



Chapter 3

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 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.*
- *Understand different types of dimensional constraints.*
- *Measure the distance value between objects in a sketch.*
- *Measure the angle between entities.*

CONSTRAINING SKETCHES

In the previous chapter, you learned to draw sketches in the Sketcher environment of NX. In this chapter, you will 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 them are automatically applied while drawing. 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.

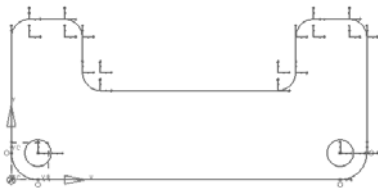


Figure 3-1 An under-constrained sketch

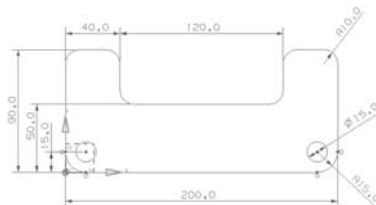


Figure 3-2 A fully-constrained sketch

Over-Constrain

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

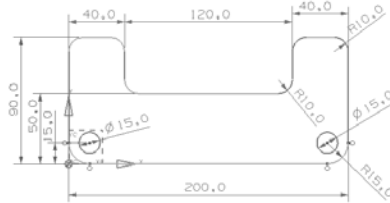


Figure 3-3 An over-constrained sketch

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 needed to remove one or more constraints applied.

DEGREE OF FREEDOM ARROWS

The degree of freedom arrows 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 constraint tool (geometrical or dimensional). These tools are discussed later in this chapter.

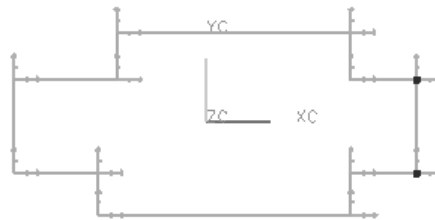


Figure 3-4 The degree of freedom arrows displayed on points

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.

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 Sketcher 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. This is discussed next.

Applying Additional Constraints Individually

Menu: Insert > Constraints
Toolbar: Sketch Tools > Constraints



In NX, you can apply additional constraints manually by using the **Constraints** tool in the **Sketch Tools** toolbar. To apply constraints, invoke the **Constraints** tool and then select the entities to which you want to add constraints; the constraints that can be applied to the selected sketched entities will be displayed

in the **Constraints** dialog box, as shown in Figure 3-5.

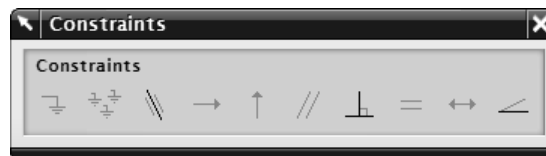


Figure 3-5 The Constraints dialog box

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

Fixed Constraint



The **Fixed** constraint is used to fix some of the characteristics of the geometry. These characteristics 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.

To apply this constraint, select the entity and choose the **Fixed** button from the **Constraints** dialog box; the selected entity will be fixed.

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, select the entity and choose the **Fully Fixed** button from the **Constraints** dialog box; 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, select the entity and then choose the **Horizontal** button from the **Constraints** dialog box; 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 being 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 **Constraints** tool and then select the keypoints of the sketched entities. Next, choose the **Coincident** button from the **Constraints** dialog box. Note that if you select the sketched entities other than the keypoints, this constraint will not be available in the **Constraints** dialog box. Figure 3-6 shows the endpoints of the two lines selected to be made coincident and Figure 3-7 shows the lines after applying constraint.

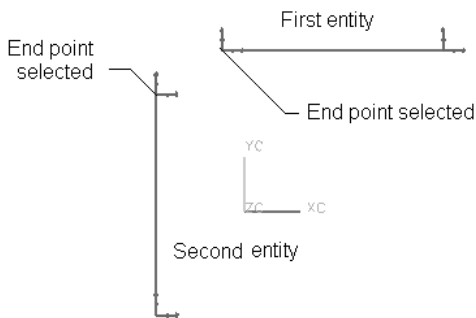


Figure 3-6 The endpoints of the first and second entities selected

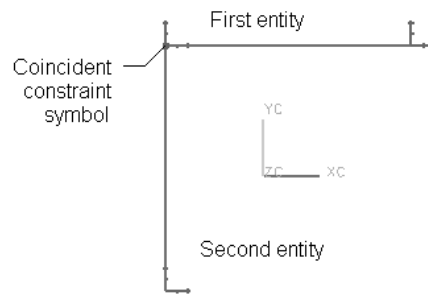


Figure 3-7 The resulting sketch after applying the **Coincident** constraint

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 **Constraints** tool and select a keypoint, such as the endpoint or the center point. Next, select a curve; the **Point On Curve** constraint button will be displayed in the **Constraints** dialog box. Choose this button; the point will be placed on

the curve. Figure 3-8 shows a sketch before applying this constraint and Figure 3-9 shows a sketch after applying this constraint.

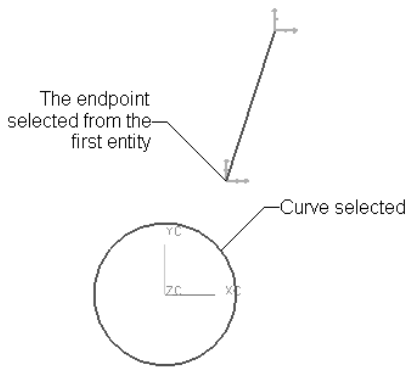


Figure 3-8 The reference elements selected from the entity to apply the **Point On Curve** constraint

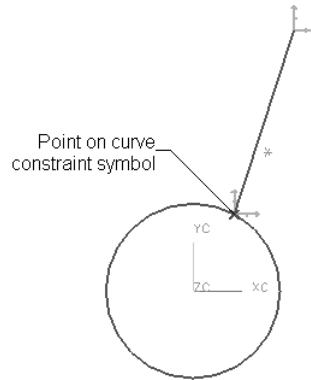


Figure 3-9 The resulting sketch after applying the **Point On Curve** constraint

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 **Constraints** tool and then select a set of line segments or ellipses; the **Parallel** constraint button will be displayed in the **Constraints** dialog box. Choose this button; the selected line segments or the axes of the ellipse will become parallel to each other. Figure 3-10 shows two line segments before and after applying this constraint.

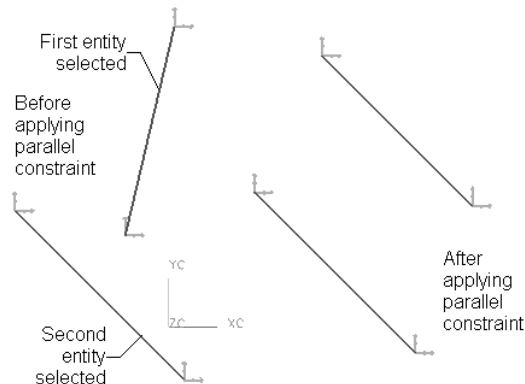


Figure 3-10 Applying the **Parallel** constraint

Perpendicular Constraint



The **Perpendicular** constraint forces a set of selected line segments or ellipse axes to become normal to each other. Figure 3-11 shows two line segments before and after applying this constraint.

