** edit boxes, respectively.

Spherical

This option allows you to create a point according to the spherical coordinate system with respect to the pre-selected point by specifying the **Radius**, **Angle 1**, and **Angle 2** in their respective edit boxes.

Along Vector

This option allows you to create a point along the specified vector direction at a distance specified in the **Distance** edit box.

Along Curve

This option allows you to create a point on the specified curve. The distance of the point on arc can be specified by entering the **Arc Length** or **Percentage** value in the respective edit box.



Tip: By default, some of the tools are not available in the toolbars. However, you can customize the toolbars to add these tools. To customize the toolbars, choose the black arrow at the bottom of the vertical toolbar or on the right of the horizontal toolbar. When you choose this arrow; a cascading menu will be displayed. Select **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 in which you need to add more tools; another cascading menu will be displayed with all the tools that can be added to the selected toolbar. Note that all the tools that are currently available in the toolbar will have a check mark on their left. Select the name of the tools which you want to add to the toolbar; the selected tools will be added at the end of the toolbar.

Drawing Ellipses or Elliptical Arcs

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



In NX, you can draw ellipses or elliptical arcs by using the **Ellipse** tool. To invoke this tool, choose **Insert > Curve > Ellipse** option from the menu bar; the **Ellipse** dialog box will be displayed, as shown in Figure 2-26. Also, you will be prompted to select the object to infer a point. Select the **Point Constructor** button from the **Center** rollout; the **Point** dialog box will be displayed, refer to Figure 2-25. Using the **Point** dialog box, you can define the center point of ellipse. Alternatively, you can define the center point of the ellipse by selecting the required option from the drop-down list available on the right of the **Point Constructor** button. After defining the center point by using the **Point** dialog

box, choose the **OK** button from it; the **Ellipse** dialog box will be displayed again. Also, the preview of the ellipse will be displayed. Next, specify the major and the minor radii of the ellipse in the **Major Radius** and **Minor Radius** edit boxes in the **Ellipse** dialog box, respectively. If you want to draw an elliptical arc, clear the **Closed** check box in the **Limits** rollout; the **Ellipse** dialog box will be modified and the **Start Angle** and **End Angle** edit boxes for the arc will appear in it. You can specify the start and end angles in their respective edit boxes. Figure 2-27 shows the parameters related to an ellipse and Figure 2-28 shows the parameters related to an elliptical arc. If you want to retain the complement of the elliptical arc, choose the **Complement** button below the **End Angle** edit box in the **Limits** rollout; the preview of the complement of the elliptical arc will be displayed. Figure 2-29 shows an elliptical arc and Figure 2-30 shows the complement of the elliptical arc. Note that Figure 2-27 shows an inclined ellipse. To create an inclined ellipse, you need to enter rotation angle in the **Angle** edit box of the **Rotation** rollout. The specified angle value will be measured with respect to X-axis in counterclockwise direction.

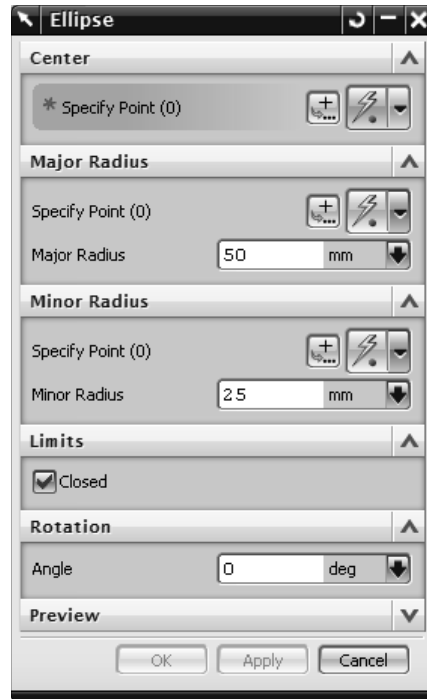


Figure 2-26 The *Ellipse* dialog box

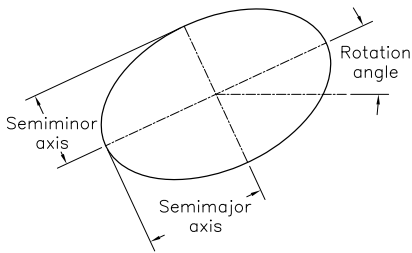


Figure 2-27 Parameters related to an ellipse

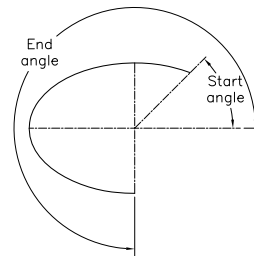


Figure 2-28 Parameters related to an elliptical arc

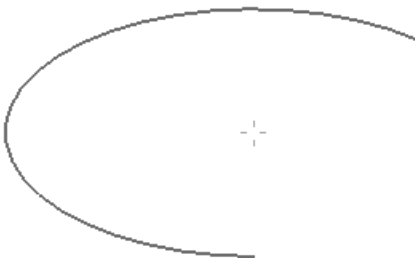


Figure 2-29 An elliptical arc



Figure 2-30 Complement of the elliptical arc shown in Figure 2-29



Tip: Sometimes while placing points or drawing an ellipse, some red cross marks are displayed on the screen. To remove them, refresh the screen by pressing the F5 key.

Drawing Conics

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



The **Conic** tool allows you to create a conic section in the Sketcher environment using three points. The first two points define the endpoints of the conic and the third point defines the apex of the conic. Also, you need to specify the projective discriminant value, termed as rho value. To invoke the **Conic** tool, choose **Insert > Curve > Conic** from the menu bar; the **Conic** dialog box will be displayed, as shown in Figure 2-31. In this dialog box, you can specify the start point and endpoint of the conic using the options in the **Limits** rollout. After specifying the start point and the endpoint of the conic, you need to specify the apex of the conic as the third point. Specify the apex of the conic by using the options in the **Specify Control Point** area of the **Control Point** rollout. Next, enter the Rho value in the **Value** edit box. This rho value will define the exact shape of conics.

