



# Chapter 3

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## Adding Constraints and Dimensions to Sketches

### Learning Objectives

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

- Add geometric constraints to the sketch.
- Control constraint inference.
- View and delete constraints from the sketch.
- Dimension the sketch.
- Modify dimensions of the sketch.
- Measure distances, angles, loops, and areas in the sketch.

## ADDING GEOMETRIC CONSTRAINTS TO THE SKETCH

Constraints are applied to the sketched entities to define their size and position with respect to other elements. Also, they are useful for capturing the design intent. As mentioned in Chapter 1, there are twelve types of geometric constraints that can be applied to the sketched entities. These constraints restrict their degrees of freedom and make them stable. Most of these constraints are automatically applied to the entities while drawing. However, sometimes you may need to apply some additional constraints to the sketched entities. These constraints are discussed next.

### Perpendicular Constraint

**Toolbar:** 2D Sketch Panel > Perpendicular  
**Panel Bar:** 2D Sketch Panel > Perpendicular



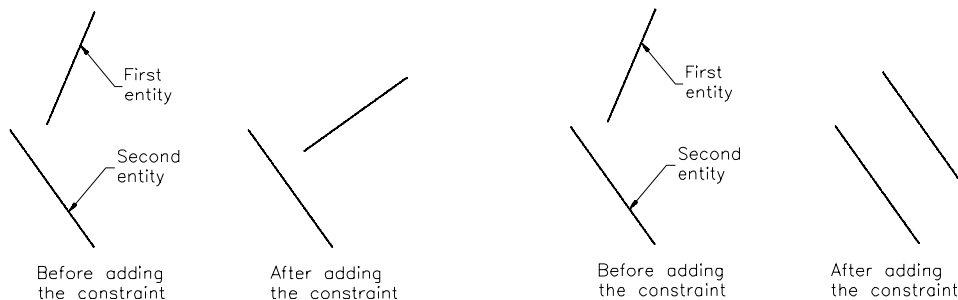
The **Perpendicular** constraint forces the selected entity to become perpendicular to the specified entity. To use this constraint, choose the **Perpendicular** button from the **2D Sketch Panel** panel bar; you will be prompted to select the first line or an ellipse axis. Once you select an entity, you will be prompted to select the second line or ellipse axis. On selecting the second entity, the first entity becomes normal to the second entity. Figure 3-1 shows two lines before and after adding this constraint.

### Parallel Constraint

**Toolbar:** 2D Sketch Panel > Perpendicular > Parallel  
**Panel Bar:** 2D Sketch Panel > Perpendicular > Parallel



The **Parallel** constraint forces the selected entity to become parallel to the specified entity. The entities to which this constraint can be applied are lines and ellipse axes. To apply this constraint, choose the down arrow on the right of the **Perpendicular** button in the **2D Sketch Panel** panel bar and then choose the **Parallel** button to invoke this constraint. On doing so, you will be prompted to select the first line or ellipse axis. After you select an entity, you will be prompted to select the second line or ellipse axis. On selecting the second entity, the first entity becomes parallel to the second entity. Figure 3-2 shows two lines before and after adding this constraint.



**Figure 3-1** Applying the **Perpendicular** constraint      **Figure 3-2** Applying the **Parallel** constraint

## Tangent Constraint

**Toolbar:** 2D Sketch Panel > Perpendicular > Tangent

**Panel Bar:** 2D Sketch Panel > Perpendicular > Tangent



The **Tangent** constraint forces the selected line segment or curve to become tangent to another curve. Choose the down arrow on the right of the **Perpendicular** button in the **2D Sketch Panel** panel bar and then choose **Tangent** to apply this constraint. On invoking this constraint, you will be prompted to select the first curve. After you select the first curve, you will be prompted to select the second curve. The curves that you can select include lines, circles, ellipses, or arcs. Figures 3-3 and 3-4 show the use of the **Tangent** constraint.

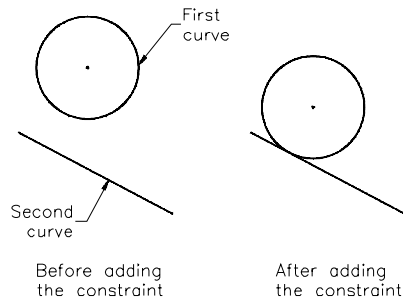


Figure 3-3 Applying the **Tangent** constraint

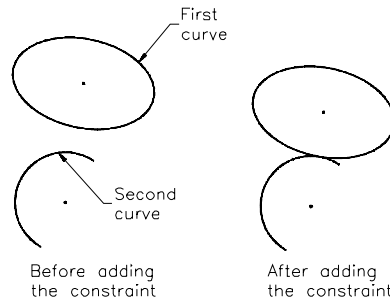


Figure 3-4 Applying the **Tangent** constraint

## Coincident Constraint

**Toolbar:** 2D Sketch Panel > Perpendicular > Coincident

**Panel Bar:** 2D Sketch Panel > Perpendicular > Coincident



The **Coincident** constraint is used to force two points or a point and a curve to become coincident. Choose the down arrow on the right of the **Perpendicular** button in the **2D Sketch Panel** panel bar and then choose **Coincident** to apply this constraint. When you invoke this constraint, you will be prompted to select the first curve or point. Once you specify the first curve or point, you will be prompted to specify the second curve or point. Note that either the first or the second entity selected should be a point. The points include sketch points, the endpoints of a line or an arc, or the center points of circles, arcs, or ellipses.

## Concentric Constraint

**Toolbar:** 2D Sketch Panel > Perpendicular > Concentric

**Panel Bar:** 2D Sketch Panel > Perpendicular > Concentric



The **Concentric** 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. When you invoke this constraint, you will be prompted to select the first arc, circle, or ellipse. After making the first selection, you will be prompted to select the second arc, circle, or ellipse.

**Note**

If you apply a constraint that is not required in the sketch, Autodesk Inventor will display a message box informing you that adding this constraint will over-constrain the sketch, see Figure 3-5. Over-constrained is a situation where the number of dimensions or constraints exceeds the required number that can be applied to the sketch.

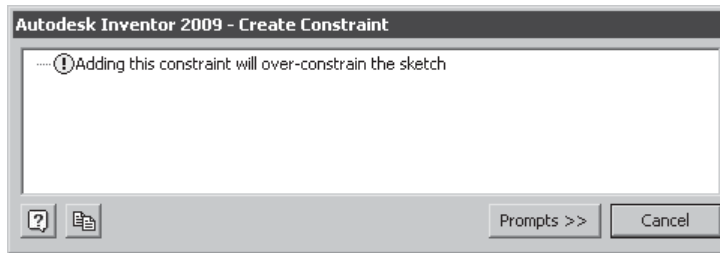


Figure 3-5 Message box informing that the sketch will be over-constrained

## Collinear Constraint

**Toolbar:** 2D Sketch Panel > Perpendicular > Collinear  
**Panel Bar:** 2D Sketch Panel > Perpendicular > Collinear



The **Collinear** constraint forces the selected line segments or ellipse axes to be placed in the same line. When you invoke this constraint, you will be prompted to select the first line or ellipse axis. After making the first selection, you will be prompted to select the second line or ellipse axis.



**Tip.** To select an ellipse axis, move the cursor close to the ellipse. Depending on whether the cursor is close to the major axis or the minor axis, the particular axis will be highlighted. When the required axis is highlighted, select it using the left mouse button.

## Horizontal Constraint

**Toolbar:** 2D Sketch Panel > Perpendicular > Horizontal  
**Panel Bar:** 2D Sketch Panel > Perpendicular > Horizontal



The **Horizontal** constraint forces the selected line segment, ellipse axis, or two points to become horizontal, irrespective of their original orientation. When you invoke this constraint, you will be prompted to select a line, an ellipse axis, or the first point. If you select a line or an ellipse axis, it will become horizontal. If you select a point, you will then be prompted to select a second point. The points, in this case, can also include the center points of arcs, circles, or ellipses.

## Vertical Constraint

**Toolbar:** 2D Sketch Panel > Perpendicular > Vertical  
**Panel Bar:** 2D Sketch Panel > Perpendicular > Vertical



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



**Tip.** You can use the **Horizontal** or the **Vertical** constraint to line up arcs, circles, or ellipses in the same horizontal or vertical direction by selecting their center points.

## Equal Constraint

**Toolbar:** 2D Sketch Panel > Perpendicular > Equal  
**Panel Bar:** 2D Sketch Panel > Perpendicular > Equal



The **Equal** constraint can be used either for line segments or for curves. If you select two line segments, this constraint will force the length of one of the selected line segment to become equal to the length of the other selected line segment. In case of curves, this constraint will force the radius of one of the selected curves to become equal to that of the other selected curve. Note that if the first selection is a line, the second selection also has to be a line. Similarly, if the first selection is a curve, the second selection also has to be a curve.

## Fix Constraint

**Toolbar:** 2D Sketch Panel > Perpendicular > Fix  
**Panel Bar:** 2D Sketch Panel > Perpendicular > Fix



This constraint is used to fix the orientation or location of the selected curve or point with respect to the coordinate system of the current drawing. If you apply this constraint to a line or an arc, you cannot move them from their current location. However, you can change their length by selecting one of their endpoints and then dragging it. If you apply this constraint to a circle or an ellipse, you cannot edit them by dragging. Once you apply this constraint to an entity, its color changes from black to blue.

## Symmetric Constraint

**Toolbar:** 2D Sketch Panel > Perpendicular > Symmetric  
**Panel Bar:** 2D Sketch Panel > Perpendicular > Symmetric



This constraint is used to force the selected sketched entities to become symmetrical about a selected sketched line segment. On invoking this constraint, you will be prompted to select the first sketched entity. Note that you can select only one entity at a time to apply this constraint. Once you have selected the first sketched entity, you will be prompted to select the second sketched entity. After doing so, you will be prompted to select the symmetry line. As soon as you select it, the second selected entity will be modified such that its distance from the line of symmetry becomes exactly equal to that of the first entity. After you have applied this constraint to one set of entities, you will again be prompted to select the first and second sketched entities. However, this time you will not be prompted to select the line of symmetry. The last line of symmetry will be automatically selected to add this constraint. Similarly, you can apply this constraint to other entities.

If the line of symmetry is different for applying the symmetric constraint to different entities in the sketch, you will have to restart the process of applying this constraint by right-clicking and choosing the **Restart** option from the shortcut menu. This is because the first symmetry

line is used to apply this constraint to all the sets of entities you select. However, if you restart applying this constraint, you will be prompted to select the line of symmetry again.

## Smooth Constraint

**Toolbar:** 2D Sketch Panel > Perpendicular > Smooth (G2)  
**Panel Bar:** 2D Sketch Panel > Perpendicular > Smooth (G2)



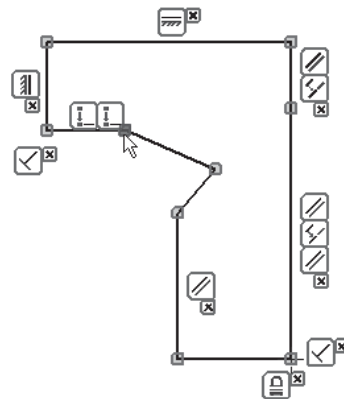
This constraint is used to apply a curvature continuity between a spline and an entity connected to it. The entities that can be selected to apply this constraint include a line, arc, or another spline. Note that these entities should be connected to the spline.

## VIEWING THE CONSTRAINTS APPLIED TO A SKETCHED ENTITY

**Toolbar:** 2D Sketch Panel > Show Constraints  
**Panel Bar:** 2D Sketch Panel > Show Constraints



You can view all constraints that are applied to the entities of a sketch by choosing the **Show Constraints** button in the **2D Sketch Panel** panel bar. When you invoke this tool and move the cursor close to any sketched entity, it will be highlighted and boxes will be displayed after a pause. These boxes show the symbols of all constraints that are applied to the entity. The symbols of the constraints will be highlighted in yellow color in this box. Select the entity to retain the constraint box; the constraint box will be displayed with the constraints in a white background. Figure 3-6 shows the constraint box along with the constraints applied to the lines. You can move this box by selecting it and dragging. To close this box, choose the cross (X) on the extreme right of this box. In the case of **Coincident** constraint, the constraint applied on a point is highlighted in yellow color, instead in a box. To view the symbols, move the cursor over the highlighted yellow point; the yellow point will turn red and the symbols will be displayed in boxes, refer to Figure 3-6.



**Figure 3-6** The constraint box showing the constraints applied to the sketch

If you move the cursor close to a constraint in the constraint box, it will be highlighted in yellow and the entities to which the constraint is applied are highlighted in red. For example, if you take the cursor close to the perpendicular constraint, the vertical line will also be highlighted along with the horizontal line, suggesting that the horizontal line is perpendicular to the vertical line.



**Tip.** You can also display the constraints applied to all the entities in the drawing. To display all the constraints, right-click to display the shortcut menu and choose the **Show All Constraints** option; separate boxes will be displayed showing the constraints on all the entities. Similarly, to hide all constraints, right-click and choose the **Hide All Constraints** option from the shortcut menu.

