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 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 and 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 geometrical constraints are applied first, some of which get automatically applied while drawing. After applying the remaining geometrical constraints, you need to add dimensional constraints using dimension tools. You will learn more about dimension tools later in this chapter.

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 the 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 light purple color are under-constrained. You need to apply additional constraints to them in order to constrain 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 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 sketching environment. Figure 3-3 shows an over-constrained sketch.

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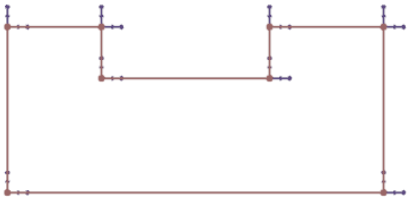


Figure 3-1 *An under-constrained sketch*

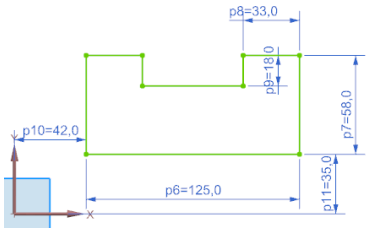


Figure 3-2 *A fully-constrained sketch*

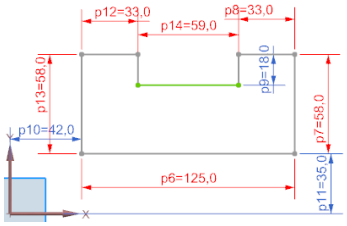


Figure 3-3 *An over-constrained sketch*

While applying the geometric and dimensional constraints, the status area in the Status Bar displays the number of constraints needed to fully constrain the sketch. After fully constraining the sketch, a message will be displayed 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 such a case, you need to remove the unwanted constraints.

DEGREES OF FREEDOM ARROWS

Degrees of Freedom can be defined as the number of independent variables or coordinates required to ascertain the position of any system or its components. In other words, you can say, it is the number of variables that are required to determine the position of a mechanism in space. In NX, the degrees 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.

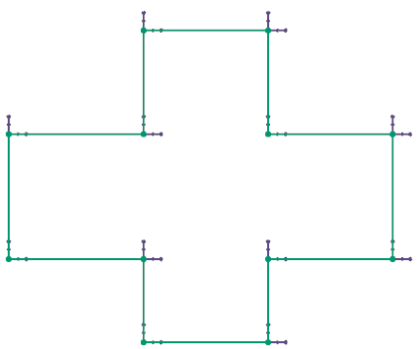


Figure 3-4 *The degrees of freedom arrows displayed on points*

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The direction of an 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 in this chapter.

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 get dimensioned automatically while drawing a sketch. However, the automatically applied dimensions do not constrain the sketch. Therefore, you need to lock them to fully constrain the sketch. Also, sometimes you may need to add additional dimensions to the sketch to fully constrain it. In NX, you can apply the dimensions by using the tools grouped together in the **Dimension** group of the **Home** tab in the sketching environment. Note that all dimension tools are not visible in this group by default. You can click on the **Group Options** arrow at the bottom of the **Dimension** group to display a flyout and then choose the required tools to add them in the group. Figure 3-5 shows the **Dimension** group after adding all the tools. The tools available in this group are listed below:

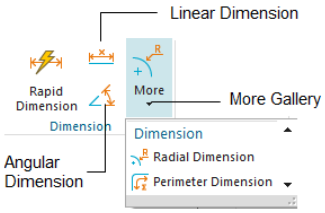


***Figure 3-5** The **Dimension** group*

1. Rapid Dimension
2. Linear Dimension
3. Angular Dimension
4. Radial Dimension
5. Perimeter Dimension

Note

*As the **Continuous Auto Dimensioning** tool is active by default in the **Constraint Tools Drop-down** of the **Constrain** group in the **Home** tab, the sketched entities will be dimensioned automatically in the sketching environment. If you do not want the sketch to be auto dimensioned, you can deactivate this tool by clicking on it. Also, you can control the display of auto dimensions by using the **Display Auto Dimensions** toggle button available in the **Constraint Tools Drop-down** of the **Constrain** group.*

The procedure of locking the automatically applied dimensions and applying various dimensions are discussed next.

Locking the Automatically Applied Dimensions

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. Also, a dialog box will be displayed for respective dimension. Enter the required value in the edit box and press ENTER; the dimension will be locked and displayed in blue color. Next, choose the **Close** button from the dialog box or press ESC key to exit the dialog box.

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Note

1. NX provides automatic and manual scale methods to scale sketches. When you add the first distance, radius, or diameter driving dimension, NX automatically scales the sketch to that value while maintaining the overall shape and inferred geometric constraints.

Note that the sketch is only scaled once and the next dimension you add or modify acts like any driving dimension.

2. You can dynamically edit and display the driving dimensions in a better manner to check the design intent. When you modify sketch geometry, NX moves the dimensions with the geometry. This behavior gives a neat look to the sketch by relocating the dimensions relative to the geometry.

Applying Dimensions by Using the Rapid Dimension Tool

Ribbon: Home > Dimension > Rapid Dimension
Menu: Insert > Dimensions > Rapid



The **Rapid Dimension** tool is used to apply dimension depending upon the entity selected. 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 and on selecting a line, linear dimension will be applied. Note that if you select an inclined line and move the cursor parallel to that line; an aligned 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 horizontally (right or left), a vertical dimension will be applied.

It is recommended that you use this tool to apply different type of dimensions as it saves the time required for selecting various dimensioning tools. To apply rapid dimensions, choose the **Rapid Dimension** tool from the **Dimension** group of the **Home** tab; the **Rapid Dimension** dialog box will be displayed and you will be prompted to select the object to be dimensioned. Now the dimension will be applied based on the selection procedure adopted while selecting objects. Figure 3-6 shows the radial and linear dimensions created by using the tool.

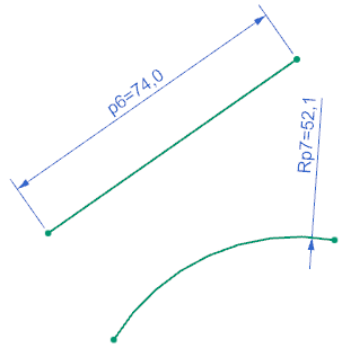


Figure 3-6 *The radial and linear dimensions created by using the* ***Rapid Dimension*** *tool*

Applying Linear Dimensions

Ribbon: Home > Dimension > Linear Dimension *(Customize to Add)*
Menu: Insert > Dimensions > Linear



The **Linear Dimension** tool is used to apply horizontal, vertical, or aligned dimension to a selected line or between two points. The points can be the endpoints of line or arc, or the center points of two circles, arcs, ellipses, or parabola, or any set of points that can be identified. To apply linear dimension to a sketch, choose the **Linear Dimension** tool from the **Dimension** group of the **Home** tab, refer to Figure 3-5; the **Linear Dimension** dialog box will be displayed, as shown in Figure 3-7.

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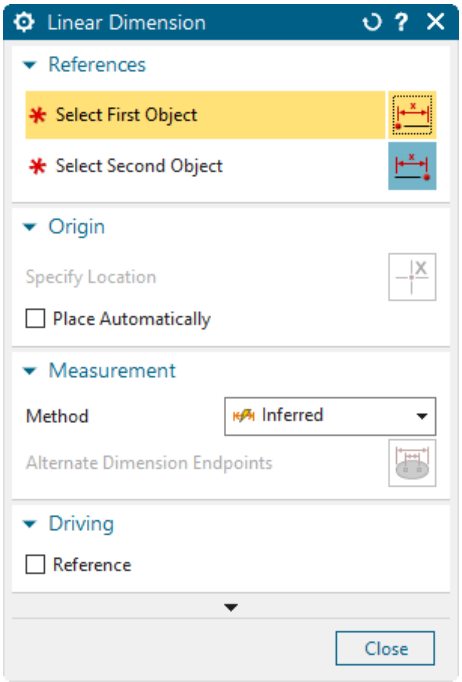


Figure 3-7 *The* ***Linear Dimension*** *dialog box*

The options in this dialog box are discussed next.