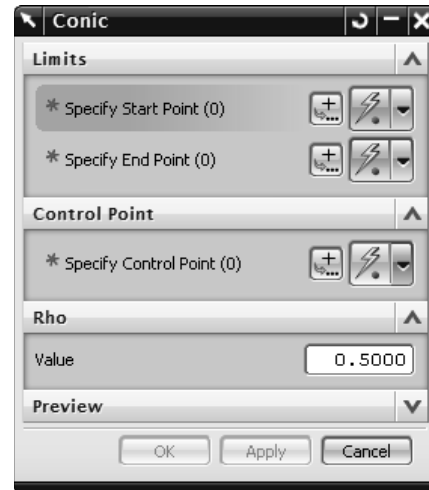


Figure 2-31 The Conic dialog box

If $0 < \text{Rho} < 0.5$, then conics of elliptical shape will be created.
 If $\text{Rho} = 0.5$, then conics of parabolic shape will be created.
 If $0.5 < \text{Rho} < 1$, then conics of hyperbolic shape will be created.

Figure 2-32 shows conics with different rho values.

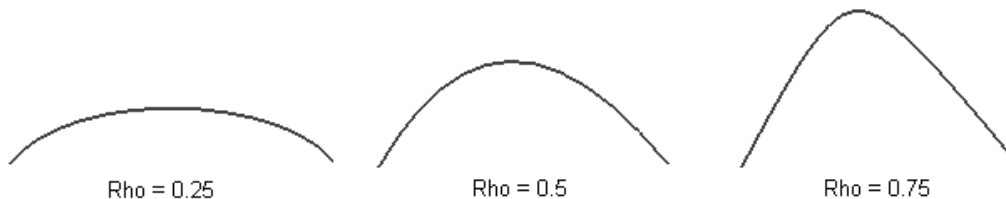


Figure 2-32 Conics with different rho values

Drawing Studio Splines

Menu: Insert > Curve > Studio Spline
Toolbar: Sketch Tools > 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 2-33. The various rollouts in this dialog box are discussed next.

Spline Setting Rollout

This rollout contains different methods and options to create the studio splines.

Method Area

There are two methods of drawing studio splines. 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 2-34 shows a spline being drawn by 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 2-35 shows a spline being drawn by using this method. Remember that the display of poles is automatically removed when you finish drawing the spline.



Figure 2-34 Drawing a spline by using the *Through Points* method

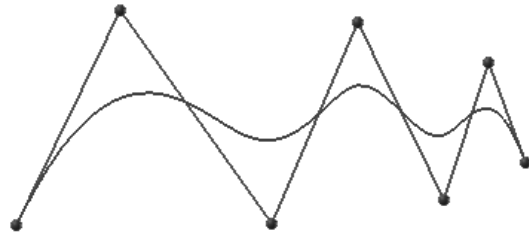


Figure 2-35 Drawing a spline by using the *By Poles* method



Figure 2-33 The *Studio Spline* dialog box

Degree Spinner

The **Degree** spinner is used to specify the degree of a spline. Figures 2-36 and 2-37 show splines of various degrees. Note that the degree of a spline cannot be more than the number of poles used to draw it.

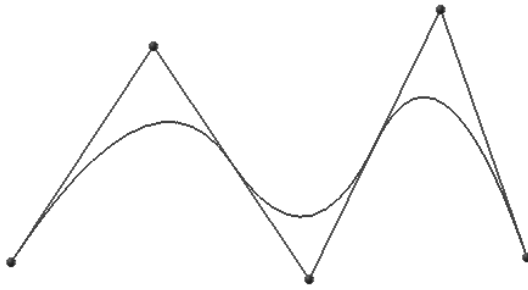


Figure 2-36 Spline of degree 2

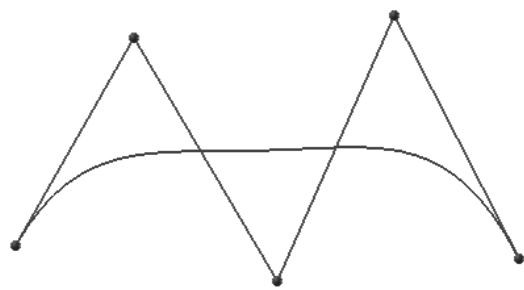


Figure 2-37 Spline of degree 4

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 numbers of poles as you require. If you select this check box, the **Degree** spinner will not be available.

Matched 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. If you select this check box, the **Closed** check box will not be available.

Closed

This check box is available for both the methods and is used to create closed splines. Figure 2-38 shows a closed spline.

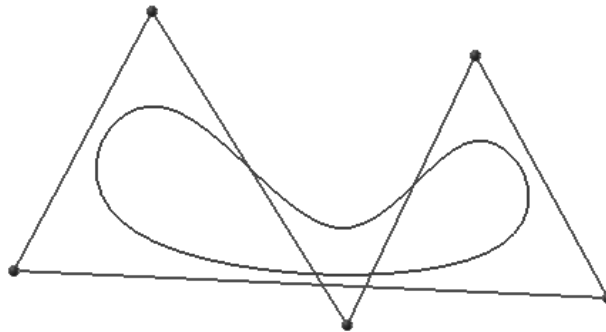


Figure 2-38 A closed spline

Associative

The **Associative** check box is used to make the spline associative to its parent feature. This means that if you modify the parent feature, the spline will also be modified. By default, this check box is cleared, which means there is no associativity of the spline with its parent feature.

Filleting Sketched Entities

Menu: Insert > Curve > Fillet
Toolbar: Sketch Tools > Fillet



Filleting is defined as the process of rounding the sharp corners of a profile to reduce the stress concentration. Fillets are created by removing the sharp corners and replacing them by round corners. In NX, you can create a fillet between any two sketched entities. You can also create a fillet using three sketched entities.

To create fillets, invoke the **Fillet** tool; the **Create Fillet** dialog box will be displayed, as shown in Figure 2-39. Also, you will be prompted to select or drag the cursor over curves to create a fillet.