## CONTROLLING CONSTRAINTS AND APPLYING THEM AUTOMATICALLY WHILE SKETCHING

You can control and select the constraints that need to be applied automatically, as well as, select the geometry to which they will be applied. The tools and the procedure of selecting the constraints and the geometry are discussed next.

### Selecting the Constraints

**Toolbar:** Inventor Standard > Constraint Inference



By default, all possible constraints will be applied automatically on the sketching entities while drawing the sketches. However, you can also specify the constraints that need to be applied automatically, and also the geometry to which they will be applied while sketching. To do so, invoke the **Constraint Inference** tool from the **Inventor Standard** toolbar and right-click anywhere in the drawing window; a shortcut menu will be displayed. Choose **Constraint Options** from the shortcut menu; the **Constraint Options** dialog box will be displayed, as shown in Figure 3-7. The options in this dialog box are discussed next.

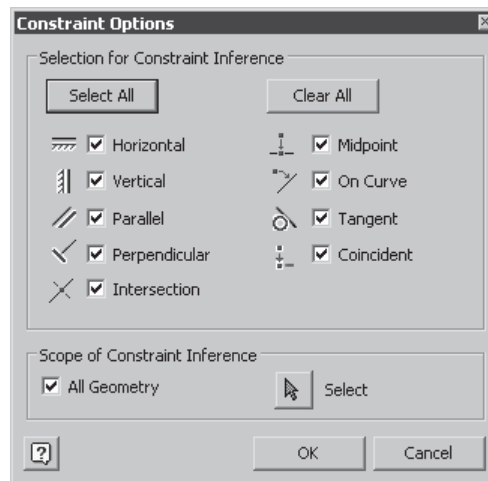


Figure 3-7 The *Constraint Options* dialog box

### Selection for Constraint Inference Area

In this area, nine types of constraints and two buttons are available. All the constraints in this area are selected by default. However, if you need to clear all the constraints, choose the **Clear All** button. You can manually select or clear the required constraint by selecting the



corresponding check box provided on the right of the constraint symbols. The selected constraints will be applied automatically to the geometry while sketching.

### Scope of Constraint Inference Area

This area is used to set the geometry to which constraint is applied while drawing. By default, the **All Geometry** check box is selected to apply the constraint to all the active sketches. If you clear this check box, the **Select** button will be activated automatically. You can use this button to select the geometry to which the constraints will be applied.

### Applying the Constraints

**Toolbar:** Inventor Standard > Constraint Persistence



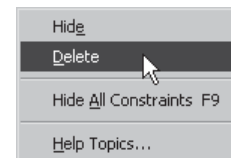
To apply the constraints selected in the **Constraint Options** dialog box, you need to choose the **Constraint Persistence** button from the **Inventor Standard** toolbar. This is a toggle button. If you choose this button, the **Constraint Inference** button gets activated and all the constraints selected earlier in the **Constraint Options** dialog box will be applied to the sketches while drawing. You can also switch off the inference of the constraint by choosing this button again.



**Tip.** The remaining three constraints **Coincident**, **Fix**, and **Symmetric** are not provided in the **Constraint Options** dialog box. By default, the **Coincident** constraint is applied whenever it is possible. You can apply the **Fix** and **Symmetric** constraints manually according to your requirement.

### DELETING GEOMETRIC CONSTRAINTS

Autodesk Inventor allows you to delete constraints applied to the selected entities. To delete the constraints, you first need to invoke the constraint box using the **Show Constraints** button. Once the constraint box is displayed, exit the **Show Constraints** tool by pressing the ESC key. Now, move the cursor over the constraint that you want to delete; it will be highlighted in yellow. Press the left mouse button to select the constraint; a red box will appear around the constraint. Now, move the cursor away, right-click, and choose **Delete** from the shortcut menu, see Figure 3-8. The selected constraint will be deleted and will also be removed from the constraint box that is displayed on the screen. Similarly, you can delete all the unwanted constraints in the sketch.



**Figure 3-8** Deleting constraints using the shortcut menu



**Tip.** When you move the cursor close to the constraint in the constraint box, its references will be highlighted in the sketch. For example, if you move the cursor over the **Perpendicular** constraint, the lines on which this constraint is applied will be highlighted. This allows you to confirm that the constraint selected is correct.



#### Note

The total number of constraints and dimensions needed to fully constrain the sketch is displayed near the bottom right corner of the drawing window.

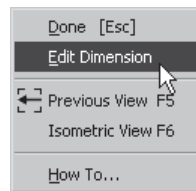


## ADDING DIMENSIONS TO SKETCHES

**Toolbar:** 2D Sketch Panel > General Dimension  
**Panel Bar:** 2D Sketch Panel > General Dimension



After drawing the sketch and adding constraints to it, dimensioning is the next most important step in creating a design. As mentioned earlier, Autodesk Inventor is a parametric solid modeling package. This property ensures that irrespective of the original size, the selected entity is driven by the dimension value you specify. Therefore, whenever you modify or apply dimension of an entity, it is forced to change its size in accordance with the specified dimension value. The type of dimension that will be applied varies depending on the type of entity selected. For example, if you select a line segment, linear dimensions will be applied and if you select a circle, diameter dimensions will be applied. Note that all these types of dimensions can be applied using the same dimensioning tool. While dimensioning, you can set the priority for editing the dimension value as soon as you place it. To set this priority, choose the **General Dimension** button from the **2D Sketch Panel** panel bar and then right-click to display the shortcut menu. From this menu, choose **Edit Dimension**, see Figure 3-9. Now, as soon as you place the dimension, the **Edit Dimension** toolbar, that allows you to modify the dimensions of the entity will be displayed, see Figure 3-10. The selected entity will be driven to the dimension value defined in this toolbar.



**Figure 3-9** Setting the priority for editing the dimensions as they are placed



**Figure 3-10** The **Edit Dimension** toolbar

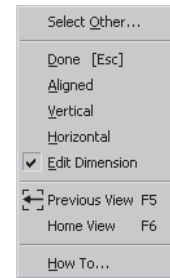
You can enter a new value for the dimension or choose the button on the right of this toolbar to accept the default value.

If you do not want to edit the dimensions after you place them, invoke the **General Dimension** tool and then right-click to display the shortcut menu. Clear the check mark on the left of the **Edit Dimension** option by choosing it again. When you place a dimension now, the **Edit Dimension** toolbar will not be displayed. To edit the dimension value in this case, click on it after placing, if the **General Dimension** tool is still active. If the tool is not active, double-click on the dimension; the **Edit Dimension** toolbar will be displayed. Enter the new dimension value in this edit box. The dimensioning techniques available in Autodesk Inventor are discussed next.

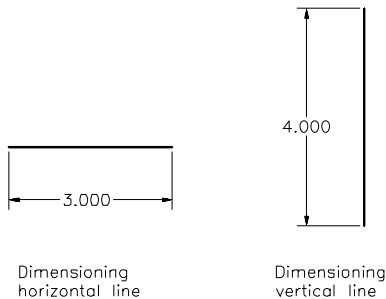
### Linear Dimensioning

Linear dimensions are defined as the dimensions that specify the shortest distance between two points. You can apply linear dimensions directly to a line or select two points or entities to apply the linear dimension between them. The points that you can select include the endpoints of lines, splines, or arcs, or the center points of circles, arcs, or ellipses. You can

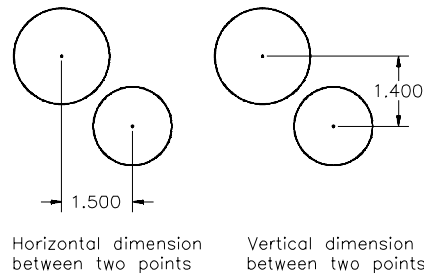
dimension a vertical or a horizontal line by directly selecting it. As soon as you select it, the dimension will be attached to the cursor. You can place the dimension at any desired location. If the priority for editing the dimensions is set, the **Edit Dimension** toolbar will be displayed as soon as you place the dimension. To place the dimension between two points, select the points one by one. After selecting the second point, right-click to display the shortcut menu, as shown in Figure 3-11. In this menu, choose the dimension type. If you choose **Horizontal**, the horizontal dimension will be placed between the two selected points. If you choose **Vertical**, the vertical dimension will be placed between the two selected points. If you choose **Aligned**, the aligned dimension will be placed between the two selected points. Figure 3-12 shows the linear dimensioning of lines and Figure 3-13 shows the linear dimensioning of two points.



**Figure 3-11** Shortcut menu displaying various options to dimension two points

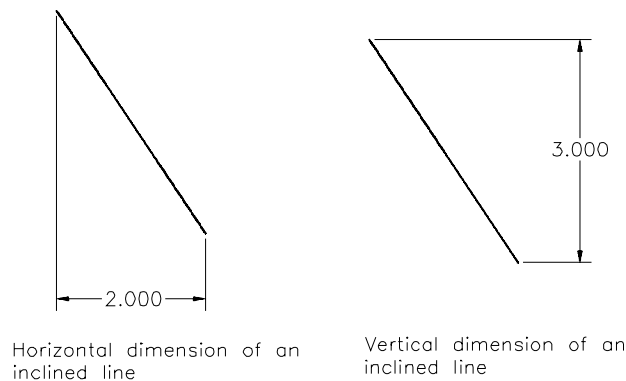


**Figure 3-12** Linear dimensioning of lines



**Figure 3-13** Linear dimensioning of two points

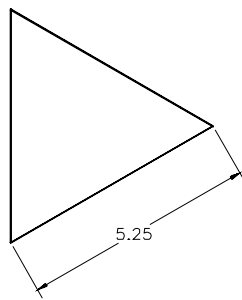
You can also apply a horizontal or vertical dimension to an inclined line, see Figure 3-14. To apply these dimensions, select the inclined line and then right-click; a shortcut menu, similar to the one shown in Figure 3-11, will be displayed. In this menu, choose **Horizontal** to place the horizontal dimension and choose **Vertical** to place the vertical dimension.



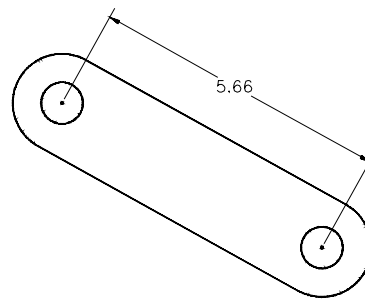
**Figure 3-14** Linear dimensioning of an inclined line

## Aligned Dimensioning

Aligned dimensions are used to dimension lines that are not parallel to the X axis or the Y axis. This type of dimension measures the actual distance of the aligned lines or the lines that are drawn at a certain angle. You can directly select the inclined line to apply this dimension. After selecting the inclined line, pick a point just below the line; the aligned dimension will be attached to the cursor. You can also select two points to apply the aligned dimension. The points include the endpoints of lines, splines, or arcs or the center points of arcs, circles, or ellipses. If you select two points to apply the aligned dimensions, right-click to display the shortcut menu and choose **Aligned**. Figures 3-15 and 3-16 show the aligned dimensions applied to various objects.



*Figure 3-15 Aligned dimension of a line*



*Figure 3-16 Aligned dimension using two points*

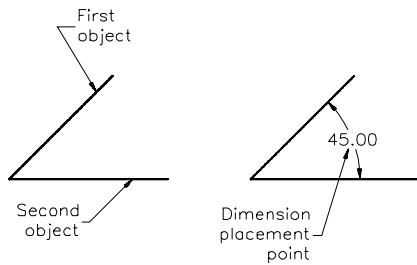
## Angular Dimensioning

Angular dimensions are used to dimension angles. You can select two line segments or use three points to apply the angular dimensions. You can also use angular dimensioning to dimension an arc. All these options of angular dimensioning are discussed next.

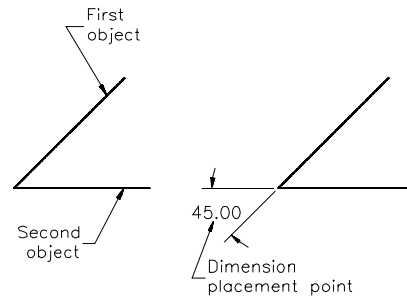
### Angular Dimensioning Using Two Line Segments

You can directly select two line segments to apply angular dimensions. Invoke the **General Dimension** tool and then select a line segment using the left mouse button. Instead of placing the dimension, select the second line segment. Now, place the dimension to measure the angle between the two lines. While placing the dimension, you need to be careful about the point where you place the dimension. This is because depending on the location of the dimension placement, the vertically opposite angles will be displayed. Figure 3-17 shows the angular dimension between two lines and Figure 3-18 shows the dimension of the vertically opposite angle between two lines.

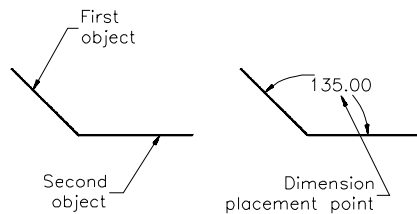
Also, depending on the location of the dimension, the major or the minor angle value will be displayed. Figure 3-19 shows the major angle dimension between two lines and Figure 3-20 shows the minor angle dimension between the same set of lines.