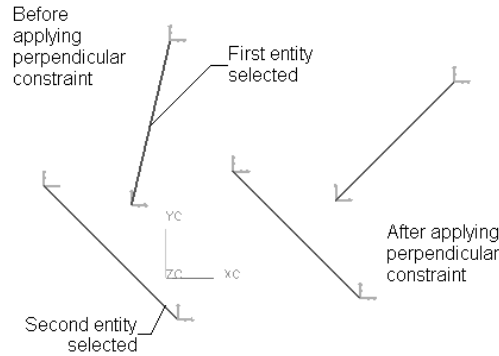


Figure 3-11 Sketch before and after applying the **Perpendicular** constraint

Tangent Constraint



The **Tangent** constraint forces the selected line segment or curve to become tangent to another curve. To apply this constraint, invoke the **Constraints** tool and then select a line and a curve or select two curves; the **Tangent** constraint button will be displayed in the **Constraints** dialog box. Choose this button; the selected sketched line or curve will become tangent to the other curve. Figures 3-12 and 3-13 show the use of the **Tangent** constraint.

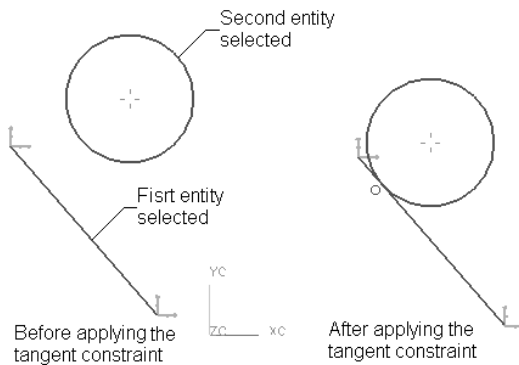


Figure 3-12 Sketch before and after applying the **Tangent** constraint

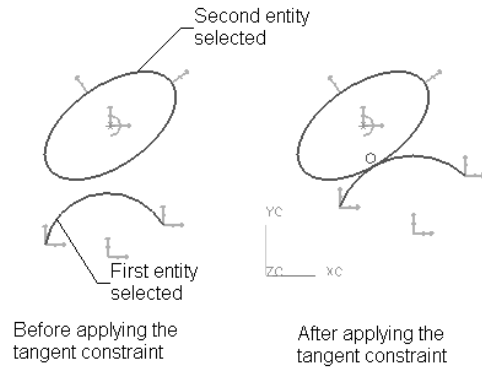


Figure 3-13 Sketch before and after applying the **Tangent** constraint



Note

By default, symbols of all constraints are not displayed in the sketch. You can turn on the display of constraints using the **Show All 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 **Constraints** tool and then select the line segments that you want to make equal in length; the **Equal Length** constraint button will be displayed in the **Constraints** dialog box. Choose this button; the selected line segments will become equal in length.

Equal Radius Constraint

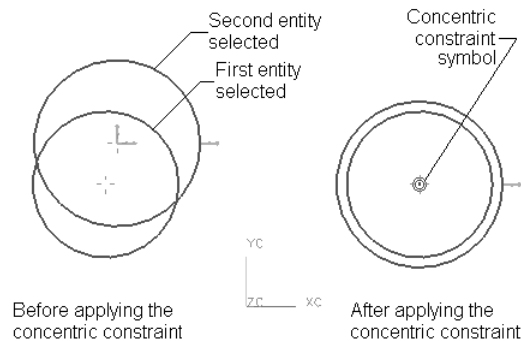


The **Equal Radius** constraint forces the selected arcs or circles to become equal in radius. To apply this constraint, invoke the **Constraints** tool and then select the arcs or circles that you want to make equal in radii; the **Equal Radius** constraint button will be displayed in the **Constraints** dialog box. Choose this button; the selected arcs or circles will be 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 3-14 shows two circles before and after adding this constraint.



*Figure 3-14 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 button is available only when you select a control point of a spline along with a line, an arc, or a spline segment. This constraint will force the slope of the spline at the selected control point to be equal to the slope of the selected line, arc, or spline segment.

Uniform Scale Constraint



The **Uniform Scale** constraint button is displayed only when you select a spline. This constraint ensures that if you modify the distance between the endpoints of the splines, the entire spline will be scaled uniformly.

Non-Uniform Scale Constraint



The **Non-Uniform Scale** constraint button is displayed only when you select a spline. This constraint ensures that if you modify the distance between the endpoints of splines, it will be scaled non-uniformly and appears to stretch.

Applying Automatic Constraints to a Sketch

Menu: 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 3-15. The options in this dialog box are discussed next.

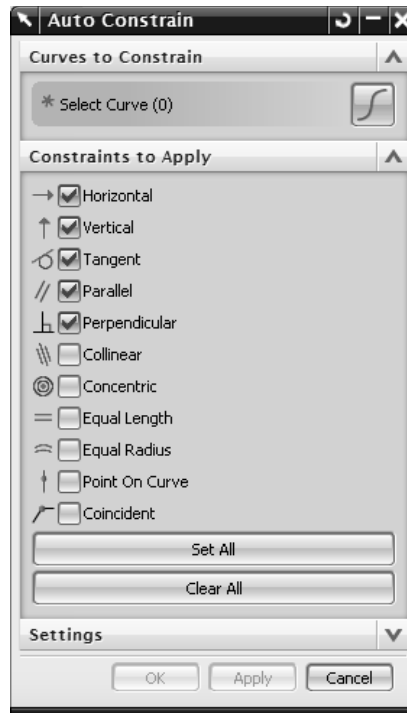


Figure 3-15 The Auto Constrain dialog box

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. These options 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: Tools > Constraints > Inferred Constraints
Toolbar: Sketch Tools > Inferred Constraints



As mentioned earlier, some of the constraints are automatically applied to the sketched entities while they are being sketched. These settings are controlled by the **Inferred Constraints** tool. When you invoke this tool, the **Inferred Constraints** dialog box will be displayed, as shown in Figure 3-16.

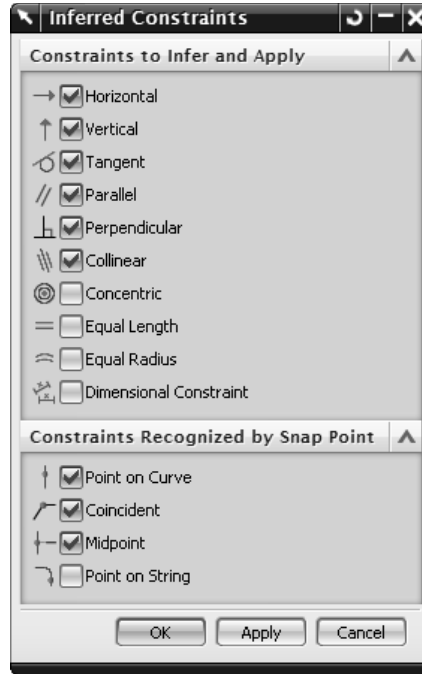


Figure 3-16 The *Inferred Constraints* dialog box

This dialog box provides you the check boxes of the main constraints that are available in NX. You can select the check boxes of the constraints that should be applied to the sketch while sketching.

