

# Chapter 3

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## Adding Geometric and Dimensional Constraints to Sketches

### Learning Objectives

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

- *Understand the concept of under-constrained, fully-constrained, and over-constrained sketches.*
- *Understand different types of dimensions.*
- *Measure the distance value between entities in a sketch.*
- *Measure the angle between entities.*
- *Understand different types of geometric constraints.*
- *Configure settings for applying constraints automatically while sketching.*
- *Force additional geometric constraints to sketches.*
- *View and delete geometric constraints from sketches.*
- *Animate a fully-constrained sketch.*

## CONSTRAINING SKETCHES

In the previous chapter, you learned to draw sketches. You must have noticed that the dimensions are applied automatically to the sketches when they are drawn but the dimensions are updated when you drag the sketched entities. Therefore, these dimensions do not constrain the sketch. In this chapter, you will learn to completely constrain the sketches to restrict their degrees of freedom to make them stable. The stability ensures that the size, shape, and location of the sketches do not change unexpectedly with respect to the surrounding. Therefore, it is always recommended to constrain the sketches. The first step is to apply the geometrical constraints to the sketch. Some of the constraints are automatically applied to sketches while drawing them. After applying the geometrical constraints, you need to add dimensional constraints using the tools in the **Sketch Tools** toolbar.

## CONCEPT OF CONSTRAINED SKETCHES

After drawing and applying the constraints, the sketch can attain any one of the following three stages:

1. Under-Constrain
2. Fully-Constrain
3. Over-Constrain

These stages are described next.

### Under-Constrain

An under-constrained sketch is the one in which all degrees of freedom of each entity are not completely defined using the geometric and dimensional constraints. The elements of the sketch that are displayed in maroon color are under-constrained. You need to apply additional constraints to them in order to constraint their degree of freedom. The under-constrained sketches tend to change their position, size, or shape unexpectedly. Therefore, it is necessary to fully define the sketched elements. Figure 3-1 shows an under-constrained sketch.

### Fully-Constrain

The fully-constrained sketch is the one in which all degrees of freedom of each element are defined using the geometric and dimensional constraints. As a result, the sketch cannot change its position, shape, or size unexpectedly. These dimensions can change only if they are modified deliberately by the user. The elements of a fully-constrained sketch are displayed in dark green color. Figure 3-2 shows a fully-constrained sketch.

### Over-Constrain

An over-constrained sketch is the one in which some additional constraints are applied. The entities that are affected due to over-constraining are displayed in magenta color. The over-constrained dimensions and constraints are displayed in red color. It is always recommended to delete additional constraints and make the sketch fully-constrained before exiting the Sketch task environment. Figure 3-3 shows an over-constrained sketch.

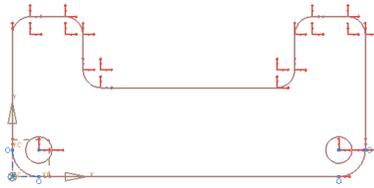


Figure 3-1 An under-constrained sketch

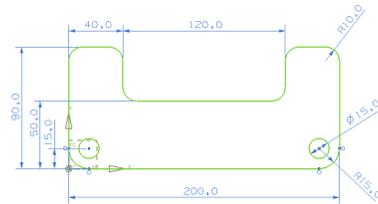


Figure 3-2 A fully-constrained sketch

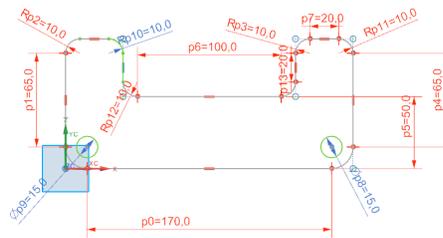


Figure 3-3 An over-constrained sketch

While applying the geometric and dimensional constraints, the status area of the **Status Bar** displays the number of constraints needed to fully constrain the sketch. After fully constraining the sketch, you will be informed in the status area of the **Status Bar** that the sketch is fully constrained.

Also, if the sketch is over-constrained, you will be informed that the sketch contains over constrained geometry. In this case, you need to remove one or more constraints applied.

## DEGREE OF FREEDOM ARROWS

The degree of freedom arrows are displayed on the points that are free to move (under-constrain), refer to Figure 3-4. Note that the degree of freedom arrows will be displayed only when you choose any constraining tool (geometrical or dimensional). These tools are discussed later in this chapter.

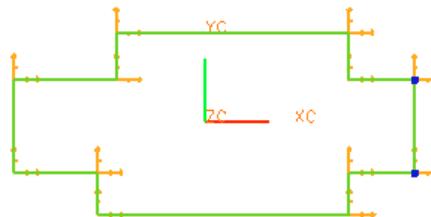


Figure 3-4 The degree of freedom arrows displayed on points

The direction of arrow at a particular point indicates that you need to constrain the movement along that direction. When you constrain a point, NX will remove the degree of freedom arrows. The sketch will be fully-constrained only when all the arrows disappear. The horizontal and vertical arrows indicate that the point is free to move in the X and Y directions, respectively.

Various geometric and dimensional constraint tools used to fully-constrain the sketch are discussed next.

## DIMENSIONING SKETCHES

After creating a sketch, you need to apply different types of dimensions (dimensional constraints) to it. The purpose of dimensioning is to control the size of the sketch and to place it with reference to some other entity. As discussed earlier, by default, sketched entities are dimensioned automatically while drawing a sketch. However, the automatically applied dimensions do not constrain the sketch. Therefore, you need to lock the automatically applied dimensions to fully constrain the sketch. The method of locking the automatically applied dimensions is discussed later in this chapter. Also, sometime you may need to add additional dimensions to the sketch to fully constrain it. The **Inferred Dimensions** tool from the **Sketch Tools** toolbar is used to dimension the entities manually. You can also apply the dimensions listed below by using their respective tools.

1. Horizontal Dimensions
2. Vertical Dimensions
3. Parallel Dimensions
4. Perpendicular Dimensions
5. Angular Dimensions
6. Diameter Dimensions
7. Radius Dimensions
8. Perimeter Dimensions



### Note

*As the **Continuous Auto Dimensioning** tool is chosen by default in the **Sketch Tools** toolbar, the sketched entities will be dimensioned automatically.*



The procedure of locking the automatically applied dimensions is discussed next.

## Locking the Automatically Applied Dimensions of the Sketch

As discussed earlier, when you draw a sketch, some dimensions are automatically applied to it. However, these dimensions do not constrain the sketch as they get modified when you drag the sketched entities. To lock a dimension of the sketch, you need to double-click on it. On doing so, an edit box will be displayed. Enter the required value in the edit box and press ENTER; the dimension will be locked and displayed in blue color.

NX is a parametric software and therefore, in this software you can modify the dimension created any time by entering the Sketch task environment. The methods of applying various dimensions are discussed next.

## Applying Horizontal Dimensions

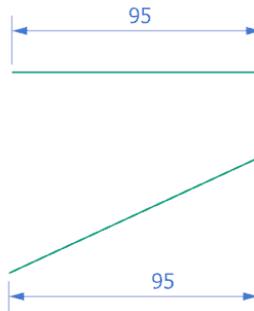
**Menu bar:** Insert > Dimensions > Horizontal

**Toolbar:** Sketch Tools > Inferred Dimensions > Horizontal Dimension



Horizontal  
Dimension

The **Horizontal Dimension** tool is used to apply a horizontal dimension between any two points. Even if you select an entity with a slant angle, the dimension will always be applied horizontally between the endpoints of the object selected. To apply the horizontal dimension, choose **Inferred Dimensions > Horizontal Dimension** from the **Sketch Tools** toolbar; the **Dimensions** dialog box will be displayed and you will be prompted to select an object to be dimensioned or select a dimension to be edited. Select the object to be dimensioned; the horizontal dimension of the selected object will be attached to the cursor. Next, you need to place it at the required location. Place the dimension above or below the selected object by pressing the left mouse button inside the drawing window; an edit box will be displayed. Enter the required value in this edit box, and then press ESC. Figure 3-5 shows the horizontal dimensioning of lines.



*Figure 3-5 The horizontal dimension created for a horizontal line and an inclined line*



### Note

*If you are creating a sketch in the Modeling environment, you can apply horizontal dimension by choosing the **Horizontal Dimension** tool from the **Direct Sketch** toolbar.*

## Applying Vertical Dimensions

**Menu bar:** Insert > Dimensions > Vertical

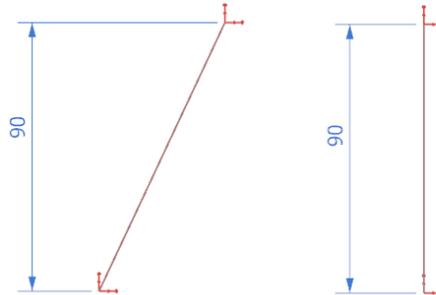
**Toolbar:** Sketch Tools > Inferred Dimensions > Vertical Dimension



Vertical  
Dimension

The **Vertical Dimension** tool is used to apply vertical dimension between any two points. Even if you select linear object with a slant angle, the dimension between the endpoints of the selected object will always be applied vertically. To apply the vertical dimension to an object, choose **Inferred Dimensions > Vertical Dimension** from the **Sketch Tools** toolbar; the **Dimensions** dialog box will be displayed and you will be prompted to select an object to be dimensioned or select a dimension to be edited. Select an object; the vertical dimension of the selected object will be attached to the cursor. Place the dimension to the left or right of the selected object by pressing the left mouse button in the

drawing window; an edit box will be displayed. Enter the required value in this edit box, and then press ESC. Figure 3-6 shows the vertical dimensioning of lines.



**Figure 3-6** The vertical dimension created for a vertical line and an inclined line



#### Note

If you are creating a sketch in the Modeling environment, you can apply vertical dimension by choosing the **Vertical Dimension** tool from the **Direct Sketch** toolbar.

In the figures, some of the dimension properties such as decimal places and labels have been modified for better display of dimensions. To modify the dimension label, choose **Task > Sketch Style** from the menu bar; the **Sketch Style** dialog box will be displayed. In this dialog box, select the **Value** option from the **Dimension Label** drop-down list. Next, choose the **OK** button; the dimensions will be modified. Alternatively, choose **Preferences > Sketch** from the menu bar; the **Sketch Preferences** dialog box will be displayed. In this dialog box, choose the **Sketch Style** tab; the **Sketch Preferences** message box will be displayed. Choose **OK**. Next, in the **Dimension Label** drop-down list of the **Sketch Preferences** dialog box, select the **Value** option, and then choose the **OK** button; the dimensions will be modified.

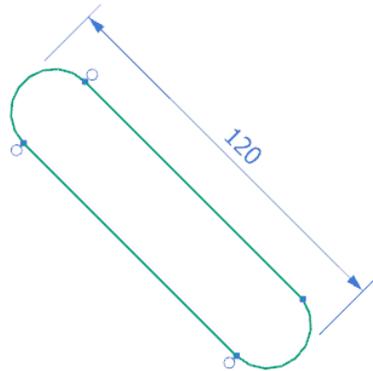
## Applying Parallel Dimensions

**Menu bar:** Insert > Dimensions > Parallel

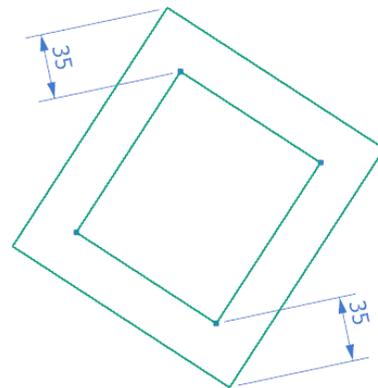
**Toolbar:** Sketch Tools > Inferred Dimensions > Parallel Dimension



The **Parallel Dimension** tool is used to measure the actual distance of a line (straight or inclined). You can apply dimension either by selecting a line or by selecting points, endpoints, or center points. To apply the parallel dimension, choose **Inferred Dimensions > Parallel Dimension** from the **Sketch Tools** toolbar; the **Dimensions** dialog box will be displayed and you will be prompted to select an object to be dimensioned or select a dimension to be edited. Select the objects between which the dimension has to be applied; the dimension will be attached to the cursor. Place the dimension at the desired location by clicking the left mouse button; an edit box will be displayed. Enter the required value in this edit box, and then press ESC. Figures 3-7 and 3-8 show the parallel dimension applied to the sketches.