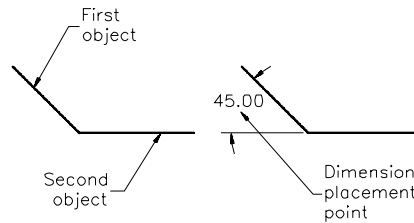
**Figure 3-17** Angular dimensioning



**Figure 3-18** Vertically opposite angle



**Figure 3-19** Major angle dimension



**Figure 3-20** Minor angle dimension

### Angular Dimensioning Using Three Points

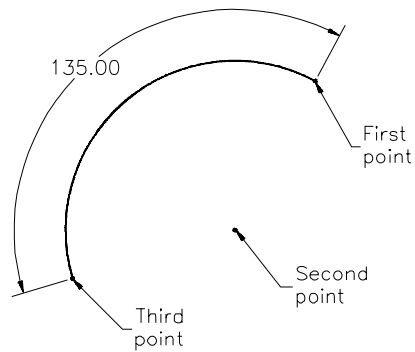
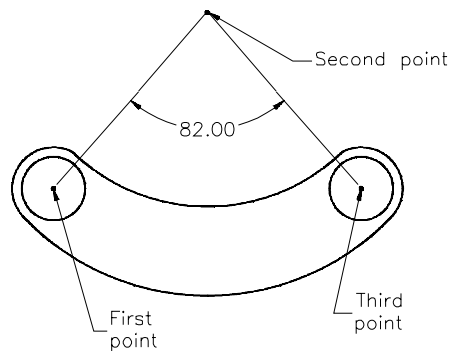
You can also apply angular dimensions using three points. Remember that the three points should be selected in the clockwise or counterclockwise sequence. The points that can be used to apply the angular dimensions include the endpoints of lines or arcs, or the center points of arcs, circles, and ellipses. Figure 3-21 shows angular dimensioning using three points.

### Angular Dimensioning of an Arc

You can use angular dimensions to dimension an arc. In case of arcs, the three points are the endpoints and the center point of the arc. Note that the points should be selected in the clockwise or counterclockwise sequence, but the center point should always be the second selection point. Figure 3-22 shows the angular dimensioning of an arc.

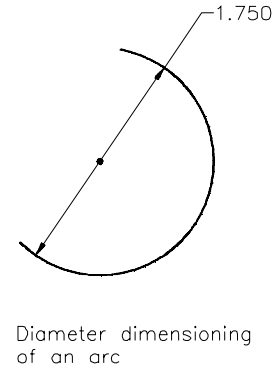
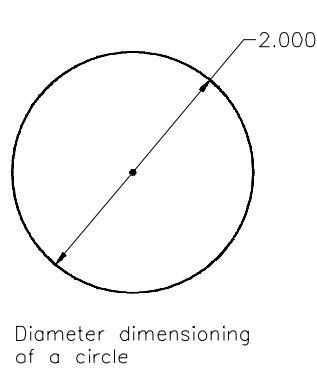
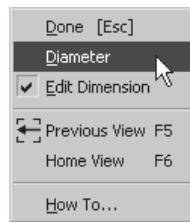
### Diameter Dimensioning

Diameter dimensions are applied to dimension a circle or an arc to specify its diameter. In Autodesk Inventor, when you select a circle to dimension, the diameter dimension is applied to it by default. However, if you select an arc to dimension, the radius dimension is applied to it. You can also apply the diameter dimension to an arc by invoking the **General Dimension** tool and selecting the arc. Now, right-click to display the shortcut menu, see Figure 3-23. Choose **Diameter** from this menu to apply the diameter dimension. Figure 3-24 shows a circle and an arc with diameter dimensions.



**Figure 3-21** Angular dimensioning using three points

**Figure 3-22** Angular dimensioning of an arc

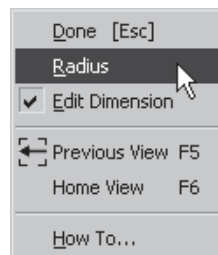


**Figure 3-23** Shortcut menu to apply a diameter dimension to an arc

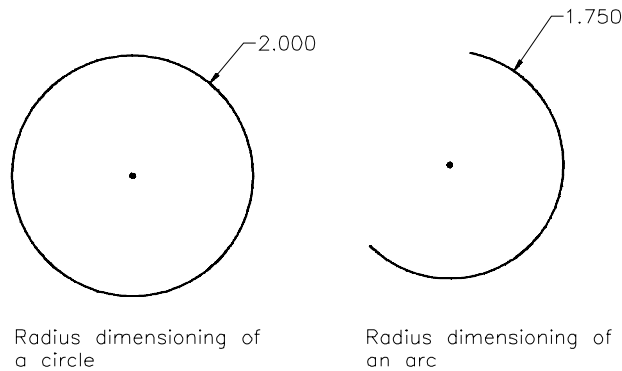
**Figure 3-24** Diameter dimensioning of a circle and an arc

## Radius Dimensioning

Radius dimensions are applied to dimension an arc or a circle to specify its radius. As mentioned earlier, by default, the circles will be assigned diameter dimensions and the arcs will be assigned radius dimensions. However, you can also apply the radius dimension to a circle. To do so, invoke the **General Dimension** tool and then select the circle. Now, right-click to display the shortcut menu, as shown in Figure 3-25. Choose **Radius** from this menu to apply the radius dimension. Figure 3-26 shows an arc and a circle with radius dimensions.



**Figure 3-25** Shortcut menu to apply a radius dimension to a circle



**Figure 3-26** Radius dimensioning of a circle and an arc



**Tip.** After invoking the **General Dimension** tool, as you move the cursor close to the sketched entities, a small symbol will be displayed close to the cursor. This symbol displays the type of dimension that will be applied. For example, if you select a line, the linear dimensioning or aligned dimensioning symbol will be displayed. If you move the cursor close to another line after selecting the first, the symbol of angular dimensioning will be displayed. These symbols help you in determining the type of dimensions that will be applied.

In Autodesk Inventor, the ellipses are dimensioned as half of the major and minor axes distances. To dimension an ellipse, invoke the **General Dimension** tool and then select the ellipse. Now, if you move the cursor in the vertical direction, the axis of the ellipse along the X axis will be dimensioned in terms of its half length. Similarly, if you move the cursor in the horizontal direction, the axis of the ellipse along the Y axis will be dimensioned equal to its half length.

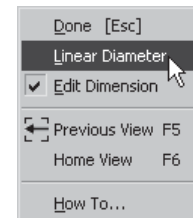
To distinguish whether the dimension applied to an arc or a circle is a radius or a diameter, try to locate the number of arrowheads in the dimension. If there are two arrowheads in the dimension and the dimension line is placed inside the circle or the arc, it is a diameter dimension. The radius dimension has one arrow head and the dimension line is placed outside the circle or the arc.

## Linear Diameter Dimensioning

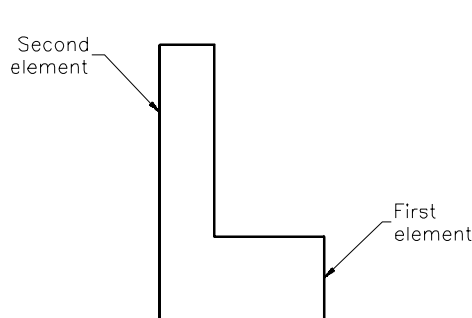
Linear diameter dimensioning is used to dimension the sketches of the revolved components. The sketch for a revolved component is drawn using simple sketcher entities. For example, if you draw a rectangle and revolve, it will result in a cylinder. Now, if you dimension the rectangle using the linear dimensions, the same dimensions will be displayed when you generate the drawing views of the cylinder. Also, the same dimensions will be used while manufacturing the component. But these linear dimensions will result in a confusing

situation in manufacturing. This is because while manufacturing a revolved component, the dimensions have to be specified as the diameter of the revolved component. The linear dimensions will not be acceptable in manufacturing a revolved component. To resolve this problem, the sketches for the revolved features are dimensioned using the linear diameter dimensions. These dimensions display the distance between the two selected line segments as a diameter, that is, double the original length. For example, if the original dimension between two entities is 10 mm, the linear diameter dimension will display it as 20 mm. This is because when you revolve a rectangle with 10 mm width, the diameter of the resultant cylinder will be 20 mm. In this type of dimension, Autodesk Inventor assumes that if you select two lines, the line selected first will act as the axis of revolution for the sketch and the line selected last will result in the outer surface of the revolved feature. This means that the line selected last will be the one that will be dimensioned. But if one of the lines is a center line drawn by choosing the **Centerline** button from the **Inventor Standard** toolbar, then the centerline will be considered as the axis of revolution.

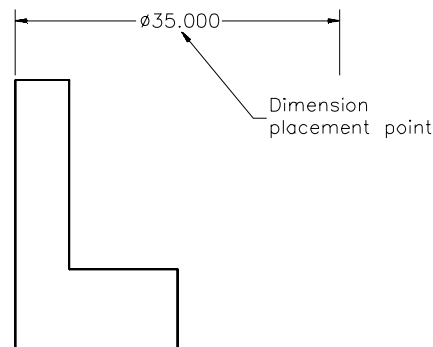
To apply linear diameter dimensions, invoke the **General Dimension** tool; you will be prompted to select the first geometry to dimension. Select the first line; you will be prompted to select the second geometry to dimension. Select the second line with reference to which you want to apply the linear diameter dimensions. If the first line you selected is a center line, the linear diameter dimension will be displayed. Else, right-click and choose **Linear Diameter** from the shortcut menu, see Figure 3-27. You will notice that the distance between the two lines is displayed as double the distance. Also, the dimension value is preceded by the  $\varnothing$  symbol, suggesting that it is a linear diameter dimension. Figures 3-28 and 3-29 show the use of linear diameter dimensioning.



**Figure 3-27** Choosing the *Linear Diameter* option



**Figure 3-28** Selecting the elements for linear diameter dimensions



**Figure 3-29** The linear diameter dimension

## CREATING DRIVEN DIMENSIONS

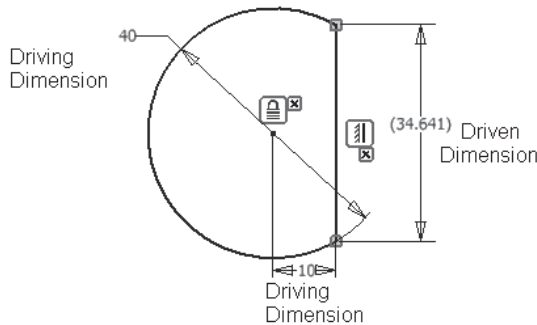
**Toolbar:** Inventor Standard > Driven Dimension



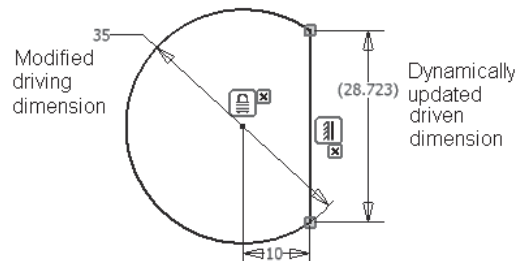
This toggle button is used to switch between the driven dimension and the sketch (driving) dimension. A dimension is called as a sketch (driving) dimension if it forces



an entity to change its length and orientation. A driven dimension is one whose value depends on the value of the sketch (driving) dimension. The driven dimensions are enclosed in a parenthesis that displays the current value of the sketched geometry and this value cannot be modified. If you change the value of the sketch (driving) dimension, the value of the driven dimension will change automatically, as shown in Figures 3-30 and 3-31. All the dimensions applied after choosing the **Driven Dimension** button will be driven dimensions. To convert the sketch (driving) dimensions into driven dimensions, select the required sketch (driving) dimension and choose the **Driven Dimension** button from **Inventor Standard** toolbar.



**Figure 3-30** Driving dimension and driven dimension in a sketch



**Figure 3-31** Modified driving dimension and updated driven dimension

## UNDERSTANDING THE CONCEPT OF FULLY CONSTRAINED SKETCHES

A fully constrained sketch is the one whose all entities are completely constrained to their surroundings using constraints and dimensions. In a fully constrained sketch, all degrees of freedom of the sketch are constrained. A fully constrained sketch cannot change its size, location, or orientation unexpectedly. Whenever you draw a sketched entity, it will be black in color. If you add dimensions and constraints to fully constrain it, the entities will turn blue. There is one more method to understand whether the sketched entities are fully constrained or not. In this method, you need to right-click in the graphic window and choose the **Show All Degrees of Freedom** option from the shortcut menu; the entities will display the available degrees of freedom such as horizontal, vertical, angular, or rotational. Note that while creating the base sketch in Autodesk Inventor, you need to dimension it with respect to a fixed point in order to fully constrain it. Therefore, you need to use some extra steps to fully constrain the sketch. These steps are given next.

1. Draw a sketch point at the origin. You can use the **Inventor Precise Input** toolbar to ensure that the point is placed exactly at the origin.
2. Apply the **Fix** constraint to the point.
3. Use this point to dimension the original sketch. You can add horizontal and vertical dimensions to the sketch from this point. You can also add a **Coincident** constraint between the point and the endpoints of one of the entities in the sketch.