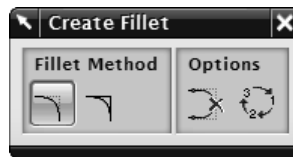


Figure 2-39 The Create Fillet dialog box

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 2-40 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. The **Create Fillet** dialog box is divided into two areas, **Fillet Method** and **Options**.

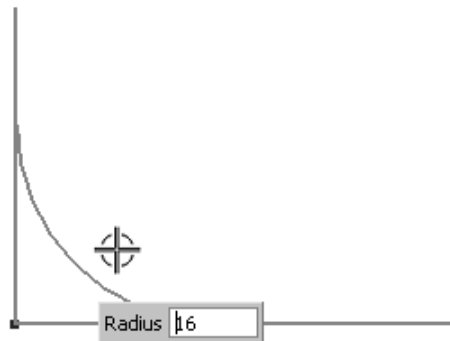


Figure 2-40 Preview of a fillet being created between two lines

Fillet Method

The first button in this area is the **Trim** button and is chosen by default. As a result, the sharp corner will automatically be trimmed after filleting, as shown in Figure 2-41. If you choose the **Untrim** button, the sharp corner will not be trimmed after filleting, as shown in Figure 2-42.



Figure 2-41 Sharp corner before and after filleting using the **Trim** button

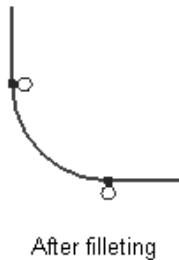
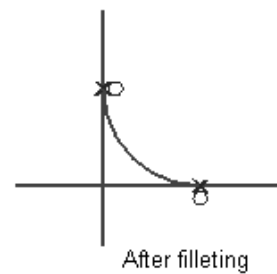


Figure 2-42 Sharp corner before and after filleting using the **Untrim** button



Tip: Ideally, the profiles created with the fillet may not give the desired result when used to create features. Therefore, they should be avoided in the **Sketcher** environment.

Options Area

The **Delete Third Curve** button in this area is useful if you are creating a fillet by using three entities. While using this option, the middle entity should be selected last. This button ensures that if the fillet is tangent to the middle entity then the middle entity is automatically deleted, as shown in Figure 2-43. If this button is deactivated, the middle entity will not be deleted, as shown Figure 2-44. The **Create Alternate Fillet** button in this area will show all alternative solutions for the fillet. It is recommended that this button should be turned off.

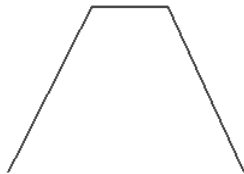


Figure 2-43 Before and after filleting with the third curve deleted

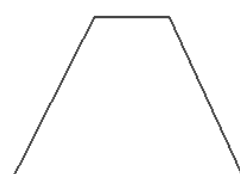
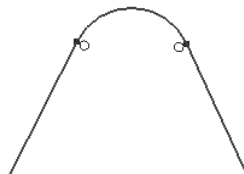
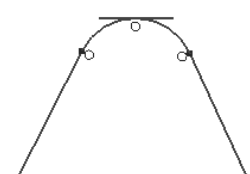


Figure 2-44 Before and after filleting with the third curve retained



Tip: In NX, you can create fillets by simply dragging the cursor across the entities that you need to fillet. For example, if you need 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 you dragged the mouse from the corner.

When you fillet two entities, and there are more than one solution for fillet. The best solution is displayed by default. If you want to view the alternate solution, press the **PAGE UP** key.

THE DRAWING DISPLAY TOOLS

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



Note

As most of the drawing display tools are transparent tools, you can use these tools at any time without exiting the other tool you are working with.

Fitting 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 also use the CTRL+F keys 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 in to a particular area by defining a box around it. When you choose this tool, 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 to 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 a '+' and a '-' sign at the center of the cursor. To zoom in, press and hold the left mouse button in the drawing window and then drag the cursor down. Similarly, to zoom out, press and hold the left mouse button and drag the cursor upward.



Tip: NX allows you to restore the view, 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 cursor is replaced by a hand cursor and you are prompted to drag the cursor to pan the view. Press and hold the left mouse button in the drawing window and then drag the mouse to pan the drawing.



Tip: In NX, you can also display the *Selection MiniBar* and the *View shortcut menu* by right-clicking in the drawing area. The *Selection MiniBar* is the compact version of the **Selection bar**.

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 is available only when an entity is selected in the drawing window.

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 environment. This tool is available only in the Sketcher environment.

SETTING SELECTION FILTERS IN THE SKETCHER ENVIRONMENT

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

Type Filter

The **Type Filter** drop-down list is used to specify the type of entity to be selected. By default, the **No Selection Filter** option is selected, refer to Figure 2-45. This option allows you to select any entity from the drawing window. These entities include curves, points, dimensions, symbols, sketch constraints, and so on. Select the required entity from the **Type Filter** drop-down list. Now, you can select only the specified entity from the drawing window.

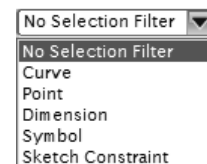


Figure 2-45 The **Type Filter** drop-down list

Selection Scope

This drop-down list allows you to filter the selection from the entire assembly, workpart only, or from the active sketch only. Select the required option from the **Selection Scope** drop-down list.

General Selection Filters



This tool is used to provide the detailed filter options. When you choose this tool, the **General Selection Filters** flyout will be displayed, as shown in Figure 2-46.

The options in this flyout are discussed next.

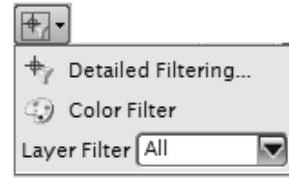


Figure 2-46 The General Selection Filters flyout

Detailed Filtering

This option is used to filter the selection using layers, types of entity, display attributes, and detailed types of entity. Select the **Detailed Filtering** option in the **General Selection Filter** flyout; the **Detailed Filtering** dialog box will be displayed. In this dialog box, you can specify layers, types of entity, detailed types of entity, and display attributes to be filtered.