**Figure 3-7** The parallel dimension applied to a sketch



**Figure 3-8** The parallel dimension applied to a sketch



**Note**

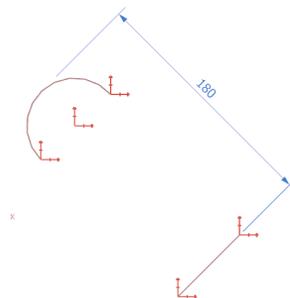
If you are creating a sketch in the Modeling environment, you can apply parallel dimension by choosing the **Parallel Dimension** tool from the **Direct Sketch** toolbar.

**Applying Perpendicular Dimensions**

**Menu bar:** Insert > Dimensions > Perpendicular  
**Toolbar:** Sketch Tools > Inferred Dimensions > Perpendicular Dimension



The **Perpendicular Dimension** tool is used to create the perpendicular dimension between a linear object and a point. It is mandatory that any one of the objects selected must be a linear object. To create the perpendicular dimension, choose **Inferred Dimensions > Perpendicular Dimension** from the **Sketch Tools** toolbar; the **DIMENSIONS** dialog box will be displayed and you will be prompted to select an object to be dimensioned or select a dimension to be edited. Select the objects between which the dimension needs to be applied and then place the dimension. As soon as you place the dimension, an edit box will be displayed. Enter the required value in this edit box, and then press ESC. Figure 3-9 shows the perpendicular dimension applied to a sketch.



**Figure 3-9** The perpendicular dimension applied between the objects

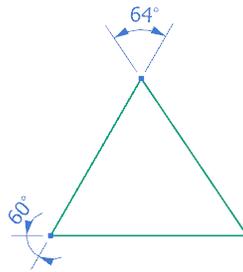
## Applying Angular Dimensions

**Menu bar:** Insert > Dimensions > Angular

**Toolbar:** Sketch Tools > Inferred Dimensions > Angular Dimension



The **Angular Dimension** tool is used to apply an angular dimension. Whenever an angular dimension is applied using the **Angular** tool, the angle is always measured in the counterclockwise direction. To create an angular dimension, choose **Inferred Dimensions > Angular Dimension** from the **Sketch Tools** toolbar; the **Dimensions** dialog box will be displayed and you will be prompted to select an object to be dimensioned or select a dimension to be edited. Select the objects between which the angular dimension needs to be applied and then place the dimension. Figure 3-10 shows different types of angular dimensions applied to a sketch.



*Figure 3-10 Angular dimensions applied between the objects*

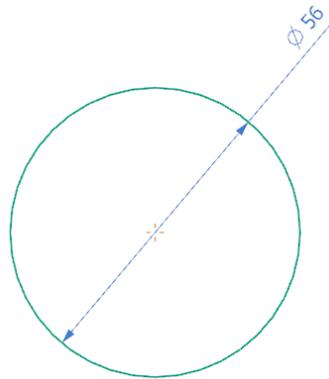
## Applying Diameter Dimensions

**Menu bar:** Insert > Dimensions > Diameter

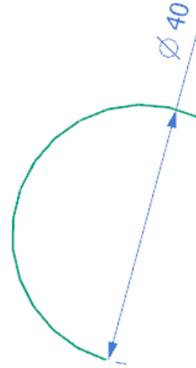
**Toolbar:** Sketch Tools > Inferred Dimensions > Diameter Dimension



The **Diameter Dimension** tool is used to apply diameter dimension to an arc or a circle. Generally, diameter dimensions are applied to circles. To apply diameter dimension, choose **Inferred Dimensions > Diameter Dimension** from the **Sketch Tools** toolbar; the **Dimensions** dialog box will be displayed and you will be prompted to select an arc to be dimensioned or a dimension to be edited. Select the object to be dimensioned and then place the dimension. As soon as you place the dimension, an edit box will be displayed. Enter the required value in this edit box, and then press ESC. Figure 3-11 shows the diameter dimension applied to a circle and Figure 3-12 shows the diameter dimension applied to an arc.



**Figure 3-11** The diameter dimension applied to a circle



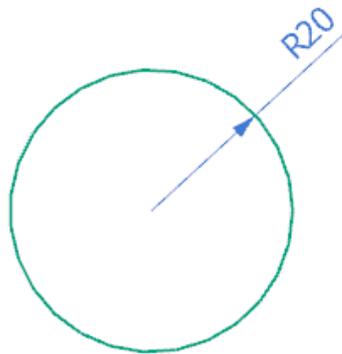
**Figure 3-12** The diameter dimension applied to an arc

### Applying Radius Dimensions

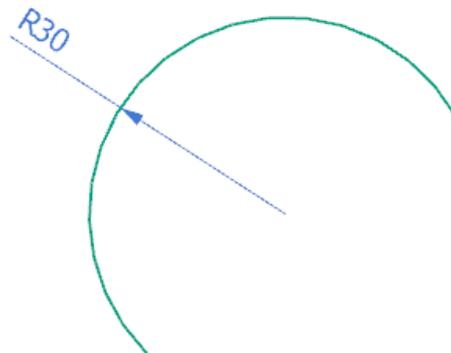
**Menu bar:** Insert > Dimensions > Radius  
**Toolbar:** Sketch Tools > Inferred Dimensions > Radius Dimension



The **Radius Dimension** tool is used to apply the radius dimension to an arc or a circle. Generally, radius dimensions are applied to arcs. To apply radius dimension, choose **Inferred Dimensions > Radius Dimension** from the **Sketch Tools** toolbar; the **Dimensions** dialog box will be displayed and you will be prompted to select an arc to be dimensioned or a dimension to be edited. Select the object to be dimensioned and then place the dimension. As soon as you place the dimension, an edit box will be displayed. Enter the required value in this edit box, and then press ESC. Figure 3-13 shows the radius dimension applied to a circle and Figure 3-14 shows the radius dimension applied to an arc.



**Figure 3-13** The radius dimension applied to a circle



**Figure 3-14** The radius dimension applied to an arc

## Applying Perimeter Dimensions

**Menu bar:** Insert > Dimensions > Perimeter  
**Toolbar:** Sketch Tools > Inferred Dimensions > Perimeter Dimension



The **Perimeter Dimension** tool is used to apply the circumferential or perimeter dimension. After applying the perimeter dimension, all dimensions of the selected objects are locked. To apply the perimeter dimension, choose **Inferred Dimensions > Perimeter Dimension** from the **Sketch Tools** toolbar; the **Perimeter Dimensions** dialog box will be displayed and you will be prompted to select lines or arcs to apply the perimeter dimension. Select the object and then choose the **OK** button from this dialog box; the perimeter dimension will be applied to the selected object. Note that this dimension will not be displayed in the drawing window.

You can also apply the perimeter dimension to a closed sketch. To do so, invoke the **Perimeter Dimensions** dialog box and then select all entities of the closed sketch one by one. Next, choose the **OK** button from the **Perimeter Dimensions** dialog box; the dimension will be applied to the sketch. Now, if you modify the dimension of any one of the entities, the dimension of other entities will also be modified such that the total perimeter of the sketch remains the same.

## Applying Dimensions by Using the Inferred Dimensions Tool

**Menu bar:** Insert > Dimensions > Inferred Dimensions  
**Toolbar:** Sketch Tools > Inferred Dimensions > Inferred Dimensions



The **Inferred Dimensions** tool is used to apply all the dimension types discussed above. A dimension is applied based on the object selected and the location of the cursor. For example, if you select an arc, the radial dimension will be applied. Similarly, if you select a circle, the diameter dimension will be applied. Select an inclined line and move the cursor parallel to that line; a parallel dimension will be applied. If you move the cursor vertically upward or downward, a horizontal dimension will be applied. Similarly, if you move the cursor in the horizontal direction (right or left), a vertical dimension will be applied.

It is recommended that you use this tool to apply dimensions as it saves the time required for selecting various dimensioning tools. To apply inferred dimensions, choose the **Inferred Dimensions** button from the **Sketch Tools** toolbar; the **Dimensions** dialog box will be displayed and you will be prompted to select the object to be dimensioned or a dimension to be edited. According to the selection procedure adopted while selecting objects, the dimensions will be applied. Figure 3-15 shows the radial and linear dimensions created.

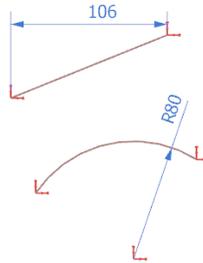


Figure 3-15 The radial and linear dimensions created by using the *Inferred Dimensions* tool

### Editing the Dimension Value and Other Parameters



To edit a dimension that has already been placed, double-click on it; an edit box will be displayed. Enter a value in this edit box and then press ESC.

You can also edit a dimension that has already been placed by using the **Dimensions** dialog box. To do so, invoke the **Dimensions** dialog box by choosing the **Inferred Dimensions** button from the **Sketch Tools** toolbar. Next, choose the **Sketch Dimensions Dialog** button from the **Dimensions** dialog box; the **Dimensions** dialog box will be modified, as shown in Figure 3-16. From the list box of this dialog box, select the dimension that you want to modify; the **Current Expression** area will be enabled. Note that the edit box on the right of this area displays the dimension value of the selected dimension. You can enter a new dimension value in the edit box and press the ESC key.

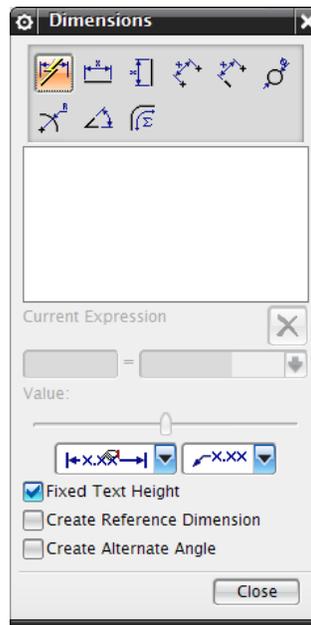


Figure 3-16 The modified *Dimensions* dialog box

You can also modify the value of the dimension dynamically by using the **Value** sliding bar in the **Current Expression** area. The first drop-down list below the **Value** sliding bar provides the options for placing the arrows with respect to the dimension line. The second drop-down list provides the options for placing the leader on the left or the right. The **Fixed Text Height** check box allows you to maintain the dimension text at a constant size when you zoom in or out a sketch. If you clear this check box, NX scales the dimension text as well as the sketch geometry. The **Create Reference Dimension** check box is used to create the reference (non-driving) dimensions. The **Create Alternate Angle** check box is used to calculate the maximum dimension between the sketch curves. Next, choose the **Close** button to reflect the changes. You can choose the **Remove Highlighted** button to delete the dimension selected from the list box. You can also apply a dimension by choosing the dimension buttons available above the list box in the **Dimensions** dialog box.