## MEASURING SKETCHED ENTITIES

Autodesk Inventor allows you to measure various parameters of the sketched entities. The parameters that you can measure are distances, angles, loops, and area. Measuring these parameters is discussed next.

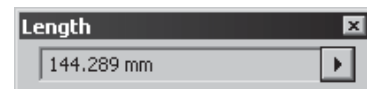
### Measuring Distances

<b>Menu Bar:</b>	Tools > Measure Distance
<b>Shortcut Menu:</b>	Measure > Measure Distance

Autodesk Inventor allows you to measure the length of a line segment, radius of an arc, diameter of a circle, minimum distance between two entities, or the coordinates of a point. All this can be done using the **Measure Distance** tool. On invoking this tool, the **Measure Distance** dialog box will be displayed and you will be prompted to select the first item. The **Measure Distance** dialog box will be modified depending on the type of entities selected to be measured. The methods of measuring the distances between various entities are discussed next.

#### Measuring the Length of a Line Segment

When you invoke the **Measure Distance** tool, the **Measure Distance** toolbar will be displayed and you will be prompted to select the first entity. If you select a line segment at this point, the **Measure Distance** dialog box will be changed to the **Length** dialog box and the length of the selected line segment will be displayed in this dialog box, see Figure 3-32.



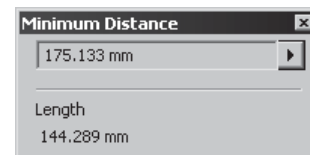
**Figure 3-32** The **Length** dialog box displaying the length of the line



**Tip.** To restart measuring the distances, right-click to display the shortcut menu and choose **Restart**; you will be prompted to select the first element to be measured.

#### Measuring Distance between a Point and a Line Segment

To measure the distance between a point and a line segment, invoke the **Measure Distance** tool and then select the point. The **Measure Distance** dialog box will show the X, Y, and Z coordinates of the point and you will be prompted to select the next entity. Select the line. The **Measure Distance** dialog box will be changed to the **Minimum Distance** dialog box. This dialog box will display the minimum distance between the point and the line and also the length of the line, see Figure 3-33.

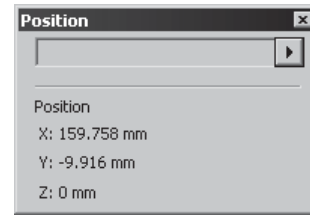


**Figure 3-33** The **Minimum Distance** dialog box displaying the distance between the lines

#### Measuring Coordinates of a Point

To measure the coordinates of a point with respect to the current coordinate system, invoke the **Measure Distance** tool; you will be prompted to select the first element. Select the point whose coordinates you want to know. The selectable points include the endpoints of lines,

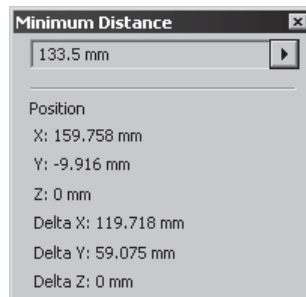
arcs, or splines, center point of arcs, circles, or ellipses, or hole centers. If you select a hole center or the center point, the **Measure Distance** toolbar is changed to the **Position** dialog box and the X, Y, and Z coordinates of the selected point with respect to the current coordinate system will be displayed, see Figure 3-34. However, if you select an endpoint, the coordinates are specified in the **Measure Distance** dialog box.



**Figure 3-34** The **Position** dialog box displaying the coordinates of a point/hole center

### Measuring Distance between Two Points

To measure the distance between two points, invoke this tool and then select the first point. The **Measure Distance** dialog box will display the coordinates of the selected point and you will be prompted to select the second element. Select the second point. The **Measure Distance** dialog box will be changed to the **Minimum Distance** dialog box. This dialog box will display the distance between the two points. This dialog box will also display the coordinates of the second point. You will also notice the **Delta X**, **Delta Y**, and **Delta Z** values in this toolbar, see Figure 3-35. These values are the distances between the two selected points along the X, Y, and Z axes. Note that if you move the cursor after selecting the second point, the **Minimum Distance** dialog box will be replaced by the **Measure Distance** dialog box.



**Figure 3-35** The **Minimum Distance** dialog box displaying the minimum distance between two points, coordinates of the second points, and the delta X, Y, and Z distances between the two points



**Tip.** You would have noticed that the Z coordinates or the Z distances are zero at all places. This is because by default, when you start a new drawing, the sketches are drawn in the XY plane. You can also draw the sketches on other planes. You will learn more about these sketching the planes in the later chapters.

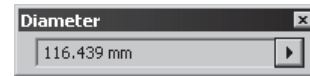
### Measuring Radius of an Arc or Diameter of a Circle

You can also measure the radius of an arc or the diameter of a circle using the **Measure Distance** tool. When you invoke this tool, the **Measure Distance** dialog box will be displayed and you will be prompted to select the first element. If you select an arc, this dialog box will be momentarily changed to the **Radius** dialog box and will display the radius of the arc, see Figure 3-36. If you select a circle, this toolbar will be momentarily changed to the **Diameter** dialog box and will display the diameter of the circle, see Figure 3-37. Note that

when you select an arc or a circle, you are not prompted to select the second element. This is because, you cannot calculate any value other than their radius or diameter.



**Figure 3-36** The **Radius** dialog box displaying the radius of an arc



**Figure 3-37** The **Diameter** dialog box displaying the diameter of a



**Tip.** Make sure that no sketching tool is selected in the **2D Sketch Panel** panel bar. If a sketching tool is active, choose the **Select** button from the **Inventor Standard** toolbar to exit that tool, or right-click and choose **Done** to exit the active sketching tool.

## Measuring Angles

<b>Menu Bar:</b>	Tools > Measure Angle
<b>Shortcut Menu:</b>	Measure > Measure Angle

To measure an angle, right-click in the drawing window and choose **Measure > Measure Angle** from the shortcut menu; the **Measure Angle** dialog box will be displayed. This tool can be used to measure the angle between two line segments or between three points. Both these methods for measuring the angles are discussed next.

### Measuring Angle between Two lines

To measure the angle between two lines, invoke the **Measure Angle** tool; the **Measure Angle** dialog box will be displayed and you will be prompted to select the first item. Select the first line; you will be prompted to select the second line. The **Measure Angle** dialog box will change to the **Angle** dialog box and the angle between the selected line segments will be displayed, see Figure 3-38.



**Figure 3-38** The **Angle** dialog box displaying the angle between two lines

### Measuring Angles Using Three Points

You can also measure the angle using three points. Note that the points must be selected in the clockwise or counterclockwise sequence. When you invoke this tool, you will be prompted to select the first item. Select the first point; you will be prompted to select the next point. After you select the second point, you will again be prompted to select the next point. Select the third point. Once you have selected the three points, Autodesk Inventor draws imaginary lines between the first and second points as well as between the second and third points. The angle between these two imaginary lines will be measured and displayed in the dialog box, as shown in Figures 3-39 and 3-40.

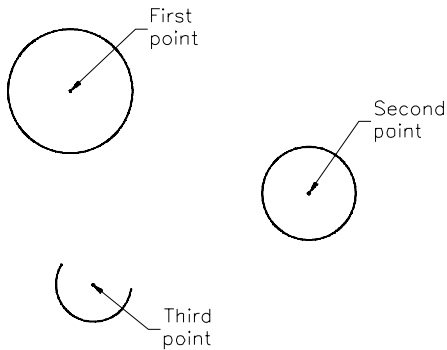


Figure 3-39 Selecting three points

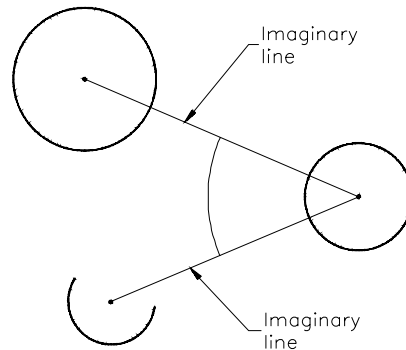


Figure 3-40 Angle between the imaginary lines



**Tip.** Autodesk Inventor allows you to switch from one measuring tool to another. This is done using the shortcut menu displayed on choosing the arrow on the right of the dialog box of any measuring tools. When you choose this arrow, the shortcut menu will be displayed with the options for invoking other measuring tools.

## Measuring Loops

**Menu Bar:** Tools > Measure Loop  
**Shortcut Menu:** Measure > Measure Loop

Autodesk Inventor allows you to measure closed loops. To measure closed loops, right-click in the drawing window and choose **Measure > Measure Loop** from the shortcut menu; the **Measure Loop** dialog box will be displayed and you will be prompted to select a face or a loop. Select the loop to be measured; the **Measure Loop** dialog box will be momentarily changed to the **Loop Length** dialog box and the measurement will be displayed in it. Figure 3-41 shows the **Measure Loop** toolbar with the measurement of a loop.



Figure 3-41 The Measure Loop dialog box

## Measuring Area

**Menu Bar:** Tools > Measure Area  
**Shortcut Menu:** Measure > Measure Area

To measure the area of closed loops, right-click in the drawing window and choose **Measure > Measure Area** from the shortcut menu. The **Measure Area** dialog box will be displayed and you will be prompted to select a face or a loop. Select the closed loop to measure the area; the **Measure Area** dialog box will change to the **Area** dialog box and the area of the loop will be displayed in it. Figure 3-42 shows the **Area** dialog box with the area of a closed loop.

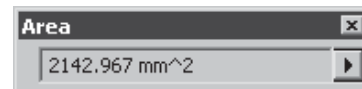


Figure 3-42 The Area dialog box



**Tip.** You can also measure the loop or the area defined by a face of an existing feature. You will learn more about features in the later chapters.

## Adding the Linear Measurements

<b>Menu Bar:</b>	Tools > Measure Distance > Add to Accumulate
<b>Shortcut Menu:</b>	Measure > Measure Distance > Add to Accumulate

You can calculate the total measurement of several linear measurements by adding their values. To do so, invoke the **Measure Distance** or **Measure Angle** tool. Next, select the geometry or the distance to be measured; the measurement will be displayed in the **Measure Distance** dialog box. Next, click the arrow on right side of the display box; a flyout will be displayed. Choose the **Add to Accumulate** option from the flyout and again choose the same arrow; the same flyout will be displayed. Choose the **Restart** option from the flyout. Next, choose another measurement and follow the same procedure till you add all the desired measurements. Then click on the arrow and choose the **Display Accumulate** option from the flyout; the sum of the measurements will be displayed in the display box.

## Clearing the Accumulated Dimensions

<b>Menu Bar:</b>	Tools > Measure Distance > Clear Accumulate
<b>Shortcut Menu:</b>	Measure > Measure Distance > Clear Accumulate

On choosing this option, you can clear all the accumulated measurements and reset the sum to zero. This option is located below the **Add to Accumulate** option.

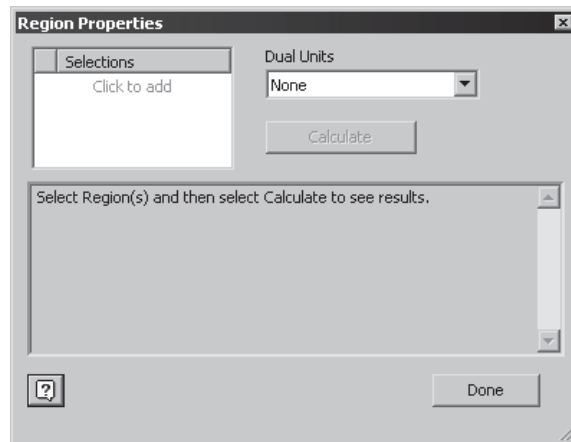
## Evaluating the Region Properties

<b>Menu Bar:</b>	Tools > Region Properties
<b>Shortcut Menu:</b>	Measure > Region Properties

This tool is used to evaluate the properties of the closed sketch loop such as area, perimeter, and display the region properties of the sketch such as Area and Moment of Inertia by taking measurements from the sketch coordinate system. To invoke this tool, right-click in the drawing window and choose **Measure > Measure Region Properties** from the shortcut menu; the **Region Properties** dialog box will be displayed, as shown in Figure 3-43. The options in this dialog box are discussed next.

## Selections

When you invoke the **Region Properties** dialog box, this option gets chosen by default and you will be prompted to select one or more closed sketch loops. Select one or more closed sketch loops from the drawing window.



*Figure 3-43 The Region Properties dialog box*

### Dual Units

You can select the required unit of measurement from this drop-down list to display the results of measurements in the selected unit. You can view the results in two different units.

### Calculate

After setting the options in the **Selections** and the **Dual Units** area, choose the **Calculate** button; the results will be displayed in the display box. In case you add or remove a closed loop in the **Selections** area or change the unit in the **Dual Units** drop-down list, the recalculation will occur and the updated results will be displayed in the display box on choosing the **Calculate** button.