Color Filter

This option allows you to filter the selection using a specific color. Only the entities in the specified color will be selected.

Layer Filter

This drop-down list allows you to filter the selection using a specific layer. You need to select the layer from the **Layer Filter** drop-down list and the entities in this layer can only be selected. By default, the **All** option is selected, which allows to select the entities from all the layers.

Reset Filters



This tool is used to reset all filtering options defined in the **General Selection Filters** flyout and the **Type Filter** drop-down list to their default states.

Allow Selection of Hidden Wireframe



This tool allows you to select the hidden wireframe geometries such as curves and edges.

Deselect All



This tool, if chosen, deselects all currently selected entities.

Find in Navigator



This tool is used to highlight the selected entities in the **Part** or **Assembly Navigator** and will be activated only when you select an entity. Select the entities that you want to highlight in the **Part** or **Assembly Navigator** and choose the **Find in Navigator** tool in the **Selection Bar**. Next, choose the **Part Navigator** tab from the **Resource Bar** to view the highlighted entities.

SELECTING OBJECTS

After setting the selection filters, you can select objects in the Sketcher environment of NX. When there is no tool active in the Sketcher environment, the select mode will be invoked. In this mode, you can select individual sketched entities from the drawing window by clicking on them. To select multiple entities, you can use the following two methods:

Rectangle



If you choose this tool from the **Selection Bar** and drag the cursor in the drawing window, temporary rectangle will be created according to the movement of the cursor. Also, all entities lying completely within the temporary rectangle will get selected.

Lasso



If you choose this tool from the **Selection Bar** and drag the cursor in the drawing window, a temporary free form curve will be created according to the movement of the cursor. Also, all entities lying completely within the free form curve will get selected.

DESELECTING OBJECTS

By default, the selected objects are displayed in orange color. If you want to deselect the individual objects from the selection, press and hold the SHIFT key and click on it; the entity will be deselected. If you want to deselect all the selected entities, press the ESC key. Alternatively, press and hold the SHIFT key and drag a box around the entities; all entities that lie completely inside the box will get deselected. Also, you can choose the **Deselect All** tool from the **Selection Bar** to deselect all the selected entities.

USING SNAP POINTS OPTIONS WHILE SKETCHING

While drawing in the Sketcher 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 coincide with the endpoint of the line.

NX allows you to control these snap settings using the snap points option tools in the **Selection Bar**, as shown in Figure 2-47. In this bar, some tools are chosen, by default. You can choose more tools to turn on the respective snapping option.

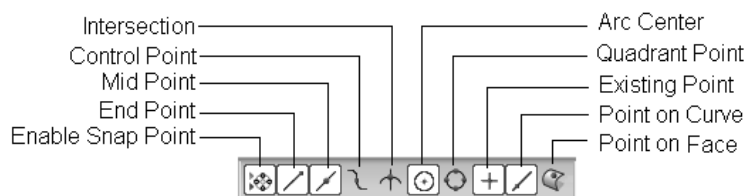


Figure 2-47 Buttons used for snap settings

**Note**

The tools to control these snap settings are available only in the Sketcher environment.

DELETING SKETCHED ENTITIES

Menu: Edit > Delete
Toolbar: Standard > Delete (Customize to add)



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 the **Delete** button, the selected entities will be deleted. However, if you choose this button without selecting any sketched entity, the **Sketcher Delete** dialog box will be displayed, as shown in Figure 2-48. You can now select the entities to be deleted and then choose the **OK** button in this dialog box. To close the dialog box, choose the **Back** or **Cancel** button.

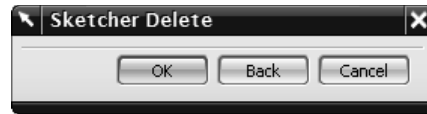


Figure 2-48 The Sketcher Delete dialog box

EXITING THE SKETCHER ENVIRONMENT

Menu: Sketch > Finish Sketch
Toolbar: Sketcher > Finish Sketch



After drawing the sketch, you need to exit the Sketcher environment to convert the sketch into a feature. To exit the Sketcher environment, choose the **Finish Sketch** button from the **Sketcher** toolbar. Alternatively, right-click in the drawing area and select the **Finish Sketch** option from the shortcut menu. When you exit the Sketcher environment, the **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 a sketch of any dimensions and then modify its size by changing the values of dimensions. However, in this chapter, you will use the dynamic input boxes to draw the sketch of exact dimensions. This will help you in improving your sketching skills.

Tutorial 1

In this tutorial, you will draw a profile for the base feature of the model shown in Figure 2-49. The profile to be drawn is shown in Figure 2-50. Do not dimension the profile because the dimensions are only for reference.

(Expected time: 30 min)

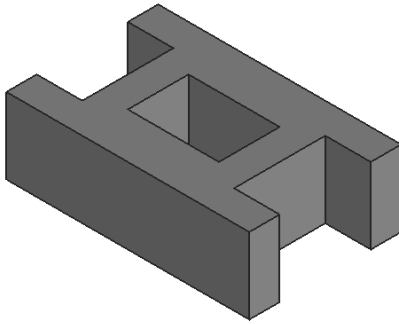


Figure 2-49 Model for Tutorial 1

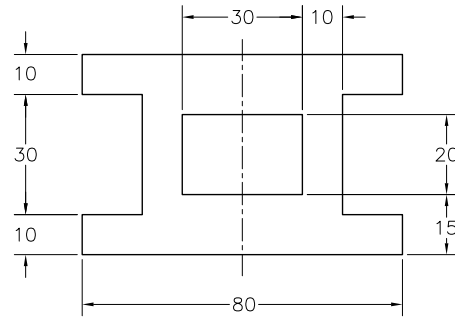


Figure 2-50 Sketch for Tutorial 1

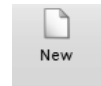
The following steps are required to complete this tutorial:

- Start a new file.
- Select the XC-YC plane as the sketching plane and invoke the Sketcher environment.
- Draw the sketch of the model by using the **Profile** and **Rectangle** tools.
- Finish the sketch and save the file.