If you select the **Dimensional Constraint** check box, then the dimensions of the entities created by entering the values inside the input boxes will be displayed.

Showing All Constraints in a Sketch

Menu: Tools > Constraints > Show All Constraints
Toolbar: Sketch Tools > Show All 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 using the **Show All 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.

Turning off the Display of All Constraints in a Sketch

Menu: Tools > Constraints > Show No Constraints
Toolbar: Sketch Tools > Show No Constraints (*Customize to Add*)



NX also allows you to turn off the display of all constraints in the sketch. This can be done by using the **Show No Constraints** tool. This tool is a toggle with the **Show All Constraints** tool. Note that with this tool turned on, you can continue working with the other sketching or constraining tools. However, if this tool is turned on and you apply constraints to the sketched entities, the constraints will not be displayed.

Showing/Removing Constraints

Menu: 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 3-17. The options in this dialog box are discussed next.

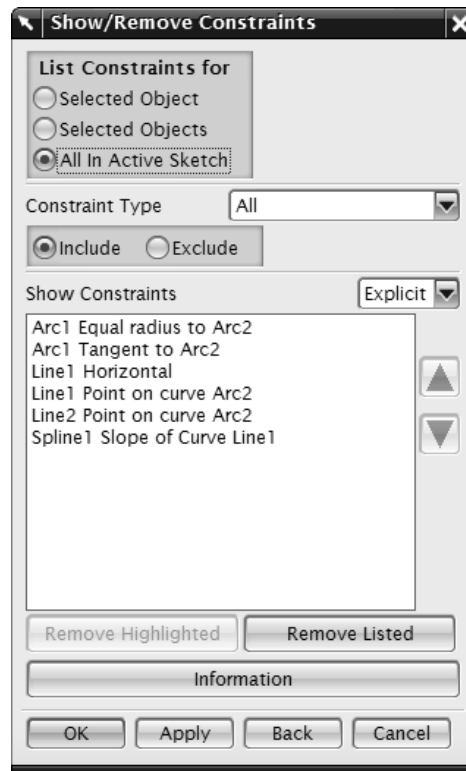


Figure 3-17 The *Show/Remove Constraints* dialog box

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** area. 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.



Tip: To select multiple constraints from the list box, press the **CTRL** key and select the constraints one by one.

Tip: Move the cursor over a constraint; all entities to which that particular constraint is applied will be highlighted. You can also highlight the constraints that are applied to the active sketched entities. To do so, select the **Sketch Constraint** option from the **Type Filter** drop-down list and then press **CTRL+A** keys; all constraints applied will be selected.

*In order to select the sketched entities, make sure the selection filter is set to **No Selection Filter**.*

Converting a Sketch Entity into a Reference Entity

Menu: 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. 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 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 3-18.

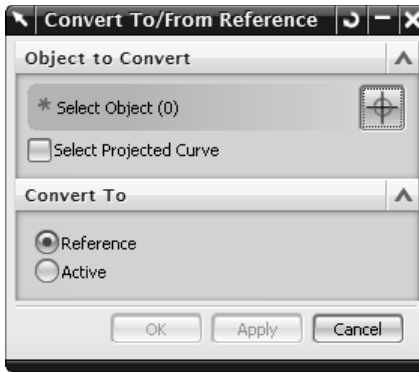


Figure 3-18 The **Convert To/From Reference** dialog box

By default, the **Reference** radio button is selected and you are prompted to select the reference entities. Select one or more objects, choose the **OK** button from the same dialog box to convert the selected objects into reference elements.

To convert the reference element into an active sketch, choose the **Active** radio button from the dialog box and select one or more reference elements. Choose the **OK** button.



Note

You can also convert dimension constraints into reference elements. When dimension constraints are converted into reference element, they are displayed in gray.

*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.*

DIMENSIONAL CONSTRAINTS

After creating the sketch, you need to apply different types of dimensions (dimensional constraints) to it. Various types of dimensions in NX are:

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

You can apply the dimensions listed above by using their respective tools or by using the **Inferred Dimensions** tool from the **Sketch Tools** toolbar. NX is a parametric software and so you can modify the dimension created at any time by entering the Sketcher environment. The methods for applying these dimensions are discussed next.

Applying Horizontal Dimensions

Menu: Insert > Dimensions > Horizontal
Toolbar: Sketch Tools > Inferred Dimensions > Horizontal



Horizontal

The **Horizontal** 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 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 ENTER. Figure 3-19 shows the horizontal dimensioning of lines.

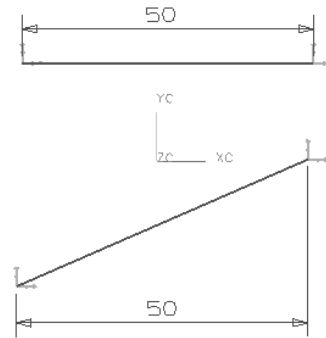


Figure 3-19 The horizontal dimension created for a horizontal line and an inclined line

Applying Vertical Dimensions

Menu: Insert > Dimensions > Vertical
Toolbar: Sketch Tools > Inferred Dimensions > Vertical



Vertical

The **Vertical** tool is used to apply a vertical dimension between any two points. Even if you select a linear object with a slant angle, the dimension will always be applied vertically between the endpoints of the selected object. To apply the vertical dimension to an object, choose **Inferred Dimensions > Vertical** 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 left or right 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 ENTER. Figure 3-20 shows the vertical dimensioning of lines.

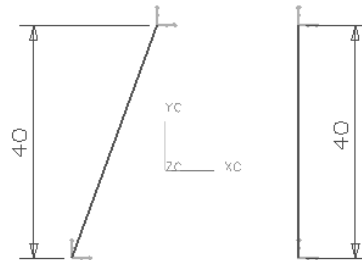


Figure 3-20 The vertical dimension created for a vertical line and an inclined line



Note

In the figures, some of the dimension properties such as decimal places and label have been modified for better display of dimensions. To modify the dimension label, choose **Sketch > 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: Insert > Dimensions > Parallel
Toolbar: Sketch Tools > Inferred Dimensions > Parallel



The **Parallel** 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** 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 ENTER. Figures 3-21 and 3-22 show the parallel dimension applied to the sketches.

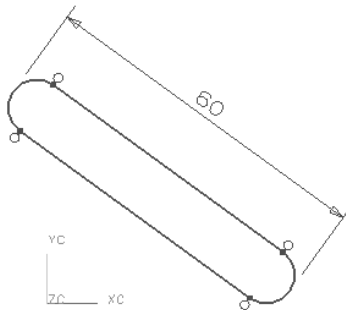


Figure 3-21 The parallel dimension applied to a sketch

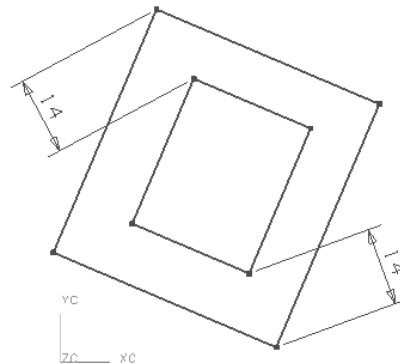


Figure 3-22 The parallel dimension applied to a sketch

Applying Perpendicular Dimensions

Menu: Insert > Dimensions > Perpendicular
Toolbar: Sketch Tools > Inferred Dimensions > Perpendicular



The **Perpendicular** 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** 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 ENTER. Figure 3-23 shows the perpendicular dimension applied to a sketch.