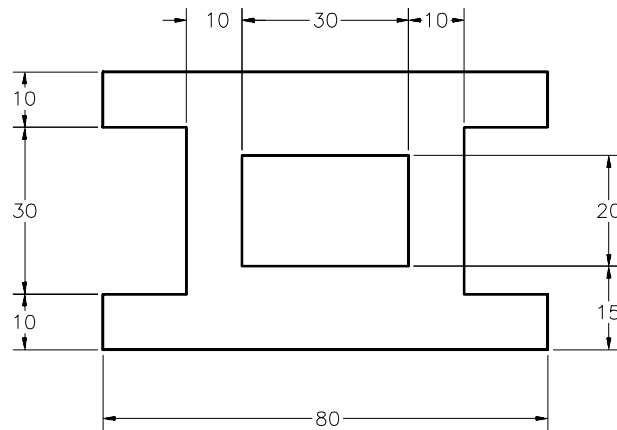
## TUTORIALS

From this chapter onward, you will use the parametric nature of Autodesk Inventor for drawing and dimensioning the sketches. The following tutorials will explain the method of drawing sketches to some arbitrary dimensions and then driving them to the dimension values required in the model.

### Tutorial 1

In this tutorial, you will draw the sketch shown in Figure 3-44. This sketch is the same as the one drawn in Tutorial 2 of Chapter 2. In this tutorial, you will not use the **Inventor Precise Input** toolbar while drawing the initial sketch. After drawing the sketch, add the required constraints and then dimension it. You will place a point at the origin and fix it at that location. Then you will dimension the sketch using this point also to fully constrain the sketch. **(Expected time: 30 min)**





**Figure 3-44** Dimensioned sketch for Tutorial 1

The following steps are required to complete this tutorial:

- Start a new metric standard part file.
- Draw the initial sketch using the **Line** and **Two point rectangle** tools, refer to Figure 3-45.
- Add the required constraints and dimensions to complete the sketch, refer to Figure 3-47.
- Place a sketch point at the origin and add the **Fix** constraint to it.
- Dimension the sketch using this point to fully constrain it, refer to Figure 3-48.
- Save the sketch with the name *Tutorial1.ipt* and then close the file.

### Starting Autodesk Inventor

- Start Autodesk Inventor by double-clicking on its shortcut icon on the desktop of your computer or by using the **Start** menu.
- Choose the **New** button in the **Inventor Standard** toolbar and start a new metric standard part file using the **Metric** tab of the **New File** dialog box.

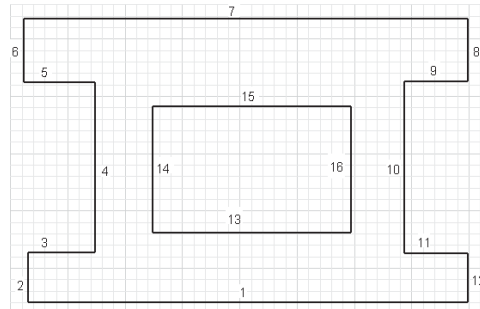
### Drawing the Initial Sketch

- Using the **Line** tool and the **Two point rectangle** tool, draw the required sketch similar to the one shown in Figure 3-44. You do not need to draw the sketch to the exact length. Use the temporary tracking option for drawing the sketch. For your reference, all the lines in the sketch are numbered, see Figure 3-45.

Note that in this sketch, the display of the X and Y axes is turned off.

### Adding Constraints to the Sketch

It is evident in Figure 3-45 that some of the lines need to be of the same length. For example, lines 1 and 7, lines 2 and 6, lines 8 and 12, and so on need to be of the same length. One option is that you assign dimensions to all these lines. But this will increase the number of dimensions in the sketch. The other option is that you apply constraints that will force the lines to maintain an equal length.



**Figure 3-45** Initial sketch drawn using the sketching tools

In this case, you can apply the **Equal** constraint to all the lines that should have the same length. The **Equal** constraint will relate the length of one of the lines with respect to the other line. Now, if you dimension any one of the related lines, all other lines that are related to it are forced to acquire the same dimension value. The **Equal** constraint is applied in pairs.

1. Choose the down arrow on the right of the **Perpendicular** button in the **2D Sketch Panel** panel bar to display the other constraints. Choose **Equal** to invoke this constraint.



When you invoke this constraint, you are prompted to select the first line, circle, or arc.



#### Note

By default, the **Perpendicular** constraint will be displayed in the **2D Sketch Panel** panel bar or toolbar. However, when you select any other constraint by choosing the down arrow on the right of the **Perpendicular** constraint, the selected constraint will become active and will be displayed in the **2D Sketch Panel** panel bar or toolbar.

2. Select line 2; the color of this line is changed to blue and you are prompted to select the second line, circle, or arc. Select line 6. The **Equal** constraint is applied to lines 2 and 6. You are again prompted to select the first line, circle, or arc. Select line 6 as the first line and then select line 8 as the second line.

If the Autodesk Inventor warning message box is displayed while applying any of these constraints, choose **Cancel** to exit that box.

3. Similarly, select lines 8 and 12, 1 and 7, 3 and 5, 5 and 9, and then lines 9 and 11. This applies the **Equal** constraint to all these pairs of lines. Right-click in the drawing window and choose **Done**.



#### Note

If while drawing the sketch, the vertical or horizontal constraint was not applied to any line, you need to apply it manually using the respective buttons.

### Dimensioning the Sketch

Once all the required constraints are applied to the sketch, you can dimension it. As mentioned earlier, when you add dimensions to the sketch and modify their value, the entity will be forced to the dimension values you have specified.

1. Choose **General Dimension** from the **2D Sketch Panel** panel bar; you are prompted to select the geometry to be dimensioned. Select line 1.



As soon as you move the cursor close to line 1, it turns red and a small symbol is displayed, suggesting that a linear dimension will be applied to this line. It is important to modify the value of the dimensions after it is placed so that the geometries are driven to the values that you require. Therefore, after selecting line 1, right-click to display the shortcut menu. In this menu, choose **Edit Dimension**. If it is already chosen, press ESC once. This will make sure that the **Edit Dimension** toolbar is displayed when you place the dimension. This toolbar allows you to modify the dimension value.

2. Place the dimension below line 1; the **Edit Dimension** toolbar is displayed. Enter **80** as the length of line 1 in this toolbar and then choose the check mark button on the right of this toolbar.

You will notice that the length of the line is modified to 80 units. Also, notice that the length of line 7 is also modified because of the **Equal** constraint (refer to Figure 3-47).

3. Because you are still in the **General Dimension** tool, you are again prompted to select the geometry to dimension. Select line 2 and place the dimension on the left of this line; the **Edit Dimension** toolbar is displayed. Change the length of this line in this toolbar to **10**.

You will notice that the length of lines 6, 8, and 12 is also forced to 10 units. This is because the **Equal** constraint is applied to all these lines.

4. Select line 4 and place it along the previous dimension. Modify the dimension value to **30** in the **Edit Dimension** toolbar. Notice that the length of line 10 is also modified.
5. Select line 16 and place the dimension outside the sketch on the right side. Modify the dimension value in the **Edit Dimension** toolbar to **20**.
6. Select line 15 and place the dimension outside the sketch on the top. Modify the dimension value to **30** in the **Edit Dimension** toolbar.
7. Now, to dimension the distance between lines 4 and 14, select them one by one. Place the dimension outside the sketch on the top and then change the dimension value to **10** in the **Edit Dimension** toolbar.
8. Similarly, select lines 16 and 10 to dimension the distance between these two lines and place the dimension outside the sketch on the top. Change the dimension value to **10** in the **Edit Dimension** toolbar. You will notice that the length of lines 5, 9, 3, and 11 is automatically adjusted due to the **Equal** constraint.

9. To locate the inner rectangle vertically from the outer loop, select lines 1 and 13 and then place the dimension on the right of the sketch. Modify the dimension value in the **Edit Dimension** toolbar to 15.

With this, you have applied all the required constraints and dimensions to the sketch. Now, this sketch is ready to be converted into a feature. If you try to add more constraints or dimensions to this sketch, Autodesk Inventor will display an error message dialog box informing you that adding this dimension or constraint will over-constrain the sketch, see Figure 3-46. If you still want this dimension to be displayed, choose the **Accept** button in this box. The dimension will be added as a **driven dimension**. A driven dimension is placed inside parentheses and is not used during the manufacturing process. These dimensions are used only for reference. Note that you cannot edit the value of a driven dimension.

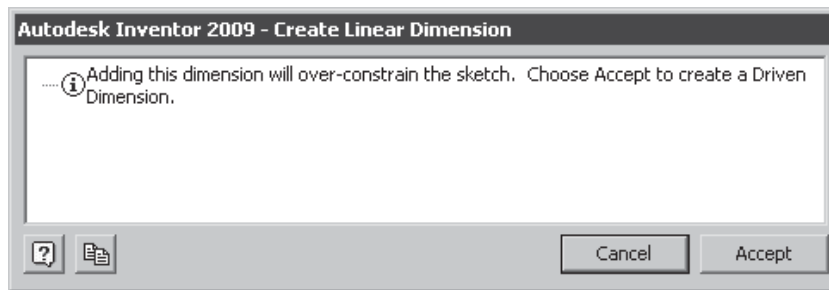


Figure 3-46 Autodesk Inventor message box

10. The sketch, after applying all dimensions and constraints, should look similar to the one shown in Figure 3-47.

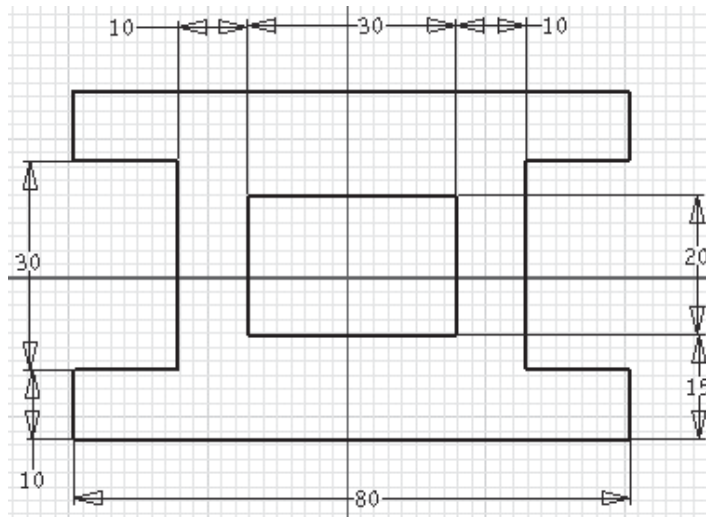


Figure 3-47 Sketch after adding dimensions

Even after adding all the dimensions, the color of the entities in the sketch is still black. This is because the sketch is not fully constrained. As mentioned in the tutorial description, you need to place a sketch point at the origin and then use it to fully constrain the sketch.

11. Using the **Inventor Precise Input** toolbar, place a sketch point at the origin.

Because you have applied the **Equal** constraint to the entities, the button of this will be displayed in the **2D Sketch Panel** panel bar instead of the default **Perpendicular** button.



12. Choose the down arrow on the right of the **Equal** button in the **2D Sketch Panel** panel bar and choose the **Fix** button; you are prompted to select a curve or a point to be fixed. 
13. Select the sketch point that you placed at the origin. This will fix the point at the origin and you can now use it to fully constrain the initial sketch.
14. Choose the down arrow on the right of the **Fix** button in the **2D Sketch Panel** panel bar and choose the **Coincident** button. You are prompted to select the first curve or point. 
15. Select the intersection point of line 1 and 2. This is the lower left vertex of the sketch. You are prompted to select the second curve or point.
16. Select the sketch point placed at the origin. The entire sketch shifts itself such that the lower left vertex of the sketch is now at the origin. But the sketch is not completely visible in the drawing window.
17. Choose the **Zoom All** button from the **Inventor Standard** toolbar to fit the sketch in the drawing window. You will notice that all the entities in the sketch turn blue in color, suggesting that the sketch is fully constrained. Press the ESC key to exit the **Coincident** constraint tool.

Figure 3-48 shows the fully constrained sketch for Tutorial 1.