## Animating a Fully-Constrained Sketch

**Menu bar:** Tools > Constraints > Animate Dimension

**Toolbar:** Sketch Tools > Animate Dimension (*Customize to Add*)



The **Animate Dimension** tool is used to animate a sketch by selecting any one of the dimensions as the driving dimension from the same sketch. Generally, this type of animation is used while creating basic mechanisms and links. When a dimension from a fully constrained sketch is animated, the whole sketch gets mechanized by the possible movements. The dimension selected from the sketch for animating is known as the driving dimension. To animate a fully constrained sketch, choose the **Animate Dimension** button from the **Sketch Tools** toolbar; the **Animate** dialog box will be displayed, as shown in Figure 3-17, and you will be prompted to select a dimension to animate. The dimensions that are applied to the sketch are listed in the list box of the same dialog box. You can select the driving dimension directly from the sketch or from the list box. After selecting the driving dimension, enter the lower limit value for the dimension inside the **Lower Limit** edit box. Similarly, enter the upper limit value for the dimension inside the **Upper Limit** edit box. The selected dimension will be animated between the lower and upper limits specified. You can also divide an animation cycle into a number of steps and then animate the design. The number of steps per cycle should be entered inside the **Steps/Cycle** edit box. To display the dimension applied during the animation, select the **Display Dimensions** check box. For example, a fully constrained sketch from which the driving dimension is selected is shown in Figure 3-18. Figure 3-19 shows the sketch while animating. Note that at an instance, only one dimension can be selected as the driving dimension. If the sketch is not fully constrained, an undesired animation may occur.

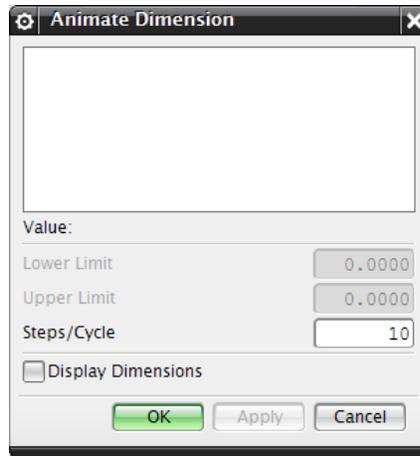


Figure 3-17 The *Animate* dialog box

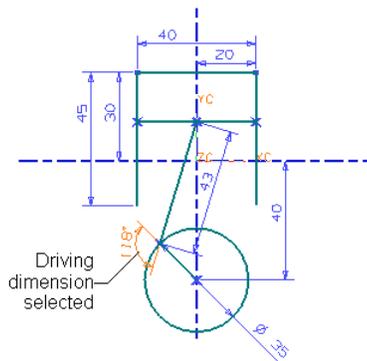


Figure 3-18 Driving dimension selected from the sketch

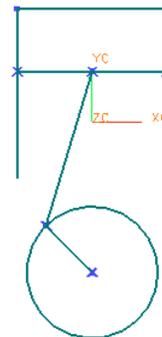


Figure 3-19 The sketch being animated

## MEASURING THE DISTANCE VALUE BETWEEN OBJECTS IN A SKETCH

**Menu bar:** Analysis > Measure Distance  
**Toolbar:** Utility > Measure Distance



While sketching, you may need to measure the dimension of various sketched entities. To do so, choose the **Measure Distance** button from the **Utility** toolbar; the **Measure Distance** dialog box will be displayed, as shown in Figure 3-20. Using this dialog box, you can measure the distance value between sketched entities through a number of methods. The methods for measuring the dimension of various sketched entities are discussed next.



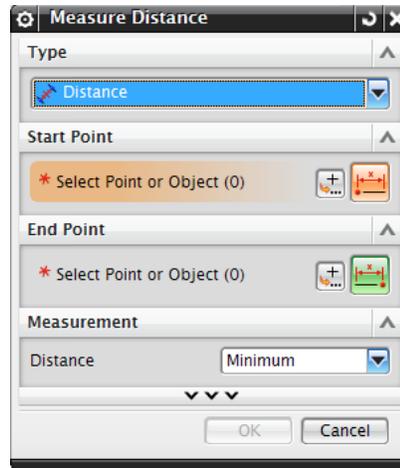


Figure 3-20 The Measure Distance dialog box

### Measuring the Distance between Two Objects in a Sketch

By default, the **Distance** option is selected in the **Type** drop-down list of the **Measure Distance** dialog box. Therefore, you are prompted to select the objects to measure the length or distance between them. Using this option, you can measure the distance between any two linear and inclined entities of a sketch. Select the start point; a ruler will be displayed, as shown in Figure 3-21. This ruler stretches along with the cursor. Now, move the cursor and specify the endpoint; the distance measured will be displayed in the display box, as shown in Figure 3-22.

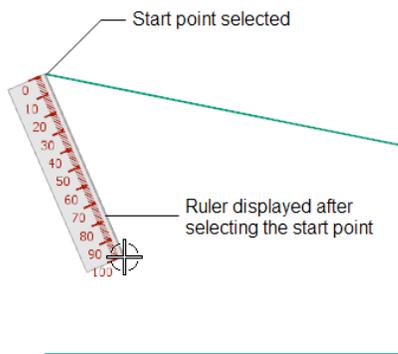


Figure 3-21 The ruler displayed after selecting the start point

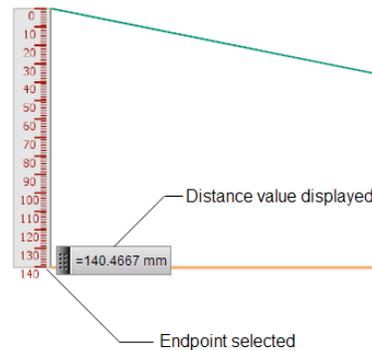


Figure 3-22 The distance value displayed in the display box



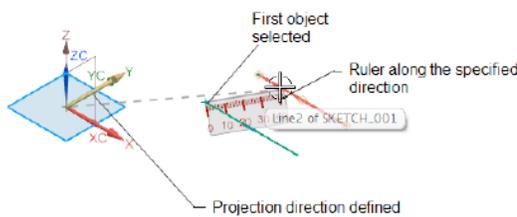
#### Note

You can also measure the distance between two objects by using the **Simple Distance** tool. To do so, choose the **Simple Distance** tool from the **Simple Measure** drop-down in the **Utility** toolbar. The procedure to measure the distance is same as discussed previously.

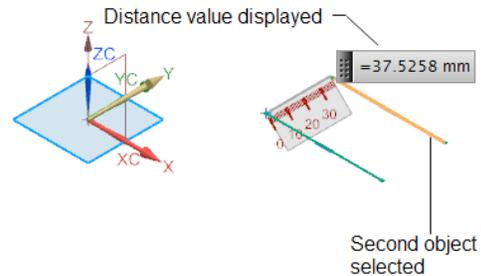


### Measuring the Projected Distance between Two Objects

To measure the distance between two objects along a predefined projected direction, select the **Projected Distance** option from the **Type** drop-down list in the **Measure Distance** dialog box; the dialog box will be modified and you will be prompted to select the objects to infer vector. Select the object or use the **Inferred Vector** drop-down list to specify the direction of projection. You can also specify the direction of projection by using the **Vector Dialog** button. Once you have specified the direction of projection, you will be prompted to select the start point or the first object to measure the distance. Select the start point; a ruler will be displayed, as shown in Figure 3-23, and you will be prompted to select the second point or the second object to measure the distance. Select the second point to measure the distance; the measured distance value will be displayed in the display box. Figure 3-24 shows the distance value displayed in the display box.



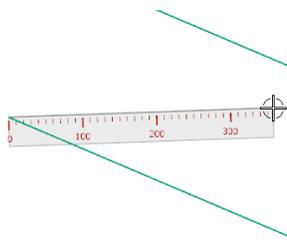
**Figure 3-23** The ruler locked to the specified projection direction



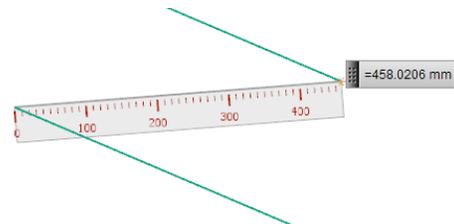
**Figure 3-24** The distance value displayed in the display box

### Measuring the Screen Distance between Two Objects

The screen distance is the distance between any two objects in a particular orientation on the screen. To measure the screen distance between two objects, select the **Screen Distance** option from the **Type** drop-down list in the **Measure Distance** dialog box; you will be prompted to select the start point or the first object to measure the distance. Select the first object; a ruler will be displayed, as shown in Figure 3-25, and you will be prompted to select the second point or the second object to measure the distance. Select the second object; the distance value between the two objects selected for the particular view or orientation will be displayed in the display box, as shown in Figure 3-26. Note that if you measure the distance between two same objects by changing the orientation, the distance value will also be changed.



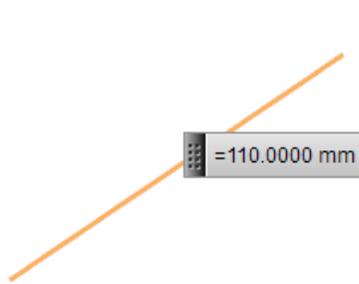
**Figure 3-25** The ruler displayed after selecting the first object



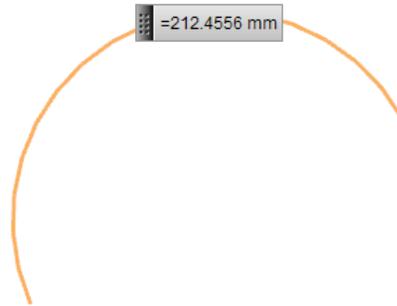
**Figure 3-26** The distance value displayed on the display box

## Measuring the Length of an Arc or a Line

To measure the length of an arc or a line, select the **Length** option from the **Type** drop-down list in the **Measure Distance** dialog box; you will be prompted to select the curve or the edge. Select an object (an arc or a line); the length of the selected object will be displayed instantly in the display box, as shown in Figures 3-27 and 3-28. Note that if you continue selecting the entities, the total arc length displayed will be the sum of the arc lengths of all the selected entities.



**Figure 3-27** The length measurement displayed for a line



**Figure 3-28** The arc length measurement displayed for an arc



### Note

You can also measure the length of an arc or a line by using the **Simple Length** tool that is available in the **Simple Measure** drop-down of the **Utility** toolbar. The procedure to measure the length is same as discussed above.



## MEASURING THE ANGLE BETWEEN ENTITIES

**Menu bar:** Analysis > Measure Angle



After creating a sketch, sometimes you may need to measure the angle between entities. To do so, choose **Analysis > Measure Angle** from the menu bar; the **Measure Angle** dialog box will be displayed, as shown in Figure 3-29. Using this dialog box, you can measure the angle values between the sketched entities. There are three methods to measure the angle, and they are discussed next.

### Measuring the Angle Value Using the By Objects Option

The **By Objects** option is used to measure the angle value subtended between any two selected objects. By default, this option is selected in the **Type** drop-down list of the **Measure Angle** dialog box and you are prompted to select the first object for the angle measurement. Select the first object; an arrow will appear on the selected object, as shown in Figure 3-30, and you will be prompted to select the second object for the angle measurement. Select the second object; the selected object will be highlighted and an arrow will be displayed on it. Note that the angle value is always subtended between the directions of arrows displayed on the two objects selected. After you select the second object, the angular ruler will be displayed along with the angle value in the display box, refer to Figure 3-30.