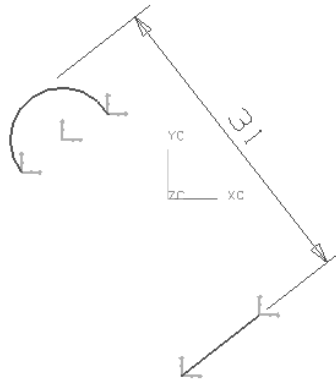


Figure 3-23 The perpendicular dimension applied between the objects

Applying Angular Dimensions

Menu: Insert > Dimensions > Angular
Toolbar: Sketch Tools > Inferred Dimensions > Angular



The **Angular** 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** 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 angle dimension needs to be applied and then place the dimension. Figure 3-24 shows different types of angular dimensions applied to a sketch.

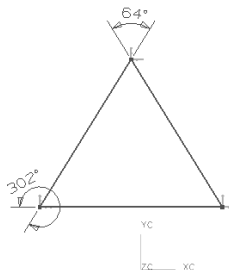


Figure 3-24 Angular dimensions applied between the objects

Applying Diameter Dimensions

Menu: Insert > Dimensions > Diameter
Toolbar: Sketch Tools > Inferred Dimensions > Diameter



The **Diameter** tool is used to apply the diameter dimension to an arc or a circle. Generally, diameter dimensions are applied to circles. To apply the diameter dimension, choose **Inferred Dimensions > Diameter** 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 ENTER. Figure 3-25 shows the diameter dimension applied to a circle and Figure 3-26 shows the diameter dimension applied to an arc.

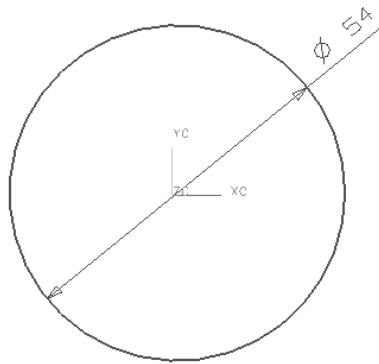


Figure 3-25 The diameter dimension applied to a circle

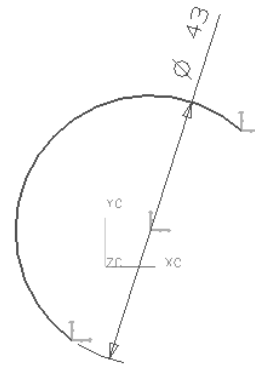


Figure 3-26 The diameter dimension applied to an arc

Applying Radius Dimensions

Menu: Insert > Dimensions > Radius
Toolbar: Sketch Tools > Inferred Dimensions > Radius



The **Radius** 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** 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 ENTER. Figure 3-27 shows the radius dimension applied to a circle and Figure 3-28 shows the radius dimension applied to an arc.

Applying Perimeter Dimensions

Menu: Insert > Dimensions > Perimeter
Toolbar: Sketch Tools > Inferred Dimensions > Perimeter

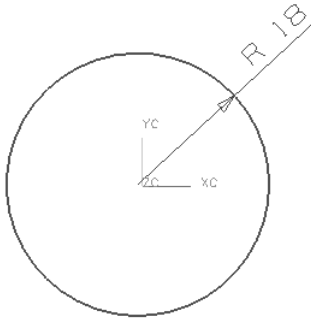


Figure 3-27 The radius dimension applied to a circle

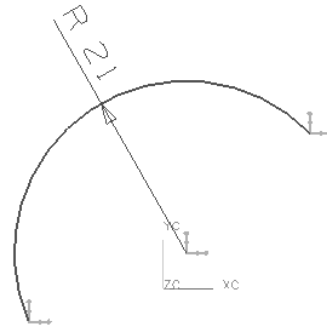


Figure 3-28 The radius dimension applied to an arc



The **Perimeter** 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** 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. However, you can change the dimension value of the object uniformly. To do so, choose **Edit > Sketch Parameters** from the menu bar; the **Sketch Parameters** dialog box will be displayed. Select the required dimension from the list area of this dialog box; the value of the selected dimension will be displayed in the **Current Expression** edit box. In this edit box, you can enter a new dimension value. You can also dynamically change the dimension value by moving the slider available below this edit box.

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.

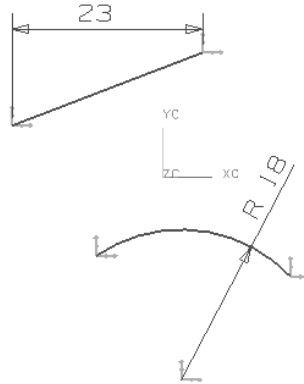
Applying Dimensions by Using the Inferred Dimensions

Menu:	Insert > Dimensions > Inferred Dimensions
Toolbar:	Sketch Tools > 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 3-29 shows the radial and linear dimensions created.



*Figure 3-29 The radial and linear dimensions created by using the **Inferred Dimensions** tool*

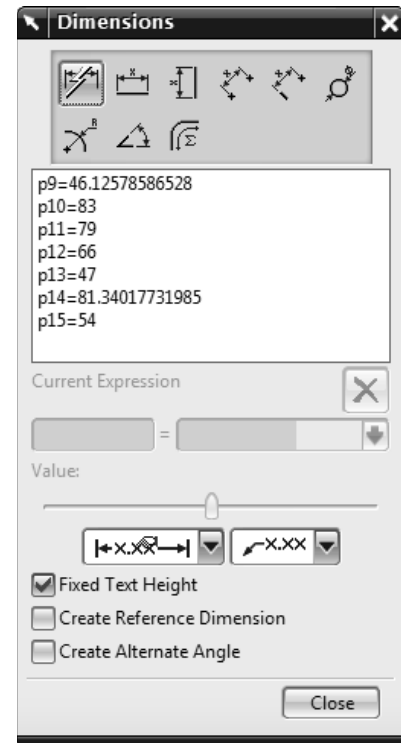
Editing the Dimension Value and Other Parameters



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

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 3-30. 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 ENTER key.

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



*Figure 3-30 The modified **Dimensions** dialog box*

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: 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 3-31, 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 3-32. Figure 3-33 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.