References Rollout

The options in this rollout are used to select the object to be dimensioned. Using the options in this rollout, you can select points or linear entities for applying the dimension.

Origin Rollout

This rollout is used to define the position of the dimension text. You can place the dimension at the required location. Place the dimension above or below (in case of horizontal) and left or right (in case of vertical) of the selected object by clicking on the desired place inside the drawing window. 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. Next, press ESC to exit the dialog box. Figure 3-8 shows the horizontal, vertical, and aligned dimensions applied to a line. The **Place Automatically** check box is used to place the dimension automatically according to the object.

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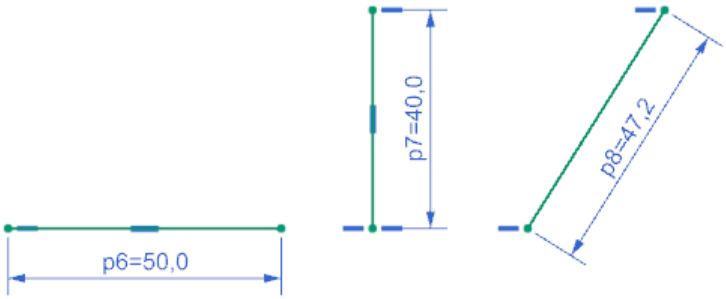


Figure 3-8 *The linear dimension created for a horizontal line, vertical line, and an inclined line*

Measurement Rollout

The options available in the **Method** drop-down list of this rollout are discussed next.

Inferred

The **Inferred** option is used to apply dimensions between entities based on the entities selected for dimensioning and the placement point.

Horizontal

The **Horizontal** option is used to apply horizontal dimension between entities. Note that if you select an entity that has a slant angle, only horizontal dimension can be applied.

Vertical

The **Vertical** option is used to apply vertical dimension between entities. Even if you select an entity that has a slant angle, a vertical dimension will be applied.

Point-to-Point

The **Point-to-Point** option is used to apply dimension between two points. You can select a set of points or a linear entity for applying dimension by using this option. Note that on selecting a linear entity, the dimension between its ends points will be applied, depending upon the entity selected, refer to Figure 3-8.

Perpendicular

The **Perpendicular** option is used to create the perpendicular dimension between a linear object and a point. It is mandatory that any one of the objects selected is a linear object.

Cylindrical

The **Cylindrical** option is used to apply dimension between cylindrical sections of objects.

Driving Rollout

In the **Driving** rollout, the **Reference** check box is available. This check box is used to convert the dimension into a reference dimension instead of a driving dimension.

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Note

If you create a redundant dimension, NX may inform you that the sketch is over-constrained. When you convert the dimension into a reference dimension, the sketch will no longer be over-constrained.

Settings Rollout

There are two buttons in this rollout and they are discussed next.

Settings



The **Settings** button is used to change the style of the dimensions that are being applied.

Inherit Dimension Settings



This button allows you to apply the style setting of an existing dimension to the dimensions being applied.

The **Enable Dimension Scene Dialogs** check box allows you to select access handles directly on different parts of the dimension to edit dimension settings.



Note

In the figures of this chapter, some of the dimension properties such as decimal places and labels have been modified for better display of dimensions. You can modify the dimension labels only when you are in the sketching environment. As discussed, you can invoke the sketching environment by using the ***Sketch*** *tool. To modify the dimension label, choose* ***Home*** *>* ***Construction*** *>* ***Sketch*** *from the* ***Ribbon****; the* ***Create Sketch*** *dialog box will be displayed. Next, choose the* ***OK*** *button from this dialog box; the sketching environment will be invoked. Now, choose* ***Menu >Task > Sketch Settings*** *from the* ***Top Border Bar****; the* ***Sketch Settings*** *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 labels will be modified.*

Applying Radial Dimensions

Ribbon:	Home > Dimension > More Gallery > Dimension Gallery > Radial Dimension *(Customize to Add)*
Menu:	Insert > Dimensions > Radial



The **Radial Dimension** tool is used to apply the radial dimension to an arc or diametral dimension to a circle. To apply radial dimension, choose the **Radial Dimension** tool from the **More Gallery** of the **Dimension** group of the **Home** tab, refer to Figure 3-5; the **Radial Dimension** dialog box will be displayed and you will be prompted to select an arc or circle to be dimensioned. 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. Next, press ESC to exit the dialog box. You can also select the type of dimension to be applied from the **Method** drop-down list in the **Measurement** rollout. In this drop-down list, the **Radial** option is used to define the radius and the **Diametral** option is used to define the diameter dimensions. However, if you select the **Inferred** option from the drop-down list, radius dimensions will be applied to arcs and diameter dimensions will be applied to circles automatically. Figure 3-9 shows the diametral dimension applied to a circle and Figure 3-10 shows the radial dimension applied to an arc. Rest of the options in the **Radial Dimension** dialog box are the same as those discussed in the Linear Dimension.

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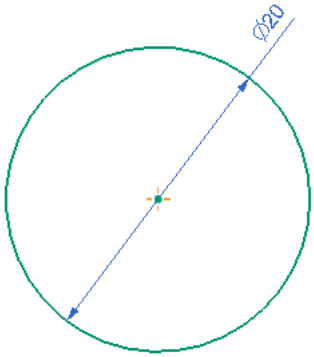


Figure 3-9 The diametral dimension applied to a circle

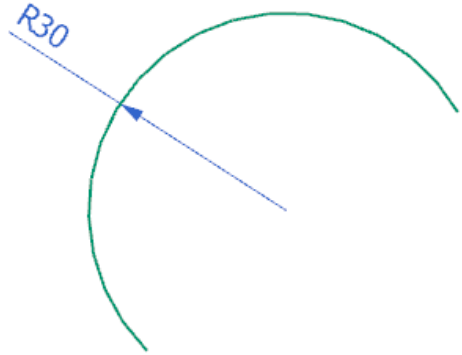


Figure 3-10 The radial dimension applied to an arc

Applying Angular Dimensions

Ribbon: Home > Dimension > Angular Dimension *(Customize to Add)*
Menu: Insert > Dimensions > Angular

The **Angular Dimension** tool is used to apply angular dimension between entities. Whenever an angular dimension is applied using the **Angular Dimension** tool, the angle is always measured in the counterclockwise direction.

To apply angular dimension, choose the **Angular Dimension** tool from the **Dimension** group of the **Home** tab, refer to Figure 3-5; the **Angular Dimension** dialog box will be displayed and you will be prompted to select an object to be dimensioned. Select the entities between which the angular dimension needs to be applied and then place the dimension. Figure 3-11 shows different types of angular dimensions applied to a sketch. Rest of the options in the **Angular Dimension** dialog box are same as discussed earlier.

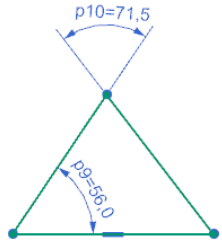


Figure 3-11 Angular dimensions applied between the entities

Applying Perimeter Dimensions

Ribbon: Home > Dimension > More Gallery > Dimension Gallery > Perimeter Dimension *(Customize to Add)*
Menu: Insert > Dimensions > Perimeter

The **Perimeter Dimension** tool is used to apply the circumferential or perimeter dimension. After applying the perimeter dimension, all the dimensions of the selected objects are locked. To apply the perimeter dimension, choose the **Perimeter Dimension** tool from the **Dimension** group of the **Home** tab; the **Perimeter Dimension** dialog box will be displayed. 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 Dimension** dialog box and then select all the entities of the closed sketch one by one. Next,

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choose the **OK** button from the **Perimeter Dimension** 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.

Editing the Dimension Value and Other Parameters

To edit a dimension, double-click on the dimension to be edited; the respective dimension dialog box will be invoked. Also, edit boxes will be displayed in the graphics window. By using the options available in the **Driving** rollout of the dialog box, you can specify the new dimension value and control the display of expression. You can also specify the new dimension value in the edit box displayed in the graphic window. Once you have specified the new dimension value, press the ESC key to apply the change and exit the dialog box.