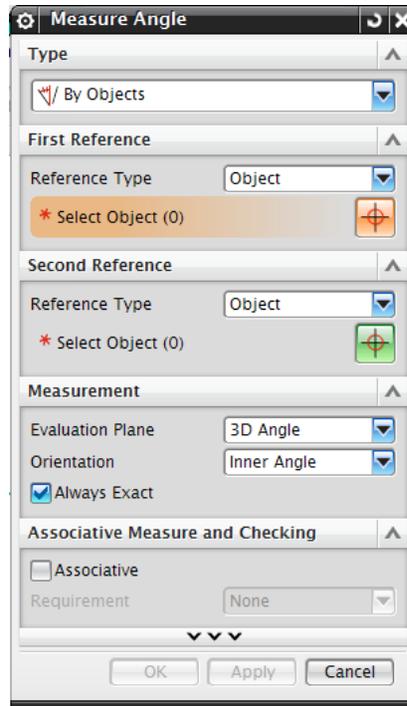


Figure 3-29 The Measure Angle dialog box



**Note**

You can also measure the angle between two objects by using the **Simple Angle** tool that is available in the **Simple Angle** tool from the **Simple Measure** drop-down in the **Utility** toolbar. The procedure to measure the angle is same as discussed above.

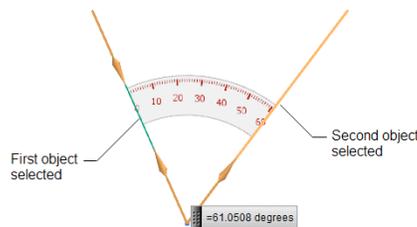


Figure 3-30 The angular measurement displayed using the **By Objects** option

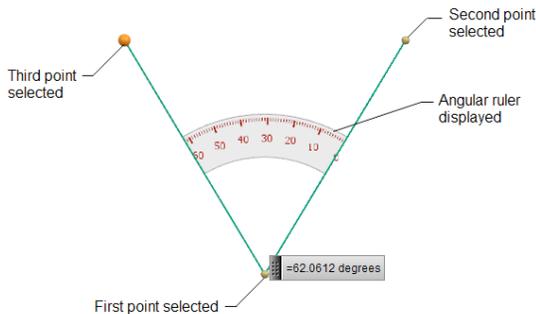
**Measuring the Angle Value Using the By 3 Points Option**

The **By 3 Points** option is used to measure the angle value subtended between three selected points. To measure the angle value by using this option, select the **By 3 Points** option from the **Type** drop-down list in the **Measure Angle** dialog box; you will be prompted to select the

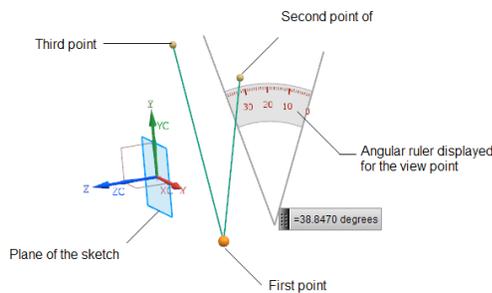
start point for the angle measurement. Select the start point; you will be prompted to select the second point for the angle base line. Select the second point; you will be prompted again to select the third point to measure the angle. On selecting the third point, the angle value along with the angular ruler will be displayed in the display box, refer to Figure 3-31.

## Measuring the Angle Value Using the By Screen Points Option

The **By Screen Points** option is used to measure the angle value between the three selected points for a particular orientation on the screen. The angle displayed between the selected objects is always subtended with respect to the view point (the point from which you are viewing the objects). To measure the angle value using this method, select the **By Screen Points** option from the **Type** drop-down list. You will be prompted to select the start point for the angle measurement. Select the start point; you will be prompted to select the second point for the angle base line. Select the second point; you will be prompted to select the third point to measure the angle. Select the third point; the angle value enclosed between the three points will be displayed in a display box along with the angular ruler. Figure 3-32 shows a sketch that was used to measure the angle using this method. However, in this figure, the view of the sketch is modified. As a result, the angle measurement has been modified on the basis of the current orientation of the view.



**Figure 3-31** The angle measured using the **By 3 Points** option



**Figure 3-32** The angular measurement displayed using the **By Screen Points** option

## GEOMETRIC CONSTRAINTS

Geometric constraints are the logical operations that are performed on the sketched entities to relate them to the other sketched entities using the standard properties such as collinearity, concentricity, tangency, and so on. These constraints reduce the degrees of freedom of the sketched entities and make the sketch more stable so that it does not change its shape and location unpredictably at any stage of the design. All geometric constraints have separate symbols associated with them. These symbols can be seen on the sketched entities when the constraints are applied to them. In the Sketch task environment of NX, you can add eleven types of geometric constraints. Some of these constraints are added automatically while sketching. Additionally, you can add more constraints to the sketch manually. These are explained next.

## Applying Additional Constraints Individually



**Menu bar:** Insert > Geometric Constraints  
**Toolbar:** Sketch Tools > Geometric Constraints



In NX, you can apply additional constraints manually by using the **Geometric Constraints** tool in the **Sketch Tools** toolbar. To apply constraints, invoke the **Geometric Constraints** tool; the **Geometric Constraints** dialog box will be displayed, as shown in Figure 3-33. Select the required constraint from the **Constraint** rollout of the dialog box and then select the entities from the drawing area to apply the selected constraint.



### Note

*By default, only few constraints are available in the **Constraint** rollout of the **Geometric Constraints** dialog box. You can customize to add or remove the constraints from the **Constraint** rollout. To do so, click on the down arrows located at the bottom of the dialog box; the **Settings** rollout will be displayed. Select the check boxes next to the required constraints; the constraints will be displayed in the **Constraints** rollout.*

In NX, you can also apply constraints directly without invoking the **Geometric Constraints** dialog box. To do so, select the entities from the drawing area; a contextual toolbar will be displayed with all possible constraints that can be applied to the selected entities. Select the required constraint from the contextual toolbar; the selected constraint will be applied to the selected entities.

Various constraints that can be applied to the sketched entities in NX are discussed next.

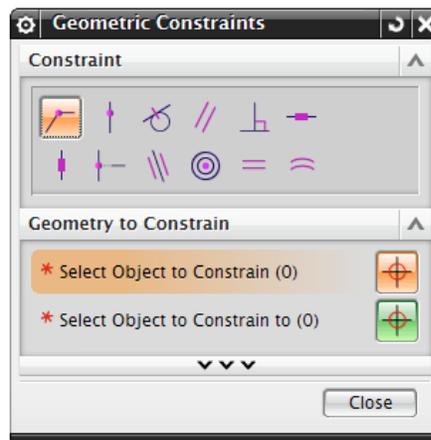


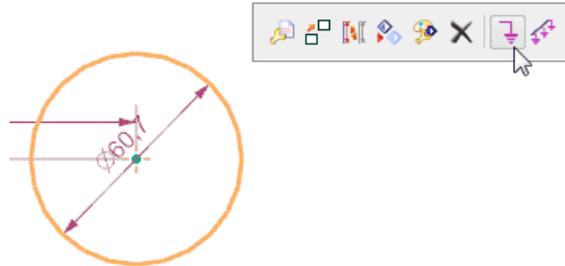
Figure 3-33 The **Geometric Constraints** dialog box

### Fixed Constraint



The **Fixed** constraint is used to fix some of the characteristics of the geometry. To apply this constraint, choose the **Fixed** button from the **Constraint** rollout of the **Geometric Constraints** dialog box. Next, select the entity to be fixed from the drawing area; the selected entity will be fixed. Alternatively, you can apply the fixed constraint using

the contextual toolbar. To do so, select an entity from the drawing area; the contextual toolbar will be displayed with the possible constraints that can be applied to the selected entity. Next, choose the **Fixed** option from the contextual toolbar, refer to Figure 3-34; the selected entity will be fixed.



*Figure 3-34 Applying the **Fixed** constraint to an entity*

The characteristics of the geometry after applying the **Fixed** constraint depend on the type of geometry selected. Following are some of the examples of **Fixed** constraint:

1. If you apply this constraint to a point, the point will be fixed and cannot be moved.
2. If you apply this constraint to a line, its angle will be fixed; however, you can move and stretch the line.
3. If you apply this constraint to the circumference of a circular arc or an elliptical arc, the radius of the arc and the location of the center point will be fixed. However, you can change the arc-length.

### Fully Fixed Constraint



This constraint is same as the **Fixed** constraint with the only difference being that this constraint fixes all characteristics of a geometry. For example, if you apply this constraint to a line, the line will be fully-constrained and it cannot be moved or stretched. To apply this constraint, choose the **Fully Fixed** button from the **Constraint** rollout and then select the entity; the selected entity will be fully fixed.

### Horizontal Constraint



The **Horizontal** constraint forces the selected line segment to become horizontal, irrespective of its original orientation. To apply this constraint, choose the **Horizontal** button from the **Geometric Constraints** dialog box and then select the entity; the selected line segment will be forced to become horizontal.

### Vertical Constraint

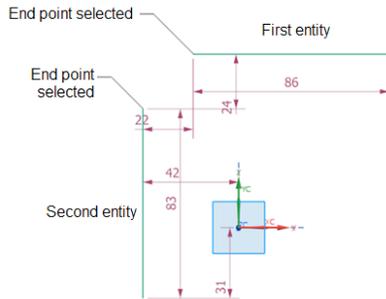


The **Vertical** constraint is similar to the **Horizontal** constraint with the only difference that this constraint will force the selected line to become vertical.

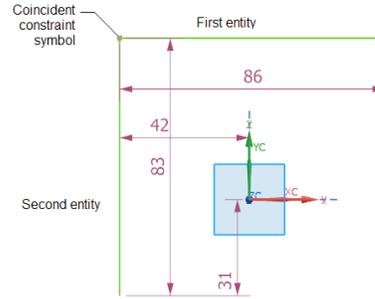
### Coincident Constraint



The **Coincident** constraint forces two or more keypoints to share the same location. The keypoints that can be used to apply this constraint include the endpoints, center points, control points of splines, and so on. To apply this constraint, invoke the **Geometric Constraints** tool and then choose the **Coincident** button from the **Constraint** rollout. Next, select the keypoints of the sketched entities that are to be made coincident. Figure 3-35 shows the endpoints of the two lines selected to be made coincident and Figure 3-36 shows the lines after applying constraint.



**Figure 3-35** The endpoints of the first and second entities selected

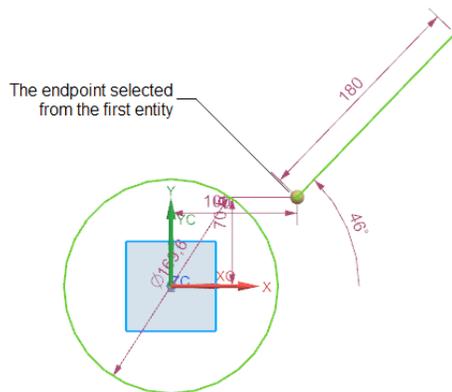


**Figure 3-36** The sketch after applying the **Coincident** constraint

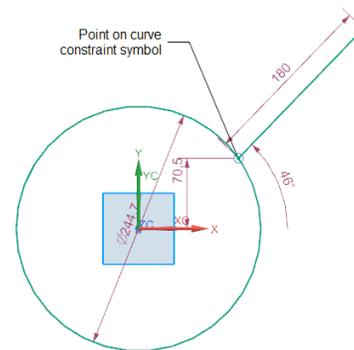
### Point On Curve Constraint



The **Point On Curve** constraint is used to place a selected keypoint on a selected curve or line. As a result, the selected point always lies on the selected curve. To apply this constraint, invoke the **Geometric Constraints** tool and then choose the **Point On Curve** button from the **Constraint** rollout; you will be prompted to select the entities to be constrained. First, select a point such as the endpoint or the center point. Next, select a curve; the point will be placed on the curve. Figure 3-37 shows a sketch before applying this constraint and Figure 3-38 shows a sketch after applying this constraint.