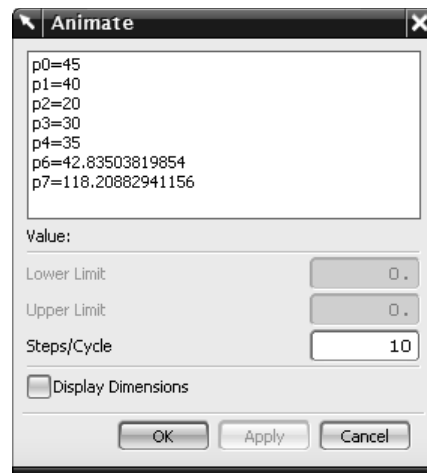


Figure 3-31 The *Animate* dialog box

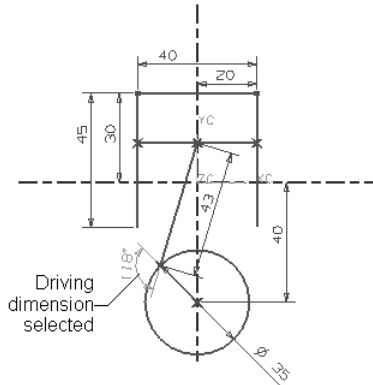


Figure 3-32 Driving dimension selected from the sketch

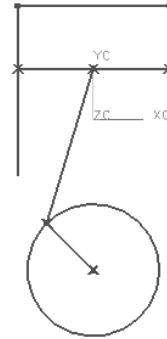


Figure 3-33 The sketch being animated

MEASURING THE DISTANCE VALUE BETWEEN OBJECTS IN A SKETCH

Menu: 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 3-34. 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.

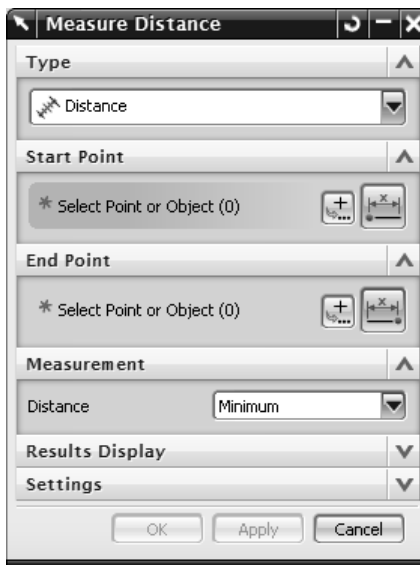


Figure 3-34 The Measure Distance dialog box

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. Also, 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 3-35. 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 3-36.

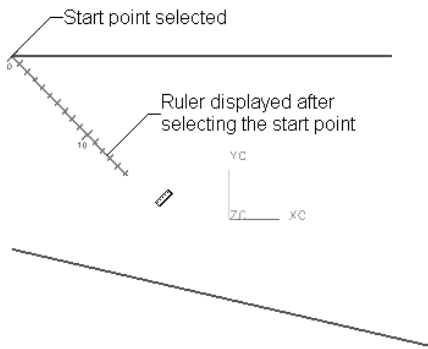


Figure 3-35 The ruler displayed after selecting the start point

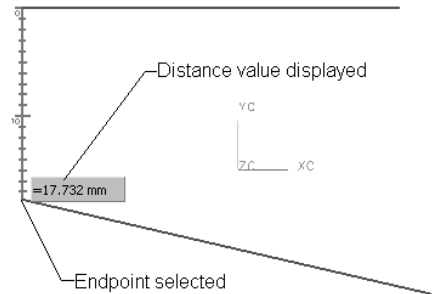


Figure 3-36 The distance value displayed in the display box

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 Constructor** 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 3-37, 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 3-38 shows 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 3-39, 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 3-40. Note that if you measure the distance between two same objects by changing the orientation, the distance value will also be changed.

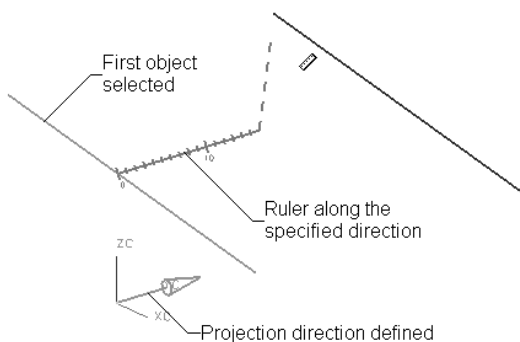


Figure 3-37 The ruler locked to the specified projection direction

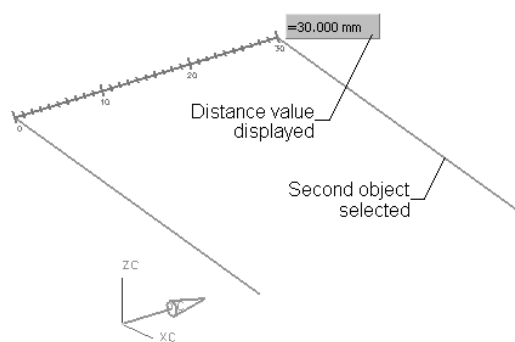


Figure 3-38 The distance value displayed in the display box

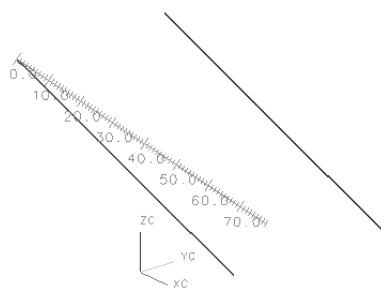


Figure 3-39 The ruler displayed after selecting the first object

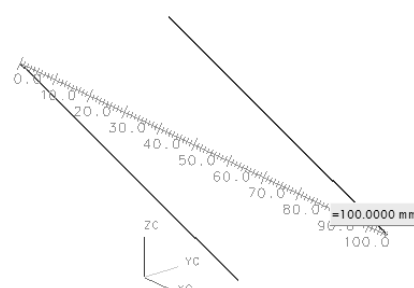


Figure 3-40 The distance value displayed on the display box

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 3-41 and 3-42. 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.

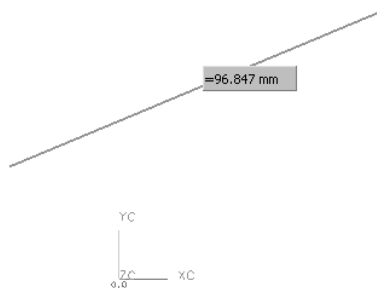


Figure 3-41 The length measurement displayed for a line

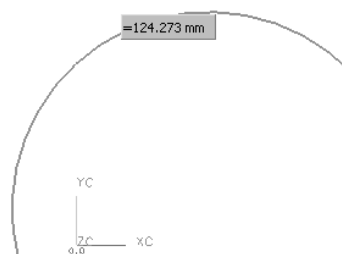


Figure 3-42 The arc length measurement displayed for an arc

MEASURING THE ANGLE BETWEEN ENTITIES

Menu: Analysis > Measure Angle
Toolbar: Utility > Measure Angle



After creating sketch, sometimes you may need to measure the angle between entities. To do so, choose the **Measure Angle** button from the **Utility** toolbar; the **Measure Angle** dialog box will be displayed, as shown in Figure 3-43. Using this dialog box, you can measure the angle values between the sketched entities by three methods. These methods 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 3-44, 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 3-44.