### Saving the Sketch

1. Choose the **Return** button in the **Inventor Standard** toolbar to exit the sketching environment. You can also right-click in the drawing window and choose **Finish Sketch** from the shortcut menu to exit the sketching environment.
2. Choose the **Save** button from the **Inventor Standard** toolbar and save this sketch with the name given below:

*C:\Inventor\_2009\c03\Tutorial1.ipt*

3. Choose **File > Close** from the menu bar to close the file.

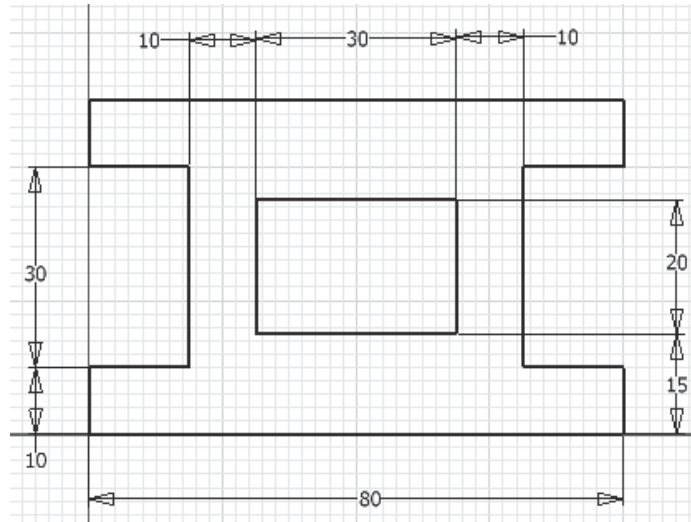


Figure 3-48 Fully constrained sketch for Tutorial 1

## Tutorial 2

In this tutorial, you will draw the sketch shown in Figure 3-49. This sketch is the same as the one that was drawn in Tutorial 4 of Chapter 2. You will not use the **Inventor Precise Input** toolbar to draw the initial sketch in this tutorial. After drawing it, apply the required constraints and dimensions to fully constrain it. **(Expected time: 30 min)**

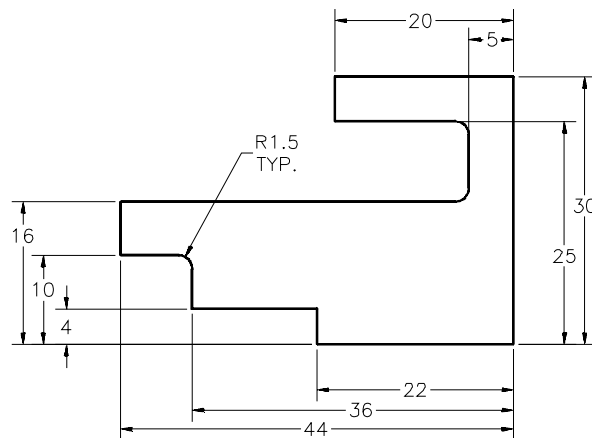


Figure 3-49 Sketch for Tutorial 2

The following steps are required to complete this tutorial:

- Start a new metric standard part file and draw the initial sketch using the **Line** tool, refer to Figure 3-50.

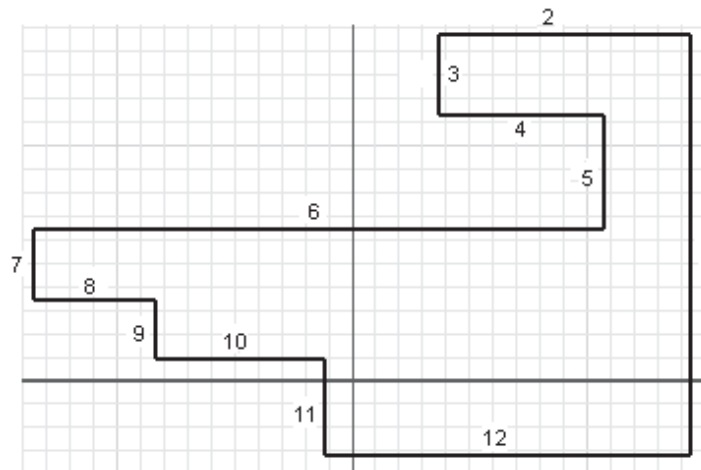
- b. Place a point at the origin and fix it using the **Fix** constraint.
- c. Add linear diameter dimensions to the sketch by using the **General Dimension** tool.
- d. Apply the **Coincident** relation between the fixed sketch point and the lower left vertex of the sketch to make it a fully constrained sketch, refer to Figure 3-51.
- e. Add fillets and then save the sketch with the name *Tutorial2.ipt* and close the file.

### Starting a New File

1. Choose the **New** button in the **Inventor Standard** toolbar and start a new metric standard part file using the **Metric** tab of the **New File** dialog box.

### Drawing the Initial Sketch

1. Draw the initial sketch, as shown in Figure 3-50, using the **Line** tool. The lines in the sketch are numbered for your reference.



*Figure 3-50* Numbering the lines in the sketch

2. Place a sketched point at the origin using the **Inventor Precise Input** toolbar.

### Dimensioning and Constraining the Sketch

The dimensions shown in Figure 3-49 are linear dimensions. But because the sketch is for a revolved feature, you need to add linear diameter dimensions to the sketch. It is recommended to first apply all the dimensions and then add the fillets. This is because the sketch generally changes its size after dimensioning. Before proceeding with adding dimensions to a revolved section, it is important for you to determine which line segment of the sketch will act as the revolution axis for revolving the sketch. If you refer to Figures 2-46 and 2-47 in Chapter 2, you will notice that for this model, line 12 will act as the axis for revolving the sketch. Therefore, while applying linear diameter dimensions, line 12 should be selected first.

1. Choose **General Dimension** from the **2D Sketch Panel** panel bar; you are prompted to select the geometry to be dimensioned. Right-click to display





the shortcut menu and choose **Edit Dimension**, if it is not already chosen. If it is already chosen, press the ESC key once to exit the shortcut menu.

2. Select line 12. You are again prompted to select the geometry to be dimensioned. Select line 10 and then right-click to display the shortcut menu. In this menu, choose **Linear Diameter**.

You will notice that the dimension is displayed as the double of the actual length. Also, the dimension value is preceded by the Ø symbol, suggesting that it is a linear diameter dimension.

3. Place the dimension on the left of the sketch. The **Edit Dimension** toolbar is displayed.
4. Figure 3-49 shows this value as 4. Because the linear diameter dimensions are placed as the double of the original length, enter **8** in the **Edit Dimension** toolbar. The vertical distance between lines 12 and 10 will be automatically adjusted to match the value entered.
5. As the **General Dimension** tool is still active, you are again prompted to select the geometry to be dimensioned. Select line 12 and then select line 8. Now, right-click to display the shortcut menu and choose **Linear Diameter**. The linear dimension is changed to the linear diameter dimension. Place the dimension on the left of the previous dimension. Modify its value in the **Edit Dimension** toolbar to **20**.
6. Select lines 12 and 6. Right-click to display the shortcut menu and then choose **Linear Diameter**. Place the dimension on the left of the previous dimension and change the value in the **Edit Dimension** toolbar to **32**.



**Tip.** Autodesk Inventor allows you to invoke the drawing display options even when a sketching environment tool is active. This is done using a combination of the hot keys and the left mouse button. For example, if the **General Dimension** tool is active, you can use the **Pan** option by holding the F2 key and then pressing the left mouse button and dragging. Similarly, you can dynamically zoom in and out of the sketch by holding the F3 key and then pressing the left mouse button and dragging.

7. Select lines 12 and 4 and then right-click to display the shortcut menu. In this menu, choose **Linear Diameter**. Place the dimension on the right of the sketch and change the value in the **Edit Dimension** toolbar to **50**.
8. Select lines 12 and 2 and then right-click to display the shortcut menu. In this menu, choose **Linear Diameter**. Place the dimension on the right of the previous dimension and change its value in the **Edit Dimension** toolbar to **60**.

With this, you have applied all the required linear diameter dimensions. Now, you need to add the linear dimensions.

9. Select lines 1 and 5 and then place the dimension above the sketch. Modify its value in the **Edit Dimension** toolbar to **5**.



**Tip.** Sometimes while dimensioning the sketch, some existing dimensions move from the location where you placed them. In this case, you need to exit the **General Dimension** tool and then drag the existing dimensions back to their original location. To continue dimensioning, invoke the **General Dimension** tool again.

10. Select line 2 and then place the dimension above the previous dimension. Modify its value in the **Edit Dimension** toolbar to **20**.
11. Select line 12 and then place the dimension below the sketch. Modify its value in the **Edit Dimension** toolbar to **22**.
12. Select lines 1 and 9 and place the dimension below the previous dimension. Modify its value in the **Edit Dimension** toolbar to **36**.
13. Select lines 1 and 7 and place the dimension below the previous dimension. Modify its value in the **Edit Dimension** toolbar to **44**.

With this, all the dimensions are added to the sketch. However, the entities in the sketch are still displayed in black, suggesting that the sketch is not fully constrained. Therefore, you need to add more dimensions or constraints. In this sketch, you will first add the **Fix** constraint to the sketched point placed at the origin and then add the **Coincident** constraint to the sketched point and the intersection point of line 11 and 12.

14. Invoke the **Fix** constraint using the **2D Sketch Panel** panel bar and select the sketch point placed at the origin to fix it at the origin.
15. Invoke the **Coincident** constraint; you are prompted to select the first curve or point. Select the intersection point of line 11 and 12. This is the lower left vertex of the sketch; you are prompted to select the second curve or point.
16. Select the sketched point placed at the origin.



The entire sketch shifts from its original location and is relocated such that the lower left vertex of the sketch now lies at the origin.

17. Choose the **Zoom All** button from the **Inventor Standard** toolbar to fit the sketch in the drawing window. All entities in the sketch are displayed in blue, suggesting that the sketch is fully constrained. The fully constrained sketch, after adding all the dimensions, is shown in Figure 3-51.

### Adding Fillets to the Sketch

After dimensioning the sketch, you need to add fillets.

1. Choose the **Fillet** button from the **2D Sketch Panel** panel bar; the **2D Fillet** dialog box is displayed. Set the value of fillet in this toolbar to **1.5**. Now, select lines 8 and 9; the fillet is automatically added between these two lines and the fillet dimension is displayed.

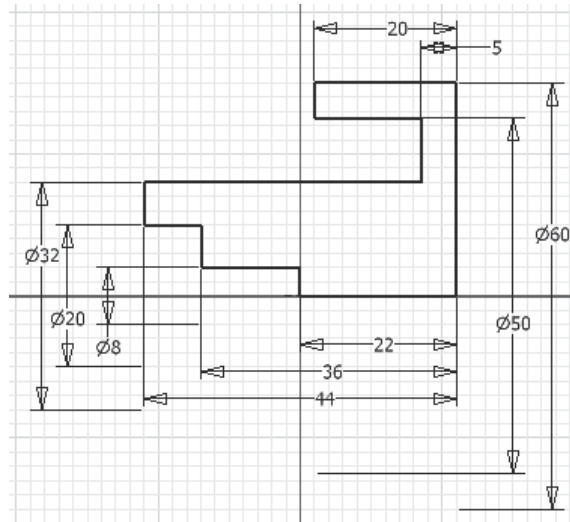


Figure 3-51 Fully constrained sketch for Tutorial 2

2. Similarly, select lines 5 and 6, and lines 4 and 5 to add fillets between these lines. Exit the **2D Fillet** toolbar. The final sketch, after adding the fillets, is shown in Figure 3-52.

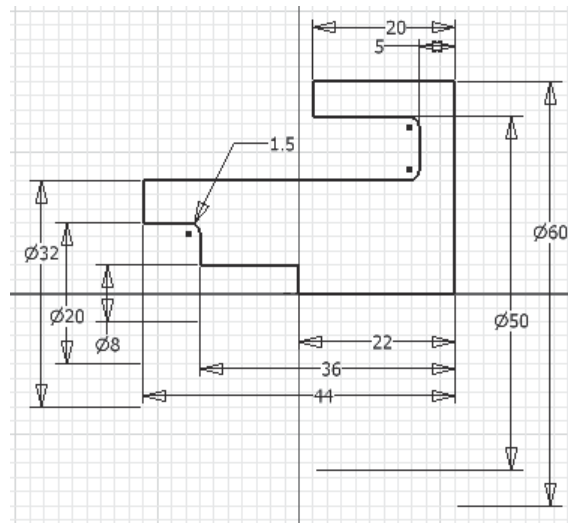


Figure 3-52 Fully dimensioned sketch after adding fillets



#### Note

To modify the fillet radius, double-click on it; the **Edit Dimension** toolbar is displayed. Modify the value of the fillet in this edit box.

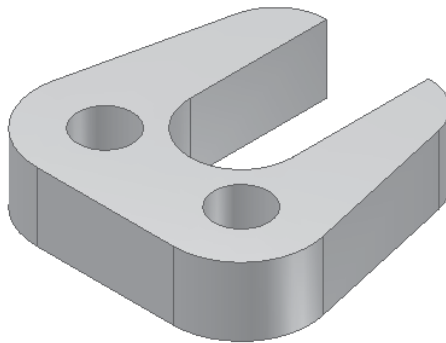
### Saving the Sketch

1. Choose **Return** from the **Inventor Standard** toolbar to exit the sketching environment.

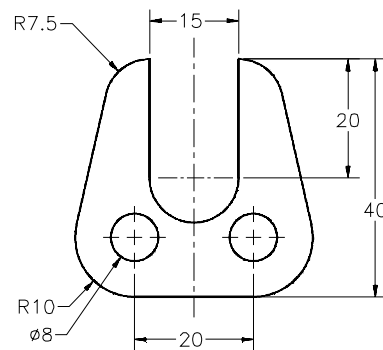
2. Save this sketch with the name *C:\Inventor\_2009\c03\Tutorial2.ipt*.
3. Choose **File > Close** from the menu bar to close the file.