Starting NX 7 and Starting a New File

First, you need to start NX 7 and then start a new file.

- Choose the **Start** button at the lower left corner of the screen to display a menu with additional options. Next, choose **All Programs (or Programs) > UGS NX 7.0 > NX 7.0** to start NX 7. Alternatively, double-click on NX 7.0 shortcut icon on the desktop of your computer.
- To start a new file, choose the **New** button from the **Standard** toolbar or choose **File > New** from the menu bar; the **New** dialog box is displayed.
- Select the **Model** template from the **Templates** rollout.
- Enter **c02tut1** as the name of the document in the **Name** text box of the dialog box.
- Choose the button on the right of the **Folder** text box; the **Choose Directory** dialog box is displayed.



It is recommended that you create a folder with the name NX 7 in the hard drive of your computer and then create separate folders for each chapter inside it for saving the tutorial files of this textbook.

- In this dialog box, browse to **NX 7/c02** folder and then choose the **OK** button twice; the new file is started in the Modeling environment.

Invoking the Sketcher Environment in the Modeling Environment

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

1. Choose the **Sketch** button from the **Feature** toolbar; the **Create Sketch** dialog box is displayed. By default, the XC-YC plane is selected.
2. Choose the **OK** button from the **Create Sketch** dialog box; the Sketcher 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 can be drawn by using the **Profile** tool.

1. By default, the **Profile** tool is invoked and the **Profile** dialog box is displayed. The **Line** button is active, by default in this dialog box. 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 modified dynamically in the dynamic input boxes.

3. Enter **80** in the **Length** dynamic input box and press the TAB key. Next, enter **0** in the **Angle** dynamic input box and press the ENTER key.
4. Choose the **Fit** button from the **View** toolbar to fit the sketch into the drawing window.
5. Enter **10** as the length and **90** as the angle in the **Length** and **Angle** dynamic input boxes, respectively. Press the ENTER key.
6. Enter **15** as the length and **180** as the angle in the **Length** and **Angle** dynamic input boxes, respectively. Press the ENTER key.
7. Enter **30** as the length and **90** as the angle in the **Length** and **Angle** dynamic input boxes, respectively. Press the ENTER key.
8. Enter **15** as the length and **0** as the angle in the **Length** and **Angle** dynamic input boxes, respectively. Press the ENTER key.
9. Enter **10** as the length and **90** as the angle in the **Length** and **Angle** dynamic input boxes, respectively. Press the ENTER key.
10. Enter **80** as the length and **180** as the angle in the **Length** and **Angle** dynamic input boxes, respectively. Press the ENTER key.

11. Enter **10** as the length and **-90** as the angle in the **Length** and **Angle** dynamic input boxes, respectively. Press the ENTER key.
12. Enter **15** as the length and **0** as the angle in the **Length** and **Angle** dynamic input boxes, respectively. Press the ENTER key.
13. Enter **30** as the length and **-90** as the angle in the **Length** and **Angle** dynamic input boxes, respectively. Press the ENTER key.
14. Enter **15** as the length and **180** as the angle in the **Length** and **Angle** dynamic input boxes, respectively. Press the ENTER key.
15. Move the cursor to the start point of the first line and click when the cursor snaps to the start point of the line.
16. Press the ESC key twice to exit the **Profile** tool. The outer profile of the sketch is shown in Figure 2-51.

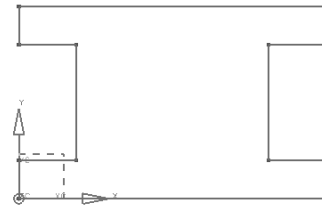


Figure 2-51 Outer profile of the sketch

Drawing the Rectangle

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 the rectangle.

1. Choose the **Rectangle** button from the **Sketch Tools** toolbar; the **Rectangle** dialog box is displayed and the **By 2 Points** button is chosen by default in this dialog box.
2. Enter **25** and **15** as the coordinates of the first point of the rectangle in the **XC** and **YC** dynamic input boxes, respectively. Next, press the ENTER key.
3. Enter **30** and **20** as the width and height of the rectangle in the **Width** and **Height** dynamic input boxes, respectively. Next, press the ENTER key. 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 also moves.
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 to exit the tool. The final sketch for Tutorial 1 is shown in Figure 2-52.

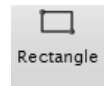


Figure 2-52 Final sketch for Tutorial 1

Finishing the Sketch and Saving the File

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

1. Choose the **Save** button from the **Standard** toolbar to save the sketch.
2. Choose the **Finish Sketch** button from the **Sketcher** toolbar; the Modeling environment is invoked.
3. Choose **File > Close > Selected Parts** from the menu bar; the **Close Part** dialog box is displayed.
4. Select the name of the current file from the list area in the **Part** rollout and then choose the **OK** button to close the current file.



Tutorial 2

In this tutorial, you will draw a sketch for the model shown in Figure 2-53. The sketch to be drawn is shown in Figure 2-54. Do not dimension the sketch because the dimensions are given only for reference.
(Expected time: 30 min)

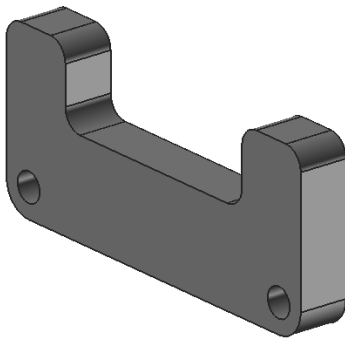


Figure 2-53 Model for Tutorial 2

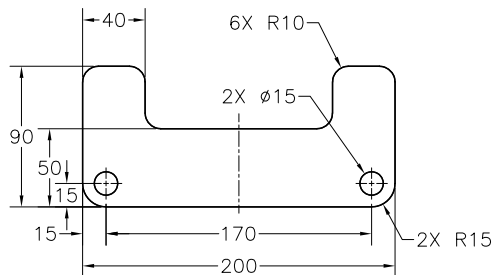


Figure 2-54 Sketch for Tutorial 2

The following steps are required to complete this tutorial:

- a. Start NX and then start a new file.
- b. Invoke the Sketcher environment by using the XC-ZC plane as the sketching plane.
- c. Draw the outer loop of the profile by using the **Profile** tool.
- d. Fillet the sharp corners of the outer loop using the **Fillet** tool.
- e. Draw circles by using the centers of fillets to complete the profile.
- f. Finish the sketch and save the file.