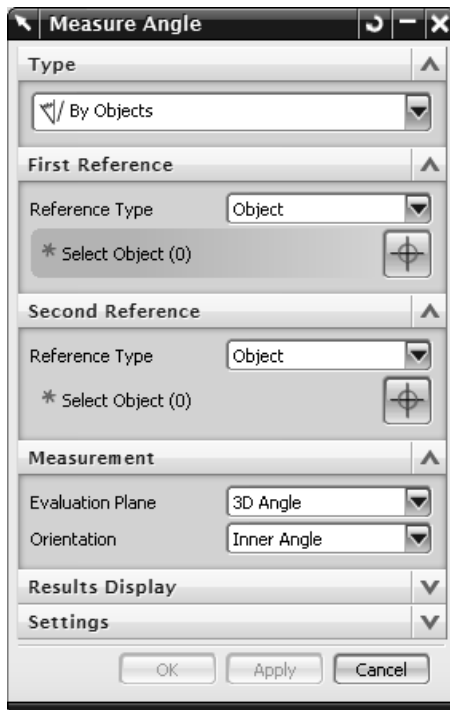


Figure 3-43 The **Measure Angle** dialog box

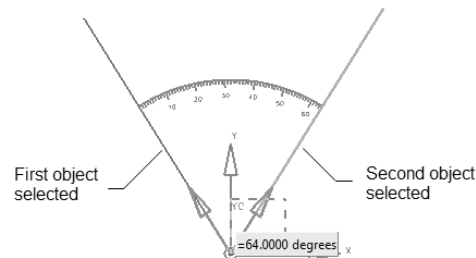


Figure 3-44 The angular measurement displayed using the **By Objects** option

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 3-45.

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 3-46 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.

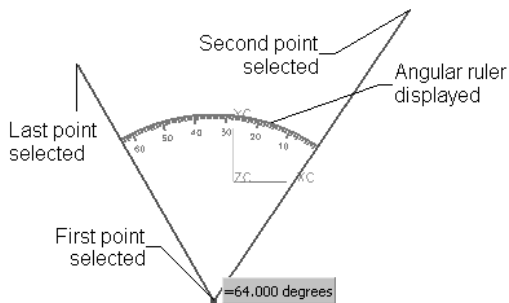


Figure 3-45 The angle measured using the **By 3 Points** option

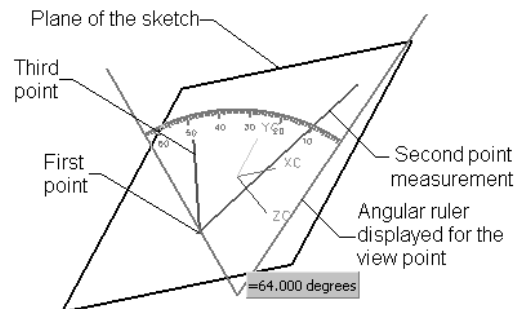


Figure 3-46 The angular measurement displayed using the **By Screen Points** option

TUTORIALS

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

Tutorial 1

In this tutorial, you will draw the profile of the model shown in Figure 3-47. The profile is shown in Figure 3-48. The profile should be symmetric about the origin. Also, you will use the parametric dimensions to complete the sketch. **(Expected time: 30 min)**

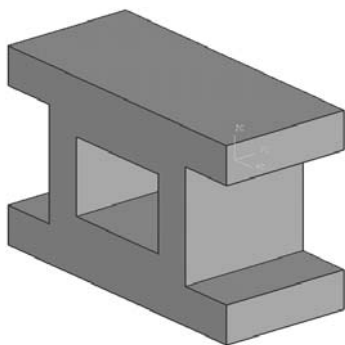


Figure 3-47 Model for Tutorial 1

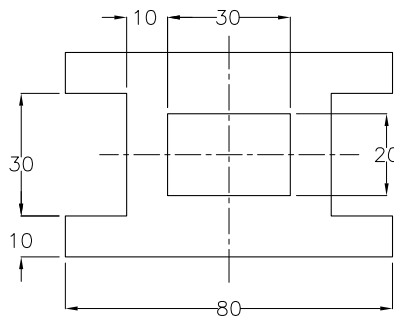


Figure 3-48 Sketch for Tutorial 1

The following steps are required to complete this tutorial:

- Start a new file, and invoke the sketcher environment.
- Draw the outer profile of the sketch using the **Profile** tool.
- Add the geometric and dimensional constraints to the outer loop.
- Draw a rectangle inside the outer loop using the **Rectangle** tool.
- Add dimensions to the rectangle to complete the sketch.
- Save the sketch and close the file.

Starting a New File and Invoking the Sketcher Environment

Start a new file by using the **Model** template.

- 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.
- Choose the button on the right of the **Folder** text box; the **Choose Directory** dialog box is displayed. Next, browse to the **C:\NX 7\c03** folder, and then choose the **OK** button twice; the new file is started in the Modeling environment.
- Turn on the display of WCS by choosing the **Display WCS** button from the **Utility** toolbar.
- Invoke the Sketcher environment by using the XC-ZC plane as the sketching plane.

Drawing the Outer Loop and Adding Sketcher 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 sketcher 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 3-49. The sketch is non-symmetric at this stage. But, after adding the sketcher constraints and the required dimensions, it will become symmetric. You can use the help lines to draw the sketch. For your reference, the sequence in which the lines to be drawn in the sketch is indicated by numbers.

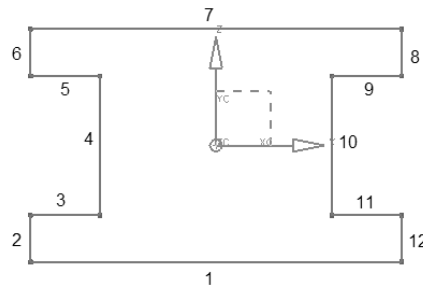
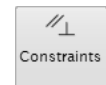
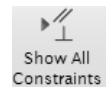
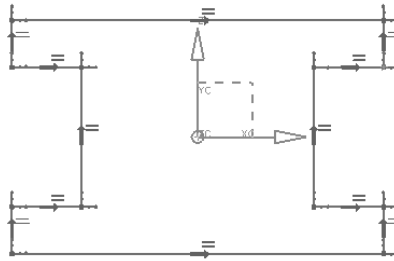


Figure 3-49 The outer loop of the profile and the sequence in which lines to be drawn

Next, you need to apply the geometric and dimensional constraints to the sketch members. 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 **Show All Constraints** button from the **Sketch Tools** toolbar to view the constraints applied to the sketch.
4. Next, choose the **Constraints** button from the **Sketch Tools** toolbar; you are prompted to select curves to create constraints.
5. Select lines 1 and 7; the **Constraints** dialog box is displayed with all possible constraints that can be applied to the selected entities. Next, choose the **Equal Length** button from the **Constraints** dialog box. 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. The sketch after applying the equal length constraint to all these line entities is shown in Figure 3-50.






*Figure 3-50 The outer profile after adding the **Equal Length** constraint*

Adding Dimensions to Sketch Members

Next, you need to add 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 **Sketch > 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 3-51. 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.
4. Select the line 2 and place the dimensions on the left of the sketch. Enter **10** in the edit box displayed and press ENTER.
5. Select the line 4 and place the dimension on the left of the sketch. Next, modify the dimension value to **30** and press ENTER.
6. Select the line 5 and place the dimension below the line. Next, modify the dimension value to **15** and press ENTER.

To make the sketch symmetric, you need to apply the dimension between line 7 and the horizontal datum axis and between line 10 and the vertical datum axis.

7. Select the line 7 and the horizontal axis and place the dimension on the right of the sketch. Next, modify the dimension value to **25** and press ENTER.

8. Select the line 10 and the vertical axis and place the dimension below the sketch. Next, modify the dimension value to **25** and press ENTER.