**Figure 3-37** The reference elements selected from the entity to apply the **Point On Curve** constraint



**Figure 3-38** The resulting sketch after applying the **Point On Curve** constraint

### Midpoint Constraint



NX provides you with an extension of the **Point on Curve** constraint, which is the **Midpoint** constraint. This constraint forces the selected point to be placed in line with the midpoint of the selected curve. Note that this constraint is available only when the selected curve is an open entity such as a line segment or an arc. Also, it is important to note that you need to select the curve anywhere other than at its endpoints.

### Parallel Constraint



The **Parallel** constraint forces a set of selected line segments or ellipse axes to become parallel to each other. To apply this constraint, invoke the **Geometric Constraints** tool and then choose the **Parallel** button from the **Constraint** rollout; you will be prompted to select the objects to constrain. Select the set of line segments or ellipses; the selected line segments or axes of the ellipse will become parallel to each other. Figure 3-39 shows two line segments before and after applying this constraint.

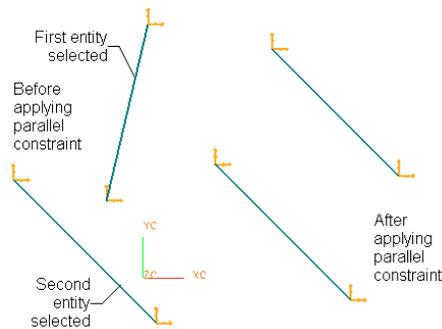


Figure 3-39 Applying the **Parallel** constraint

### Perpendicular Constraint



The **Perpendicular** constraint forces a set of selected line segments or ellipse axes to become normal to each other. Figure 3-40 shows two line segments before and after applying this constraint.

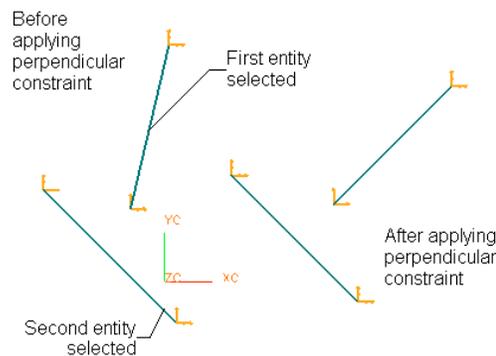
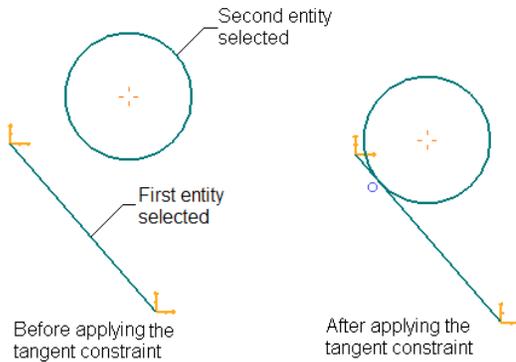


Figure 3-40 Sketch before and after applying the **Perpendicular** constraint

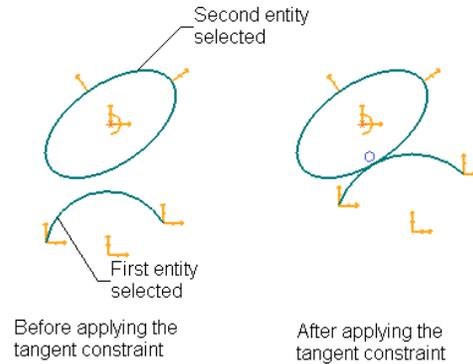
## Tangent Constraint



The **Tangent** constraint forces the selected line segment or curve to become tangent to another curve. To apply this constraint, invoke the **Geometric Constraints** tool and then choose the **Tangent** button from the **Constraint** rollout; you will be prompted to select the entities to be constrained. Select a line and a curve or select two curves; the selected line or curve will become tangent to the other selected curve. Figures 3-41 and 3-42 show the use of the **Tangent** constraint.



**Figure 3-41** Sketch before and after applying the **Tangent** constraint



**Figure 3-42** Sketch before and after applying the **Tangent** constraint



### Note

By default, symbols of all constraints are not displayed in the sketch. You can turn on the display of constraints using the **Display Sketch Constraints** tool, which is discussed later in this chapter.

## Equal Length Constraint



The **Equal Length** constraint forces the length of the selected line segments to become equal. To apply this constraint, invoke the **Geometric Constraints** tool and then choose the **Equal Length** button from the **Constraint** rollout. Next, select the line segments that you want to make equal in length; the selected line segments will become equal in length.

## Equal Radius Constraint



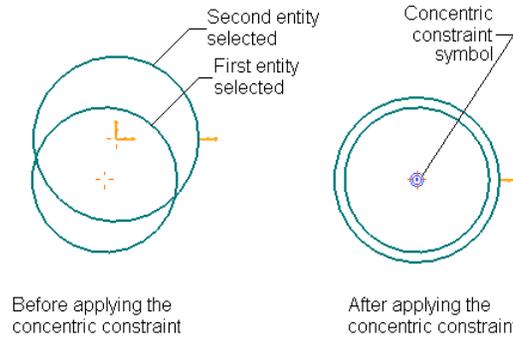
The **Equal Radius** constraint forces the selected arcs or circles to become equal in radius. To apply this constraint, invoke the **Geometric Constraints** tool and then choose the **Equal Radius** button from the **Constraint** rollout. Next, select the arcs or circles that you want to make equal in radii; the selected arcs or circles will become equal in radii.

## Concentric Constraint



This constraint is used to force two curves to share the same location of the center points. The curves that can be made concentric include arcs, circles, and ellipses. The

ellipses can be made concentric with a circle or an arc also. Figure 3-43 shows two circles before and after adding this constraint.



*Figure 3-43 Sketch before and after applying the **Concentric** constraint*

### Collinear Constraint



This constraint forces the selected line segments to be placed along the same line.

### Constant Length Constraint



The **Constant Length** constraint makes the length of the selected line segments constant. As a result, you will not be able to modify the length of the line by using the dimension constraints or by dragging.

### Constant Angle Constraint



The **Constant Angle** constraint makes the angle between the selected line segments constant. As a result, you will not be able to modify the angle between the lines by using the dimension constraints or by dragging.

### Slope of Curve Constraint



The **Slope of Curve** constraint will force the slope of the spline at the selected control point to become equal to the slope of the selected line, arc, or spline segment.

### Uniform Scale Constraint



The **Uniform Scale** constraint ensures that if you modify the distance between the endpoints of a spline, the entire spline will be scaled uniformly.

### Non-Uniform Scale Constraint



The **Non-Uniform Scale** constraint ensures that if you modify the distance between the endpoints of a spline, it will be scaled non-uniformly and it appears to stretch.

### Point On String



The **Point On String** constraint is used to place a selected keypoint on a projected curve or line. As a result, the selected point always lies on the projected curve.

## Applying Symmetry constraint



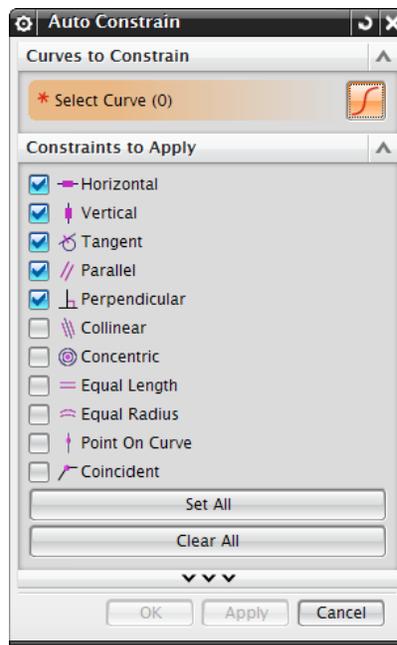
In NX, you can make two selected entities symmetric about a line by using the **Make Symmetric** tool. To make two entities symmetric, invoke the **Make Symmetric** tool from the **Sketch Tools** toolbar; the **Make Symmetric** dialog box will be displayed and you will be prompted to select the first object. Select the first object; you will be prompted to select the second object. Select the second object; you will be prompted to select a line or plane as the centerline. After selecting the centerline, you will notice that the selected objects have become symmetric about the centerline. Also, the selected centerline has converted into a reference element. This is because the **Make Reference** check box is selected by default in the dialog box. Next, choose the **Close** button; the **Make Symmetric** dialog box will be closed.

## Applying Automatic Constraints to a Sketch

**Menu bar:** Tools > Constraints > Auto Constrain  
**Toolbar:** Sketch Tools > Auto Constrain



The **Auto Constrain** tool allows you to apply the possible constraints automatically to the entire sketch. This tool is mainly used when you import geometry from another CAD system. To apply automatic constraints, choose the **Auto Constrain** button from the **Sketch Tools** toolbar; the **Auto Constrain** dialog box will be displayed, as shown in Figure 3-44. The options in this dialog box are discussed next.



*Figure 3-44 The Auto Constrain dialog box*

### Curves to Constrain Rollout

This rollout is used to select a line, a circle, or a curve to which constraints will be applied. Select the sketched entities to which the constraints are to be applied. As you select the sketched entities, the **Apply** and **OK** buttons of this dialog box will be enabled.

### Constraints to Apply Rollout

This rollout consists of various check boxes for major geometric constraints in NX. You can select the check boxes of the constraints that should be applied to the sketch. After selecting all the required check boxes of the constraints, choose the **Apply** button and then exit from the dialog box by choosing the **Cancel** button. After you exit the **Auto Constrain** dialog box, all possible constraints from the selected constraints will be applied to the sketch. Also, this rollout contains the options to set and clear all the geometric constraint check boxes. These options are discussed next.

#### Set All

If you choose this button, the check boxes of all constraints will be selected. As a result, all the possible constraints will be automatically applied to the sketch after you exit this dialog box.

#### Clear All

If you choose this button, the check boxes of all constraints will be cleared.

### Settings Rollout

This rollout provides the options to set the tolerance for applying the constraints. Note that this rollout is hidden by default. To view this rollout, click on the down arrows at the bottom of the **Auto Constrain** dialog box. The options in this rollout are discussed next.

#### Distance Tolerance

In this edit box, you can specify the maximum distance between the endpoints of two entities to be considered for applying the **Coincident** constraint.

#### Angle Tolerance

In this edit box, you can specify the angle tolerance value that will control whether the Horizontal, Vertical, Parallel, and Perpendicular constraints should be applied to the lines in the sketch after you exit the **Auto Constrain** dialog box. For example, if the deviation of lines from the X and Y axes is more than that specified value in this edit box, the **Horizontal** and **Vertical** constraints will not be applied to them.

#### Apply Remote Constraints

This check box is selected to apply constraints between the objects that are separated by a distance or angle more than the value entered in the **Distance Tolerance** and the **Angle Tolerance** edit boxes.

## Controlling Inferred Constraints Settings

**Menu bar:** Tools > Constraints > Inferred Constraints and Dimensions

**Toolbar:** Sketch Tools > Inferred Constraints and Dimension



As mentioned earlier, some of the constraints and dimensions are automatically applied to the sketched entities while they are being sketched. These settings are controlled by the **Inferred Constraints and Dimensions** tool. When you invoke this tool, the **Inferred Constraints and Dimensions** dialog box will be displayed, as shown in Figure 3-45.

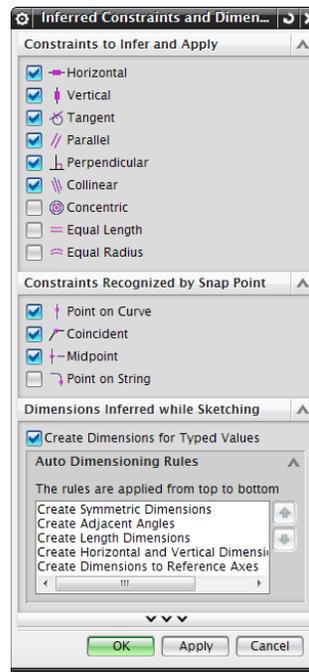


Figure 3-45 The *Inferred Constraints and Dimensions* dialog box

This dialog box displays all constraints that are available in NX. You can select the check box corresponding to those constraints that should be applied automatically to the sketch while the sketch is being drawn.