Animating a Fully-Constrained Sketch

Ribbon:	Home > Constrain > Constraint Tools Drop-down > Animate Dimension
Menu:	Tools > Constraints > Animate Dimension

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** tool from the **Constraint Tools Drop-down** of the **Constrain** group of the **Home** tab; the **Animate Dimension** dialog box will be displayed, as shown in Figure 3-12, 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 3-13. Figure 3-14 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.

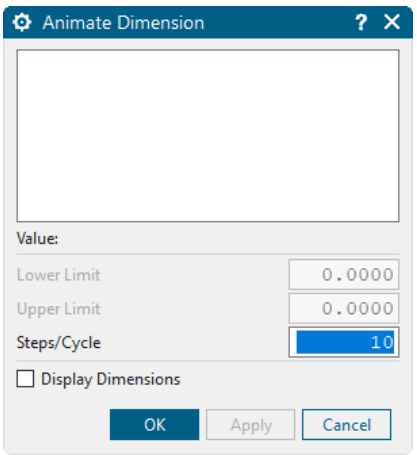


Figure 3-12 *The* ***Animate Dimension*** *dialog box*

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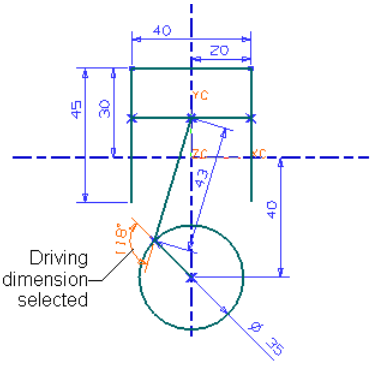


Figure 3-13 *Driving dimension selected from the sketch*

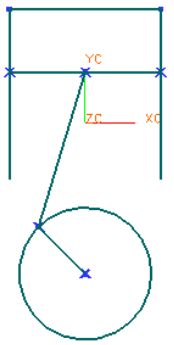


Figure 3-14 *The sketch being animated*

Note

You can hide and show the auto dimensions whenever you want. To do so, use the ***Display Auto Dimensions*** *toggle button available in the* ***Constraint Tools Drop-down*** *of the* ***Constrain*** *group of the* ***Home*** *tab to toggle on/off the auto dimensions.*

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 unexpectedly 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 sketching environment of NX, you can add various 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

Ribbon:	Home > Constrain > Geometric Constraints
Menu:	Insert > Geometric Constraints

In NX, you can apply additional constraints manually by using the **Geometric Constraints** tool available in the **Constrain** group of the **Home** tab. To apply constraints, invoke the **Geometric Constraints** tool; the **Geometric Constraints** message box will be displayed. Select the **Don't display this message again** check box in it and then choose the **OK** button; the **Geometric Constraints** dialog box will be displayed, as shown in Figure 3-15. 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 constraints to add or remove the constraints from the* ***Constraint*** *rollout. Select the check boxes next to the required constraints from the* ***Settings*** *rollout; the selected constraints will be displayed in the* ***Constraint*** *rollout.*

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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, as shown in Figure 3-16. Select the required constraint from the contextual toolbar; the selected constraint will be applied to the selected entities.

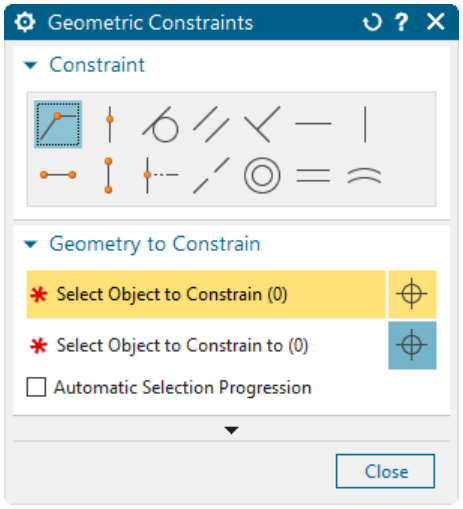


Figure 3-15 *The* ***Geometric Constraints*** *dialog box*



Figure 3-16 *The contextual toolbar*

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. To apply this constraint, choose the **Fixed** button *(Customize to add)* 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 3-17; the selected entity will be fixed.

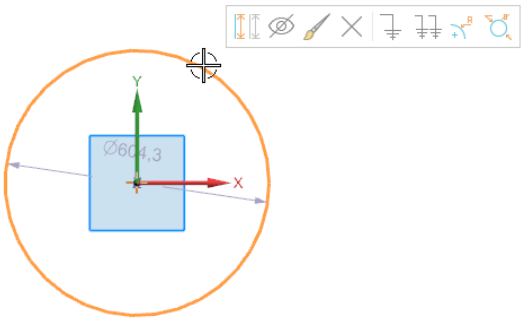


Figure 3-17 *Applying the Fixed constraint to an entity*

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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 *(Customize to add)* from the **Constraint** rollout and then select the entity; the selected entity will be fully fixed.

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 3-18 shows the endpoints of the two lines selected to be made coincident and Figure 3-19 shows the sketch after applying the constraint.

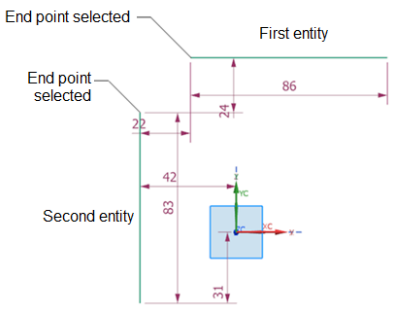


Figure 3-18 *The endpoints of the first and second entities selected*

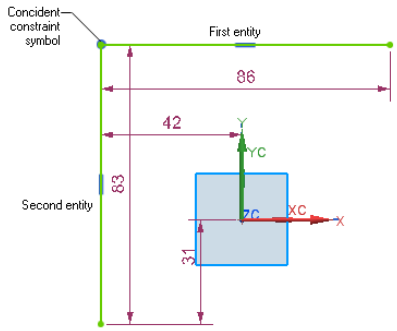


Figure 3-19 *The sketch after applying the Coincident constraint*

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.

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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.

Point on Curve Constraint

The Point on Curve constraint is used to place selected keypoints on selected curve or line. 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 3-20 shows a sketch before applying this constraint and Figure 3-21 shows a sketch after applying this constraint.

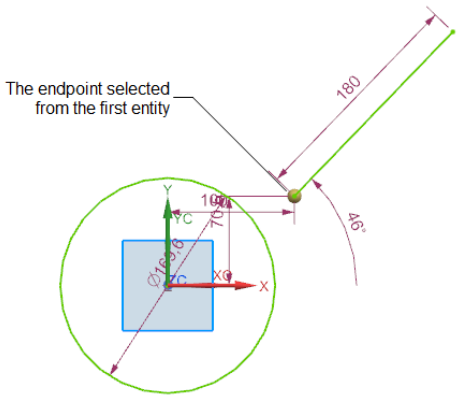


Figure 3-20 *The reference elements selected from the entity to apply the Point on Curve constraint*

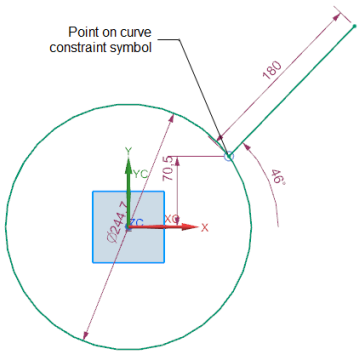


Figure 3-21 *The sketch after applying the Point on Curve constraint*

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 line or 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 3-22 shows two line segments before and after applying this constraint.

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Perpendicular Constraint

The Perpendicular constraint forces a set of selected line segments or ellipse axes to become normal to each other. Figure 3-23 shows two line segments before and after applying this constraint.