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

9. 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 3-51.

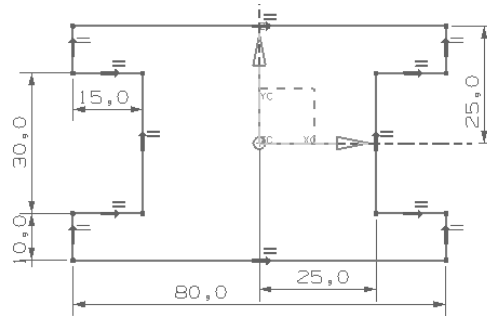


Figure 3-51 The outer profile after adding the required dimensions

Drawing the Inner Loop and Adding the Constraints and 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 and you are prompted to select the first point of the rectangle.
2. Draw the rectangle inside the outer profile by specifying its first point and a point to create the rectangle, refer to Figure 3-52.

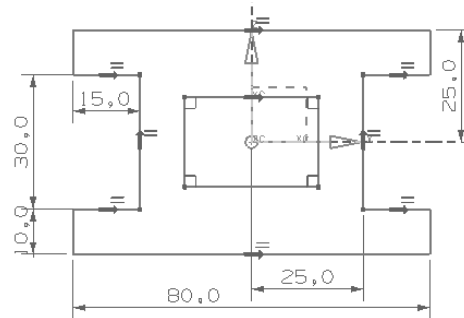
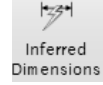


Figure 3-52 The sketch after drawing the inner loop and turning on the display of constraints

Next, you need to add dimensions to the inner profile.

3. Choose the **Inferred Dimensions** button from the **Sketch Tools** toolbar and select the upper horizontal line of the rectangle. Place the dimension above the sketch. Next, modify the value to **30** and press ENTER.
4. Select the right vertical line of the rectangle and place the dimension on the right of the sketch. Next, modify the value to **20** and press ENTER, refer to Figure 3-53.



To make the rectangle symmetric, you need to apply the dimension between any of the horizontal lines and the horizontal datum axis, and between any of the vertical lines and the vertical datum axis.

5. Select the lower horizontal line of the rectangle and the horizontal axis. Place the dimension on the right of the sketch. Next, modify the dimension value to **10** and press ENTER, refer to Figure 3-53.
6. Select the right vertical line of the rectangle and the vertical axis. Place the dimension above the sketch. Next, modify the dimension value to **15** and press ENTER, refer to Figure 3-53. The completed sketch is shown in Figure 3-53.

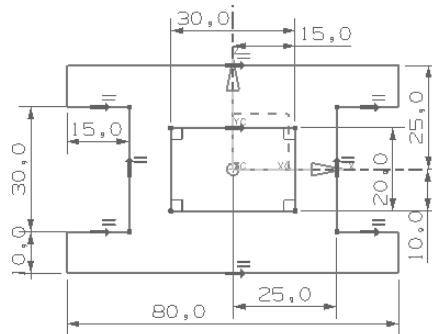


Figure 3-53 The resulting sketch after adding the required dimensions and constraints

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 Sketcher environment by choosing the **Finish Sketch** button from the **Sketcher** toolbar. Next, 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 3-54. The profile is shown in Figure 3-55. You will use the geometric and dimensional constraints to complete this sketch. (Expected time: 30 min)

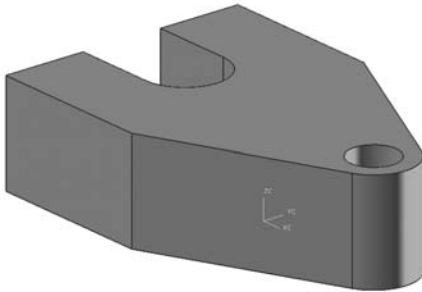


Figure 3-54 Model for Tutorial 2

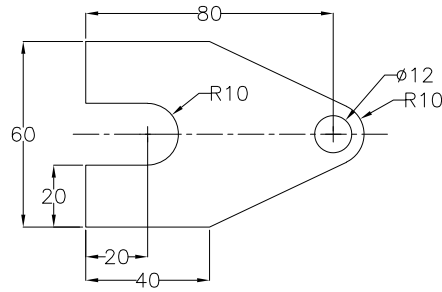


Figure 3-55 Sketch for Tutorial 2

The following steps are required to complete this tutorial:

- Start a new file in NX and invoke the sketcher 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 Sketcher Environment

- Start a new file with the name *c03tut2.prt* using the **Model** template and specify its location as *C:\NX 7\c03*.
- Turn on the display of WCS by choosing the **Display WCS** button from the **Utility** toolbar.
- Invoke the Sketcher environment by using the XC-ZC 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 3-56. You can draw the first line with exact dimensions 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 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 3-56; the center point of the arc is highlighted.

**Note**

*If the center point 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 3-56.

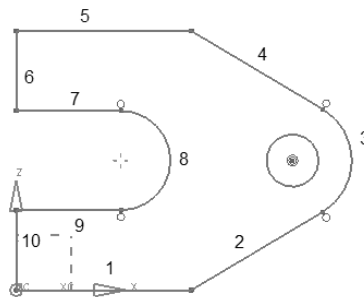


Figure 3-56 The sequence to be followed for drawing the sketch

Adding Constraints to the Sketch

- Choose the **Constraints** button from the **Sketch Tools** toolbar.
- Select the line 1 and apply the horizontal constraint by choosing the **Horizontal** button from the **Constraints** dialog box, if this constraint has not already been applied. Similarly, apply the horizontal constraints to lines 5, 7, and 9.
- Similarly, apply the **Vertical** constraints to lines 6 and 10, if this constraint has not already been applied.
- Next, select the circle and the arc 3 to apply the concentric constraint; the **Constraints** dialog box is displayed. Choose the **Concentric** button from it.
- Select lines 1 and 5 and then choose the **Equal Length** button from the **Constraints** dialog box 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 3-57.

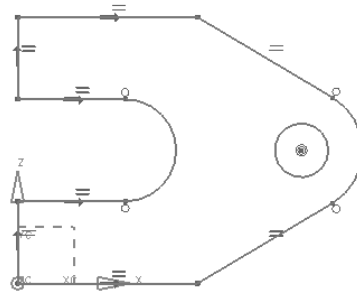


Figure 3-57 The sketch displayed after adding the required constraints

Adding Dimensions to the Sketch

Next, you need to add the dimensions to the sketch.



1. Choose the **Inferred Dimensions** button from the **Sketch Tools** toolbar.
2. Select the line 5 and place the dimension above the line. Modify the dimension value to **40** and press ENTER, refer to Figure 3-58.
3. Select the line 6 and the center point of the circle. Place the dimension above the previous dimension and modify the dimension value to **80**. Press the ENTER key.
4. Select the line 9 and place the dimension, refer to Figure 3-58. Next, modify the dimension value to **20** and press ENTER.
5. Select the line 10 and place the dimension on the left of the sketch. Next, modify the dimension value to **20** and press ENTER.
6. Select the arc 8 and place the dimension, refer to Figure 3-58. Next, modify the dimension value to **10** and press ENTER.
7. Select the arc 3 and place the dimension, refer to Figure 3-58. Next, modify the dimension value to **10** 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 3-58.