If you select the **Create Dimensions for Typed Values** check box, then the dimensions of the entities created by entering the values inside the input boxes will be displayed.

The **Auto Dimensioning Rules** area in the **Dimensions Inferred while Sketching** rollout displays rules for applying auto dimensions to the sketch. You can modify the order of these rules by using the **Move up** and **Move down** buttons.

## Showing All Constraints in a Sketch

**Menu bar:** Tools > Constraints > Display Sketch Constraints  
**Toolbar:** Sketch Tools > Display Sketch Constraints



By default, all the constraints that are applied to the sketch are not displayed automatically. For example, constraints such as Concentric, Coincident, and so on are displayed by default. However, constraints such as Parallel, Perpendicular, and so on are not displayed by default. You can turn on the display of these constraints by using the **Display Sketch Constraints** button. This is a toggle button and when you turn it on, it remains on until you turn it off. With this tool turned on, you can continue working with the other sketching tools.

## Showing/Removing Constraints

**Menu bar:** Tools > Constraints > Show/Remove Constraints  
**Toolbar:** Sketch Tools > Show/Remove Constraints



If you want to temporarily highlight or permanently delete the constraints applied to the sketch, you can use the **Show/Remove Constraints** tool. When you invoke this tool, the **Show/Remove Constraints** dialog box will be displayed, as shown in Figure 3-46. The options in this dialog box are discussed next.

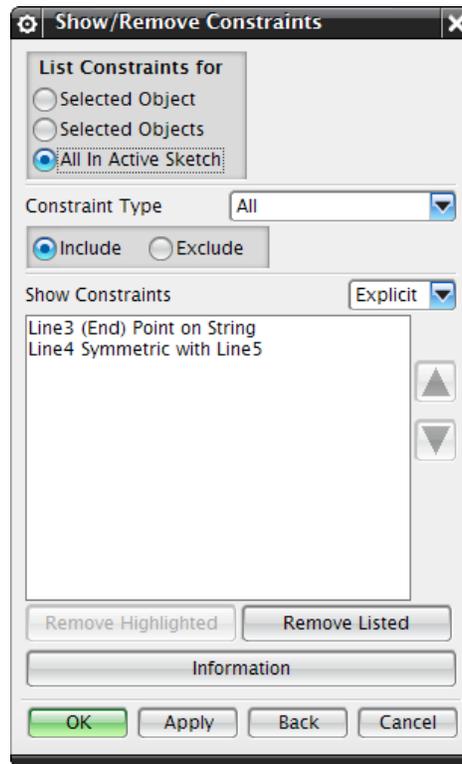


Figure 3-46 The Show/Remove Constraints dialog box

## List Constraints for Area

This area provides three radio buttons, which are discussed next.

### Selected Object

Selecting this radio button ensures that the constraints applied only to the currently selected entity are listed in the list box in the **Show Constraints** area. For example, when you select a sketched entity from the drawing window, the constraints applied to that entity are listed in the list box in the **Show Constraints** area. If you select another sketched entity, the constraints applied to the previously selected entity are removed and the constraints applied to the entity selected now are listed.

### Selected Objects

This radio button is used to list the constraints applied to more than one entity. You can select any number of entities by picking them from the graphics window or by creating a temporary rectangle around them by dragging the cursor. The constraints applied to all selected entities will be listed in the list box in the **Show Constraints** list box. You can select a constraint from this list box to highlight it in the drawing window.

### All In Active Sketch

This radio button is selected to list the constraints applied to all the sketched entities in the current sketch.

## Constraint Type Drop-down List

The **Constraint Type** drop-down list is used to select the type of constraints that are to be listed. By default, the **All** option is selected from this drop-down list. As a result, all constraints that are applied to the sketch are listed. However, you can select a particular type of constraint from this drop-down list. By doing so, you can ensure that only the specified type of constraint applied to the selected entity is highlighted. Below this drop-down list, there are two radio buttons and they are discussed next.

### Include

This radio button ensures that the specified types of constraints selected from the **Constraint Type** drop-down list are included for listing in the dialog box.

### Exclude

This radio button ensures that the specified type of constraints selected from the **Constraint Type** drop-down list are excluded for listing in the dialog box.

## Show Constraints Area

The options in this area are discussed next.

### Drop-down List

The drop-down list in the **Show Constraints** area allows you to specify whether you want to display the explicit constraints, the inferred coincident constraints, or both. Explicit constraints such as horizontal, vertical, midpoint, and so on are those constraints that are applied by the user while drawing the sketch, whereas, the Inferred coincident constraints such as coincident are those constraints that are applied automatically. For displaying

the explicit constraints, you need to select the **Explicit** option, whereas for the inferred constraints, you need to select the **Inferred** option. If you select the **Both** option from the drop-down list, both types of constraints will be listed in the dialog box.

### List Box

The constraints applied to the sketch are listed in this list box based on the specified selections. If you select a constraint from this list box, the sketched entities related to the constraints are highlighted in the graphics window. You can scroll the selection intent upward or downward over the constraints listed by choosing the **Step Up The List** or the **Step Down The List** buttons, respectively.

### Remove Highlighted

When you choose this button, the selected constraints in the list box are deleted.

### Remove Listed

Choosing this button deletes only the constraints that are listed in the list box. For example, if you have listed only the inferred constraints, they will be deleted and the explicit constraints will be retained. You can view them by listing the explicit constraints.

### Information

This button is used to open a new window, which provides the information about the constraints listed in the **Show/Remove Constraints** dialog box.

## Converting a Sketch Entity or Dimension into a Reference Entity or reference dimension

**Menu bar:** Tools > Constraints > Convert To/From Reference

**Toolbar:** Sketch Tools > Convert To/From Reference



The **Convert To/From Reference** tool in the **Sketch Tools** toolbar is used to convert or retain the reference property of a sketched entity or a dimension. Generally, reference elements are created for assigning the axis of revolution or for applying dimensions with reference to an entity. To convert any of these sketched entities or dimensions into a reference element, choose the **Convert To/From Reference** button from the **Sketch Tools** toolbar; the **Convert To/From Reference** dialog box will be displayed, as shown in Figure 3-47.

By default, the **Reference Curve or Dimension** radio button is selected and you are prompted to select the entity or dimension. Select one or more objects, or dimensions. Next, choose the **OK** button from the dialog box to convert the selected objects or dimensions into reference elements or dimensions.

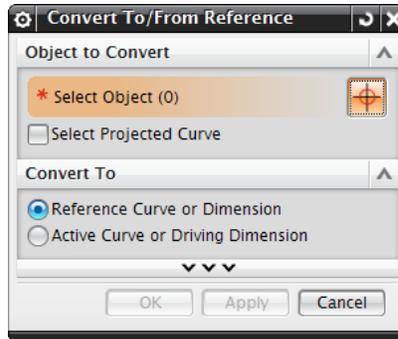


Figure 3-47 The *Convert To/From Reference* dialog box

To make the reference elements or dimensions active, choose the **Active Curve or Driving Dimension** radio button from the dialog box and select the reference elements or dimensions. Next, choose the **OK** button.



**Note**

You can also convert geometric entities into reference entities and vice-versa, without opening the **Convert To/From Reference** dialog box. Select the entities to be converted from the drawing window and then choose the **Convert To/From Reference** button from the **Sketch Tools** toolbar; the selected geometric entities will be converted into reference entities and vice-versa.

**TUTORIALS**

From this chapter onward, you will use Sketch constraints and parametric dimensions to complete the model.

**Tutorial 1**

In this tutorial, you will draw the profile of the model shown in Figure 3-48. The profile is shown in Figure 3-49. The profile should be symmetric about the origin. Also, you will use the parametric dimensions to complete the sketch. **(Expected time: 30 min)**

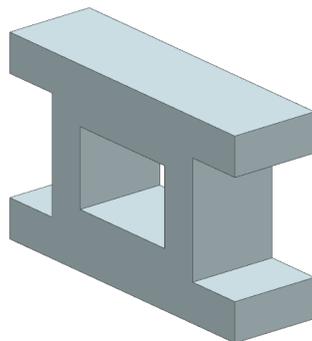


Figure 3-48 Model for Tutorial 1

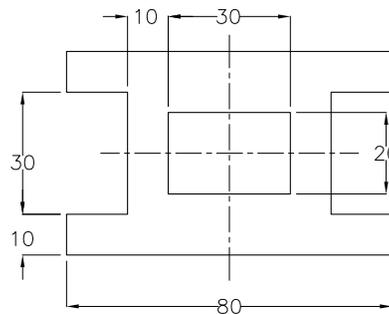


Figure 3-49 Sketch for Tutorial 1

The following steps are required to complete this tutorial:

- a. Start a new file and invoke the Sketch task environment.
- b. Draw the outer profile of the sketch using the **Profile** tool.
- c. Add the geometric constraints to the outer loop and modify its dimensional constraints.
- d. Draw a rectangle inside the outer loop using the **Rectangle** tool.
- e. Add dimensions to the rectangle to complete the sketch.
- f. Save the sketch and close the file.

### Starting a New File and Invoking the Sketch Task Environment

1. Start a new file by using the **Model** template.
2. Choose the **New** button from the **Standard** toolbar; the **New** dialog box is displayed. Next, select the **Model** template from the **Templates** rollout, and then enter **c03tut1** as the name of the document in the **Name** text box.
3. Choose the button on the right of the **Folder** text box; the **Choose Directory** dialog box is displayed. Next, browse to the *C:\NX 8.5\c03*, and then choose the **OK** button twice; the new file is started in the Modeling environment.
4. Invoke the Sketch task environment by using the XC-ZC plane as the sketching plane.



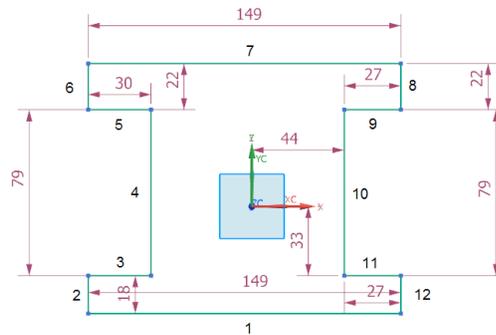
#### Note

*You can also start a sketch directly by choosing the **Sketch** button from the **Direct Sketch** toolbar available at the bottom of the drawing window, and then select the XC-ZC plane. In this case, you can invoke the sketch, dimension, and constraint tools from the **Direct Sketch** toolbar.*

### Drawing the Outer Loop and Adding Sketch Constraints

If the sketch consists of more than one closed loop, it is recommended that you draw the outer loop first and then add all the required Sketch constraints and dimensions to it. This makes it easier to draw and dimension the inner loops. Next, you need to draw the inner loop.

1. By default, the **Profile** tool from the **Sketch Tools** toolbar is chosen and you are prompted to select the first point of the line or press and drag the left mouse button to begin with the arc creation.
2. Draw the sketch around the origin following the sequence shown in Figure 3-50. The sketch is unsymmetrical at this stage. But, after adding the Sketch constraints and modifying the dimensions, it will become symmetrical. You can use the help lines to draw the sketch. For your reference, the sequence in which the lines need to be drawn in the sketch is indicated by numbers.



**Figure 3-50** The outer loop of the profile and the sequence in which lines have to be drawn



**Note**

The dimensions shown in Figure 3-50 are automatically applied while drawing the sketch. However, those dimensions are not driving dimensions. They will change when you drag or move a sketched entity. In order to make them driving dimensions, you need to modify them. You will learn to modify these dimensions later in this tutorial.