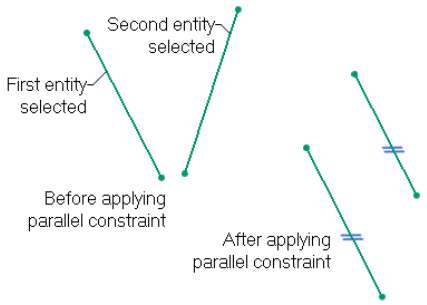


Figure 3-22 *Applying the Parallel constraint*

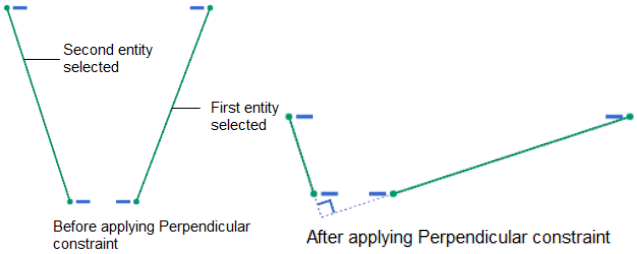


Figure 3-23 *Sketch before and after applying the Perpendicular constraint*

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 3-24 and 3-25 show the use of the Tangent constraint.

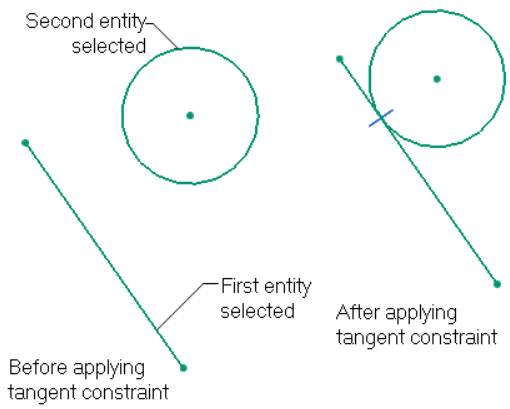


Figure 3-24 *Sketch before and after applying the Tangent constraint*

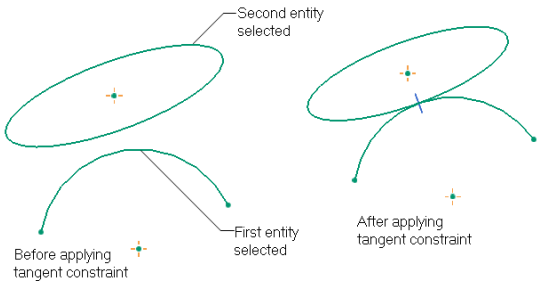


Figure 3-25 *Sketch before and after applying the Tangent constraint*

Note

By default, symbols of all the constraints are displayed in the sketch. You can turn on/off the display of constraints using the ***Display Constraints*** *tool which is discussed later in this chapter.*

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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 3-26 shows two circles before and after adding this constraint.

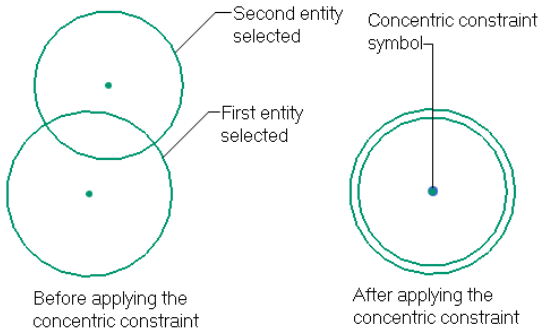


Figure 3-26 *Sketch before and after applying the Concentric constraint*

Collinear Constraint



This constraint forces the selected line segments to lie 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.

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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 Constraint



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

Horizontal Alignment Constraint



To apply this constraint, invoke the **Geometric Constraints** tool and then choose the **Horizontal Alignment** button from the **Constraint** rollout; you will be prompted to select the entities to be constrained. Select the points or midpoints of lines or curves you want to align; the aligned sketch will be displayed in the drawing area.

Vertical Alignment Constraint



To apply this constraint, invoke the **Geometric Constraints** tool and then choose the **Vertical Alignment** button from the **Constraint** rollout; you will be prompted to select the entities to be constrained. Next, select the points or midpoints of lines or curves you want to align; the aligned sketch will be displayed in the drawing area.

Perpendicular to String Constraint



This constraint is used to create a perpendicular constraint between a sketch curve and a recipe curve.

Tangent to String Constraint



This constraint is used to create a tangent constraint between a sketch curve and a recipe curve.



Note

The curves that are associatively projected to the sketch are called Recipe Curves.

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 **Constrain** group of the **Home** tab; 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

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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

Ribbon:	Home > Constrain > Constraint Tools Drop-down > Auto Constrain
Menu:	Tools > Constraints > 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** tool from the **Constraint Tools Drop-down** of the **Constrain** group; the **Auto Constrain** dialog box will be displayed, as shown in Figure 3-27. The options in this dialog box are discussed next.

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.

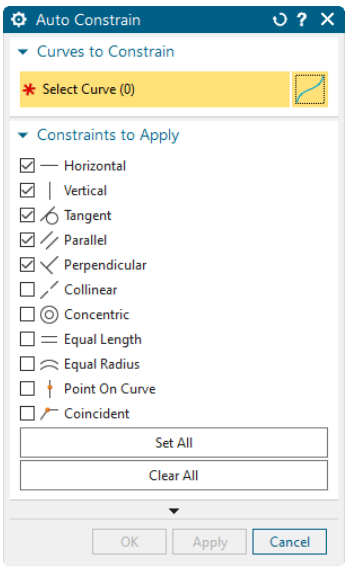


*Figure 3-27 The **Auto Constrain** dialog box*

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 for all the 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 for all the constraints will be cleared.

Settings Rollout

This rollout provides the options to set the tolerance for applying the constraints. Note that this rollout is not expanded by default. Expand this rollout to display options in it. The options in this rollout are discussed next.

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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 define 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 the 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 **Angle Tolerance** edit boxes.

Controlling Inferred Constraints Settings

Ribbon:	Home > Constrain > Constraint Tools Drop-down > Inferred Constraints and Dimensions
Menu:	Tools > Constraints > Inferred Constraints and Dimensions



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 choose this tool, the **Inferred Constraints and Dimensions** dialog box will be displayed, as shown in Figure 3-28.

This dialog box displays all the 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.

The **Create Dimensions for Typed Values** check box is selected by default, so the dimensions of the entities created by entering the values inside the input boxes will be displayed in the drawing area.

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.

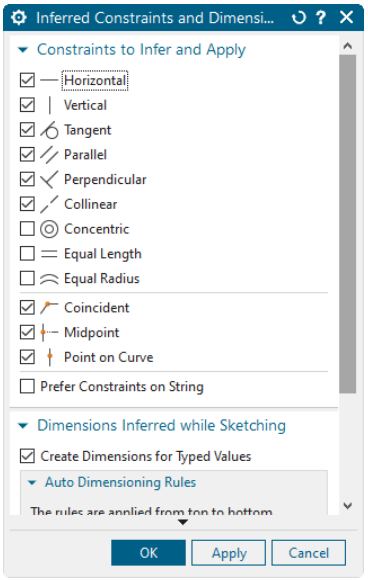


*Figure 3-28 The **Inferred Constraints and Dimensions** dialog box*

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Showing All Constraints in a Sketch

Ribbon: Home > Constrain > Constraint Tools Drop-down > Display Constraints
Menu: Tools > Constraints > Display Constraints



By default, all the constraints that are applied to a sketch are displayed in the sketch. You can turn on/off the display of these constraints by using the **Display Constraints** button. This is a toggle button and when you turn it off, it remains off until you turn it on. With this button turned on, you can continue working with the other sketching tools.

Sketch Relations Browser

Ribbon: Home > Constrain > Constraint Tools Drop-down > Constraints Browser
Menu: Tools > Constraints > Sketch Constraints Browser



You can use the **Constraints Browser** tool to interrogate sketched objects. To do so, choose the **Constraints Browser** tool from the **Constraint Tools Drop-down** in the **Constrain** group of the **Home** tab; the **Sketch Constraints Browser** dialog box will be displayed, refer to Figure 3-29. In this dialog box, you can view the constraints, dimensions, and external references associated to a part of the sketch or the entire sketch. Also, it enables you to view details of the sketched objects in an active sketch and to resolve their conflicting constraints. You can also delete dimensions and constraints using this dialog box.