Starting NX 7 and Starting a New File

First, you need to start NX 7 and then start a new file.

1. Choose the **Start** button at the lower left corner of the screen to display a menu. Next, choose **All Programs** (or **Programs**) > **UGS NX 7.0** > **NX 7.0** to start NX 7. Alternatively, double-click on **NX 7.0** shortcut icon on the desktop of your computer.
2. To start a new file, choose the **New** button from the **Standard** toolbar or choose **File > New** from the menu bar; the **New** dialog box is displayed.
3. Select the **Model** template from the **Templates** rollout.
4. Enter **c02tut2** as the name of the document in the **Name** text box of the dialog box.
5. Choose the button on the right side of the **Folder** text box; the **Choose Directory** dialog box is displayed.
6. In this dialog box, browse to **NX 7/c02** folder and then choose the **OK** button twice; the new file is started in the Modeling environment.



Invoking the Sketcher Environment in the Modeling Environment

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

1. Choose the **Sketch** button from the **Feature** toolbar; the **Create Sketch** dialog box is displayed.
2. Select the XC-ZC plane from the drawing window.
3. Choose the **OK** button from the **Create Sketch** dialog box; the Sketcher environment is invoked and the sketching plane is oriented parallel to the screen.



Drawing Lines of the Outer Loop

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

By default, the **Profile** tool is invoked and the **Profile** dialog box is displayed. The **Line** button is chosen by default in the dialog box. Also, the dynamic input boxes are displayed below the line cursor.


1. Move the cursor close to the origin; the coordinates of the point are displayed as 0,0 in the dynamic input boxes. Next, click to specify the start point of the line.

The point you specified 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 modified dynamically 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 by using the **Length** and **Angle** dynamic input boxes.

2. Enter **200** in the **Length** dynamic input box and press the TAB key. Next, enter **0** in the **Angle** dynamic input box and press the ENTER key.

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

3. 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. 

4. Enter **90** in the **Length** dynamic input box and press the TAB key. Next, enter **90** in the **Angle** dynamic input box and press the ENTER key; a vertical line of 90 mm is drawn.

5. Choose the **Fit** button again to fit the drawing into the current display. 

6. Move the cursor away from the end point of the last line and then enter **40** in the **Length** dynamic input box and press the TAB key. Next, enter **-180** in the **Angle** dynamic input box and press the ENTER key; a horizontal line of 40 mm is drawn.

7. Move the cursor away from the end point of the last line and then enter **40** in the **Length** dynamic input box and press the TAB key. Next, enter **-90** in the **Angle** dynamic input box and press the ENTER key; a vertical line of 40 mm is drawn downward.

8. Move the cursor away from the end point of the last line and then enter **120** in the **Length** dynamic input box and press the TAB key. Next, enter **180** in the **Angle** dynamic input box and press the ENTER key; a horizontal line of 120 mm is drawn.

9. 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.

10. Move the cursor once toward the vertical line of 40 mm drawn earlier and then move it back to the vertical direction from the start point of this line. When the line is vertical, the vertical constraint symbol is displayed.

11. 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 value of the length in the **Length** dynamic input box is **40** and the value of the angle is **90**. Click to specify the endpoint of this line.

12. 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.

13. Move the cursor vertically downward to the origin. If the first line is not highlighted in yellow, move the cursor over it once and then move it back to the origin; the cursor snaps to the endpoint of the first line.
14. Click to specify the endpoint of the line when the vertical constraint symbol is displayed. Choose the **Fit** button to fit the sketch into the drawing window.
15. Press the ESC key twice to exit the **Profile** tool. The sketch after drawing the lines is shown in Figure 2-55.

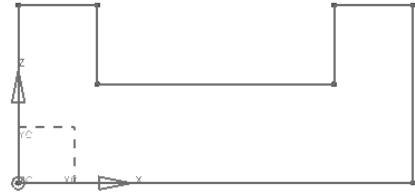


Figure 2-55 Sketch after drawing the lines

Filleting Sharp Corners

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

1. Choose the **Fillet** button from the **Sketch Tools** toolbar; the **Create Fillet** dialog box is displayed.



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

2. Enter **15** in the **Radius** dynamic input box and press the ENTER key.
3. Move the cursor over the lower left corner of the sketch; the two lines comprising this corner are highlighted in yellow. Click to select this corner; a fillet is created at the lower left corner.
4. Similarly, move the cursor over the lower right corner and click to select it when the two lines that form this corner are highlighted in yellow.

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

5. Enter **10** in the **Radius** dynamic input box and press the ENTER key.
6. Select the remaining corners of the sketch one by one and fillet them with a radius of 10.
7. Right-click, and then select the **OK** option from the shortcut menu to exit the **Fillet** tool. The fillets are created, refer to Figure 2-56.

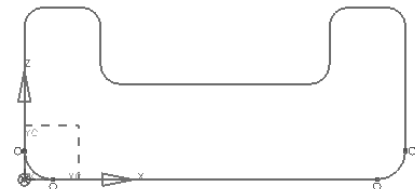


Figure 2-56 Sketch after creating fillets

Drawing Circles

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

1. Choose the **Circle** button from the **Sketch Tools** toolbar; the **Circle** dialog box is displayed. By default, the **Circle by Center and Diameter** button is chosen in this dialog box. Also, you are prompted to select the center of the circle.
2. Move the cursor towards the center point of the lower left fillet; 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 fillet to specify the center point of the circle.
4. Enter **15** in the **Diameter** dynamic input box and press the ENTER key; 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 towards the center point of the lower right fillet; 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 is drawn at the specified location.