Next, you need to apply the geometric constraints and modify dimensions. But before you do that, it is recommended that you turn on the display of constraints, if it is not already on.

3. Choose the **Display Sketch Constraints** button from the **Sketch Tools** toolbar to view the constraints applied to the sketch. 
4. Next, choose the **Geometric Constraints** button from the **Sketch Tools** toolbar; the **Geometric Constraints** dialog box is displayed and you are prompted to select curves to create constraints. 
5. Choose the **Equal Length** button from the **Geometric Constraints** dialog box and then select lines 1 and 7. The symbol for the equal length constraint is displayed on both sketch members, indicating that this constraint is applied between the two selected entities. 
6. Similarly, apply the equal length constraint between lines 8 and 6, 6 and 2, 2 and 12, 12 and 8, 3 and 11, 9 and 5, and 10 and 4. Next, exit the dialog box.

Now, you need to make this sketch symmetric about horizontal and vertical datum axes.

7. Choose the **Make Symmetric** tool from the **Sketch Tools** toolbar; the **Make Symmetric** dialog box is displayed and you are prompted to select the first object to apply symmetry.
8. Select line 2; you are prompted to select the second object. Select the line 12; you are prompted to select the centerline.

9. Select the vertical axis as the centerline; line 2 and 12 are made symmetric about vertical axis. Next, choose the **Close** button from the **Make Symmetric** dialog box to close it.
10. Similarly, make the lines 1 and 7 symmetric about the horizontal axis. The sketch after applying constraints to all these entities is shown in Figure 3-51.

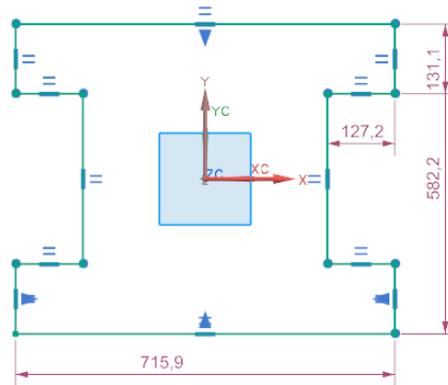


Figure 3-51 The outer profile after adding constraints

### Adding Dimensions to Sketch Members

Next, you need to add the dimensions to the sketch. As mentioned earlier, when you add dimensions to the sketch and modify their values, the entity is forced by the specified dimension value to maintain this modification. Before you start dimensioning the sketch, you need to modify some dimension display options.

1. Choose **Task > Sketch Style** from the menu bar; the **Sketch Style** dialog box is displayed. Select the **Value** option from the **Dimension Label** drop-down list. Next, choose the **OK** button to exit this dialog box.
2. Choose the **Inferred Dimensions** button from the **Sketch Tools** toolbar; you are prompted to select an object to dimension or the dimension to edit. 
3. Select the line 1; the current dimension of the line 1 is attached to the cursor. Now, you need to place the dimension at the required location. Click the left mouse button below the line 1 to place the dimension, refer to Figure 3-52. As you place the dimension, an edit box is displayed. Enter **80** in the edit box and press ENTER. Next, choose the **Fit** button from the **View** toolbar.



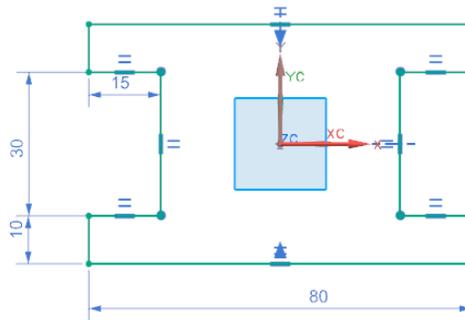
**Tip:** Instead of adding the dimensions to the sketch, you can also modify the automatically applied dimensions by double-clicking on them. On doing so, an edit box will be displayed. Enter the required value in the edit box and press ENTER; the dimensions will be locked and displayed in blue color.

4. Select line 2 and place the dimensions on the left of the sketch. Enter **10** in the edit box displayed and press ENTER.

5. Select line 4 and place the dimension on the left of the sketch. Next, modify the dimension value to **30** and press ENTER.
6. Select line 5 and place the dimension below the line. Next, modify the dimension value to **15** and press ENTER.

When you place the dimensions, they generally scatter all around the sketch. It is a good practice to arrange them properly.

7. Exit the **Inferred Dimensions** tool by pressing the ESC key twice and then drag the dimensions to place them properly around the sketch, refer to Figure 3-52.

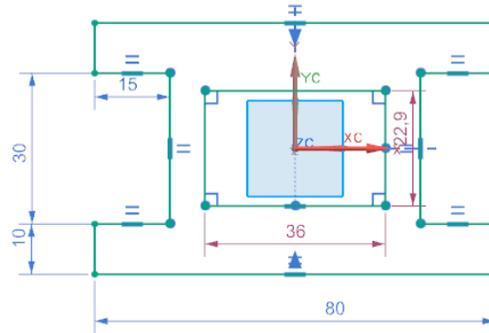


*Figure 3-52 The outer profile after adding the required dimensions*

### Drawing the Inner Loop and Adding the Dimensions

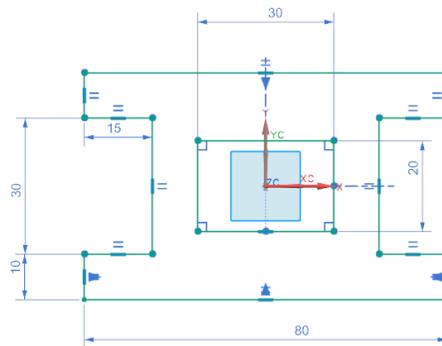
Next, you need to draw the rectangular profile inside the outer loop.

1. Choose the **Rectangle** button from the **Sketch Tools** toolbar; the **Rectangle** dialog box is displayed.
2. Choose the **From Center** button from the **Rectangle** dialog box; you are prompted to specify the center of the rectangle.
3. Select origin as the center of the rectangle; you are prompted to specify the second point of the rectangle.
4. Move the cursor horizontally toward right and click to specify the second point; you are prompted to select the third point to create the rectangle.
5. Move the cursor vertically upward and click to specify the third point; the rectangle is created, as shown in Figure 3-53. Press ESC to exit the tool.



*Figure 3-53 The sketch after drawing the inner loop and turning on the display of constraints*

6. Double-click on the horizontal dimension of the rectangle; a dynamic edit box is displayed. Modify the value of the dimension to 30 and press ENTER. Next, drag and place the dimension above the sketch.
7. Similarly, double-click on the vertical dimension of the rectangle and modify the value of the dimension to 20. Next, press ESC. Next, place the dimension on the right of the sketch, refer to Figure 3-54.



*Figure 3-54 The resulting sketch after adding the required dimensions and constraints*

### **Saving the File**

1. Exit the Sketch task environment by choosing the **Finish Sketch** button from the **Sketch** toolbar. Next, choose the **Save** button from the **Standard** toolbar to save the sketch. Note that the name and location of the document had already been specified when you started the new file.
2. Choose **File > Close > All Parts** from the menu bar to close the file.

## Tutorial 2

In this tutorial, you will create the profile for the model shown in Figure 3-55. The profile is shown in Figure 3-56. You will use the geometric and dimensional constraints to complete this sketch. **(Expected time: 30 min)**

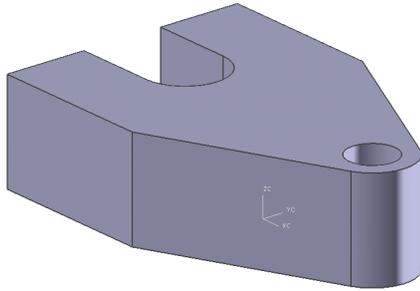


Figure 3-55 Model for Tutorial 2

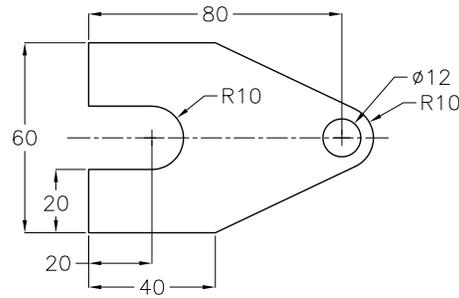


Figure 3-56 Sketch for Tutorial 2

The following steps are required to complete this tutorial:

- Start a new file in NX and invoke the Sketch task environment.
- Draw the sketch using the **Profile** tool.
- Add the geometric and dimensional constraints to the sketch.
- Save the sketch and close the file.

### Starting a New File in NX and Invoking the Sketch Task Environment

- Start a new file with the name *c03tut2.prt* using the **Model** template and specify its location as *C:\NX 8.5\c03*.
- Turn on the display of WCS by choosing the **Display WCS** button from the **WCS** drop-down in the **Utility** toolbar, if it is not already displayed.
- Invoke the Sketch task environment by using the XC-YC plane as the sketching plane.

### Drawing the Sketch

- By default, the **Profile** button is chosen in the **Sketch Tools** toolbar. Draw the outer profile of the sketch, refer to Figure 3-57. You can draw the first line with exact dimension and then draw the remaining sketched entities with dimension values close to the required dimension values. Note that the start point of the line 1 is at the origin.



#### Note

In Figure 3-57, the temporary dimensions are hidden for the clarity of the view. To hide the temporary dimensions, choose the **Show and Hide** button from the **Utility** toolbar; the **Show and Hide** dialog box is displayed. In this dialog box, select the minus sign (-) from the **Drafting Annotations** row; all the temporary dimensions are hidden.

Note that after drawing the first line, you may need to modify the drawing display area by using the **Fit** button from the **View** toolbar.

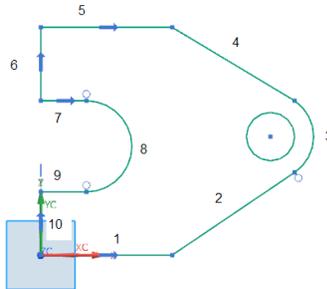
- Next, choose the **Circle** button from the **Sketch Tools** toolbar. Move the cursor over the arc that is numbered 3 in Figure 3-57; the center point of the arc is highlighted.



**Note**

*If the center of the arc is not highlighted, choose the **Arc Center** button from the **Selection Bar**.*

- After the center point gets highlighted, move the cursor over it and press the left mouse button to specify the center point of the circle. Now, move the cursor away from the center point and specify the diameter of the circle by clicking the left mouse button or by entering the diameter value in the diameter input box. The circle is created, refer to Figure 3-57.



*Figure 3-57 The sequence to be followed for drawing the sketch*

### Adding Constraints to the Sketch

- Choose the **Geometric Constraints** button from the **Sketch Tools** toolbar; the **Geometric Constraints** dialog box is displayed.
- Choose the **Horizontal** button from the **Geometric Constraints** dialog box and select line 1 to apply the horizontal constraint, if this constraint has not already been applied. Similarly, apply the **Horizontal** constraint to lines 5, 7, and 9.
- Similarly, apply the **Vertical** constraint to lines 6 and 10, if this constraint has not already been applied. 
- Choose the **Concentric** button from the **Geometric Constraints** dialog box. Next, select the circle and arc 3 from the drawing area to apply the **Concentric** constraint. 
- Choose the **Equal Length** button from the **Geometric Constraints** dialog box and select lines 1 and 5 to apply the **Equal Length** constraint. 
- Similarly, apply the **Equal Length** constraint between lines 2 and 4, 6 and 10, and 7 and 9.

The sketch after applying all constraints is shown in Figure 3-58.