Using this dialog box you can view:

1. Curves and their associated constraints
2. Constraints and their associated curves
3. Dimensions and their associated curves
4. External references attached to the sketch

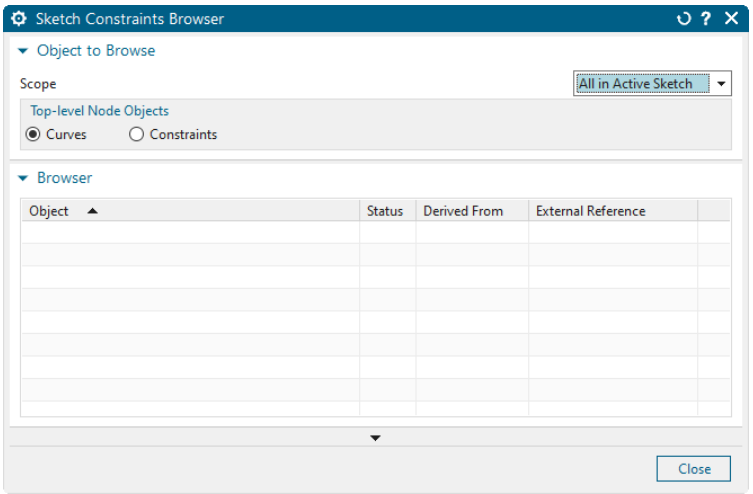


***Figure 3-29** The **Sketch Constraints Browser** dialog box*

Object to Browse Rollout

The options in this rollout are discussed next:

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Scope

The options available in the **Scope** drop-down list allow you to select the objects to be analyzed. These options are discussed next:

All in Active Sketch: This option is selected by default. It displays all the curves and constraints in the sketch in the **Browser** rollout.

Single Object: When you select this option from the **Scope** drop-down list, it displays the curves associated with the selected constraint, or the constraints associated with the selected curve in the **Browser** rollout. When you select an object from the drawing area, the current selection is replaced.

Multiple Objects: When this option is selected, the curves associated with the selected constraints, or the constraints associated with the selected curves are displayed in the **Browser** rollout.

Top-level Node Objects

This area of the dialog box contains two radio buttons, **Curves** and **Constraints** which are discussed next:

Curves: When you select this radio button, the curves are displayed as the top level nodes. Each curve node contains nodes of the constraints attached to it.

Constraints: When you select this radio button, the constraints are displayed as the top level nodes. Each constraint node contains nodes of the curves attached to it.

Browser Rollout

This area of the dialog box contains a table with four columns. They are discussed next:

Object

This column displays the names of the curves and constraints contained in the sketch.

Status

This column displays the constraint status of the curves. For example, overconstrained curves are indicated. Hover the cursor over the icons in this column for more information.

Derived From

This column displays the icon of the command/tool used to create relationship.

External Reference

This column displays a message when a sketch curve has a reference to an object in another file or in the same file but outside the sketch.

Settings Rollout

This area contains the **Display Degree-of-Freedom** check box which toggles the degree-of-freedom arrows on the sketch curves.

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Converting a Sketch Entity or Dimension into a Reference Entity or Reference Dimension

Ribbon:	Home > Constrain > Constraint Tools Drop-down > Convert To/From Reference
Menu:	Tools > Convert To/From Reference

The **Convert To/From Reference** tool 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** tool from the **Constraint Tools Drop-down** of the **Constrain** group; the **Convert To/From Reference** dialog box will be displayed, as shown in Figure 3-30.

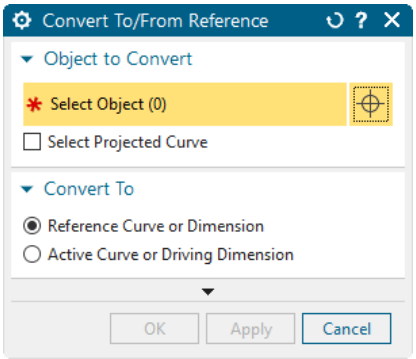


*Figure 3-30 The **Convert To/From Reference** dialog box*

By default, the **Reference Curve or Dimension** radio button is selected and therefore 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.

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

Creating Alignment Constraints

In NX, you can control the creation of alignment constraints (horizontal or vertical). In the releases prior to NX 12.0, if an entity was placed into the sketch after snapping the existing point of the sketched entity horizontally or vertically, the respective constraint would be applied between the entities by default. NX 12 onwards, these alignment constraints will not be applied by default. However, you can control the application of these alignment constraints before you start a file. To do so, choose **Menu > File > Utilities > Customer Defaults** from the Welcome screen of NX; the **Customer Defaults** dialog box will be displayed, refer to Figure 3-31.

Note

*The **Customer Defaults** dialog box can be invoked only in the Modeling environment of NX.*

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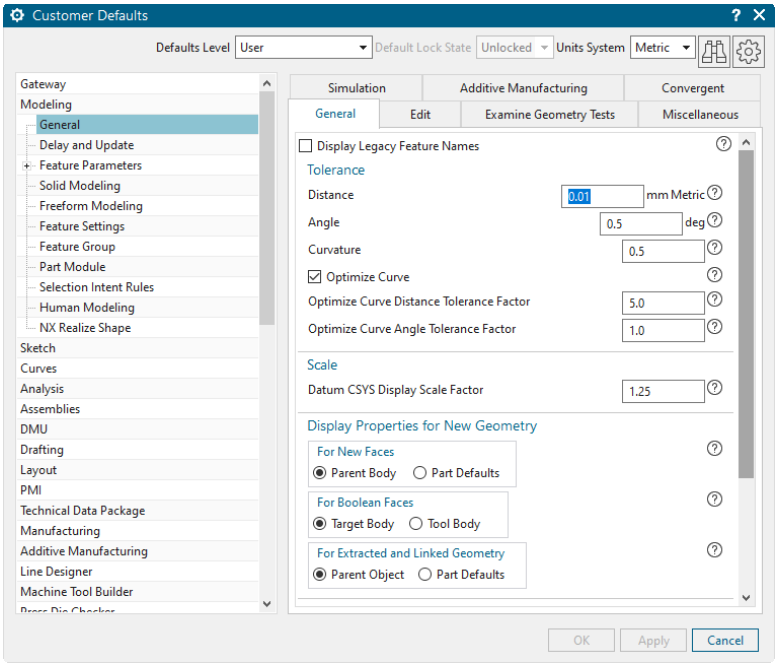


Figure 3-31 *The* ***Customer Defaults*** *dialog box*

In this dialog box, expand the **Sketch** node from the list box available at the left by clicking on it. Choose the **Inferred Constraints and Dimensions (Legacy)** sub-node; the options related it to it will be displayed on the right. Next, select the **Create Alignment Constraints** check box available under the **Constraints** tab.

After selecting this check box, the alignment constraints will be applied between entities while sketching. Figure 3-32 shows a sketch in which the right circle is placed horizontally after snapping the mid point of the left circle; the horizontal alignment constraint is applied between the centers. Notice that no alignment constraint is applied as the **Create Alignment Constraints (Legacy)** check box is cleared in the **Customer Defaults** dialog box, refer to Figure 3-33.