### Tutorial 3

In this tutorial, draw the sketch for the model shown in Figure 3-53. After drawing the sketch, add the required constraints and then dimension it. The basic dimensioned sketch that is required for the model is shown in Figure 3-54. The solid model shown in Figure 3-53 is only for your reference. **(Expected time: 30 min)**



**Figure 3-53** Model for Tutorial 3



**Figure 3-54** Dimensioned sketch for the model

The sketch shown in Figure 3-54 is a combination of multiple closed loops: the outer loop and the inner circles. As the number of loops increases, the complexity of the sketch also increases. This is because the number of constraints and dimensions in the sketch increase in case of multiple loops. Now, to draw sketches without using the **Inventor Precise Input** toolbar, it is recommended that you first draw the outer loop and then add constraints and dimensions to it. This is because once the outer loop is constrained and dimensioned, the inner circles can be easily constrained and dimensioned with reference to the outer loop.

The following steps are required to complete this tutorial:

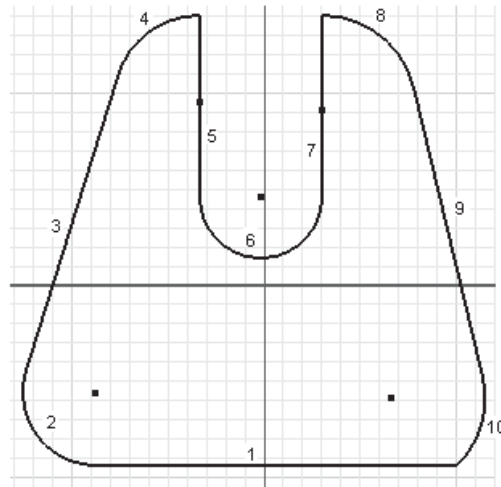
- a. Start a new metric template and draw the outer loop of the sketch, refer to Figure 3-55.
- b. Add the required dimensions and constraints to the outer loop, refer to Figure 3-57.
- c. Draw the inner circles and add constraints and dimensions to them, refer to Figure 3-58.
- d. Finally, save the sketch with the name *Tutorial3.ipt* and close the file.

### Starting a New File

1. Choose the **New** button in the **Inventor Standard** toolbar to invoke the **New File** dialog box. Choose the **Metric** tab and start a metric standard part file.

### Drawing the Outer Loop

1. Using the **Line** tool, draw the outer loop of the sketch, as shown in Figure 3-55.




*Figure 3-55 Initial sketch with numbered geometries*

You can use the option of drawing the tangent arcs using the **Line** tool for drawing this sketch. This can be done by pressing the left mouse button and then dragging in the required direction (refer to Tutorial 3 of Chapter 2 to learn more about drawing this type of arc.)


For your reference, all the geometries in the sketch are numbered. The inner holes in the sketch will be drawn after dimensioning the outer loop.

### Adding Constraints to the Sketched Entities

As evident in Figure 3-55, some of the constraints such as tangent and equal are missing in the sketch. You need to manually add these constraints to the sketch. You can view the constraints applied on the various geometries using the **Show Constraints** tool.

1. In Figure 3-55, the tangent constraint is missing between line 1 and arc 10. To add this constraint, choose the down arrow on the right of the **Coincident** button (or the button of the constraint that was used last) in the **2D Sketch Panel** panel bar and choose **Tangent**. You are prompted to select the first curve. Select arc 10 and then select line 1 as the second curve. Similarly, add this constraint at all the places where it is missing in the sketch. 

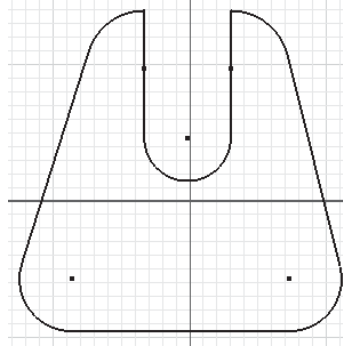
The geometries 5 and 7, and 3 and 9 are lines and must be of equal length. Also, geometries 2 and 10, and 4 and 8 are arcs that must be of equal radii. Therefore, you need to add the **Equal** constraint between the respective pairs of all these geometries.

2. Choose the down arrow on the right of the **Tangent** button in the **2D Sketch Panel** panel bar and choose **Equal**. 
3. Select line 5 as the first line and then select line 7 as the second line to apply the **Equal** constraint to them. Autodesk Inventor again prompts you to select the first entity. Select line 3 and then select line 9 to apply the **Equal** constraint to these lines.

4. You are prompted to select the first entity again. Select arc 2 and then arc 10 to apply the **Equal** constraint to these arcs.

Applying this constraint to the arcs or circles forces their radii or diameters to be equal.

5. Similarly, apply the **Equal** constraint on arcs 4 and 8. The sketch, after applying all the constraints, is shown in Figure 3-56.



**Figure 3-56** The sketch after applying all the constraints

6. Apply the **Coincident** constraint between the center point of arc 4 and line 5, and the center point of arc 8 and line 7, if it is not applied automatically.



#### Note

The shape of the sketch that you have may be a little different at this stage. This is because of the difference in specifying the points while drawing the sketch. However, once all the dimensions are applied, the shape of the sketch will become similar.

### Dimensioning the Sketch

1. Choose the **General Dimension** button from the **2D Sketch Panel** panel bar. Right-click to display the shortcut menu. From this menu, choose **Edit Dimension** if the check mark is not available at the left-side of the **Edit Dimension** option. If it shows the check mark, press the ESC key once to exit the shortcut menu; you are prompted to select the geometry to be dimensioned. Select line 1 and place the dimension below the sketch. Modify the value of this dimension in the **Edit Dimension** toolbar to 20.
2. Select arc 4 and place the dimension on the left of the sketch. This will place the radius dimension for the sketch. Modify the dimension value in the **Edit Dimension** toolbar to 7.5. The size of arc 8 will also be modified because of the **Equal** constraint applied between these two entities.



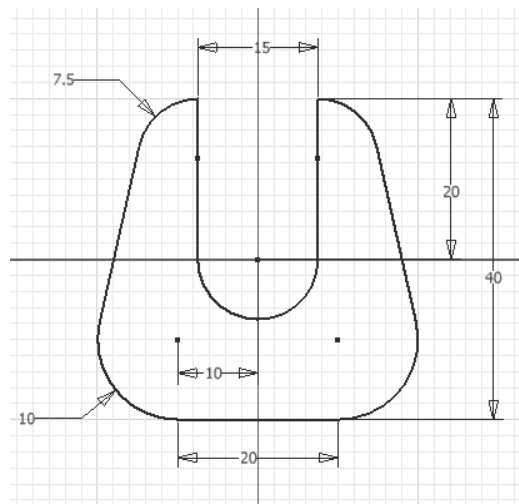
#### Note

You may need to use a combination of the hot keys as discussed in the previous tutorial, to zoom or pan the model.

3. Select arc 2 and place the radius dimension on the left of the sketch. Modify the dimension value in the **Edit Dimension** toolbar to **10**. The size of the arc 10 will also be modified due to the **Equal** constraint applied between these two entities.
4. Select line 5 and then line 7 and place the dimension above the sketch. Modify the value of this dimension in the **Edit Dimension** toolbar to **15**.
5. Select line 7 and place the dimension on the right of the sketch. Modify the value of this dimension in the **Edit Dimension** toolbar to **20**.
6. Select the upper endpoint of line 7 and then select line 1 and place the dimension on the right of the previous dimension. Modify the value of this dimension toolbar to **40**.

With this, all the dimensions are applied, except the horizontal dimension between the center points of arcs 4 and 6 or arcs 7 and 6. Their need depends on the constraints and dimensions assumed while drawing the sketch. If you are warned that adding any of these dimensions will over-constrain the sketch, choose **Cancel** in the message box. In this case, this dimension is already assumed.

7. Select the center point of the arc 2 and right-click to display a shortcut menu. Choose the **Display Degrees of Freedom** option from the shortcut menu; a horizontal line with arrows at both ends is displayed indicating that the arc can move horizontally.
8. Select the center point of the arc 2 and then the center point of the arc 6. As you have selected the two points that are not horizontally or vertically aligned, you can apply horizontal, vertical, or aligned dimensions to these points. However, in this sketch, you need only horizontal dimension. Therefore, use the shortcut menu and place the dimension on lower side of the sketch. Modify the value of this dimension toolbar to 10. The sketch after adding the required dimensions is shown in Figure 3-57.



**Figure 3-57** Dimensioned sketch for Tutorial 3



## Drawing Circles

Once all the required dimensions and constraints are applied, you will have to draw the circles. Figure 3-54 suggests that the circles are concentric with arcs 2 and 10.

1. To draw the concentric circles, choose **Center Point Circle** from the **2D Sketch Panel** panel bar; you are prompted to select the center of the circle. Move the cursor close to the center of arc 2. Specify the center point when the cursor snaps to the center point of arc 2 and turns green. Now, move the cursor away from the center and specify a point to size the circle.
2. Similarly, taking the reference of the center of arc 10, draw the other circle.

## Adding Constraints to the Circles

Because both circles have the same diameter, you can apply the **Equal** constraint to them. This way you have to apply a dimension to just one of them. The other circle will be automatically forced to the specified diameter value because of the **Equal** constraint.

1. Invoke the **Equal** constraint from the **2D Sketch Panel** panel bar. Select the first circle and then select the second circle to apply the **Equal** constraint.

## Dimensioning the Circle

1. Choose **General Dimension** from the **2D Sketch Panel** panel bar and select the left circle. Place the dimension on the left of the sketch. Change the value of the diameter of the circle in the **Edit Dimension** toolbar to 8.



Notice that the size of the right circle is automatically modified to match the dimension of the left circle. This is because of the **Equal** constraint applied between the two circles. The final sketch for Tutorial 3, after drawing and dimensioning the circles, is shown in Figure 3-58.

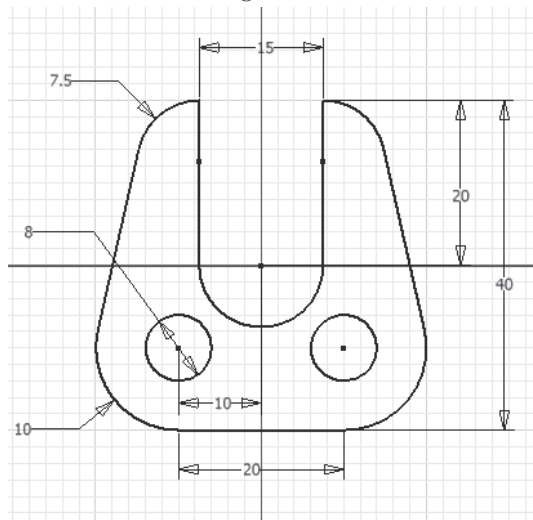


Figure 3-58 The final dimensioned sketch for Tutorial 3

### Saving the Sketch

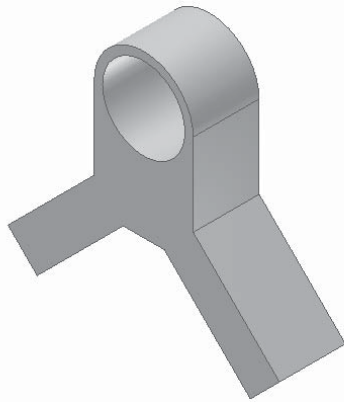
1. Choose the **Return** button in the **Inventor Standard** toolbar to exit the sketching environment. Save this sketch with the name given below:

*C:\Inventor\_2009\c03\Tutorial3.ipt*

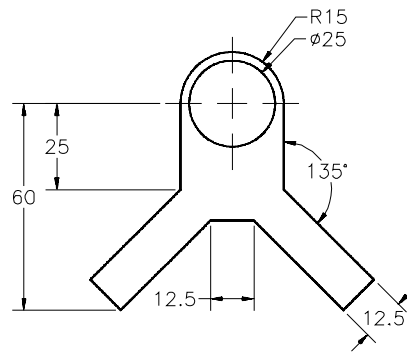
2. Choose **File > Close** from the menu bar to close the file.

## Tutorial 4

In this tutorial, you will draw the sketch for the model shown in Figure 3-59. The dimensioned sketch is shown in Figure 3-60. After drawing, add constraints and then dimension it. The solid model is given for reference only. **(Expected time: 30 min)**



**Figure 3-59** Model for the sketch of Tutorial 4



**Figure 3-60** Dimensions of the sketch

The following steps are required to complete this tutorial:

- a. Start a new metric standard part file and draw the outer loop of the sketch.
- b. Add the required dimensions and constraints to the sketch.
- c. Add the inner circle and add the dimension to it.
- d. Finally, save the sketch with the name *Tutorial4.ipt* and close the file.

### Starting a New File

1. Choose the **New** button from the **Inventor Standard** toolbar to display the **New File** dialog box. Start a new metric standard part file from the **Metric** tab of this dialog box.

### Drawing the Outer Loop

1. Choose **Line** from the **2D Sketch Panel** panel bar and draw the outer loop, as shown in Figure 3-61. As mentioned earlier, you should draw the inner loop after drawing and dimensioning the outer loop. This is because once the outer loop is dimensioned, you can draw the inner loop by taking the reference of the outer loop.

You can use the option of drawing the arc from within the **Line** tool to draw the arc in the

sketch. You can also use the temporary tracking option for drawing this sketch. For your reference, the geometries in the sketch are numbered, see Figure 3-61.