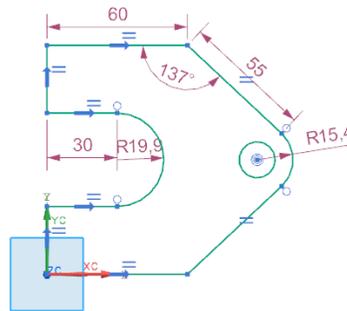


Figure 3-58 The sketch displayed after adding the required constraints

### Modifying Dimensions of the Sketch

Next, you need to modify the dimensions of the sketch.

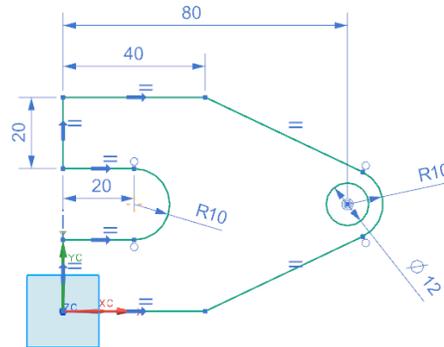
1. Double-click on the dimension of line 5; a dynamic edit box is displayed. Modify the value of the dimension to 40 and press ENTER. Next, drag and place the dimension above the line, refer to Figure 3-58.
2. Double-click on the dimension of line 7; a dynamic edit box is displayed. Modify the value of the dimension to 20 and press ENTER. Next, drag and place the dimension below the line, refer to Figure 3-58.
3. Double-click on the dimension of the arc 8; a dynamic edit box is displayed. Modify the value of the dimension to 10 and press ENTER.
4. Double-click on the dimension of the arc 3; a dynamic edit box is displayed. Modify the value of the dimension to 10 and press ENTER.

Next, you need to add some dimensions which are not added automatically.

5. Choose the **Inferred Dimensions** button from the **Sketch Tools** toolbar.
6. Select line 6 and the center point of the circle. Place the dimension above the sketch and modify the dimension value to **80**. Press the ENTER key.
7. Select line 6 and place the dimension on the left of the sketch. Next, modify the dimension value to 20 and press ENTER.
8. Select the circle and place the dimension, refer to Figure 3-58. Next, modify the dimension value to 12 and press ENTER.
9. Choose the **Fit** button from the **View** toolbar.



The final sketch after adding the required dimensions is shown in Figure 3-59.



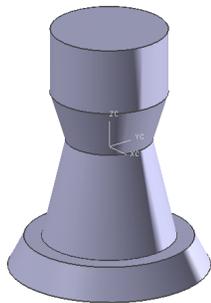
*Figure 3-59* The final sketch after adding the required dimensions

### Saving the File

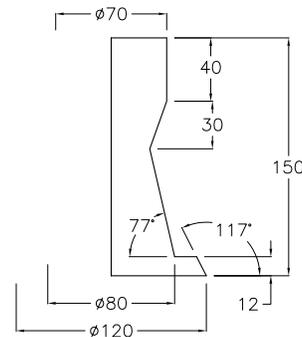
1. Choose the **Save** button from the **Standard** toolbar to save the sketch. Note that the name and the location of the document had already been specified when you started the new file.
2. Exit the Sketch task environment and then choose **File > Close > All Parts** from the menu bar to close the file.

## Tutorial 3

In this tutorial, you will create the profile for the revolved model shown in Figure 3-60. The profile is shown in Figure 3-61. You will use the geometric and dimensional constraints to complete this sketch.  
(Expected time: 30 min)



*Figure 3-60* Model for Tutorial 3



*Figure 3-61* Sketch for Tutorial 3

The following steps are required to complete this tutorial:

- Start a new file in NX and invoke the Sketch task environment.
- Draw the required profile of the sketch using the **Profile** tool.
- Add the geometric and dimensional constraints to the sketch.
- Save the sketch and close the file.

### Starting a New File in NX and Invoking the Sketch Task Environment

- Start a new file with the name *c03tut3.prt* using the **Model** template and specify its location as *C:\NX 8.5\c03*.
- Turn on the display of WCS by choosing the **Display WCS** button from the **WCS** drop-down in the **Utility** toolbar.
- Invoke the Sketch task environment using the XC-ZC plane as the sketching plane.

### Drawing the Sketch

It is recommended that you create the first sketch member with the exact measurement by entering the value in the edit box displayed. After creating the first sketch member, you can create the other sketch members by taking the first entity as the reference. After creating the entire sketch, you can modify the values by using the dimensions tool.

- By default, the **Profile** button is chosen in the **Sketch Tools** toolbar and you are prompted to specify the first point of the line. Specify the start point of the line at the origin. Next, move the cursor horizontally toward the right and enter **60** in the **Length** edit box and **0** in the **Angle** edit box. Next, press ENTER.
- Follow the sequence given in Figure 3-62 for drawing the sketch. Draw the other entities of the sketch. For better understanding, the sketch has been numbered and temporary dimensions have been hidden.

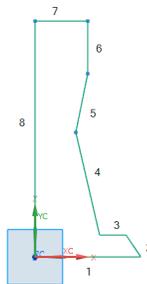


Figure 3-62 The sequence for drawing the profile

### Adding Geometric Constraints to the Sketch

After completing the sketch, you need to apply constraints to it.

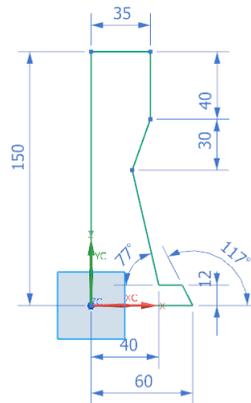


6. Select line 2 and then line 1; an angular dimension is attached to the cursor. Move the cursor outside the sketch toward the right and click the left mouse button to place the dimension. Next, modify the dimension value to 117 and press ENTER, refer to Figure 3-64.
7. Select lines 3 and 4; an angular dimension is attached to the cursor. Move the cursor inside the sketch and place the dimension. Next, modify the dimension value to 77 and press ENTER, refer to Figure 3-64.
8. Select the lower endpoint of line 4 and then select line 8; the dimension value is attached to the cursor. Place the dimension value below the sketch and then modify this value to 40. Next, press ENTER.
9. Select line 2 and place the dimension on the right of the line. Next, modify the dimension value to 12 and press ENTER, refer to Figure 3-64. Press the ESC key twice.
10. Choose the **Fit** button from the **View** toolbar. The resulting sketch after adding all dimensions is shown in Figure 3-64.



### Note

In Figure 3-64, the display of constraints is turned off to get a better display of dimensions. You can turn off the display of constraints by choosing the **Display Sketch Constraints** button. Note that it is a toggle button.



**Figure 3-64** The completed sketch displayed after adding the required constraints and dimensions

### Saving the File

1. Choose the **Save** button from the **Standard** toolbar to save the sketch. Note that the name and location of the document has already been specified when you started the new file.
2. Exit the Sketch task environment and choose **File > Close > All Parts** from the menu bar to close the file.

## Self-Evaluation Test

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

1. In NX, you can add all types of geometric constraints by using the **Geometric Constraints** tool in the **Sketch Tools** toolbar. (T/F)
2. The **Auto Constrain** tool allows you to apply all possible geometric constraints automatically to the entire sketch. (T/F)
3. The **Inferred Dimensions** tool in the **Sketch Tools** toolbar is used to add all possible dimension types. (T/F)
4. In NX, the **Diameter Dimension** tool is used to add the diameter dimension to the sketch members. (T/F)
5. The \_\_\_\_\_ constraint is used to force two curves to share the same location of the center points.
6. The \_\_\_\_\_ tool is used to dimension the radius of an arc.
7. The \_\_\_\_\_ tool is used to measure the distance between two objects.
8. The \_\_\_\_\_ tool is used to animate a fully constrained sketch.
9. The \_\_\_\_\_ option in the **Measure Distance** dialog box is used to measure the distance between the objects with respect to a view point.
10. The \_\_\_\_\_ tool is used to show all constraints applied to a sketch.

## Review Questions

Answer the following questions:

1. Which one of the following tools is used to apply geometric constraints to a sketch?
 

(a) <b>Geometric Constraints</b>	(b) <b>Automatic Constraints</b>
(c) <b>Inferred Dimensions</b>	(d) None of these
2. Which one of the following tools is used to add a radial dimension to a sketch?
 

(a) <b>Radius</b>	(b) <b>Automatic Constraints</b>
(c) <b>Inferred Dimensions</b>	(d) None of these

3. Which one of the following tools is used to make the endpoints of selected objects coincident?
- (a) **Coincident** (b) **Concentric**  
(c) **Horizontal** (d) None of these
4. Which one of the following tools is used to apply the constant length constraint between sketch members?
- (a) **Equal Length** (b) **Automatic Constraints**  
(c) **Vertical** (d) None of these
5. Which one of the following tools is used to apply a parallel dimension to a sketch member?
- (a) **Parallel** (b) **Automatic Constraints**  
(c) **Inferred Dimensions** (d) None of these
6. Which one of the following tools is used to convert a sketch member into a reference element?
- (a) **Convert To/From Reference** (b) **Automatic Constraints**  
(c) **Geometric Constraints** (d) None of these
7. While measuring an angular dimension, you can display a major or minor dimension. (T/F)
8. The **Sketch Tools** toolbar contains all tools required to draw a sketch. (T/F)
9. The **Sketch** tool in the **Feature** toolbar is used to enter the Sketch task environment. (T/F)
10. The **Finish Sketch** tool in the **Sketch** toolbar is used to exit the Sketch task environment. (T/F)

## Exercises

### Exercise 1

Draw the base sketch of the model shown in Figure 3-65. The sketch to be drawn is shown in Figure 3-66. Use the geometric and dimensional constraints to complete this sketch.

(Expected time: 15 min)

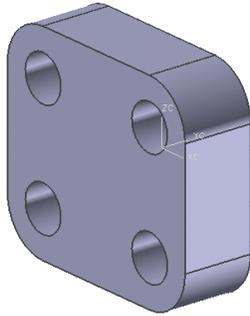


Figure 3-65 Model for Exercise 1

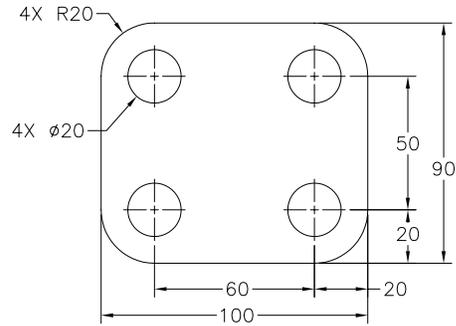


Figure 3-66 Sketch for Exercise 1

## Exercise 2

Draw the base sketch of the model shown in Figure 3-67. The sketch to be drawn is shown in Figure 3-68. Use the geometric and dimensional constraints to complete this sketch.

(Expected time: 15 min)

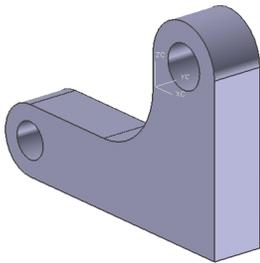


Figure 3-67 Model for Exercise 2

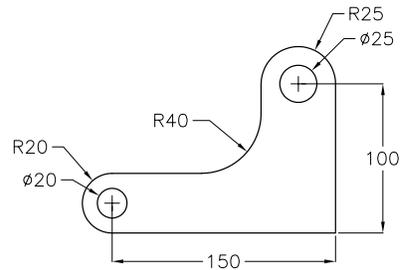


Figure 3-68 Sketch for Exercise 2

### Answers to Self-Evaluation Test

1. T, 2. T, 3. T, 4. T, 5. Concentric, 6. Radius, 7. Measure Distance, 8. Animate Dimension, 9. Screen Distance, 10. Display Sketch Constraints