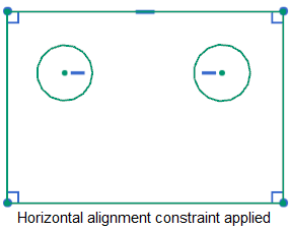


Figure 3-32 *Sketch after selecting the* ***Create Alignment Constraints*** *check box*

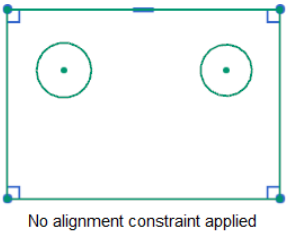


Figure 3-33 *Sketch after selecting the* ***Create Alignment Constraints*** *check box is cleared*



Note

To reflect the changes made in the ***Customer Defaults*** *dialog box, you need to restart the NX session.*

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TUTORIALS

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

Tutorial 1

In this tutorial, you will draw the profile of the model shown in Figure 3-34. The profile is shown in Figure 3-35. The profile should be symmetric about the origin. Also, you will use the parametric dimensions to complete the sketch. **(Expected time: 30 min)**

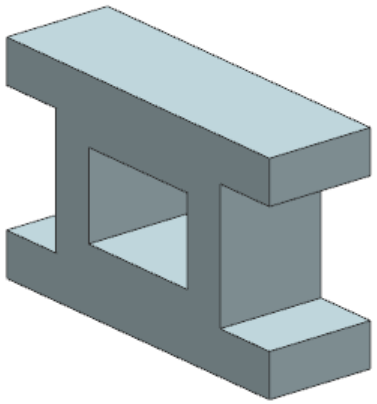


Figure 3-34 *Model for Tutorial 1*

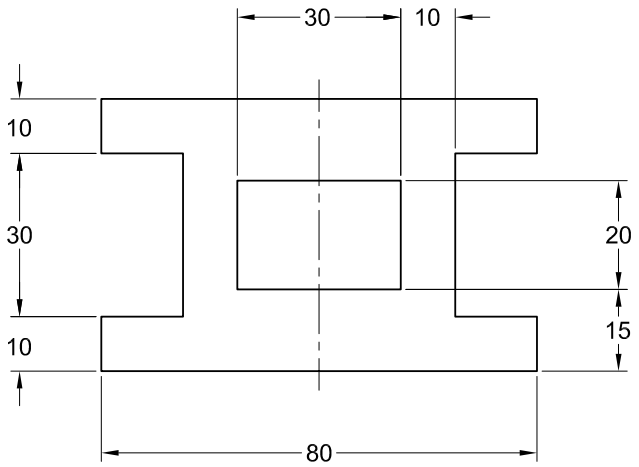


Figure 3-35 *Sketch for Tutorial 1*

The following steps are required to complete this tutorial:

a. Start a new file.
b. Draw the outer profile of the sketch using the **Profile** tool.

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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 Environment

1. Start a new file by using the **Model** template. To do so, choose the **New** button from the **File** tab of the **Ribbon**; 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.

2. Choose the button on the right of the **Folder** text box; the **Choose Directory** dialog box is displayed. Next, browse to *C:\NX\c03* and then choose the **OK** button twice; the new file is started in the Modeling environment.

3. Choose the **Sketch** tool from the **Construction** group; the **Create Sketch** dialog box is displayed. Select the XC-ZC plane as the sketching plane and choose the **OK** button from the dialog box.

Drawing the Outer Loop and Adding Sketch Constraints

If the sketch consists of more than one closed loop, it is recommended that you first draw the outer loop and then add all the required 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. Choose the **Profile** tool from the **Curve** group of the **Home** tab; 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 3-36. The sketch is unsymmetrical at this stage. But after adding the 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 are needed to be drawn in the sketch are indicated by numbers.

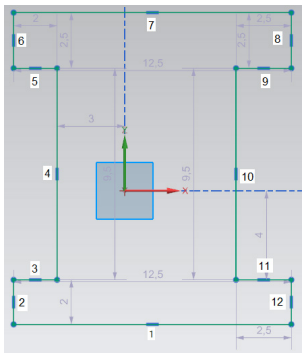


Figure 3-36 *The outer loop of the profile and the sequence in which lines have to be drawn*

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Note

The dimensions shown in Figure 3-36 get automatically applied when sketch is drawn. 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 not already on.

3. Choose the **Display Constraints** tool from the **Constraint Tools Drop-down** of the **Constrain** group of the **Home** tab to view the constraints applied to the sketch, if it is not chosen by default.

4. Choose the **Geometric Constraints** tool from the **Constrain** group of the **Home** tab; 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 the sketch members indicating that this constraint is applied between the two selected entities. Note that the **Automatic Selection Progression** check box should be selected in the **Geometry to Constrain** rollout to move to the succeeding points.

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 **Constrain** group; 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 line 12; you are prompted to select the centerline.

9. Select the vertical axis as the centerline; lines 2 and 12 become symmetric about vertical axis.

10. Similarly, make lines 1 and 7 symmetric about the horizontal axis. Next, choose the **Close** button from the **Make Symmetric** dialog box to close it. The sketch after applying constraints to all these entities is shown in Figure 3-37.

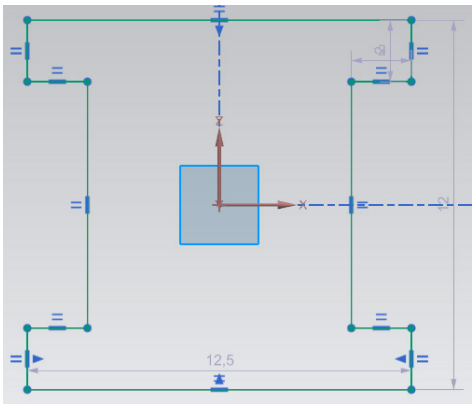


Figure 3-37 *The outer profile after adding constraints*

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Adding Dimensions to Sketch Members

Next, you need to add dimensions to the sketch. As mentioned earlier, when you add dimensions to a sketch, the sketch gets modified accordingly.

1. Choose the **Rapid Dimension** tool from the **Dimension** group of the **Home** tab; you are prompted to select an object to dimension or the dimension to edit.



2. Select line 1; the current dimension of line 1 is attached to the cursor. Now, you need to place the dimension at the required location. Click below the line 1 to place the dimension, refer to Figure 3-38. As you place the dimension, an edit box is displayed. Enter **80** in the edit box and press ENTER. Next, choose the **Fit** tool from the **Operation** group of the **View** tab.

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

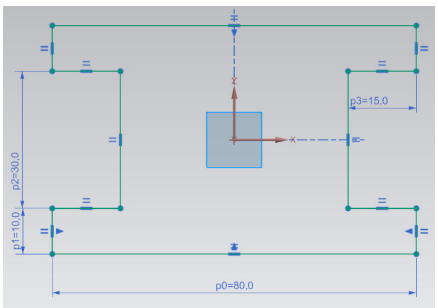


Figure 3-38 *The outer profile after adding required dimensions*

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

5. Select line 9 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.

6. Exit the **Rapid Dimension** tool by pressing the ESC key and then drag the dimensions to place them properly around the sketch, refer to Figure 3-38.



Tip

Instead of adding dimensions to a sketch, you can also modify the existing dimension. To do so, double-click on the dimension; 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.

Drawing the Inner Loop and Adding Dimensions

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

1. Choose the **Rectangle** tool from the **Curve** group of the **Home** tab; 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.

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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, refer to Figure 3-39. Press ESC to exit the tool.

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 left of the sketch, refer to Figure 3-40.

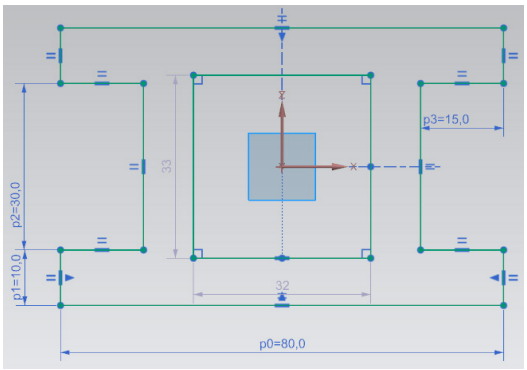


Figure 3-39 *The sketch after drawing the inner loop and turning on the display of constraints*

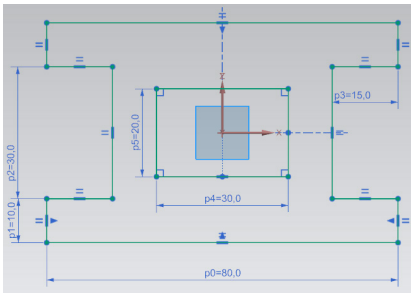


Figure 3-40 *The sketch after adding required dimensions and constraints*

Saving and Closing the File

1. Exit the sketching environment by choosing the **Finish** tool from the **Sketch** group of the **Home** tab. Next, choose the **Save** button from the **Quick Access** toolbar to save the sketch. Note that the name and location of the document had already been specified when you started new file.