Note

*If you select an incorrect point as the center point of the circle by mistake, 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 of the model for Tutorial 2. The final sketch for Tutorial 2 is shown in Figure 2-57.

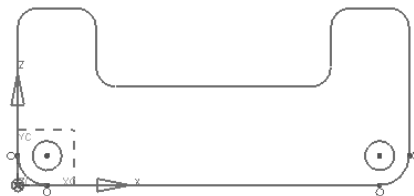


Figure 2-57 Final sketch for Tutorial 2

Finishing the Sketch and Saving the File

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

1. Choose the **Fit** button to fit the sketch into the drawing window.
2. Choose the **Save** button from the **Standard** toolbar to save the sketch.
3. Choose the **Finish Sketch** button from the **Sketcher** toolbar; the Modeling environment is invoked.
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 list area in the **Part** rollout and then choose the **OK** button to close the current file.



Note

For better visualization, the background color of graphics in this textbook is set to white. To change the background color, choose **Preferences > Background** from the menu bar; the **Edit Background** dialog box is displayed. Select the **Plain** radio button from both the **Shaded Views** and **Wireframe Views** areas. Next, choose the **Plain Color** swatch; the **Color** dialog box is displayed. In this dialog box, select the white color and choose the **OK** button twice to exit the **Edit Background** dialog box.

Tutorial 3

In this tutorial, you will draw the profile of the model shown in Figure 2-58. The profile to be drawn is shown in Figure 2-59. Do not dimension the profile because the dimensions are given only for reference. **(Expected time: 30 min)**

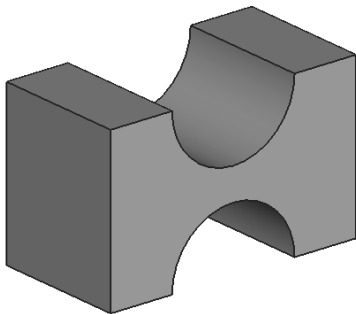


Figure 2-58 Model for Tutorial 3

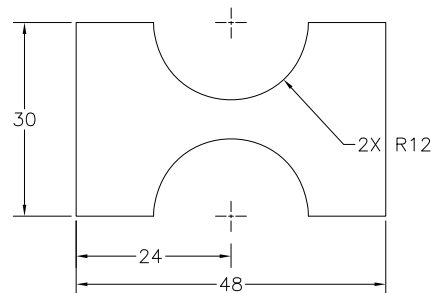


Figure 2-59 Sketch for Tutorial 3

The following steps are required to complete this tutorial:

- a. Start a new file.
- b. Select the YC-XC plane as the sketching plane and invoke the Sketcher environment.
- c. Draw the sketch of the model by using the **Profile** tool.
- d. Finish the sketch and save the file.

Starting a New File

If you continue working after completing Tutorial 2, you do not need to open a new session of NX. You can start a new part file by selecting the **Model** template from the **New** dialog box.

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



2. Select the **Model** template from the **Templates** rollout.

3. Choose the button on the right of the **Name** text box; the **Choose New File Name** dialog box is displayed.



4. In this dialog box, browse to *NX 7/c02* folder and then enter *c02tut3* in the **File name** edit box. Next, choose the **OK** button twice; the new file is started in the Modeling environment.

Invoking the Sketcher Environment in the Modeling Environment

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

1. Choose the **Sketch** button from the **Feature** toolbar; the **Create Sketch** dialog box is displayed.
2. Select the YC-ZC plane from the drawing window. Note that the Z-axis direction of the sketching plane points toward the front side of the sketching plane and the direction of Y-axis is upward.
3. Choose the **OK** button from the **Create Sketch** dialog box; the Sketcher 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 by using the **Line** and **Arc** options of the **Profile** tool.

1. By default, the **Profile** tool is invoked and the **Profile** dialog box is displayed. The **Line** button is activated, by default, in this dialog box. 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 specified 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 modified dynamically 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 by using the **Length** and **Angle** dynamic input boxes.

3. Enter **12** in the **Length** dynamic input box and press the TAB key. Next, enter **0** in the **Angle** dynamic input box and press the ENTER key. The first line is drawn and the rubber-band line is displayed with the start point at the endpoint of the previous line and the endpoint attached to the cursor. Now, you need to invoke the arc mode because the next entity to be drawn is an arc.
4. Choose the **Arc** button from the **Object Type** area of the **Profile** dialog box 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. Next, move the cursor toward the right; you will notice that a normal arc starts from the endpoint of the last line.
6. Enter **12** in the **Radius** dynamic input box and press TAB key. Next, enter **180** in the **Sweep Angle** dynamic input box and press the ENTER key.

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 dynamic input boxes.

7. Move the cursor horizontally toward the right 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, and then press ENTER key. Choose the **Fit** button from the **View** toolbar to fit the sketch into the drawing window.
9. Enter **30** as the length and **90** as the angle in the **Length** and **Angle** dynamic input boxes, respectively, and then press ENTER key.

10. Move the cursor horizontally toward the left. 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 endpoint of the arc.



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

11. Choose the **Arc** button from the **Profile** dialog box 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, refer to the Figure 2-60.

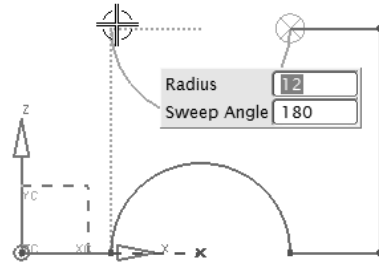


Figure 2-60 Horizontal and vertical help lines displayed to define the endpoint of the arc

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, refer to the Figure 2-60.

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, and then press ENTER key.
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 2-61.