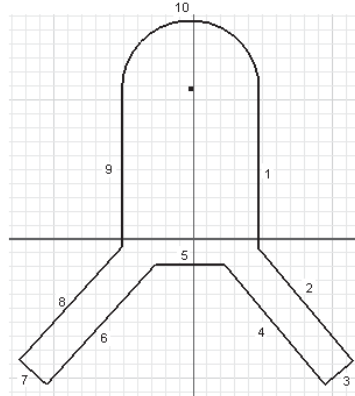


Figure 3-61 Initial sketch with the geometries numbered

### Adding Constraints to the Outer Loop

1. Add the **Equal** constraint to lines 1 and 9, lines 2 and 8, lines 3 and 5, lines 5 and 7, and lines 4 and 6.
2. Add the **Perpendicular** constraint to lines 2 and 3, and lines 7 and 8.
3. Add the **Horizontal** constraint between the lower endpoints of lines 4 and 6.



### Dimensioning the Outer Loop

1. Choose **General Dimension** from the **2D Sketch Panel** panel bar; you are prompted to select the geometry to dimension. Select line 9 and place the dimension on the left of the sketch. Modify the dimension value in the **Edit Dimension** toolbar to **25**.
2. Select the center of the arc and then select the lower endpoint of line 6. Place the dimension on the left of the previous dimension. Modify the dimension value in the **Edit Dimension** toolbar to **60**.
3. Select line 3 and then right-click to display the shortcut menu. Choose **Aligned** from this shortcut menu and then place the dimension below the sketch. Modify its value in the **Edit Dimension** toolbar to **12.5**.



Notice that the length of lines 5 and 7 is also modified due to the **Equal** constraint.

4. Select lines 1 and 2 and then place the angular dimension on the right of the sketch. Modify the value of the angular dimension in the **Edit Dimension** toolbar to **135**.
5. Select arc 10 and then place the radius dimension above the sketch. Modify the value of the radius of the arc in the **Edit Dimension** toolbar to **15**.

With this, all the required dimensions are applied to the outer loop.

### Drawing the Circle

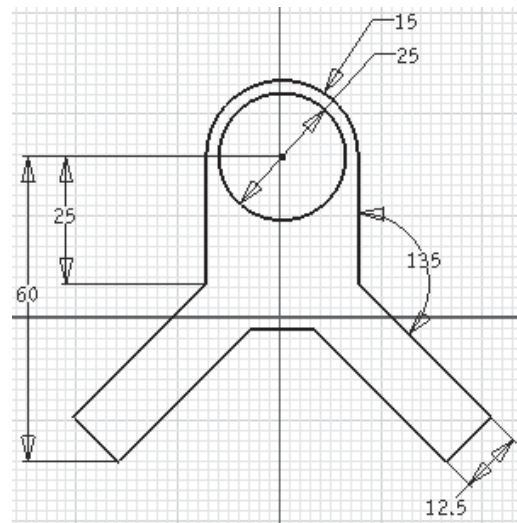
1. Choose **Center Point Circle** from the **2D Sketch Panel** panel bar; you are prompted to select the center of the circle.
2. Move the cursor close to the center of the arc; the cursor snaps to the center point and turns green. Select this point as the center of the circle and then move the cursor away from the center to size the circle. Specify a point to give it an approximate size.

### Dimensioning the Circle

1. Choose **General Dimension** from the **2D Sketch Panel** panel bar and select the circle. Place the diameter dimension below the arc dimension. Enter the diameter of the circle as **25** in the **Edit Dimension** toolbar.



This completes the sketch for Tutorial 4. The final dimensioned sketch is shown in Figure 3-62.



**Figure 3-62** Final dimensioned sketch for Tutorial 4

### Saving the Sketch

1. Choose the **Return** button in the **Inventor Standard** toolbar to exit the sketching environment and then save this sketch with the name given below:

*C:\Inventor\_2009\c03\Tutorial4.ipt*

2. Choose **File > Close** 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 the chapter:

1. The **Perpendicular** constraint forces the selected entity to become normal to another specified entity. (T/F)
2. The **Coincident** constraint can be applied to two line segments. (T/F)
3. The **Collinear** constraint can be applied only to line segments. (T/F)
4. If you apply a constraint that is not required in the sketch, Autodesk Inventor will display a message box informing you that adding this constraint will over-constrain the sketch. (T/F)
5. The \_\_\_\_\_ nature of Autodesk Inventor ensures that irrespective of the original size, the selected entity is driven to the dimension value you specify.
6. When you select a circle to be dimensioned, the \_\_\_\_\_ dimension is applied to it by default.
7. The \_\_\_\_\_ dimension has one arrow head and is placed outside the circle or the arc.
8. The \_\_\_\_\_ dimension displays the distance between two selected line segments in terms of diameter; that is, double the original length.
9. The \_\_\_\_\_ tool, can be used measure the radius of an arc.
10. To measure the angle between three points, they must be selected either in the \_\_\_\_\_ sequence or in the \_\_\_\_\_ sequence.

### Review Questions

Answer the following questions:

1. You cannot apply the **Concentric** constraint between a point and a circle. (T/F)
2. You can use the **Horizontal** or **Vertical** constraint to line up arcs, circles, or ellipses in the same horizontal or vertical direction. (T/F)
3. You can view all or some of the constraints applied to the sketch. (T/F)
4. There are twelve types of geometric constraints that can be applied additionally to the sketched entities. (T/F)
5. Linear dimensions are defined as the dimensions that define the shortest distance between two points. (T/F)

6. The situation, where the number of dimensions or constraints exceeds the number that are required in the sketch, is called
- (a) Constraint (b) Under-constrained  
(c) Over-constrained (d) None
7. When you invoke the **Measure Distance** tool and select two lines, the **Measure Distance** toolbar changes into which of the following toolbars?
- (a) **Length** (b) **Distance**  
(c) **Minimum Distance** (d) Remains the same
8. Whenever you select an arc to be dimensioned, by default, which of the following types of dimensions is applied to it?
- (a) **Radius** (b) **Diameter**  
(c) **Linear** (d) **Linear Diameter**
9. In addition to the lines, which of the following entities can be selected to apply the **Collinear** constraint?
- (a) Arc (b) Circle  
(c) Ellipse (d) Ellipse axis
10. Which of the following combination of entities cannot be used to apply the **Tangent** constraint?
- (a) Line, line (b) Line, arc  
(c) Circle, circle (d) Arc, circle

## Exercises

### Exercise 1

Draw the basic sketch for the model shown in Figure 3-63. The sketch is shown in Figure 3-64. After drawing the sketch, add the required constraints and then dimension it.

(Expected time: 30 min)

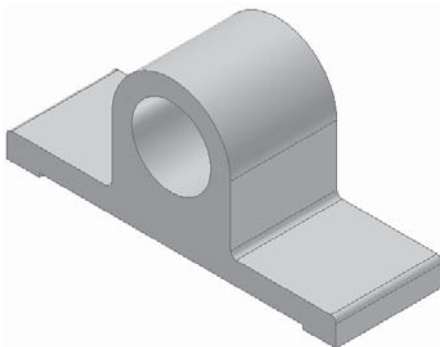


Figure 3-63 Model for Exercise 2

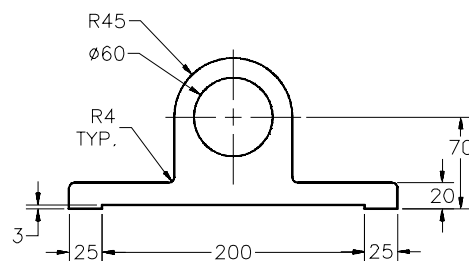


Figure 3-64 Sketch for Exercise 2

## Exercise 2

Draw the basic sketch for the model shown in Figure 3-65. The sketch is shown in Figure 3-66. After drawing the sketch, add the required constraints and then dimension it.

(Expected time: 30 min)

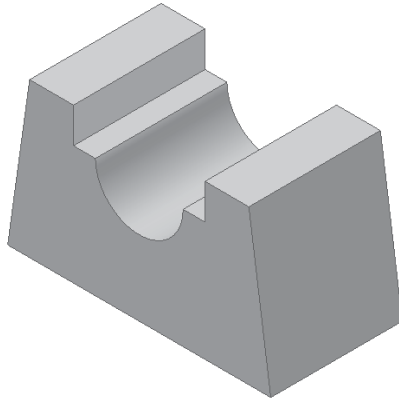


Figure 3-65 Model for Exercise 3

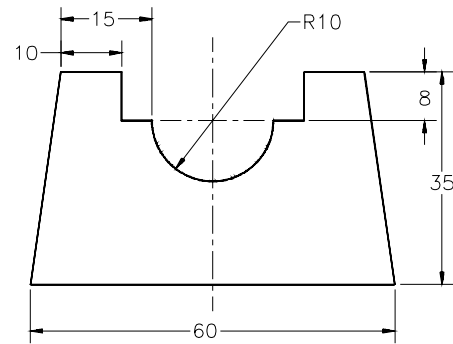


Figure 3-66 Sketch for Exercise 3

## Exercise 3

Redraw the sketch for Exercise 1 of Chapter 2 without using the **Inventor Precise Input** toolbar. After drawing the sketch, add the required constraints to it and then dimension it. The dimensioned sketch is shown in Figure 3-67.

(Expected time: 30 min)

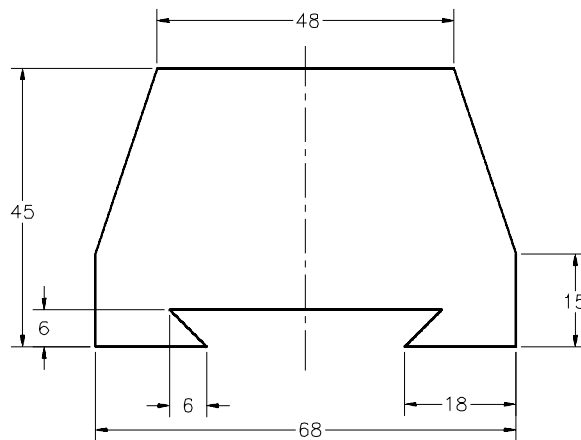


Figure 3-67 Dimensioned sketch for Exercise 4

### Exercise 4

Redraw the sketch for Exercise 2 of Chapter 2 without using the **Inventor Precise Input** toolbar. After drawing the sketch, add the required constraints to it and then dimension it. The dimensioned sketch is shown in Figure 3-68. **(Expected time: 30 min)**

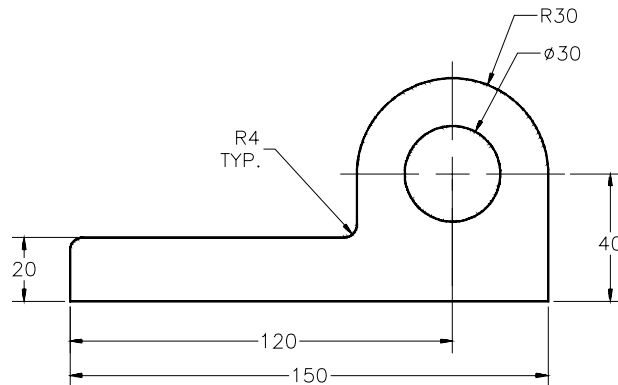


Figure 3-68 Dimensioned sketch for Exercise 5

### Exercise 5

Draw the basic sketch for the model shown in Figure 3-69. The sketch to be drawn is shown in Figure 3-70. After drawing the sketch, add the required constraints and then dimension it. **(Expected time: 30 min)**

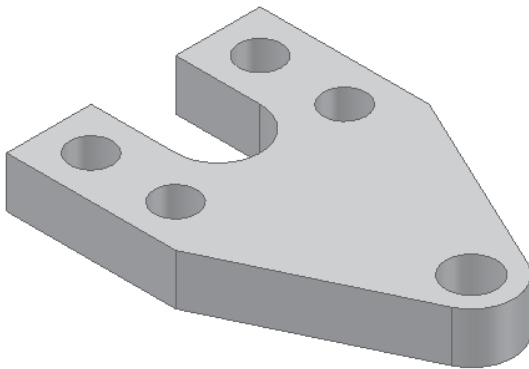


Figure 3-69 Model for Exercise 1

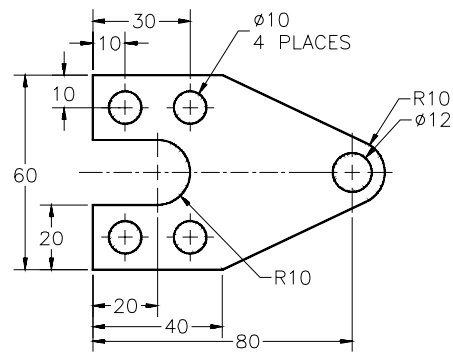


Figure 3-70 Sketch for Exercise 1

### Answers to Self-Evaluation Test

1. T, 2. F, 3. F, 4. T, 5. parametric, 6. diameter, 7. radius, 8. linear diameter, 9. Measure Distance, 10. clockwise, counterclockwise