2. Choose **Menu > File > Close > All Parts** from the **Top Border Bar** to close the file.

Tutorial 2

In this tutorial, you will create the profile for the model shown in Figure 3-41. The profile is shown in Figure 3-42. You will use the geometric and dimensional constraints to complete this sketch. **(Expected time: 30 min)**

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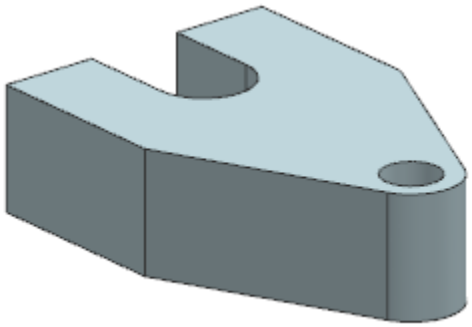


Figure 3-41 *Model for Tutorial 2*

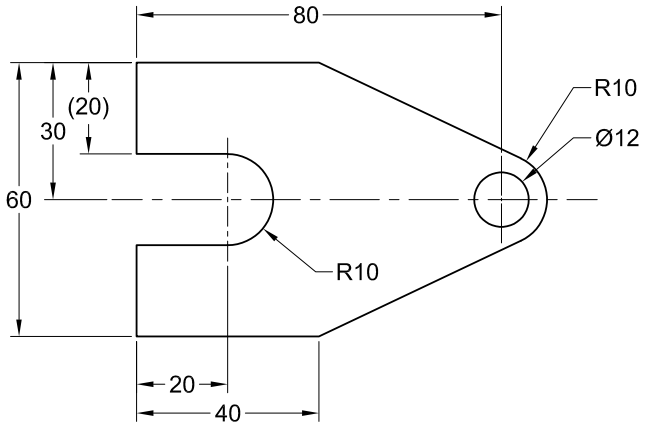


Figure 3-42 *Sketch for Tutorial 2*

The following steps are required to complete this tutorial:

a. Start a new file in NX and invoke the sketching environment.
b. Draw 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 Sketch Environment

1. Start a new file with the name *c03tut2.prt* using the **Model** template and specify its location as *C:\NX\c03*.

2. Turn on the display of WCS by choosing the **Display WCS** tool from **Tools > Utilities > More Gallery > WCS Gallery** in the **Ribbon** if not already displayed.

3. Invoke the Sketch environment by using the XC-YC plane as the sketching plane.

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Drawing the Sketch

1. Choose the **Profile** tool from the **Curve** group. Draw the outer profile of the sketch, refer to Figure 3-43. 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 line1 is at the origin.

   Note that after drawing the first line, you may need to modify the drawing display area by using the **Fit** tool from the **Operation** group of the **View** tab.

2. Next, choose the **Circle** tool from the **Curve** group. Move the cursor over the arc that is numbered 3 in Figure 3-43; 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 Group** of the **Top Border Bar**.*

3. Once 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 3-43.

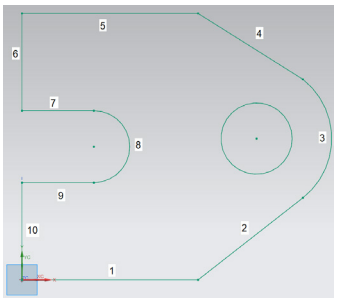


Figure 3-43 *The sequence to be followed for drawing the sketch*

Adding Constraints to the Sketch

1. Choose the **Geometric Constraints** tool from the **Constrain** group; the **Geometric Constraints** dialog box is displayed.
2. 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.
3. Similarly, apply the Vertical constraint to lines 6 and 10 if this constraint has not already been applied.
4. 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.
5. Choose the **Equal Length** button from the **Geometric Constraints** dialog box and select lines 1 and 5 to apply the Equal Length constraint.
6. Similarly, apply the Equal Length constraint between lines 2 and 4, 6 and 10, and 7 and 9.
7. Choose the **Tangent** button from the **Geometric Constraints** dialog box and select Line 4 and arc 3, and Line 2 and arc 3 to apply the Tangent constraint between them if not already applied.

   The sketch after applying all the constraints is shown in Figure 3-44.

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Modifying Dimensions of the Sketch

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

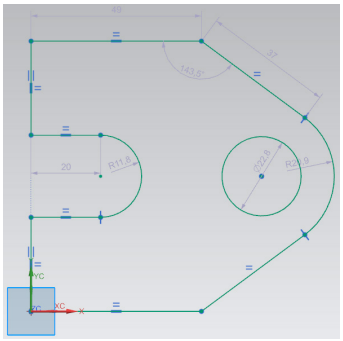


Figure 3-44 The sketch displayed after adding the required constraints

1. 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 3-45.

2. 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.

3. 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.

4. Choose the **Rapid Dimension** tool from the **Dimension** group.



5. 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.

6. Select line 6 and place the dimension on the left of the sketch. Next, modify the dimension value to **20** and press ENTER.

7. Select the circle and place the dimension, refer to Figure 3-45. Next, modify the dimension value to 12 and press ENTER. Make sure that the dimension value assigned to Line 1 is 40.

8. Choose the **Fit** tool from the **View** tab.

The final sketch after adding the required dimensions is shown in Figure 3-45.

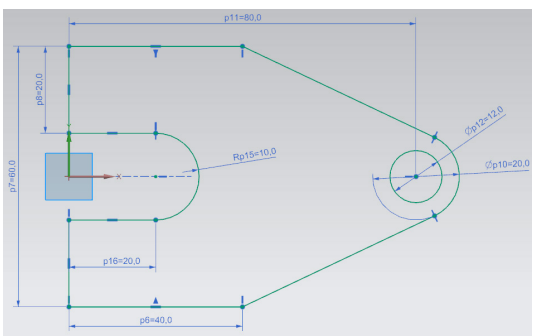


Figure 3-45 The final sketch after adding the required dimensions

Saving and Closing the File

1. Choose the **Save** button from the **Quick Access** 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 sketching environment and then choose **Menu > File > Close > All Parts** from the **Top Border Bar** to close the file.

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Tutorial 3

In this tutorial, you will create the profile of the revolved model shown in Figure 3-46. The profile is shown in Figure 3-47. You will use the geometric and dimensional constraints to complete this sketch. **(Expected time: 30 min)**

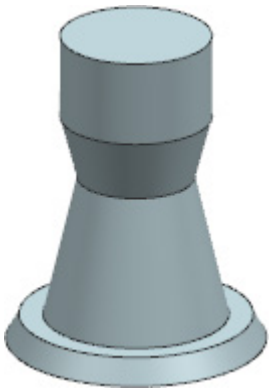


Figure 3-46 *Model for Tutorial 3*

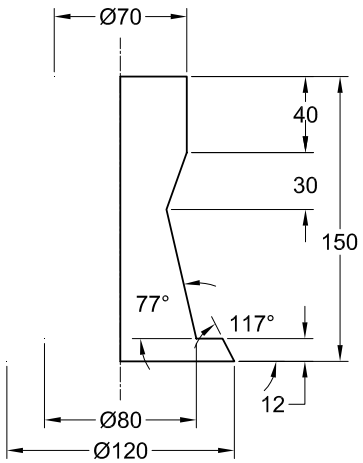


Figure 3-47 *Sketch for Tutorial 3*

The following steps are required to complete this tutorial:

a. Start a new file in NX and invoke the sketching 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.

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Starting a New File in NX and Invoking the Sketch Environment

1. Start a new file with the name *c03tut3.prt* using the **Model** template and specify its location as *C:\NX\c03*.

   If required turn on the display of WCS by choosing the **Display WCS** tool from the **Tools > Utilities > More Gallery > WCS Gallery** in the **Ribbon**.

2. Invoke the Sketch environment using the XC-ZC plane as the sketching plane.

Drawing the Sketch

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

1. Choose the **Profile** tool from the **Curve** group; 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.

2. Follow the sequence given in Figure 3-48 for drawing the sketch. Draw the rest of the entities of the sketch. For better understanding, the sketch has been numbered and temporary dimensions have been hidden.