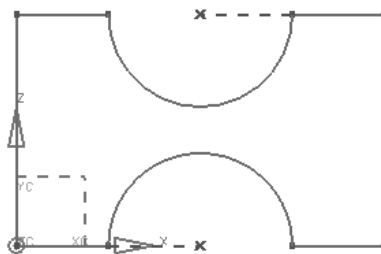


Figure 2-61 Final sketch for Tutorial 3

Finishing the Sketch and Saving the File

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

1. Choose the **Save** button from the **Standard** toolbar to save the sketch.
2. Choose the **Finish Sketch** button from the **Sketcher** toolbar; the Modeling environment is invoked.
3. Choose **File > Close > Selected Parts** from the menu bar; the **Close Part** dialog box is displayed.
4. Select the name of the current file from the list area in the **Part** rollout and then choose the **OK** button to close the current file.



Self-Evaluation Test

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

1. Most of the designs created in NX consist of sketch-based features and placed features. (T/F)
2. When you invoke the Sketcher environment, the **Profile** tool is invoked by default. (T/F)
3. You can use the dynamic input boxes to specify the exact values of sketched entities. (T/F)
4. The **Sketch** button is chosen to invoke the Sketcher environment. (T/F)
5. You can restore the original orientation of the sketching plane by 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** dialog box.
7. You can fillet corners in a sketch by using the _____ tool.
8. You can draw an elliptical arc by using the _____ tool.
9. If you choose the _____ button from the **Rectangle** dialog box, it will enable you to draw a centerpoint rectangle.
10. You can exit the Sketcher environment by choosing the _____ button from the **Sketcher** toolbar.

Review Questions

Answer the following questions:

- Which one of the following dialog boxes is displayed when you choose the **New** button from the **Standard** toolbar to start a new file?
 - New Part File**
 - New**
 - File New**
 - Part File**
- Which of the following tools in NX is used to create conics?
 - General Conic**
 - Conic**
 - Round**
 - None
- Which mode is automatically invoked from the **Profile** dialog box when you specify the start point of a line?
 - Coordinate Mode**
 - Angle Mode**
 - Parameter Mode**
 - None
- In NX, how many methods are used to start a new file?
 - 1
 - 2
 - 3
 - 5
- Which of the following methods is available in the **Studio Spline** dialog box along with the **By Poles** method to draw splines?
 - No Poles**
 - From Poles**
 - From Points**
 - Through Points**
- The files in NX are saved with *.prt* extension. (T/F)
- You can select entities by dragging a box around them. (T/F)
- You can set the selection mode to select only the sketched entities. (T/F)
- In NX, you can create fillets by simply dragging the cursor across the entities that you want to fillet. (T/F)
- In NX, you cannot draw a rectangle from its center. (T/F)

Exercises

Exercise 1

Draw a sketch for the base feature of the model shown in Figure 2-62. The sketch to be drawn is shown in Figure 2-63. Do not dimension the profile because the dimensions are given only for reference.
(Expected time: 30 min)

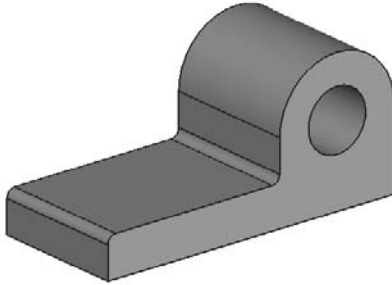


Figure 2-62 Model for Exercise 1

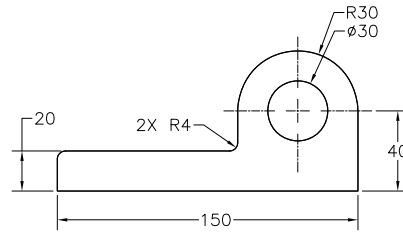


Figure 2-63 Sketch for Exercise 1

Exercise 2

Draw a sketch for the base feature of the model shown in Figure 2-64. The sketch to be drawn is shown in Figure 2-65. Do not dimension the profile because the dimensions are given only for reference.
(Expected time: 30 min)

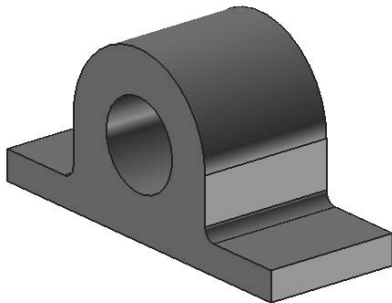


Figure 2-64 Model for Exercise 2

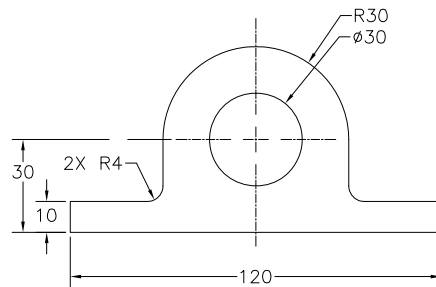


Figure 2-65 Sketch for Exercise 2

Exercise 3

Draw a sketch for the base feature of the model shown in Figure 2-66. The sketch to be drawn is shown in Figure 2-67. Do not dimension the profile because the dimensions are given only for reference.

(Expected time: 30 min)

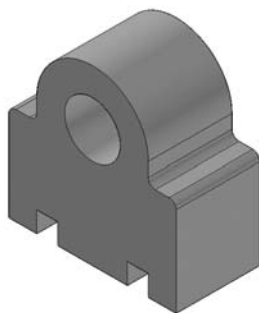


Figure 2-66 Model for Exercise 3

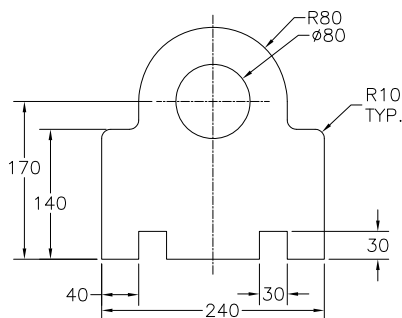


Figure 2-67 Sketch for Exercise 3

Answers to Self-Evaluation Test

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