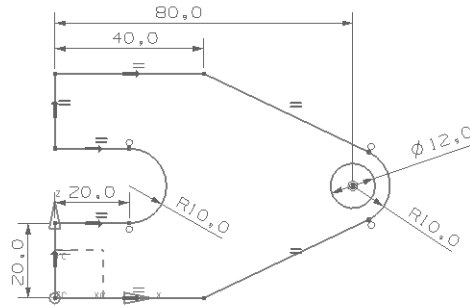


Figure 3-58 The final sketch after adding the required dimensions and constraints

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 has already been specified when you started the new file.
2. Exit the Sketcher environment and 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 3-59. The profile is shown in Figure 3-60. You will use the geometric and dimensional constraints to complete this sketch. **(Expected time: 30 min)**



Figure 3-59 Model for Tutorial 3

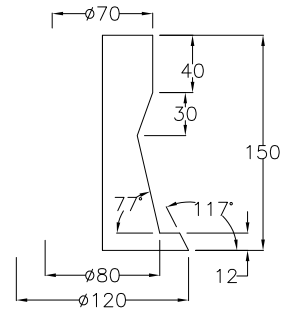


Figure 3-60 Sketch for Tutorial 3

The following steps are required to complete this tutorial:

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

Starting a New File in NX and Invoking the Sketcher Environment

1. Start a new file with the name *c03tut3.prt* using the **Model** template and specify its location as *C:\NX 7\c03*.
2. Turn on the display of WCS by choosing the **Display WCS** button from the **Utility** toolbar.
3. Invoke the Sketcher 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.

1. 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 the ENTER key.
2. Follow the sequence given in Figure 3-61 for drawing the sketch. Draw the other entities of the sketch. For better understanding, the sketch has been numbered.

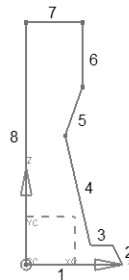
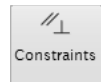


Figure 3-61 The sequence for drawing the profile

Adding Geometric Constraints to the Sketch

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

1. Choose the **Constraints** button from the **Sketch Tools** toolbar and select the line 8; the **Constraints** dialog box is displayed. Choose the **Vertical** button from this dialog box to apply the vertical dimension, if it is not applied automatically, while drawing the sketch.
2. Similarly, apply the horizontal constraint to lines 1, 3, and 7, if this constraint is not applied automatically.
3. Apply the vertical constraint to line 6, if it is not applied automatically.



- After applying the constraints, choose the **Show All Constraints** button from the **Sketch Constraints** toolbar. The resulting sketch is shown in Figure 3-62.

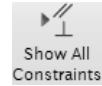
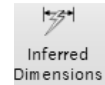


Figure 3-62 The resulting sketch displayed after adding the constraints

Adding Dimensions to the Sketch

Next, you need to apply dimensions to the sketch.

- Choose the **Inferred Dimensions** button from the **Sketch Tools** toolbar; the **Dimensions** dialog box is displayed.
- Select the line 8 and then place the dimension on the left of the sketch; an edit box with the default value is displayed. Enter **150** in this edit box and press the ENTER key; the dimension value is modified, refer to Figure 3-63.
- Select the line 1 and then place the dimension below the sketch. Modify the dimension value to **60** and press the ENTER key, refer to Figure 3-63.
- Select the line 5 and then place the dimension on the right of the sketch. Next, modify the dimension value to **30** and press the ENTER key, refer to Figure 3-63.
- Select the line 6 and then place the dimension on the right of the sketch. Next, modify the dimension value to **40** and press the ENTER key, refer to Figure 3-63.
- Select the line 7 and place the dimension above the sketch. Next, modify the dimension value to **35** and press the ENTER key, refer to Figure 3-63.
- Select the line 2 and then the 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 the ENTER key, refer to Figure 3-63.
- 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 the ENTER key, refer to Figure 3-63.



9. Select the lower endpoint of the line 4 and then select the 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.
10. Select the 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 3-63. Press the ESC key twice.
11. Choose the **Fit** button from the **View** toolbar. The resulting sketch after adding all dimensions is shown in Figure 3-63.

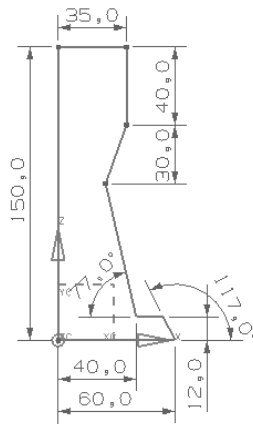


Figure 3-63 The completed sketch displayed after adding the required constraints and dimensions



Note

In Figure 3-63, 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 **Show All Constraints** button. Note that it is a toggle button.

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

(a) Radius	(b) Automatic Constraints
(c) Inferred Dimensions	(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) **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 in the Sketcher environment. (T/F)
10. The **Finish Sketch** tool in the **Sketcher** toolbar is used to exit the Sketcher environment. (T/F)

Exercises

Exercise 1

Draw the base sketch of the model shown in Figure 3-64. The sketch to be drawn is shown in Figure 3-65. Use the geometric and dimensional constraints to complete this sketch.

(Expected time: 15 min)

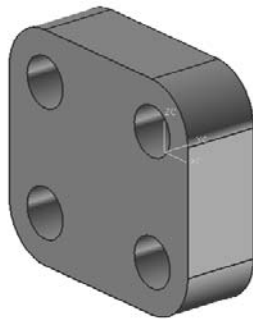


Figure 3-64 Model for Exercise 1

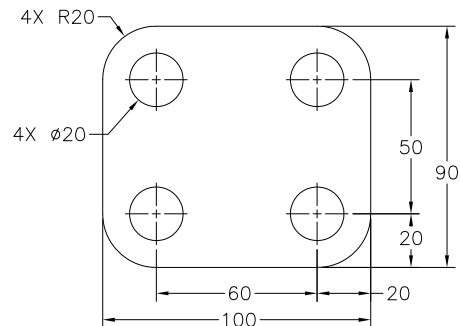


Figure 3-65 Sketch for Exercise 1

Exercise 2

Draw the base sketch of the model shown in Figure 3-66. The sketch to be drawn is shown in Figure 3-67. Use the geometric and dimensional constraints to complete this sketch.

(Expected time: 15 min)

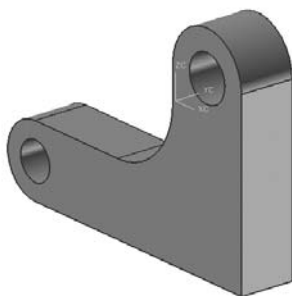


Figure 3-66 Model for Exercise 2

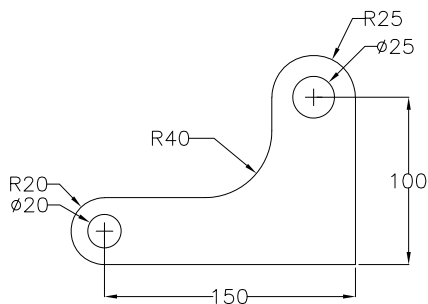


Figure 3-67 Sketch for Exercise 2

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. Show All Constraints