Adding Geometric Constraints to the Sketch

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

1. Choose the **Geometric Constraints** tool from the **Constrain** group of the **Home** tab; the **Geometric Constraints** dialog box is displayed. Choose the **Vertical** button from this dialog box and select line 8 to apply the vertical dimension if not applied automatically while the sketch is drawn.

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 not applied automatically.

4. After applying the constraints, choose the **Display Constraints** tool from the **Constraints Tools Drop-down** of the **Constrain** group, if not already chosen. The resulting sketch is shown in Figure 3-49.

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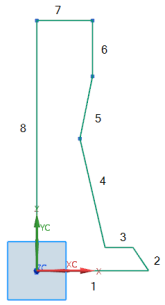


Figure 3-48 The sequence for drawing the profile

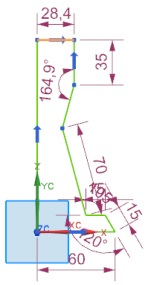


Figure 3-49 The resulting sketch displayed after adding the constraints

Modifying Dimensions of the Sketch

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

1. Double-click on the dimension of line 6; a dynamic edit box is displayed. Modify the value of the dimension to **40** and press ENTER. Next, drag and place the dimension on the right of the sketch, refer to Figure 3-50.

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

2. Choose the **Rapid Dimension** tool from the **Dimension** group; the **Rapid Dimension** dialog box is displayed.



3. Select line 8 and then place the dimension on the left of the sketch; an edit box with default value is displayed. Enter **150** in this edit box and press the ENTER key; the dimension value is modified, refer to Figure 3-50.

4. Select 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 3-50.

5. Select line 7 and place the dimension above the sketch. Next, modify the dimension value to **35** and press ENTER, refer to Figure 3-50.

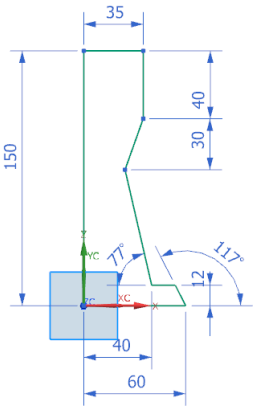


Figure 3-50 The completed sketch displayed after adding required constraints and dimensions

6. Select line 2 and then line 1; an angular dimension gets attached to the cursor. Move the cursor outside the sketch towards right and click the left mouse button to place the dimension. Next, modify the dimension value to **117** and press ENTER, refer to Figure 3-50.

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 3-50.

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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 3-50. Press the ESC key.

10. Choose the **Fit** tool from the **Operation** group of the **View** tab. The resulting sketch after adding all the dimensions is shown in Figure 3-50.



Note

*In Figure 3-50, 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 Constraints** tool.*

Saving and Closing the File

1. Choose the **Save** button from the **Quick Access** 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 sketching environment by choosing the **Finish** tool from the **Sketch** group of the **Home** tab and choose **Menu > File > Close > All Parts** from the **Top Border 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. The _____ constraint is used to force two curves to share same center point.

2. The _____ tool is used to dimension the radius of an arc.

3. The _____ tool is used to animate a fully constrained sketch.

4. The _____ tool is used to show all the constraints applied to a sketch.

5. In NX, you can add all types of geometric constraints by choosing the **Geometric Constraints** tool from the **Constrain** group of the **Home** tab. (T/F)

6. The **Auto Constrain** tool allows you to apply all possible geometric constraints automatically to an entire sketch. (T/F)

7. The **Rapid Dimension** tool in the **Dimension** group is used to add all possible dimension types. (T/F)

8. In NX, the **Radial Dimension** tool is used to add the diameter dimension to the sketch members. (T/F)

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Review Questions

Answer the following questions:

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

   (a) **Geometric Constraints** (b) **Automatic Constraints**
   (c) **Rapid Dimensions** (d) None of these

2. Which of the following tools is used to add a radial dimension to a sketch?

   (a) **Radial** (b) **Automatic Constraints**
   (c) **Rapid Dimension** (d) None of these

3. Which 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 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 of the following tools is used to apply an angular dimension to a sketch member?

   (a) **Angular** (b) **Constant Length**
   (c) **Rapid Dimensions** (d) None of these

6. Which 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. In NX, the degree of freedom arrows are displayed on the points that are free to move. (T/F)

8. The **Construction** group contains all the tools required to draw a sketch. (T/F)

9. The **Sketch** tool in the **Home** tab is chosen to switch to the sketching environment. (T/F)

10. The **Finish** tool in the **Home** tab is used to exit the Sketch environment. (T/F)

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EXERCISES

Exercise 1

Draw the base sketch of the model shown in Figure 3-51. The sketch to be drawn is shown in Figure 3-52. Use the geometric and dimensional constraints to complete this sketch.

(Expected time: 15 min)

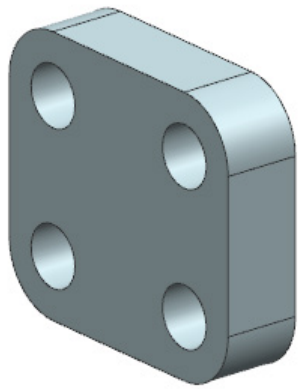


Figure 3-51 *Model for Exercise 1*

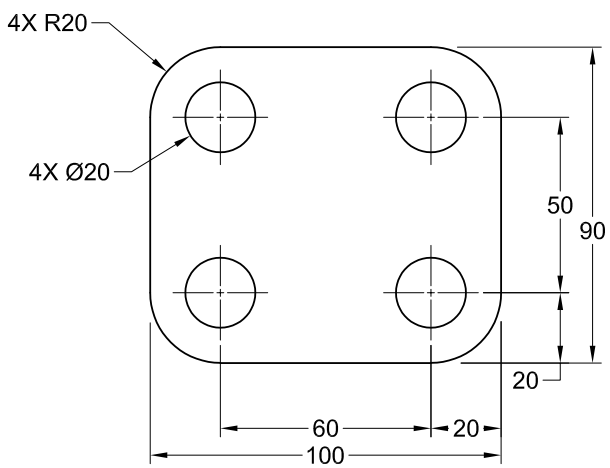


Figure 3-52 *Sketch for Exercise 1*

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Exercise 2

Draw the base sketch of the model shown in Figure 3-53. The sketch to be drawn is shown in Figure 3-54. Use the geometric and dimensional constraints to complete this sketch.

(Expected time: 15 min)

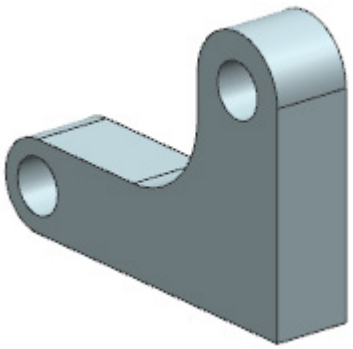


Figure 3-53 *Model for Exercise 2*

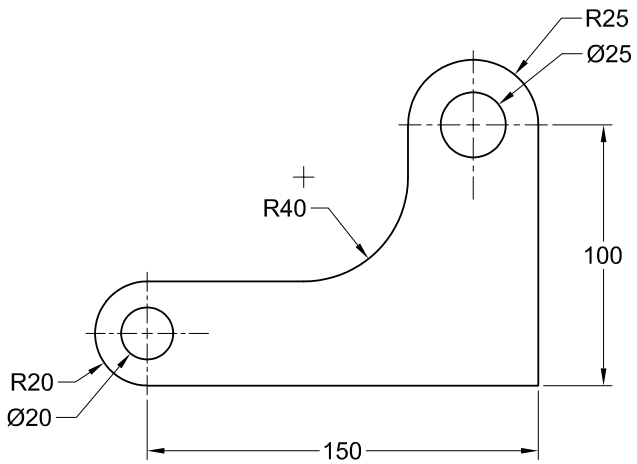


Figure 3-54 *Sketch for Exercise 2*

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Answers to Self-Evaluation Test

1. Concentric, **2. Radial**, **3. Animate Dimension**, **4. Display Sketch Constraints**, **5.** T, **6.** T, **7.** T, **8.** T